QUAVERATO

Assembly Instructions

HARMONIC TREMOLO PEDAL

ZEPPELIN DESIGN LABS

ZEPPELINDESIGNLABS.COM • 2950 N. WESTERN, CHICAGO, IL 60618

QUAVERATO
HARMONIC TREMOLO PEDAL

Assembly Instructions

This manual is for circuit board version 8.2 or above (serial number ZD3672 and above). If your circuit board version is under 8.2 (serial number ZD3644 and below) please use the previously released assembly manual located on the Quaverato page of our website: zeppelindesignlabs.com

INTRODUCTION

The Quaverato is a versatile tap-time tremolo pedal, giving you a wide range of control over the tremolo speed, depth, wave shape, and duty cycle (or spacing). The Quaverato can also operate as a harmonic tremolo, meaning it can apply tremolo independently to the high and low frequencies. The HARMONIC MIX knob allows you to blend the effect between these two frequency ranges. Further, internal controls allow you to change the cutoff frequencies, so you get to define what is high and what is low. The Quaverato has an entirely analog signal path including a true-bypass switching scheme, while still benefiting from the extended functionality and versatility of digital control.

HOW TREMOLO WORKS

Tremolo is an effect in which the amplitude (or volume) of an audio signal is turned up and down (or modulated) at a relatively slow rate. You can manually create tremolo by turning the volume knob of your guitar or amp up and down at a constant rate, but tremolo is usually created electrically with a low frequency oscillator (LFO) circuit. An LFO is a circuit that creates a relatively slow periodic waveform, usually slower than 20 oscillations per second (20Hz). In a tremolo effect, the LFO essentially controls a volume-changing component within the audio signal path.

SOME HISTORICAL CONTEXT

In the early 1940's the DeArmond company of Toledo, Ohio started manufacturing the very first stand-alone effect unit: an electro-mechanical tremolo. How this effect worked was very simple, but quite clever. A variable-speed motor caused a grounded copper can to spin around. Inside this can was a conductive liquid and an electrode connected to the audio input. As the can spun, the grounded liquid would slosh around and come into contact with the electrode, causing the audio signal to be shunted to ground. The audio signal would fade in and out at the rate of the sloshing liquid – ingenious!

DeArmond originally designed their electro-mechanical tremolo box for use with electric pianos, but by the 1950's many guitar amplifier manufacturers were implementing fully electrical tremolo effects in their amps. The Fender Company was one of the most notable amp makers to incorporate tremolo. Ironically enough, Fender mislabeled the effect as "vibrato" on their amps. Vibrato is the effect which uses an LFO to modulate the pitch (or frequency) of an audio signal, whereas tremolo uses the LFO to modulate the volume (or amplitude). They can sound very similar, but they are, in fact, two different effects. Some people speculate that Fender labeled the amps "vibrato" to distinguish them from the pitch-changing whammy bar on their Stratocaster guitar which Fender was marketing as "tremolo" – very confusing. It's no wonder people still get confused by the meaning of "tremolo" and "vibrato."

Throughout the 1960's Fender used several types of electrical tremolo circuits in their various amp models, including *bias-modulating tremolo*, *optical tremolo*, and *harmonic tremolo*.

Bias tremolo is created by modulating the bias voltage on a preamp tube, phase inverter tube, or power tube in an amplifier circuit. This has the effect of partially cutting off the current going through the tube which reduces the tubes' capacity to amplify the signal. Bias-modulated tremolo is associated with a smooth, sine wave oscillation.

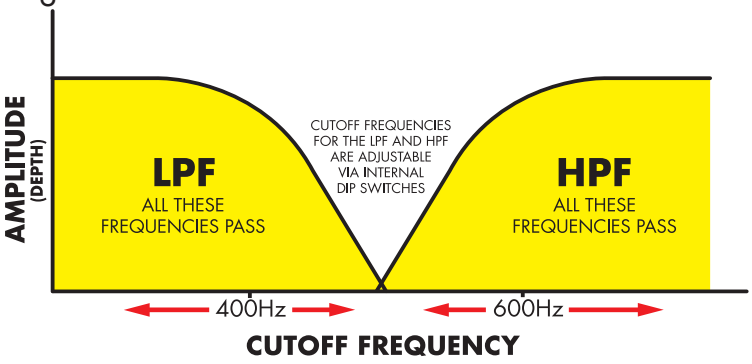
Optical tremolo is produced by using an optocoupler to modulate the signal in the preamp circuit of an amp. Optocouplers contain a light dependent resistor (LDR) which is placed next to a light bulb (or LED in modern optocouplers). The voltage to the light bulb is modulated, which in turn modulates the resistance of the LDR, causing the signal to be variably attenuated. Optical tremolos create a pulsing or throbbing sound, and tend to modulate the signal in a more lopsided manner, which tends to be quite pleasing to the ear.

In a **harmonic tremolo** circuit the audio signal passes through a crossover circuit which splits the low frequencies and high frequencies, sending them through their own circuit paths. The two frequency ranges are modulated 180 degrees out of phase from each other: while one frequency range is on, the other is off, and vice versa. Because the low frequencies and the high frequencies are separated from each other, harmonic tremolo has a wonderfully subtle phasing sound that is quite mesmerizing. This type of tremolo was found on many of Fender's early brownface amps, but was soon replaced because of the large number of expensive tubes and components that the circuit needed to operate. Due to the frequency-shifting characteristics of harmonic tremolo, the effect actually comes close to true vibrato; maybe Fender wasn't too far off after all by calling one of their earliest tremolo circuits "vibrato"!

HOW THE QUAVERATO WORKS

The Quaverato is a very versatile tremolo pedal that produces all three types of tremolo sounds mentioned above (bias, optical, and harmonic tremolo). When the guitar signal enters the pedal, it is buffered with a low-gain amplifier stage and then immediately sent to the high pass and low pass filters. The high pass filter (HPF) does exactly what it sounds like: it only lets high frequencies pass through, and blocks the lower frequencies. The low pass filter (LPF) does just the opposite: it passes low frequencies and blocks the high frequencies. See "Figure 2: Filters and Cutoff Frequency" on page 6. The specific cutoff frequency for each of these filters is adjustable via an internal trimmer potentiometer.

Figure 1: Filter Crossover



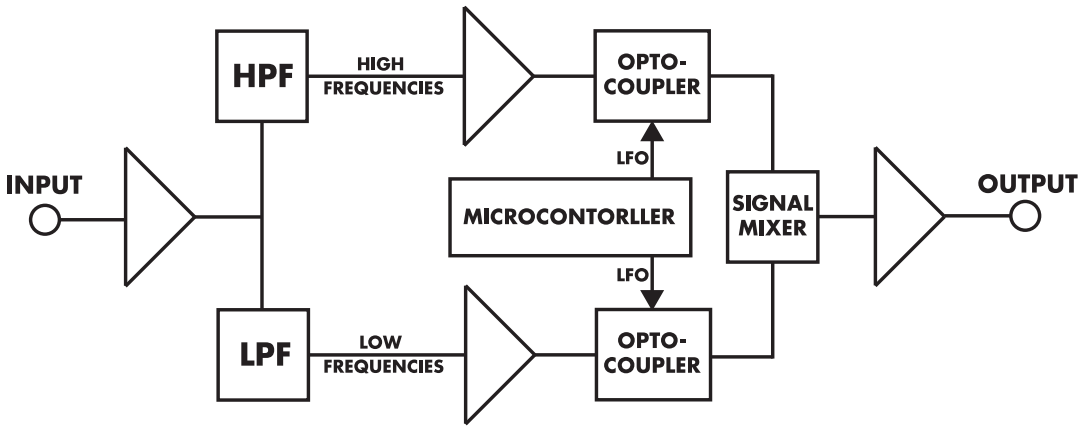
Next each signal is sent through a unity gain amplifier and through a digitally-controlled optocoupler. Each optocoupler consists of a light dependent resistor (LDR) and a light emitting diode (LED). The LDR is in series with the audio path. When the LED is dark, the LDR's resistance is at maximum, which is much too high for any audio signal to get through. As the LED gets brighter, the LDR's resistance lowers and allows signal to pass through. At full brightness the LDR's resistance is very low, allowing almost all of the signal to pass through.

The LED is digitally controlled by a microcontroller chip. This microcontroller produces an LFO which causes the LED to turn on and off. The microcontroller has been programmed to give you a wide range of control over how the LFO modulates the brightness of the LED.

Next the high and low frequencies are mixed back together. From there, the signal goes through a final amplifier stage on its way to the volume control and finally the output jack.

For a complete discussion of the Quaverato's features and controls, please see the Quaverato Owner's Manual, available from www.zeppelindesignlabs.com.

Figure 2: Quaverato Block Diagram



WHAT'S IN THE BOX

"Table 1: Quaverato Harmonic Tremolo Pedal Bill of Materials" (BOM) is a complete parts list of everything that should be present in your kit, followed by photos of each part. Print the BOM and carefully go through the kit, identifying every part. Before opening the silver static-protective bag, please read about the proper handling of integrated circuits (ICs) in paragraph 10 on page 25. Note that some of the components are difficult to tell apart. Compare them carefully with the photos. Besides verifying that nothing is missing, this will acquaint you with the parts and their names. If anything is missing, first double-check; we double-checked before sealing the box at our lab! If it's still missing, EMAIL US right away at info@zeppelindesignlabs.com. If we goofed and shorted your kit, we will get replacement parts in the mail to you as soon as possible. If you lose or damage anything, we will be glad to sell you replacements. You will need to order the unusual or custom components from us, but for more common parts, like resistors, capacitors, or screws, you may prefer to go to a local electronics or hardware store.

TIP: Empty the parts of the kit into a bowl, NOT onto the cluttered workbench, or onto the living room carpet! This will protect you from losing tiny parts.

Figure 3: What's In The Box

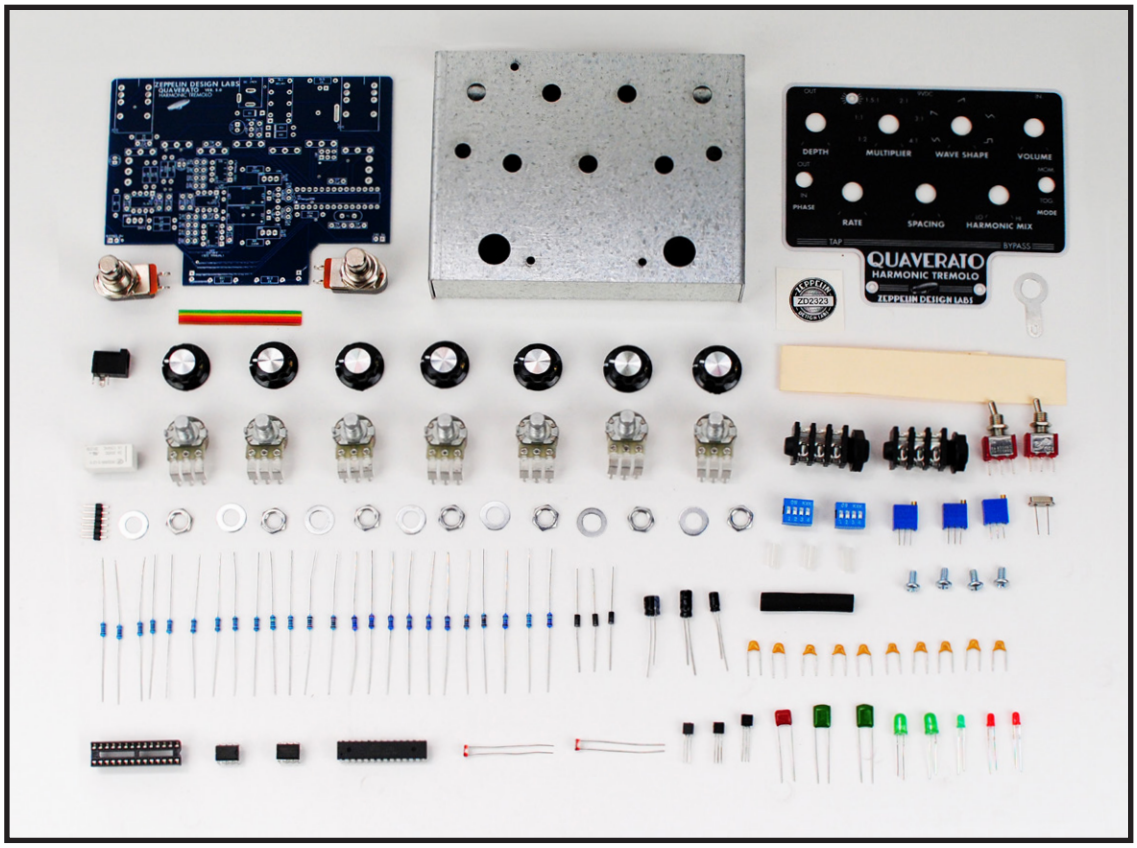
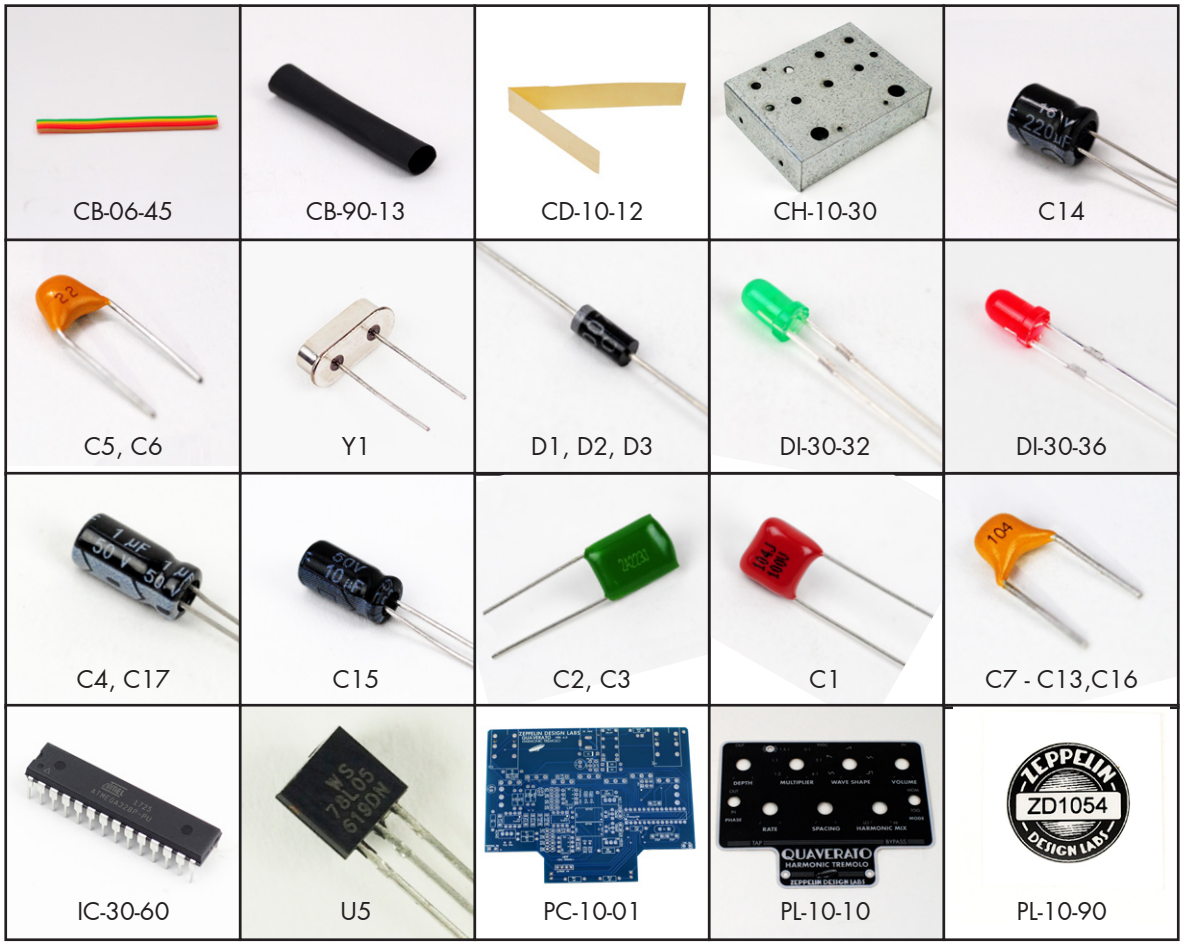


Table 1: Quaverato Harmonic Tremolo Pedal Bill of Materials

ZDL Part #	Description	Notes	Qty
CB-06-45	Hookup Wire, 2" (5cm) x 5 wires		1
CB-90-13	Heat Shrink Tube, 3/16" x 1-1/2" (5x40mm)	For Optocouplers	1
CD-10-12	Insulation Cardstock, 3/4" x 8" (2x20cm)		1
CH-10-30	Steel Chassis, Top & Bottom		2
CP-10-10	Electrolytic Cap, Radial 5x11mm 1uF	C18	1
CP-12-10	Electrolytic Cap, Radial 5x11mm 1uF Bipolar	C4, C17	2
CP-10-08	Electrolytic Capacitor 220uF	C14	1
CP-11-10	Electrolytic Cap, Radial 5x11mm 10uF	C15	1
CP-20-07	Film Capacitor 22nF	C2, C3	2
CP-20-16	Film Capacitor 100nF	C1	1
CP-30-18	Ceramic Capacitor 100nF	C7 - C13,C16	8
CP-30-19	Ceramic Capacitor 22pF	C5, C6	2
CR-10-10	Crystal Oscillator 16MHz	Y1	1
DI-20-03	Diode1N4007	D1, D2, D3	3
DI-30-32	LED, 3mm Green	TAP LED	1
DI-30-36	LED, 3mm Red	BYPASS LED, POWER LED	2
DI-30-52	LED, 5mm Green	For Optocouplers	2
FA-60-32	Screw, Phillips Pan Head #6 x 1/4”		4
HD-32-04	Knob		7
HD-40-01	¼" TRS Audio Jack	J1, J2	2
HD-40-10	DC Power Jack	P1	1
HD-50-02	Grounding Solder Lug		1
HD -62-01	MIDI IN hole plug		1
HE-20-01	Header, Single Row, 6 Pins Total	ISP	1
HE-25-28	Socket, 28 Pin	U1	1
IC-24-20	J-FET Quad Op Amp, TL074	U2	1
TA-15-20	Transistor, 2N3904	Q1, Q2, Q4, Q5	4
TA-20-04	J-FET J176	Q3	1
IC-30-60	Microcontroller, ATmega328-PU	U1	1
IC-80-50	Voltage Regulator, 78L05	U5	1
PC-10-01	Quaverato PCB		1
PL-10-10	Quaverato Faceplate Sticker		1
PL-10-90	Serial Number Sticker		1
PT-10-26	Potentiometer, 16mm 100K	VR2 - VR7	6
PT-10-27	Potentiometer, 16mm 500K	VR1	1
PT-30-10	Trim Pot, 10K	VR8, VR9	2
PT-30-30	Trim Pot, 100K	VR10	1

ZDL Part #	Description	Notes	Qty
RL-30-01	Relay	RL1	1
RS-80-32	Resistor, 220R	R7, R8	2
RS-80-40	Resistor, 1K	R4-6, R11-12, R15, R30-32	9
RS-80-50	Resistor, 14K	R17, R21	2
RS-80-51	Resistor, 10K	R2, R13, R14, R16, R27, R28	6
RS-80-52	Resistor, 18K	R18, R22	2
RS-80-55	Resistor, 36K	R20, R24	2
RS-80-56	Resistor, 24K	R19, R23	2
RS-80-61	Resistor, 100K	R9, R10, R29	3
RS-80-71	Resistor, 1M	R1, R3	2
RS-80-73	Resistor, 3.3M	R33	1
SN-30-10	Light Dependent Resistor	For Optocouplers	2
ST-60-49	LED Standoff, 9mm		3
SW-55-11	Footswitch, Momentary	S1, S2	2
SW-20-15	Toggle Switch, SPDT	S3, S4	2
SW-45-40	DIP Switch, 4 Position, SPST	S5, S6	2



HD-50-02	ISP	U1	U2	Q1, Q2, Q4, Q5
VR2 - VR7	VR1	VR8, VR9	VR10	RL1
R7, R8	R4-6, R11-12, R15, R30-32	R17, R21	R2, R13-14, R16, R27-28	R18, R22
DI-30-52	FA-60-32	HD-32-04	J1, J2	P1
R20, R24	R19, R23	R9, R10, R29	R1, R3	SN-30-10
ST-60-49	S1, S2	S3, S4	S5, S6	HD -62-01*
Q3	R33	C18		

If you ordered the MIDI upgrade with your Quaverato kit you will have an extra parts bag containing all the parts that pertain to the MIDI circuit. Throughout this assembly manual these MIDI circuit components will be designated with an asterisk(*). If you didn't opt for the MIDI upgrade, just ignore the components with the asterisk (*) beside them.

Figure 3: What's In The MIDI Upgrade Bag

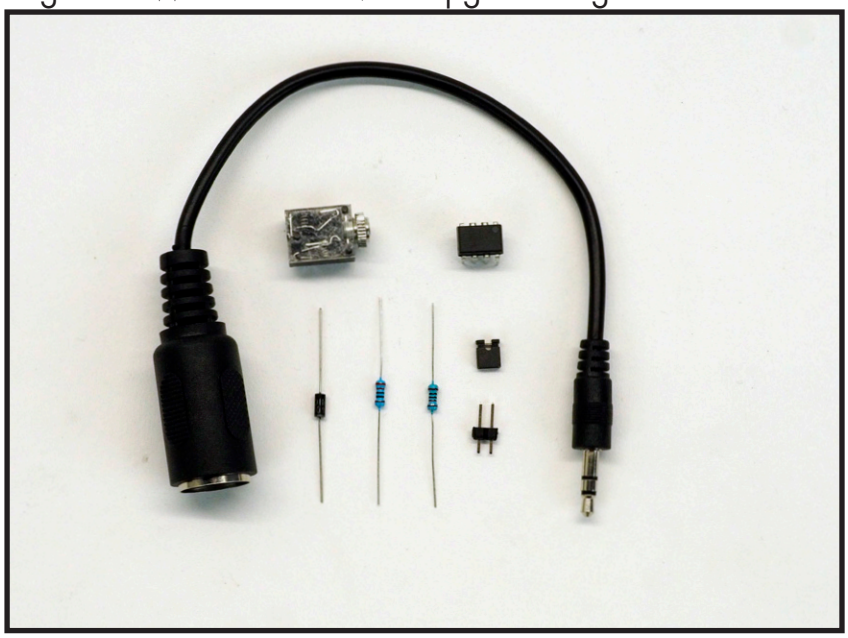
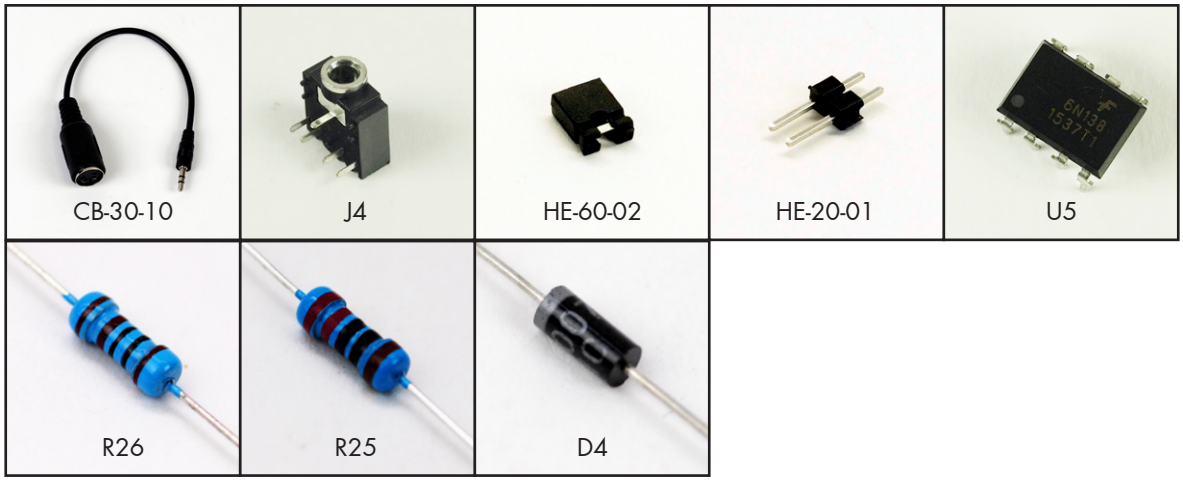


Table 2: Quaverato MIDI Mod Bill of Materials

ZDL Part #	Description	Notes	Qty
CB-30-10	Female MIDI to 1/8" Male TRS Adaptor		1
HD-40-40	1/8" TRS Jack	J4	1
HE-60-02	Jumper Header, 2 pin		1
HE-20-01	Header, Single Row, 2 Pins	MIDI Gnd Lift	1
IC-40-13	Optocoupler 6N138 DIP 8	U5	1
RS-80-40	Resistor, 1K	R26	1
RS-80-32	Resistor, 220R	R25	1
DI-20-03	1N4007 General Purpose Diode	D4	1



WHAT YOU WILL NEED

Here's everything you will need to build the Quaverato kit.

TOOLS

1. Digital multimeter
2. #2 Phillips screw driver
3. Small flat-head screwdriver
4. Needle nose pliers
5. Wire strippers
6. Small diagonal-cutting or flush-cutting pliers
7. X-ACTO® knife or hobby knife
8. 8mm, 10mm, 9/16" and 5/8" sockets and driver; in a pinch, pliers may work as well.
9. Soldering iron (not a soldering gun, or a "cold heat" iron), good quality, 15-50 watt. An iron with a temperature control and a stand is best. Use a small or medium size tip, with conical or chisel shape.
10. Solder sucker or solder braid (optional, but very handy if you have to remove or repair any components!)
11. Damp sponge or dry solder-cleaning pad
12. Clamp or vise to hold the printed circuit board while soldering (optional, but handy)

SUPPLIES

1. Solder, 60/40 rosin core, small diameter, good quality. We prefer .032" [0.8mm] Kester brand, but most brands will work fine.
2. Isopropyl alcohol and cleaning rag
3. Transparent tape or masking tape
4. Solder braid (AKA solder wick)
5. A plastic hotel key card or gift card (for making the knob-setting jig; see Appendix A)

POPULATING THE PRINTED CIRCUIT BOARD

Your work space should be well-lit, well-ventilated, and disposable; that is, don't work on the nice dining room table! Work on a utility surface that you can burn, drill and scratch. A piece of ¼" tempered masonite, or a chunk of MDF, makes an excellent surface if you don't have a utility work bench.

> **CAUTION**: Solder fumes are not healthy for you. The fumes consist of vaporized flux, which can irritate your nose, lungs, and even your skin. You MUST work in a space where the air drifts away from you as you work, so fumes do not rise straight into your face.

> **CAUTION**: Solder residue usually contains lead, which is poisonous if you ingest it. Do not breathe the fumes, do not eat the supplies, wash your hands after you handle solder, and sweep and wipe up your work space after EVERY USE.

Your Quaverato tremolo pedal contains one printed circuit board (PCB). All of the components will be installed on the "component side" of the board, which is the side with the part labels printed on it. The other side of the board is called the "solder side," which, as the name implies, is the side on which the legs of most of the components are soldered. Proper technique for installing and soldering components to a circuit board is demonstrated through several great resources on Instructables and Youtube under the search "PCB soldering tutorial." The general procedure consists of the following:

1. Install the part on the component side of the board, by threading the wire leads through the appropriate holes in the board. For your convenience, the board has silk screen outlines indicating where the components should be placed, along with text indicating the part number and the component value.

2. Hold the component in place with your finger and turn the board over.

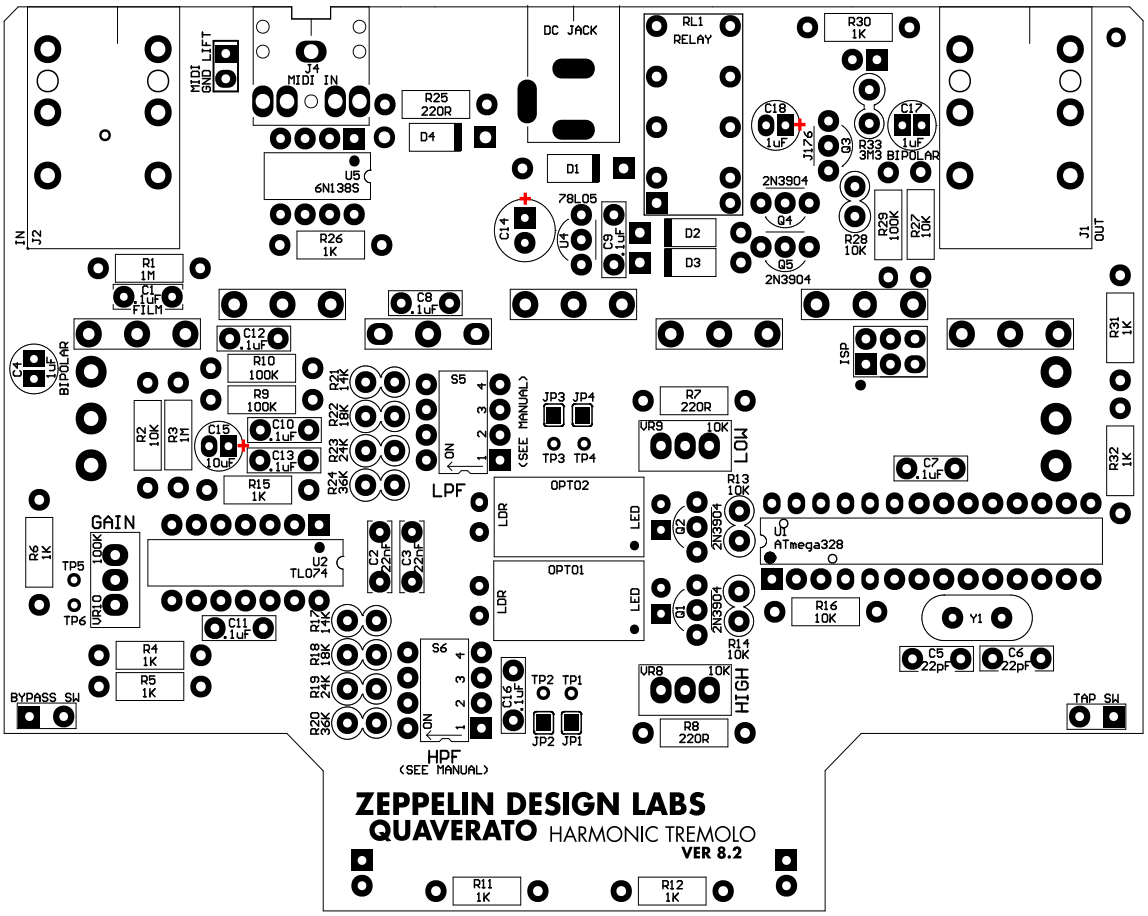
3. Gently bend the leads out at about 45 degrees to keep the component from falling out.

4. Install all of one type of component, bending each of the leads as they are installed.

5. Flip the board over solder-side-up, and solder all of the components in one pass.

6. Clip the leads off with small diagonal cutters, right at the solder joint.

Figure 4: Component Values and Locations



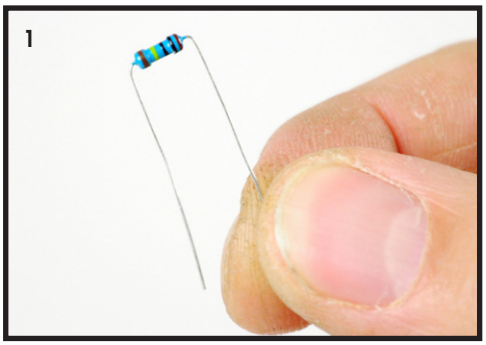
NOTE: Some photos in this manual are from a previous circuit board version, so don't be dismayed if you see some discrepancies.

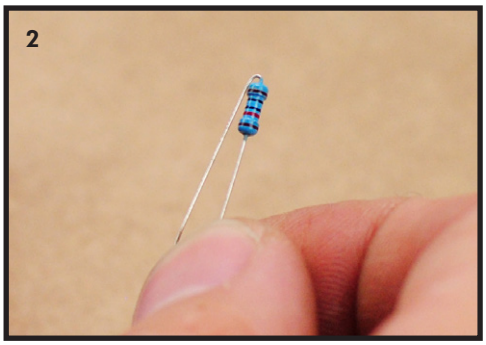
Let's begin!

1. Resistors: The values of resistors are given by a series of colored stripes. There are several tutorials online describing how to decode these stripes, but we will identify each resistor for you by simply naming the stripe colors, and giving you the value and the part number. "Figure 4: Component Values and Locations" on page 12 is a good reference. If you are color blind or can't see the stripes clearly, then you must use your digital multimeter to measure the resistance of each resistor.

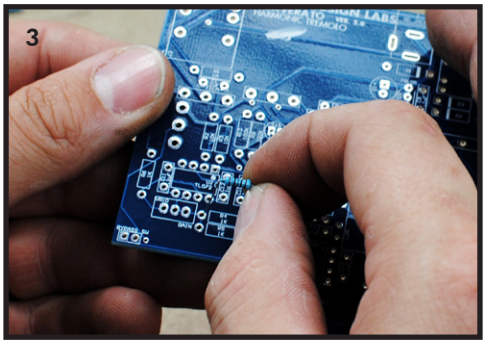
   The white graphics on the component side of the board give reference to the part number and value. Figure 4 shows the component values and locations and may be easier to read than the graphics on the PCB, so please use it to cross check your work.

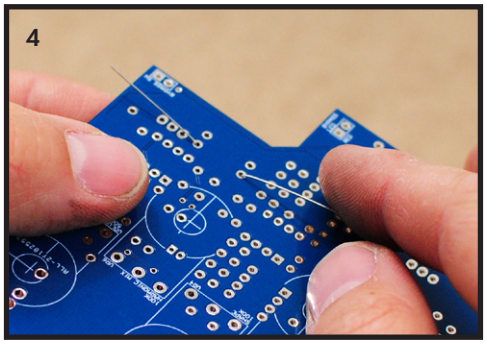
   Resistors are not polarized, meaning they can be installed into their holes in either direction. It doesn't matter which lead goes into which hole. The hole spacing of the resistors on the circuit board allows the leads to be (gently) bent 90 degrees at the body of the resistor (1). This allows the resistors to slip into their holes very easily. Resistors R13, R14, R17 - R24, R28 and R33 are exceptions. Note how those components are bent (2).

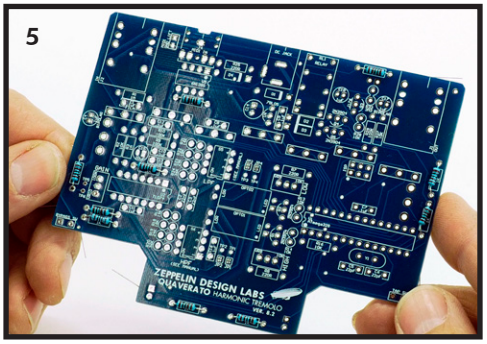


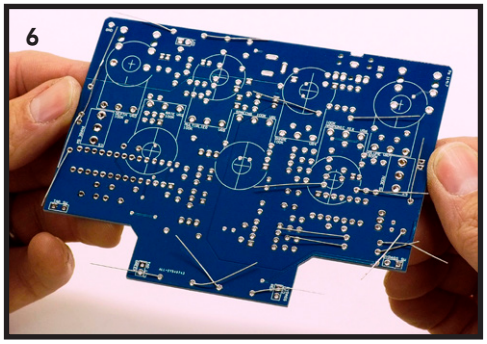


a. Start with the 1K resistors (R4, R5, R6, R11, R12, R15, R26*, R30, R31, R32), marked BROWN, BLACK, BLACK, BROWN, BROWN. Compare to their picture in the BOM. Find their locations on the circuit board; install and bend the leads out on the solder side as described above (3,4,5,6). Don't solder any of them until all 33 resistors are installed.

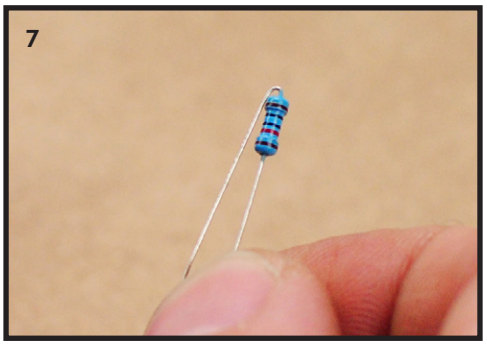


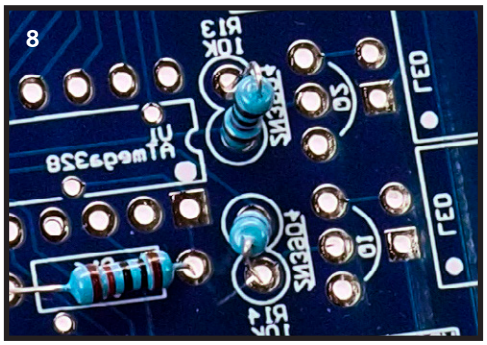






b. Install the 10K resistors (R2, R13, R14, R16, R27, R28). These resistors are marked BROWN, BLACK, BLACK, RED, BROWN. R13, R14, and R28 stand upright on the PCB so bend one lead nearly parallel with their bodies (7) and install them standing up (8). Bend the leads out on the back so they won't fall out.





c. Install the 220 ohm (220R) resistors (R7, R8, R25*), marked RED, RED, BLACK, BLACK, BROWN. Bend the leads on the back.

d. Next do the 100K resistors (R9, R10, R29), marked BROWN, BLACK, BLACK, ORANGE, BROWN. Bend the leads on the back.

e. Install the 1M resistors (R1, R3), marked BROWN, BLACK, BLACK,YELLOW, BROWN. Bend the leads on the back.

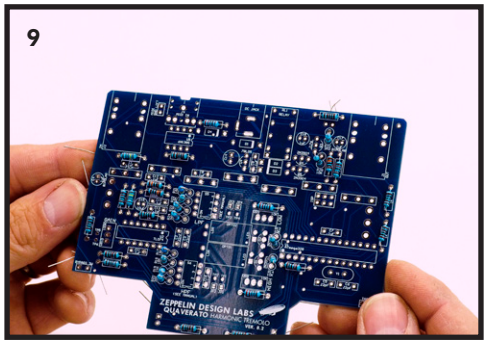
f. The rest of these resistors will be installed upright on the board. Always bend out the leads after insertion. Install the 14K resistors (R17, R21), marked BROWN, YELLOW, BLACK, RED, BROWN.

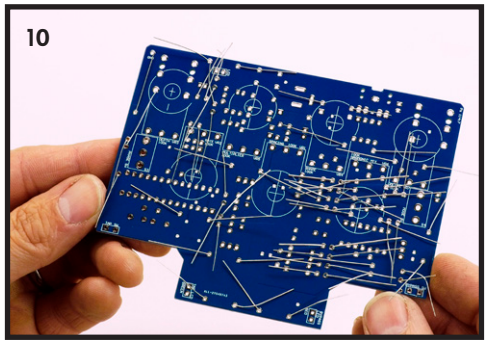
g. Install the 18K (18K) resistors (R18, R22), marked BROWN, GREY, BLACK, RED, BROWN.

h. Install the 24K resistors (R19, R23), marked RED, YELLOW, BLACK, RED, BROWN.

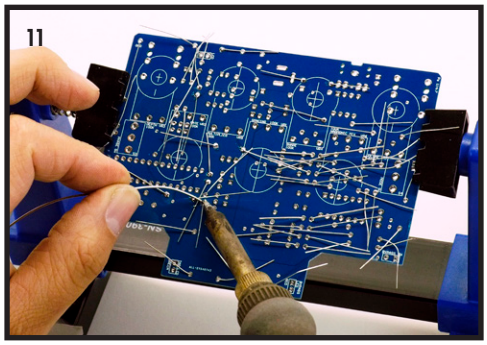
i. Install the 36K resistors (R20, R24), marked ORANGE, BLUE, BLACK, RED, BROWN.

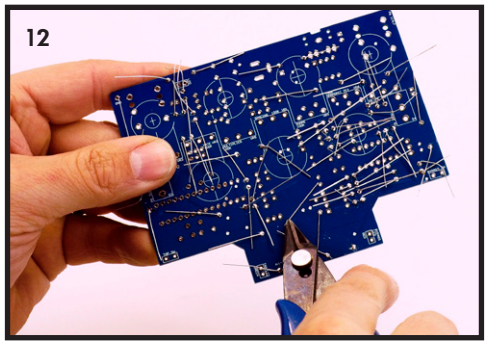
j. Last, install the 3.3M (3M3) resistor (R33), marked ORANGE, ORANGE, BLACK, YELLOW, BROWN.



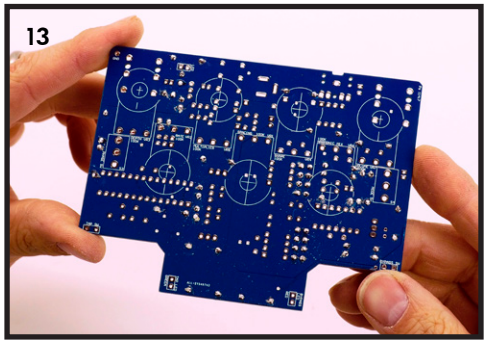


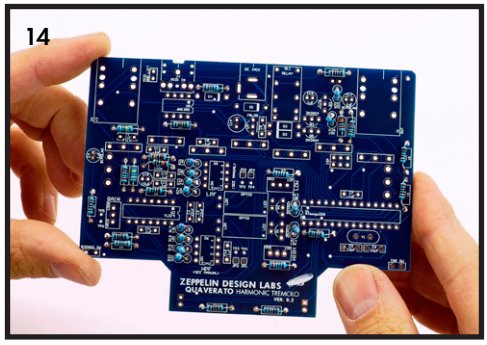
k. You should have a whole forest of bent leads coming out the solder side of the board (9, 10). Now you can turn the board solder-side-up and solder each lead to the board. Use a clamp or vise if you have one; it makes soldering much easier (11).



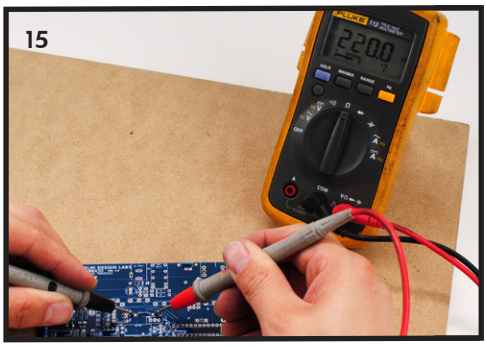


l. Now clip each lead with your flush cutters at the solder joint (12,13,14).





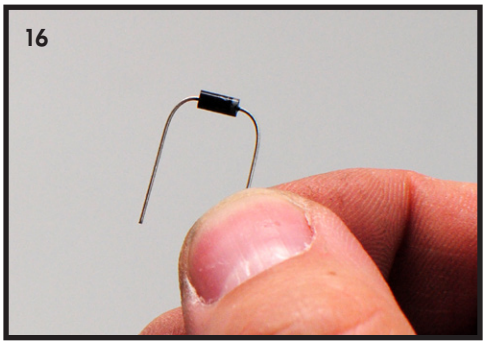
m. Before installing any more components on the circuit board, double-check the resistance values of each of the installed resistors (15). Set your digital multimeter to the “ohms” or “resistance” setting, and measure across all of the resistors. Compare the measured value to the listed value in Figure 4 on page 12. Make sure they are all correct (within 1%) before moving on!

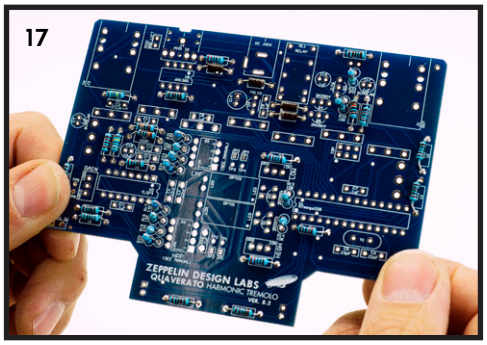


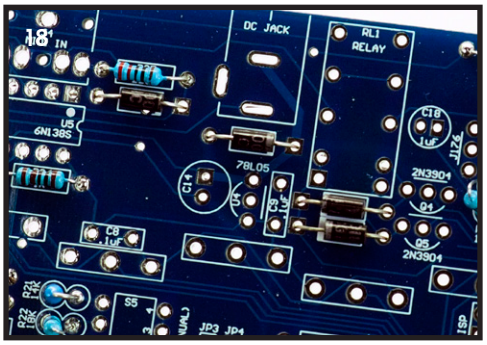
2. Diodes (D1,D2, D3, D4*): Diodes are polarized, meaning it matters which lead goes in which hole. If you get it wrong your pedal won’t work. You will notice one end of the diode body has a white stripe around it (16). The lead coming from the striped end of the diode goes in the hole with the square pad. The lead coming from the non-striped end of the diode goes in the round pad.

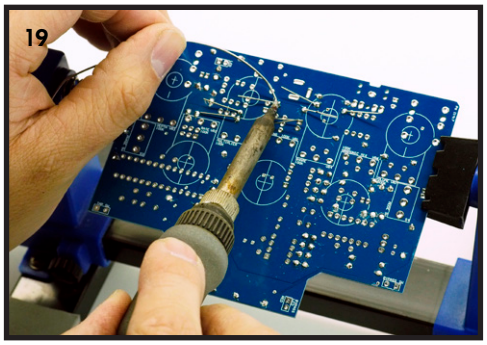
   WHITE STRIPE = SQUARE PAD
   NO STRIPE = ROUND PAD

   Place the diodes in their locations on the board (17,18). Gently bend the leads like you did for the resistors. Before you solder them, double check to make sure they are installed in the correct orientation. Solder and clip the leads (19).

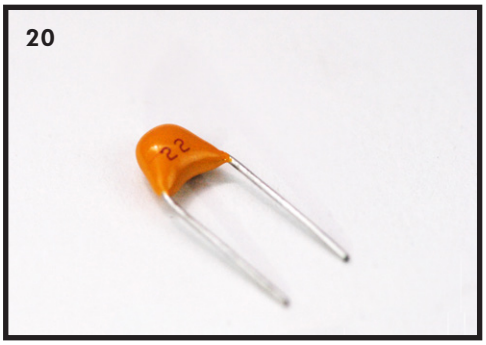


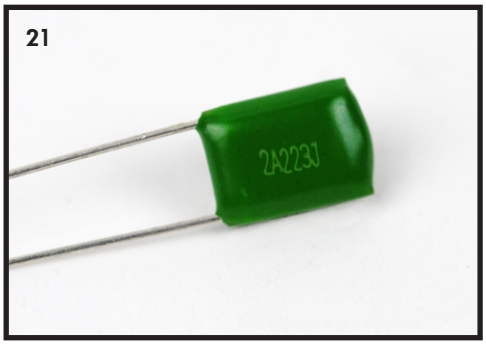


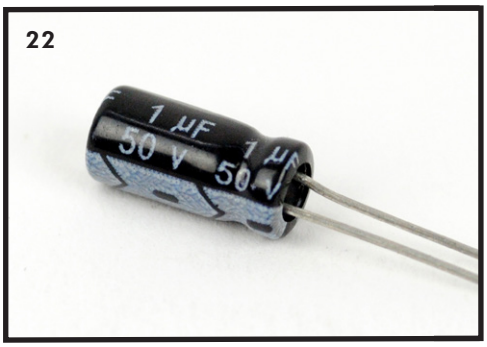




3. Capacitors: There are three types of capacitors in the kit: ceramic (20), film (21) and electrolytic (22). We will place them in the PCB one type at a time, and then solder them in groups.



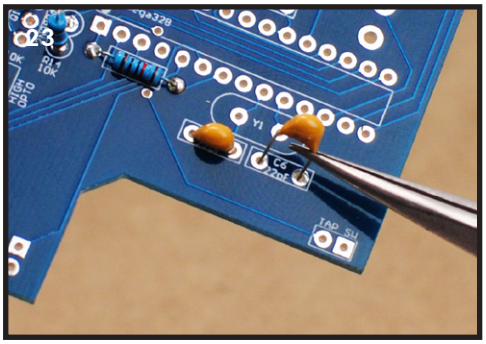


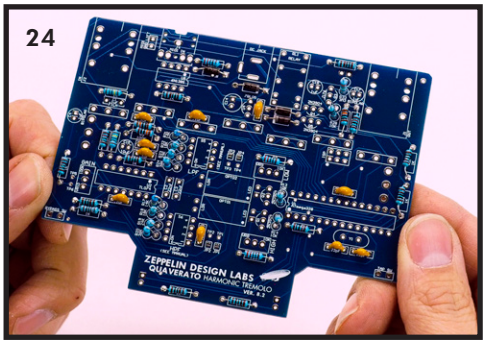


a. Ceramic Caps: These caps look like little yellow blobs with two protruding leads. The Quaverato uses two values of ceramic caps: 100nF and 22pF. The only noticeable difference between these capacitors is the tiny label printed on their yellow bodies. The 100nF caps are labeled "104" and 22pF caps are labeled "22" or "220". Please make sure that you use the correct value in the correct location. Ceramic capacitors are not polarized. It doesn't matter which lead goes into which hole.

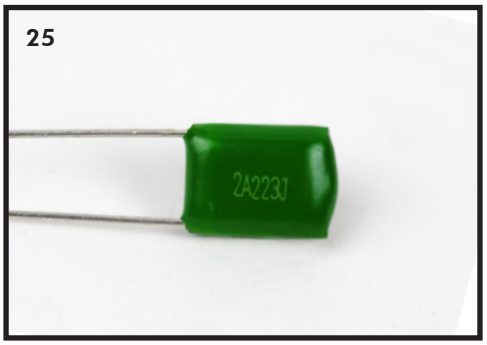
   i. There are two 22pF ceramic capacitors (C5, C6), marked "22" or "220". Install the 22pF capacitors at C5 and C6 on the PCB and bend the leads out on the back (23,24).

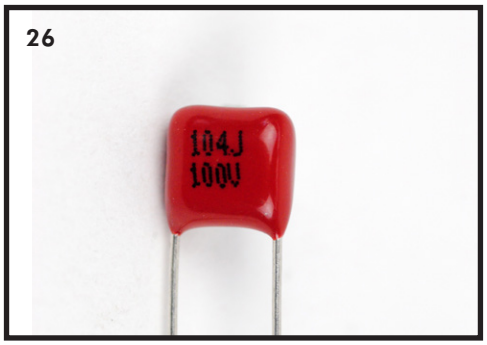
   ii. There are eight 100nF (=0.1uF) ceramic capacitors (C7-C13, C16). Place all the 100nF capacitors in their respective places on the PCB and bend the leads out on the back. Make sure you don't accidentally put one of these 100nF ceramic caps in C1, which is reserved for a .1uF film capacitor.





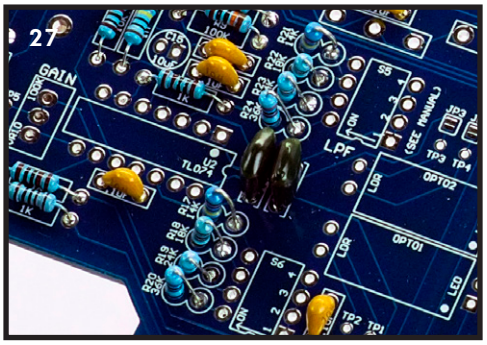
b. Film Capacitors: There are 3 film capacitors in the Quaverato. The two green film caps are 22nF (.022uF) in value and labeled "2A 223 J" (25). The red cap is 100nF (.1uF) in value and labeled "104J" (26). These caps are not polarized; the leads can go into either hole.

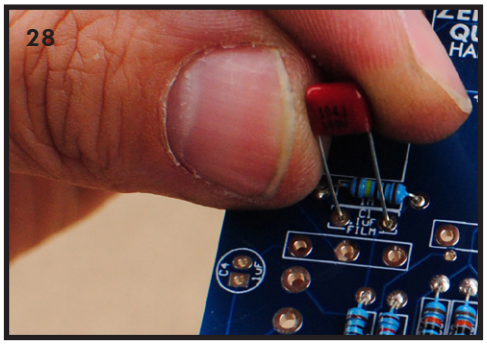


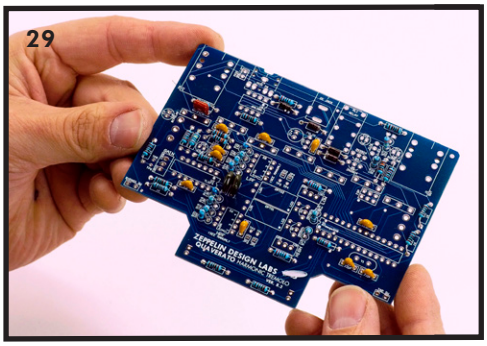


i. Place the green 22nF caps (C2, C3) in their locations (27) and bend out the leads.

ii. Place the red 100nF cap (C1) in its place (28,29) and bend out its leads.







c. Electrolytic Capacitors: We will use four different electrolytic caps. There is one 10uF cap (C15), one 220uF cap (C14), one polarized 1uF cap (C18), and 2 non-polarized (bipolar)1uF caps (C4, C17). You can read their values on their casings. All these electrolytic capacitors (except C4 and C17) ARE POLARIZED: there is a right way and a wrong way to install them. If you get it wrong, your pedal will not work and the capacitor might burst. The white stripe on the case indicates the side of the cap with the shorter, negative lead. The longer lead is positive. The longer, positive lead goes into the hole with square pad; the shorter, negative, white-stripe lead goes into the hole with the round pad.

STRIPE = NEGATIVE = SHORTER LEAD = ROUND PAD
NO STRIPE = POSITIVE = LONGER LEAD = SQUARE PAD

Make sure you orient these caps properly! For reference, Figure 4 on page 12 has little red plus signs (+) on the positive pads.

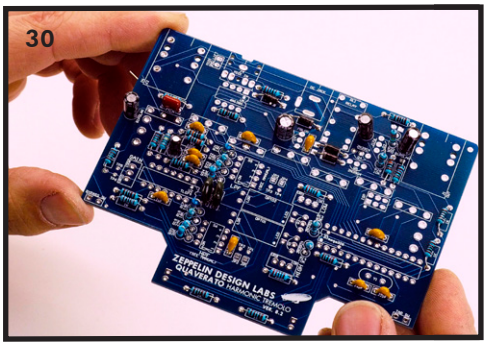
C4 and C17 are not polarized, which explains why there is no white stripe on their casing. You can install these in either direction.

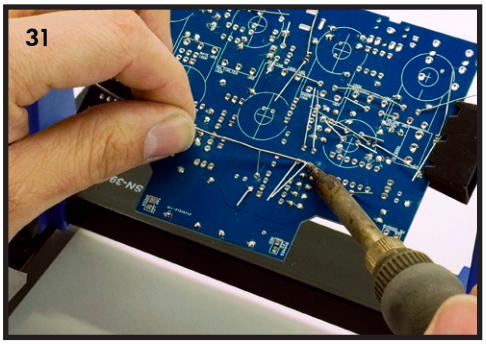
i. Install C15 in its place and bend the leads out.

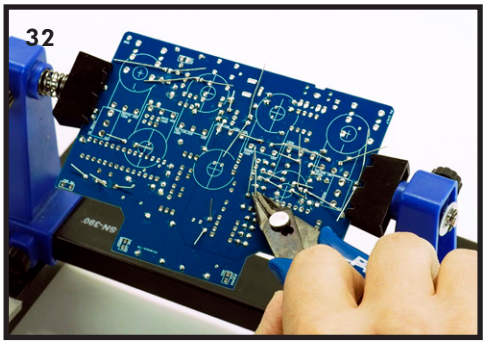
ii. Install C14 in its place and bend the leads out.

iii. Install C4 and C18 in their places and bend the leads out (30).

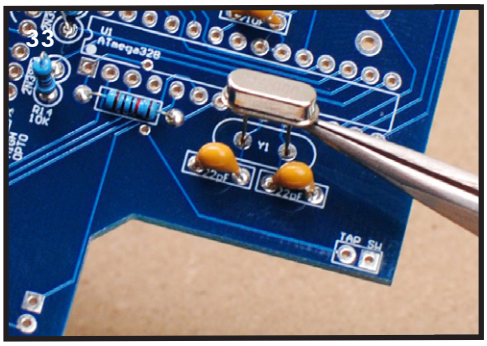
iv. Flip the board over, solder (31) and snip all the capacitors' leads (32).







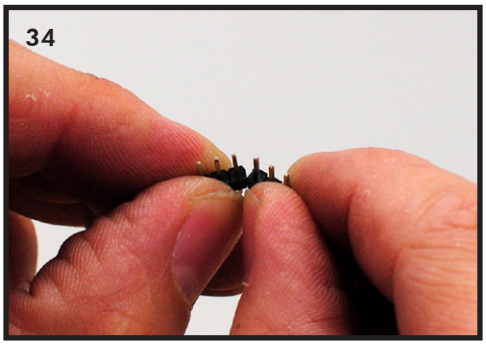
4. Crystal Oscillator (Y1): Place the crystal in the Y1 position on the PCB (the crystal is not polarized) (33). Bend the leads out on the bottom; solder and clip the leads.

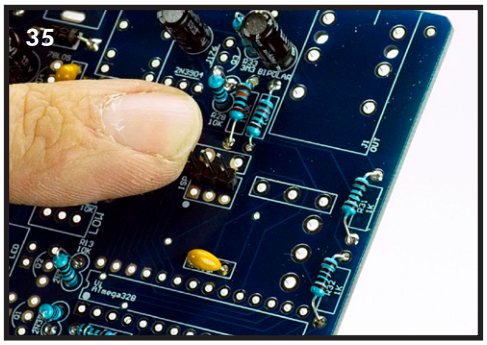


5. Headers (ISP and MIDI*): We will now install the headers on the board to allow the Quaverato to connect to other hardware.

   a. By installing a 2x3 header array, you create an ISP (in-system programming) port. This is a little socket that enables you to plug a cable into your pedal and upload (or "flash") new software onto the control chip (aka the microcontroller). The microcontroller in this kit comes pre-programmed with the software needed to make your Quaverato work, but you can mod or tweak the software and re-program the chip, if you are into that sort of thing. See the Quaverato Owner's Manual for more information about this.

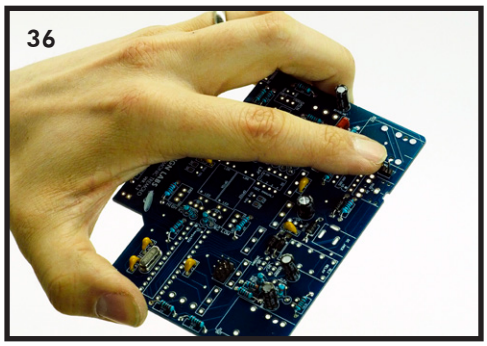
   Carefully break the header into two pieces with three pins each. (Your kit may have been packaged with pre-cut header pieces.) You can use your fingers to do this (34). Insert the two pieces in the pads marked "ISP" (35). The short pins go through the board; the long pins point up. Make sure the bottoms of the headers are flat against the circuit board.

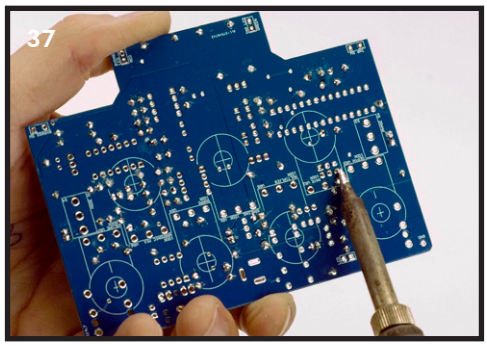


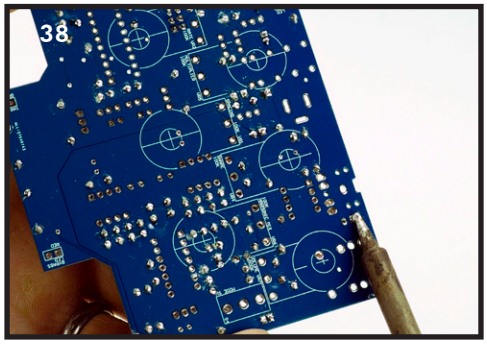


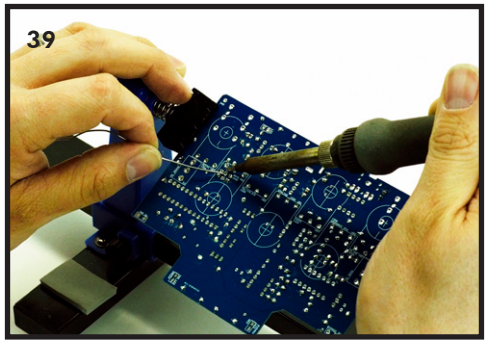
   b. Tack one pin down with solder while you hold the header in from the top (36,37). Once each row has been tacked on, you can solder the other pins in place(39).

   c. If you opted for the MIDI circuitry then install the 2 pin header* into the MIDI GND LIFT location. Install this 2 pin header* in the same manner as the ISP header: hold one pin in from the top as you tack it in from the solder side; then finish soldering it in (38,39). Leave the jumper off this header. See Appendix B for more information about what this jumper does.



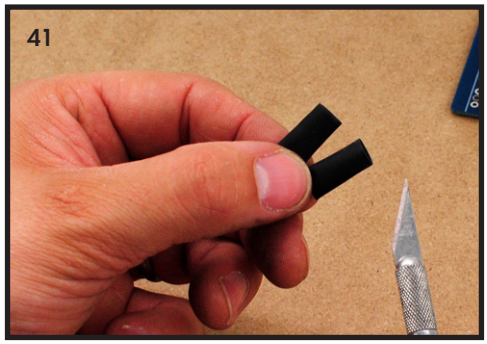




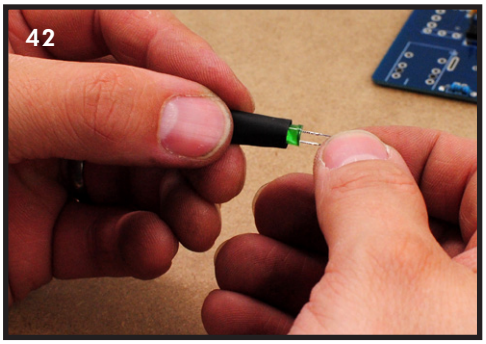


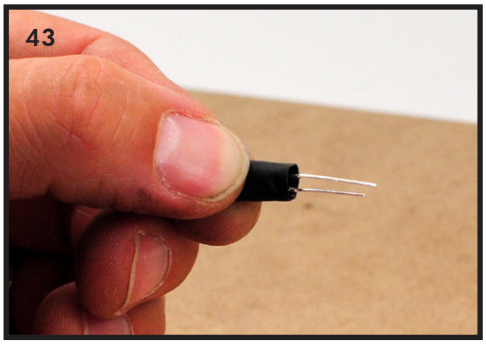
6. Optocouplers: The optocoupler is the link between the digital (control) side of the circuit and the analog signal path. There are two optocouplers: one that controls the low frequencies and one that controls the high frequencies. Each optocoupler consists of a light dependent resistor (LDR) and a light emitting diode (LED), both of which are sealed inside a length of heat-shrink tubing. The LDRs in this kit have been hand selected and matched to each other to within around 100 ohms at the same light intensity. This helps ensure that both signal paths are controlled equally. We will first construct the optocouplers and then install them on the board.

   a. First cut the heat-shrink tubing into two equal lengths. (They both should be about 11/16″ [17-18mm] long.) (40,41)

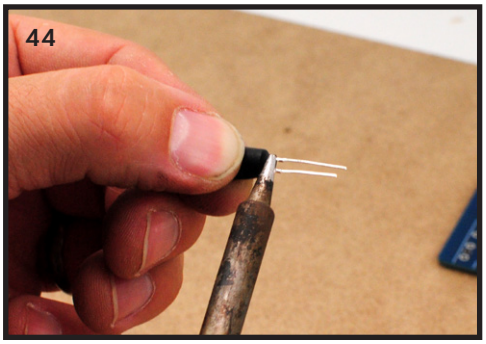


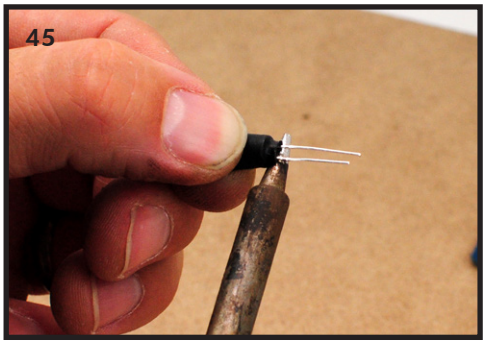
b. Insert a green 5mm LED (don't get them confused with the 3mm LEDs) into the end of a piece of heat-shrink tubing (42). Proper placement of the LED in the tube is important. Slide the LED into the tube until the back of the LED is about 5mm from the end. The LED just needs to be far enough into the tube so the end can be pinched fully closed around the backside of the LED (43). This will keep as much ambient light out of the tube as possible.

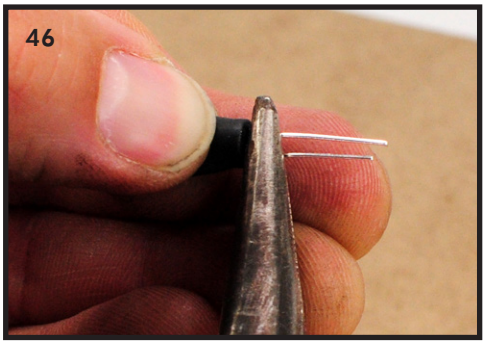


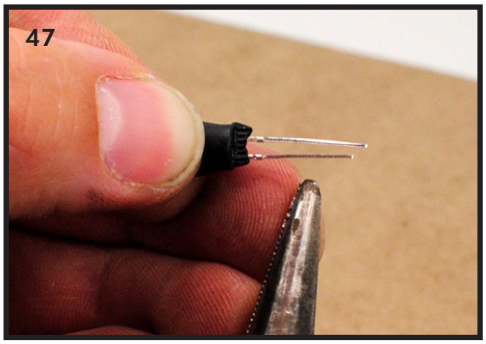


c. As you hold the LED in this position from the outside of the tube, carefully heat up the end of the tube with your soldering iron until it closes around the LED leads (44,45). Once it has stopped shrinking but while it is still very hot, use your pliers to pinch the shrunken tubing closed around the leads to help seal the opening (46,47).

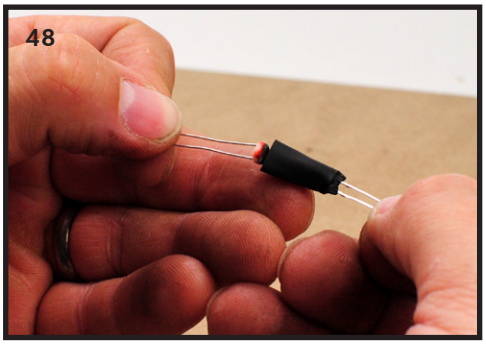


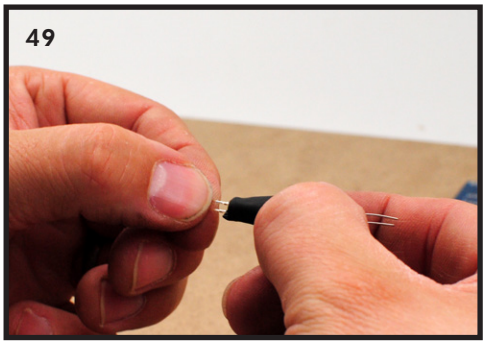




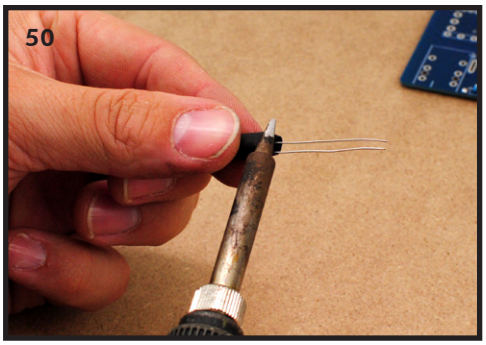


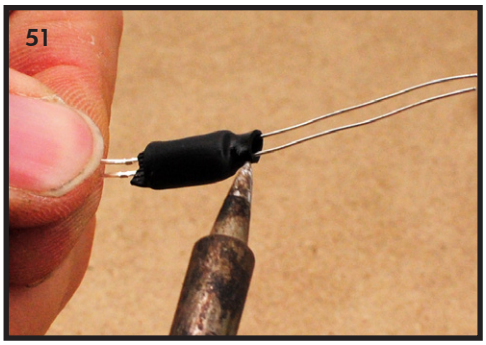
d. Slide the LDR into the other end of the heat-shrink tubing until it is touching the LED (48). Rotate the LDR to put the leads in the same plane as the LED leads (49).

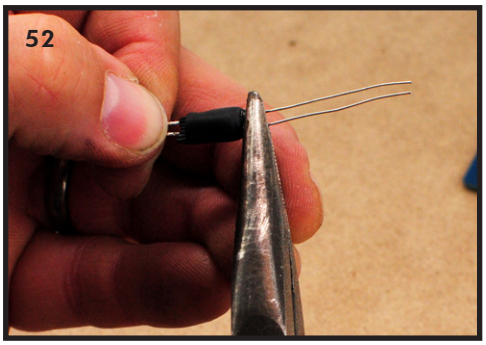


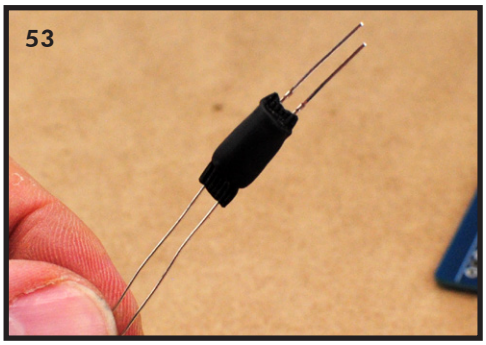


e. As you hold the LDR from the outside of the heat-shrink, carefully heat this side of the tube until it closes around the LDR leads (50,51). While it is still very hot, use your pliers to pinch the end closed around the LDR leads (52,53).



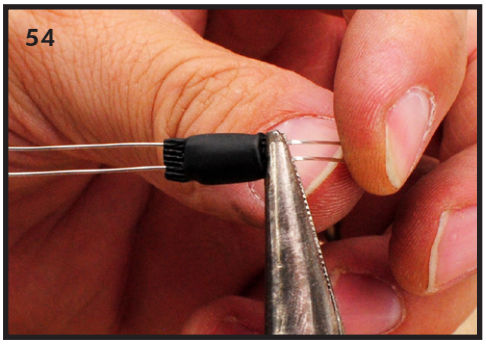


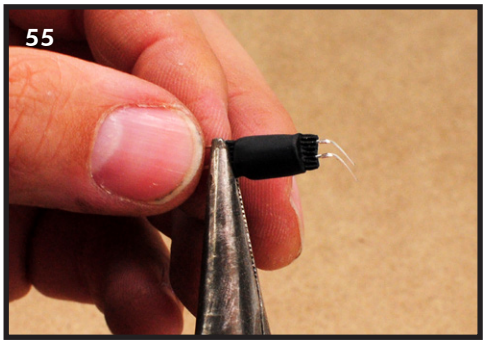


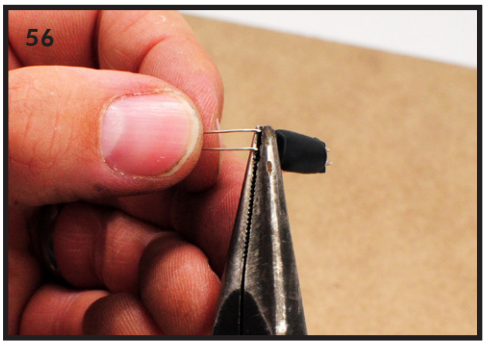


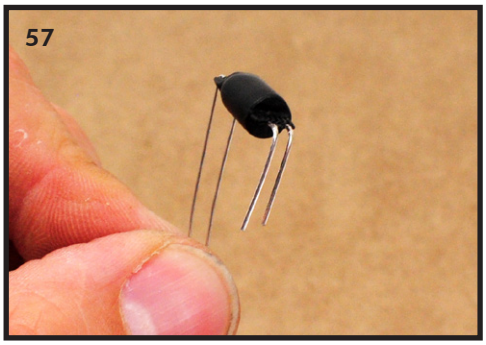
f. Repeat this process to make the second optocoupler.

g. Now that we've made the optocouplers we can install them. The leads need to be bent 90 degrees, but it matters which way you bend them for them to fit in the proper orientation. If you put the opto in backwards the tremolo effect will not work. Note the PCB graphics indicate where the LDR and LED are intended to go. On your optocoupler, the LED has one long lead and one short lead. The long lead of the LED goes into the hole with the square pad. Note which side of the optocoupler needs to be up as you bend the leads down in order for it to fit into the holes properly (54). Grip the leads with your pliers as shown (55) and bend them down 90 degrees with your fingers (56). Bend the LDR's leads in the same way, holding with pliers and bending with fingers (57). Repeat for the other optocoupler.

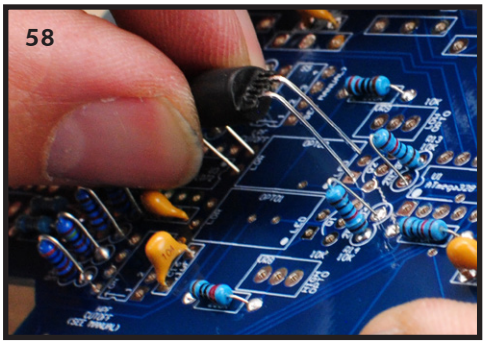


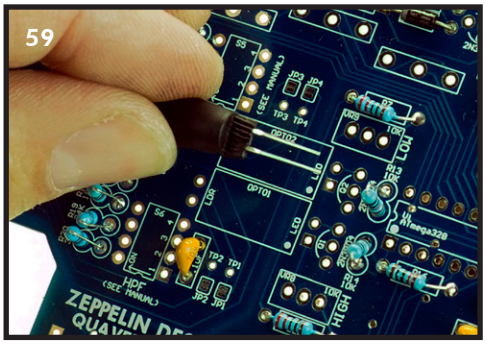




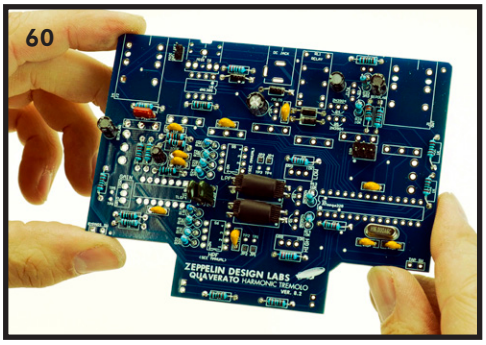


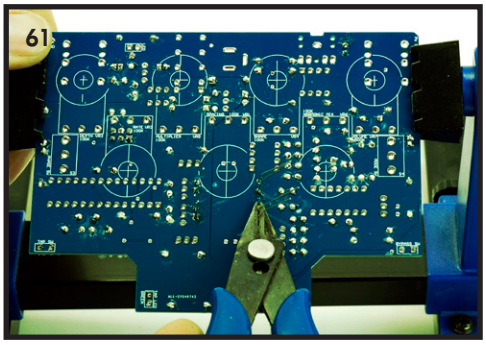
h. Carefully install both optocouplers on the PCB. Double check to make sure the long LED leads are in their correct holes – with the square pads (58,59).



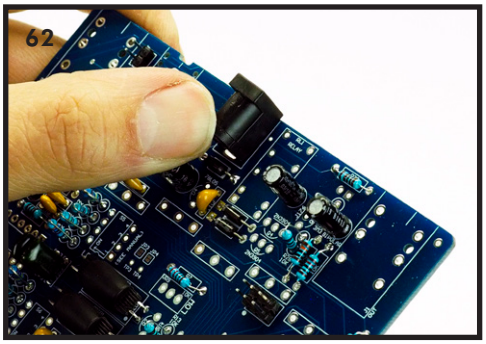


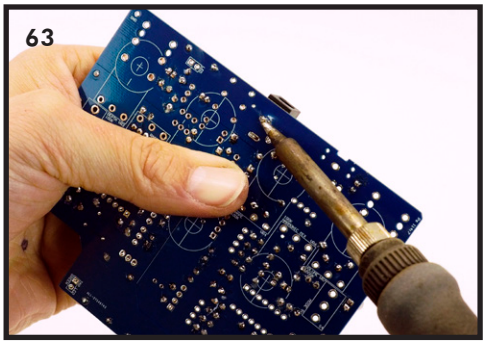
i. Bend the leads out on the back, solder all 8 leads, and snip them flush (60,61).

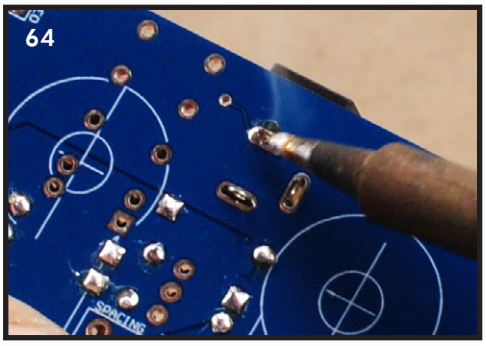


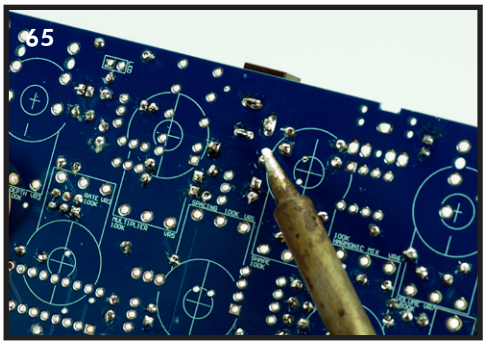


7. DC Power Jack: Press the jack snug to the board and slide it all the way forward, toward the edge of the board (62). Double check to make sure it's sitting square to the edge and flat against the face of the board. Hold the jack in place as you tack in one pin with solder to get it to stay in place. Solder the other pins and then re-solder the first pin (63,64,65).







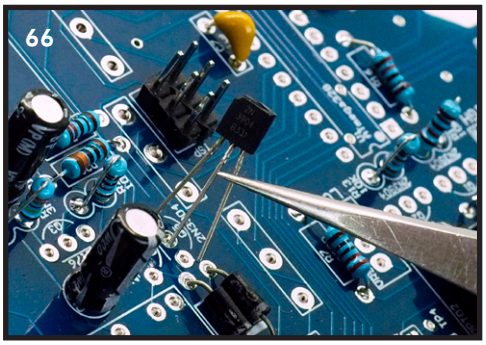


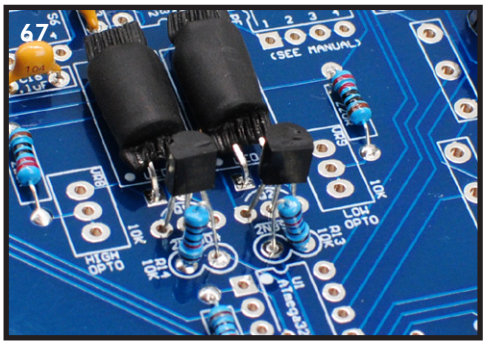
8. Transistors: The transistors are in the silver anti-static bag. Before you open this bag, review the notes in Step 10 on page 27 concerning the safe handling of ICs. There are 5 transistors in the Quaverato (Q1-Q5).
Q1, Q2, Q4, and Q5 are NPN transistors labeled 2N3904 (and possibly a few extra numbers or letters at the end). They look identical to the voltage regulator (labeled L78L05), so please don't get them mixed up – otherwise your Quaverato won't work!

The transistor with the leads shaped a little different than the others is Q3. It is actually a JFET and it should be labeled J176. JFETs are very sensitive to static electricity so please observe the precautions described in the paragraph on the safe handling of static sensitive components on page 27.

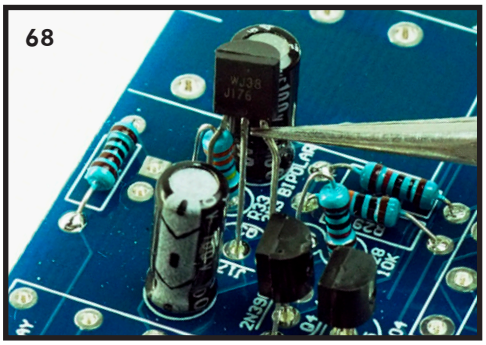
All these transistors are shaped like a three-quarter moon. Notice the PCB graphics around the holes have a similar shape to show you the component's correct orientation.

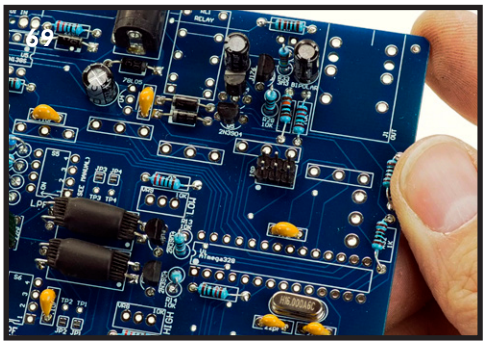
a. Install the 2N3904 transistors in locations Q1, Q2, Q4, and Q5 (66,67). You'll need to spread the leads out a little bit so they will fit in their holes. Bend the leads out on the other side of the board.



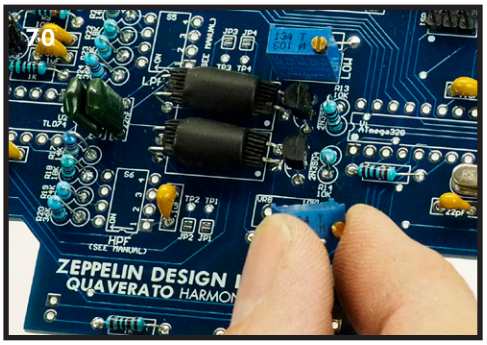


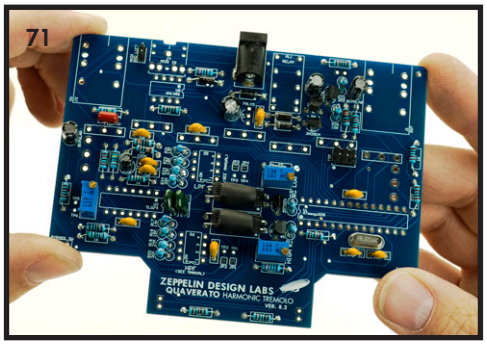
b. Install the J176 JFET in Q3. Bend the leads out on the other side of the board (68,69). When soldering, try to keep these components from getting too hot. Take your time. Clip the leads when you're done soldering them.





9. Trimmer Potentiometers: The Quaverato contains 3 trimmer potentiometers (trim pots) in two different values: 10K, (VR8, VR9, labeled "103"), and 100K (VR10, labeled "104"). You need to pay close attention to the labels because the pots are otherwise identical. Place VR10 in its location with the adjustment screw located upward toward R2 and R3 (71). Place VR8 and VR9 in their locations with the adjustment screw toward U1 (the big 28-pin chip socket) (70). Solder and clip the leads.





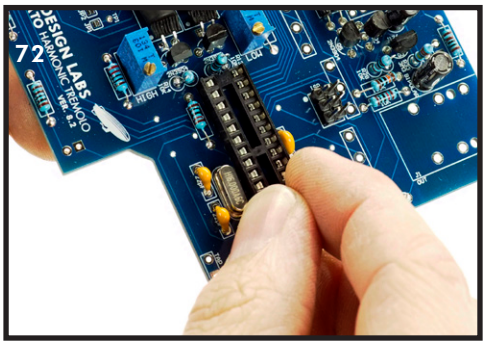
10. Integrated Circuits (ICs): This kit contains (up to) four types of IC components: the microcontroller, a quad op amp, a voltage regulator, and a opto-isolator* (if you got the MIDI upgrade). Keep in mind, the voltage regulator looks just like the transistors, described in Step 8 on page 25. These components are packed in a static-protective silver bag. In general, ICs are quite sensitive to static electricity and can easily be damaged. Humans are not sensitive to static electricity at these low, yet damaging levels; in fact, most people can't even feel a static discharge less than 1000 volts! So it is easy to damage these components without even knowing it. Before touching an IC, and often while working with them, ground yourself preferably by touching something grounded to the mains like the metal chassis of a plugged-in amplifier, or a refrigerator. At the very least touch a large conductive object like a metal desk or filing cabinet. In our lab, we wear conductive, anti-static bracelets that are connected to the electrical main's ground.

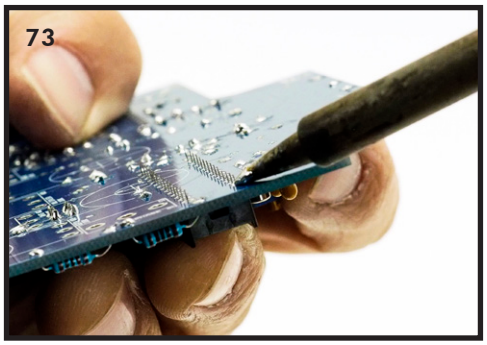
    When soldering ICs, try to keep the IC from getting too hot. As a rule of thumb don't keep your iron on any leg longer than two seconds, and make sure the chip stays cool enough to touch. Solder one leg at time and let the chip cool off before proceeding to the next leg. ICs have a specific orientation. If you install them wrong, your pedal will not work and you could damage or destroy the IC. Each IC has a dot or a divot at one corner or end. These features have a corresponding graphic in "Figure 4: Component Values and Locations" on page 12, and/or on the PCB, to show you how to orient the IC.

    a. Microcontroller, ATmega328-PU (U1): This IC is the "brain" of the Quaverato. We pre-programmed this chip with software which tells the pedal how to operate. This software is open-source and published under the creative commons license. To learn how to modify the software to make your Quaverato behave differently, please visit our GitHub page. You can use the popular development environment Arduino to edit and modify the code. We will not actually be soldering the microcontroller to the PCB itself; instead we will solder an IC socket to the board and install the microcontroller into the socket.

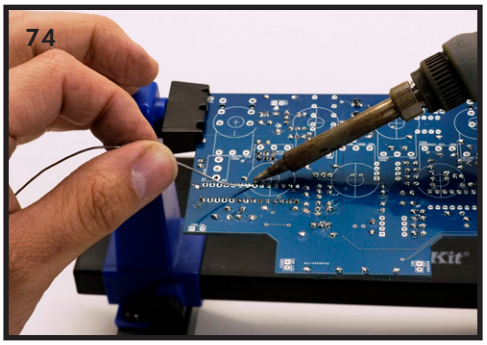
i. Install the 28-pin socket (part #HE-25-28) in the U1 location (72). Please note the divot at one end of the socket. This end should align with the dot on the PCB graphic.

ii. Hold down the socket with one finger while you flip the board over. Tack down one pin with a tiny bit of solder (73).

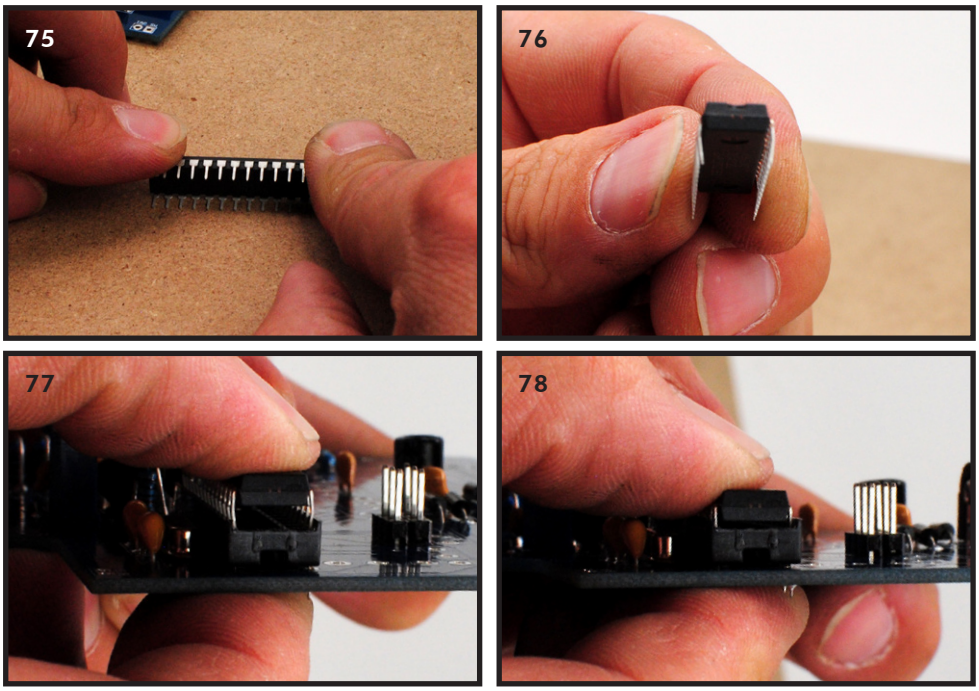




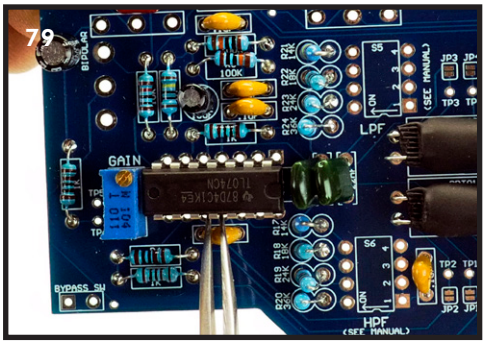
iii. Now solder the rest of the pins (74). Don't forget to go back to the original "tacked down" pin to finish soldering that correctly.

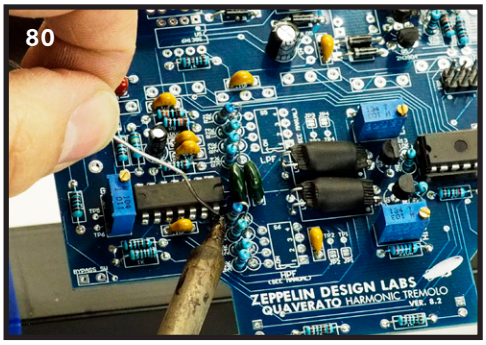


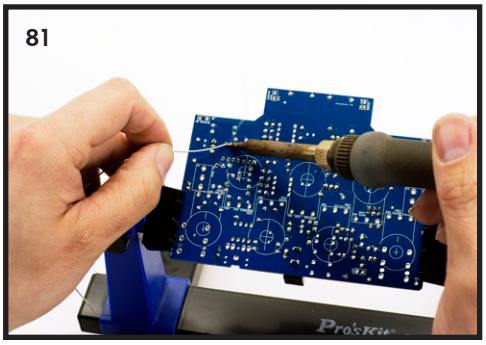
iv. Install the ATmega328-PU microcontroller in its socket. Be careful to align the divot in the micro with the divot in the socket. If the pins don't line up easily with the socket, you can gently bend the pins a little closer together by pushing each side of the chip down on the table top. (75,76). Press the IC snugly into place (77,78).



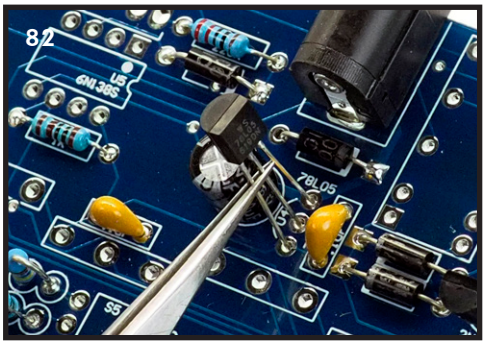
b. Quad Op-Amp, TL074 (U2): This chip actually consists of four individual operational amplifiers (op-amps for short), that drive and boost the audio signal. Install this IC in its location (79). You may need to gently flatten the pins down on each side of the chip to get them to fit more easily into their holes. Notice the divot at one end of the chip. There is a matching shape on the PCB graphic – don't get it backwards! Press the op-amp down snugly with your fingertip. Mount the PCB in your clamp and tack one pin down with solder from the component side (80). Now flip the board over and finish soldering the rest of the pins (81).

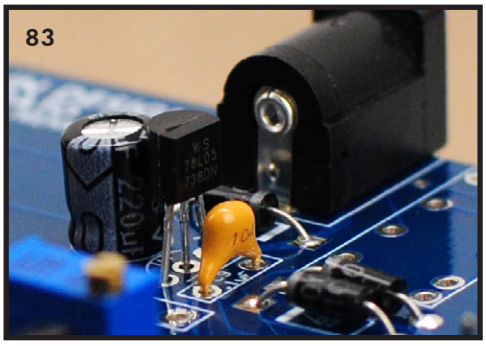




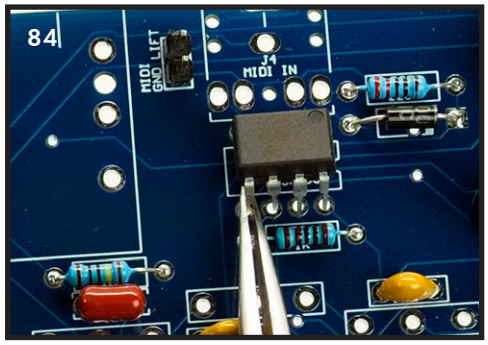


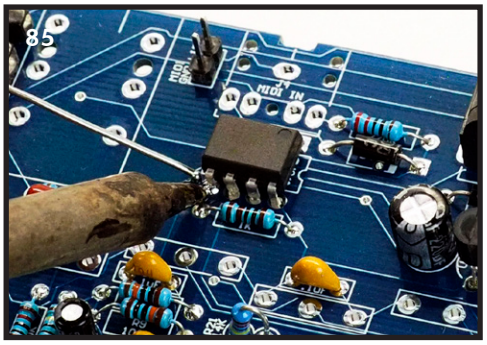
c. Voltage Regulator, 78L05 (U4): This component takes the 9 volts from the power jack and converts it to 5 volts to run the microcontroller. This component is shaped like a three-quarter moon and looks very similar to the transistors you installed earlier. Install the voltage regulator on the board in the correct orientation (it is polarized). Bend the leads out on the other side of the board. Solder and clip the leads (82,83).

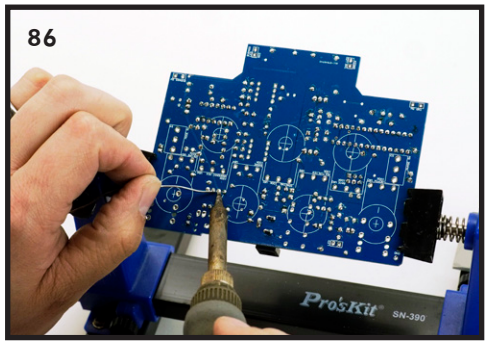




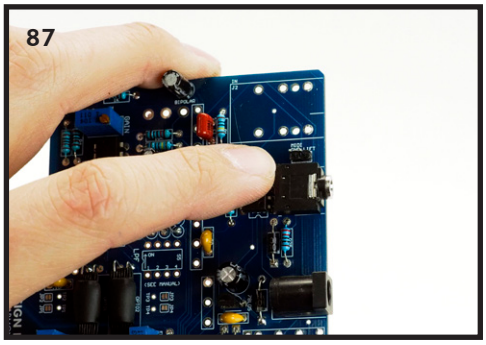
d. Opto-isolator*, 6N138S (U5*): This IC helps protect the microcontroller from static voltage spikes that may inadvertently be sent down the MIDI cable to the Quaverato. This opto-isolator has a dot near one corner. The pin nearest this dot is pin 1, and it goes into the hole with the square pad. Make sure it is installed as in the pictures (84). Solder it in the same way as the quad opamp IC (85,86).

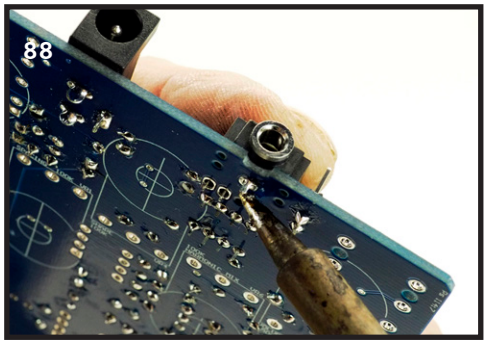




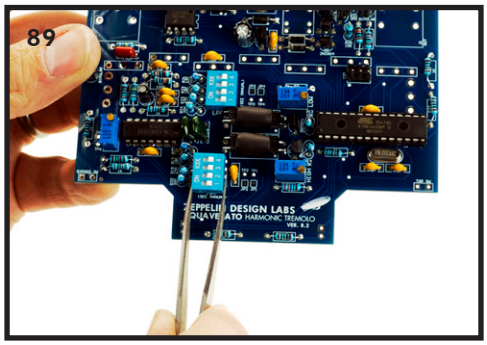


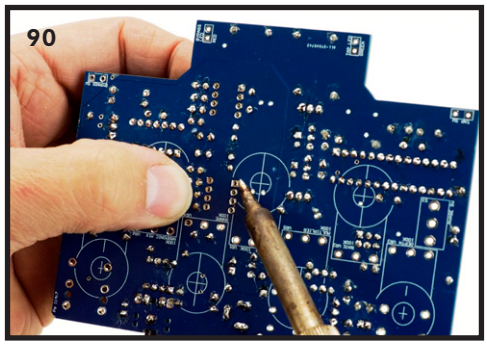
11. MIDI IN Jack* (J4*): If you are installing the MIDI components, you'll need to install this jack. Make sure this jack is in its holes all the way and sitting flat against the top of the board (87). Hold the jack in place as you tack in one pin with solder to get it to stay in place. Solder the other pins and then re-solder the first pin (88).



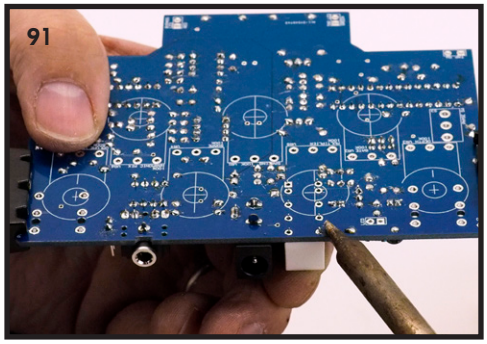


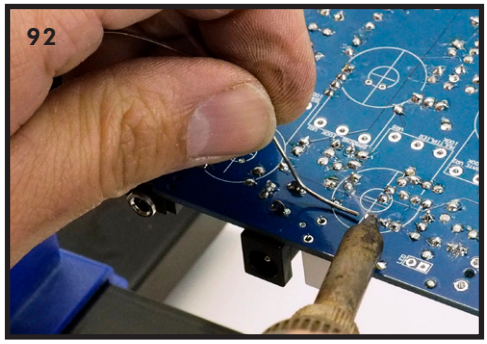
12. DIP Switches (S5, S6): "DIP" stands for "Dual In-line Package," which describes the pin layout on these miniature switches. When installing them, make sure the labels on the switches line up with the markings on the PCB (89). Hold the switch in place with your finger. Flip the board over and tack one pin in place. Now set down the board or place it in your clamp and solder the rest of the pins properly (90).



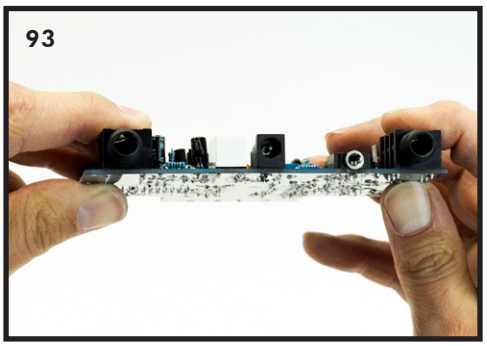


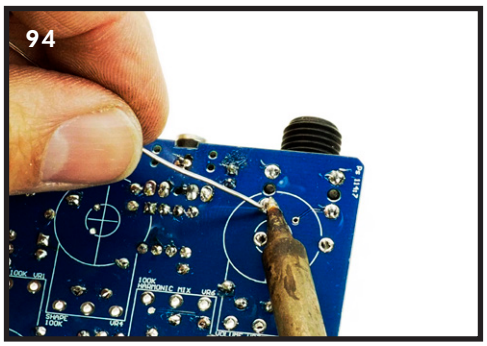
13. Relay (RL1): Relays are just electrically controlled switches. This allows us to have a true-bypass signal path that is digitally controlled. There is only one way the relay fits in its footprint. Install it the same way as the DIP switches, holding it from the top (91) while tacking down one pin. Finish soldering the rest of the pins (92).

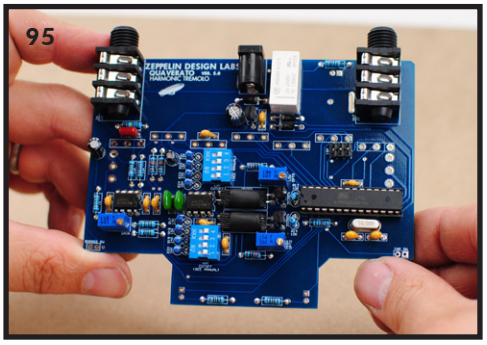




14. Audio Jacks (J1, J2): Install the 1/4" jacks into J1 and J2 (93). Make sure the bottoms of the jacks are flat against the face of the PCB. Solder the pins (94,95). The jacks in your kit may be missing a couple of it's 6 pins. This is fine; those missing pins are not used in the circuit.

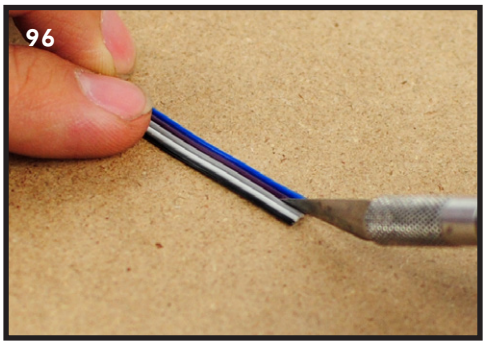

93

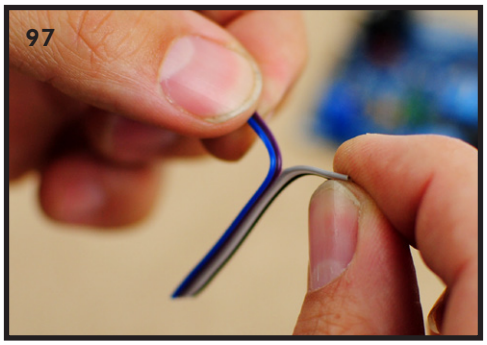

94

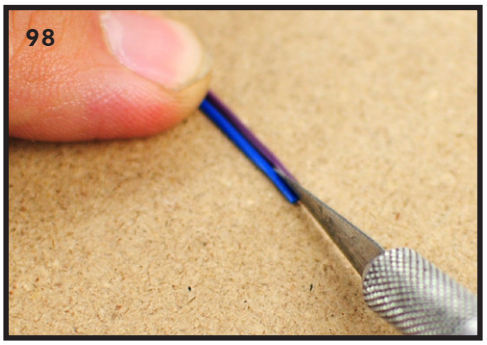


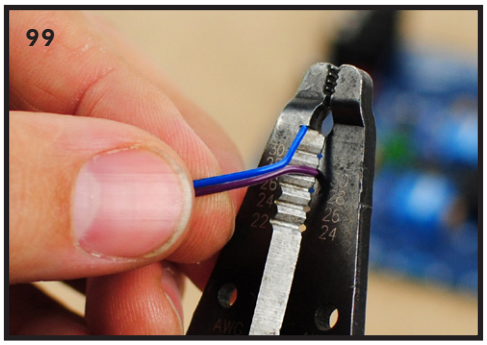
95

15. Hookup Wires: The 5-conductor ribbon wire needs to be split into three pieces: two 2-wire pieces and one 1-wire piece. Slit the wires at one end with your knife, and pull them apart (96,97). Next, split the 2-wire pairs about ¼" (6mm) on each end (98). Now strip off about 1/8" (3mm) of insulation from each end of all five wires (99,101).

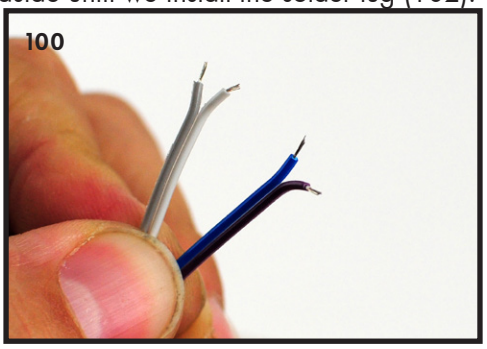

96

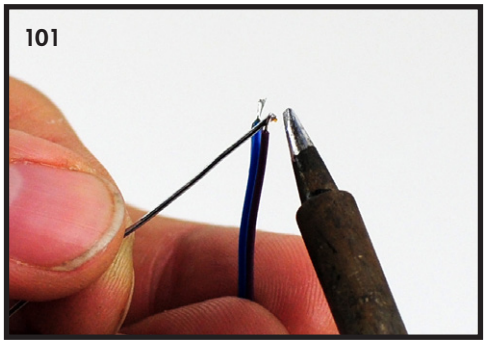

97

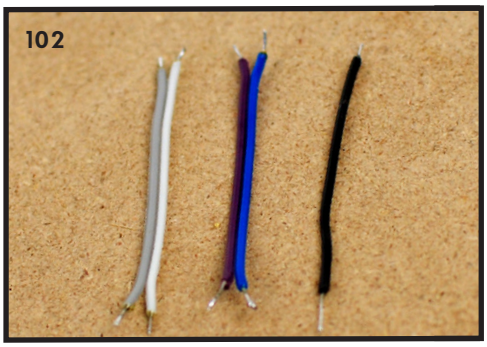

98


99

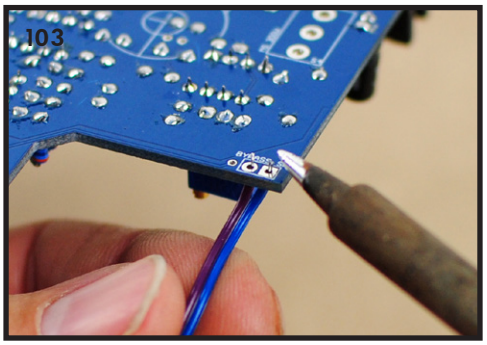
Tin each end with solder (101). We'll use the two 2-wire pairs in the next step. Set the single wire aside until we install the solder lug (102).

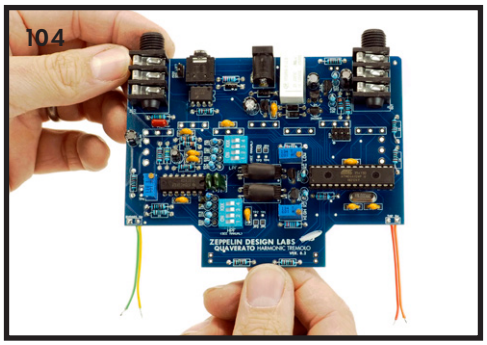






16. Switch Wires: Insert one of the 2-wire pieces into the BYPASS SW pads on the component side of the PCB. Solder them on the solder side. Repeat with the other wire pair at the TAP SW location.



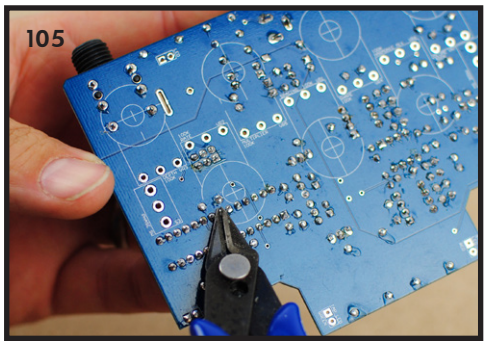


The remaining PCB components will be installed on the solder side of the board.

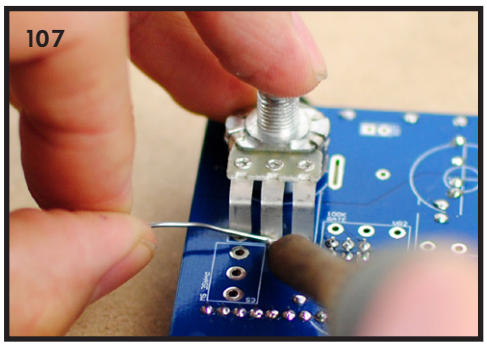
17. 16mm Potentiometers:

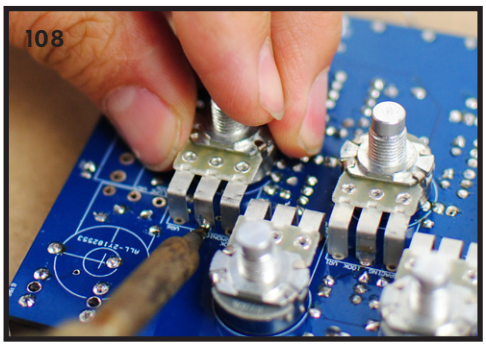
a. Before we install the potentiometers (or "pots") we need to trim some component leads as short as possible so they don't puncture the insulation paper and short out against the bottoms of the pots. The outlines of the pots are drawn on the solder side of the board. Trim down all the leads that are inside these circles (105). Cut them as short as possible, without cutting into the solder joints themselves.

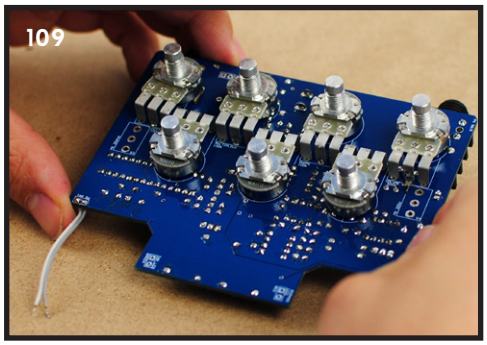
b. Next, if any of your pots have anti-rotation tabs, break them off now. (Not all pots come with these tabs.) Use your needle nose pliers and gently bend the tabs out until they snap off (106). Throw the tabs away.

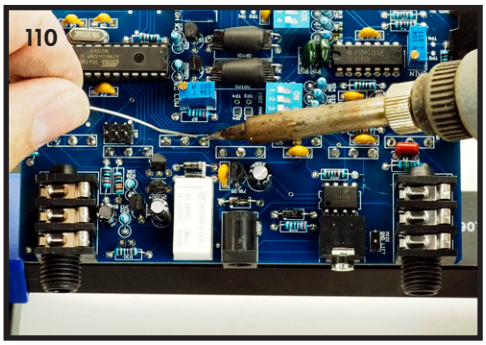


c. There are seven pots in two different values: six 100K pots (VR1-VR6, labeled "B100K") and one 500K pot (VR7, labeled "B500K"). Install the 500K pot at VR7. Hold down the pot from the solder side. Make sure the body of the pot is neatly centered in the round graphic on the board. Be sure the leads are fully inserted into their holes, then tack-solder in the center pin from the solder side (107). Repeat with the other six pots (108). Double-check to make sure the pots are lined up properly and the pins are fully seated into their holes, then go back and finish soldering them in from the component side (109,110).



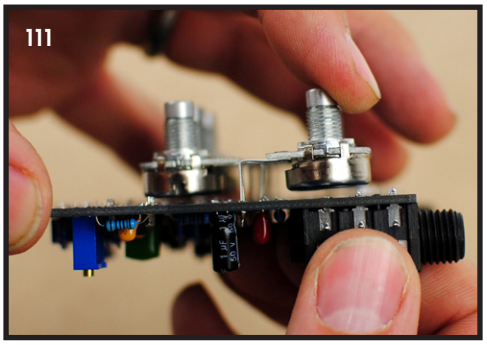


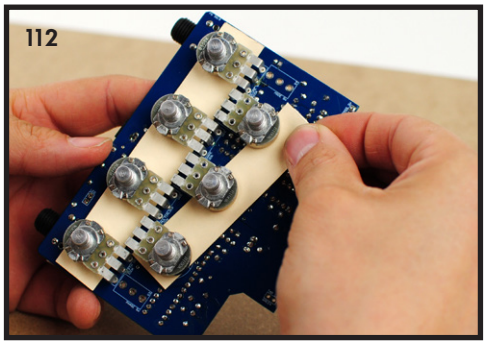




d. Gently bend the potentiometers slightly up-right, until their back is parallel with the PCB (111).

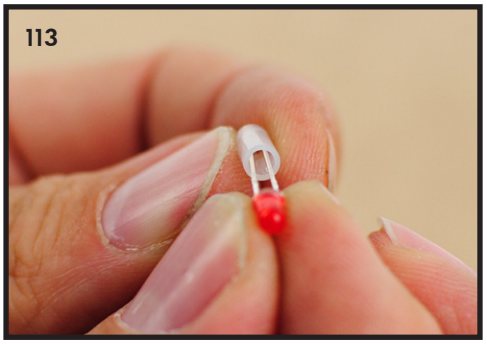
e. Cut the strip of insulation paper into two strips: 4.5″ (11.5cm) and 3.25″ (8.5cm). Slide the strips under the pots to help insulate them from the leads on the PCB (112).

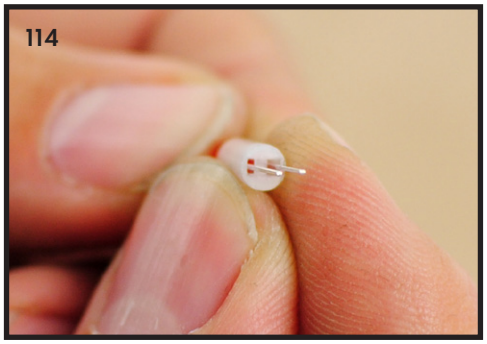




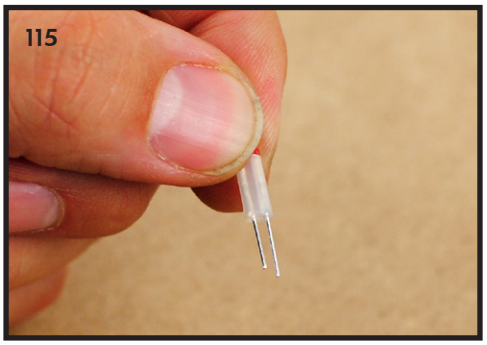
18. 3mm LEDs: The BYPASS and POWER LEDs are red and the TAP LED is green. We use 9mm standoffs to place them accurately above the PCB.

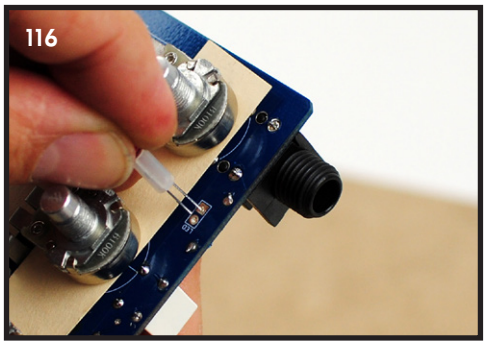
a. Insert the leads of a red LED into the open end of a standoff (115). Poke the leads out through the two holes at the other end (113,114).





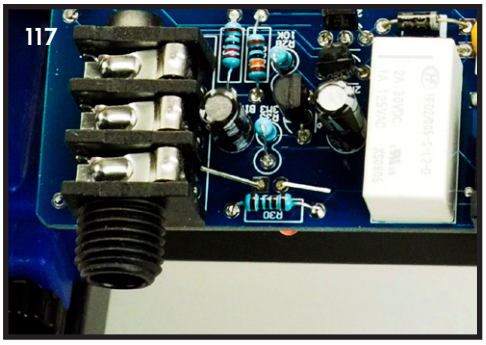
b. Insert the leads into the LED location on the solder side of the board. The long lead goes into square pad, otherwise it won't work! Press down on the LED so the standoff is tight against the surface of the board. Hold the LED perpendicular to the PCB (116).

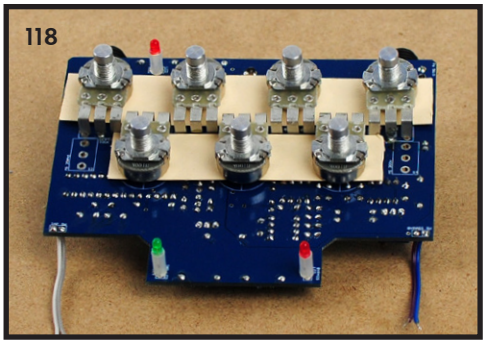




c. Bend the leads out on the component side of the board(117). Solder and clip the leads.

d. Repeat with the other red 3mm LED at the BYPASS LED location, and with the green 3mm LED at the TAP LED location (118).

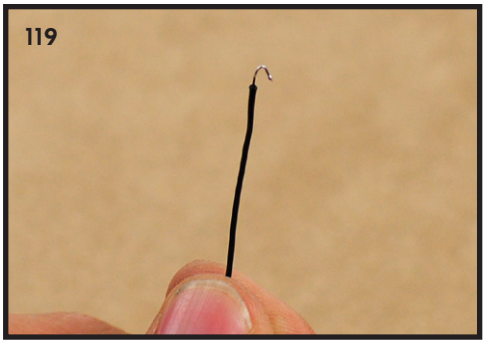


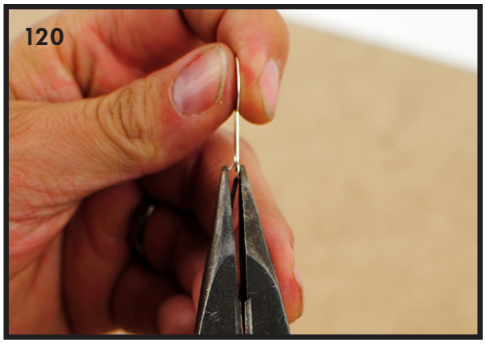


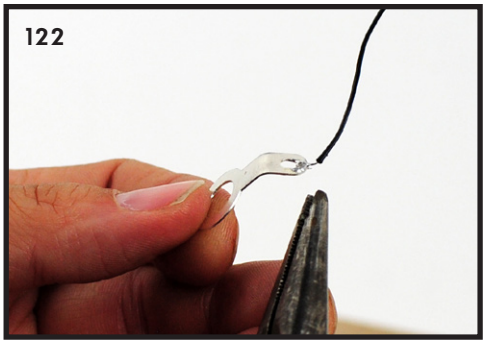
19. Ground Lug: The ground lug electrically connects the chassis to the circuit ground.

a. Solder the single remaining wire to the ground lug tab. Bend a little hook in the wire with your pliers and hook it to the tab hole (119). Use your pliers to crimp it down on the tab (120) and then solder it in place (121).

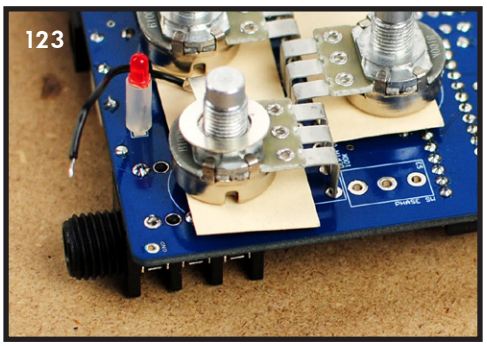
b. Bend the end of the tab down as in the picture (122).

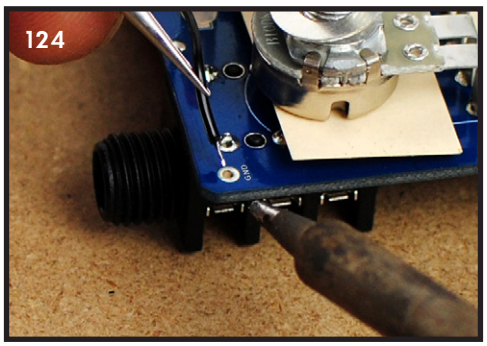




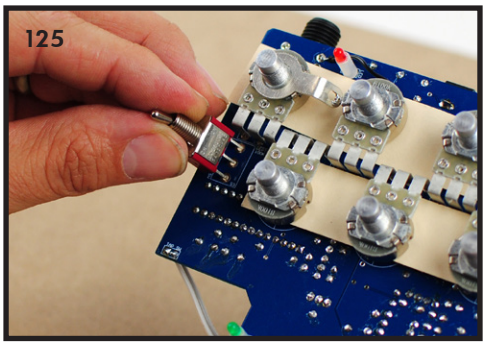


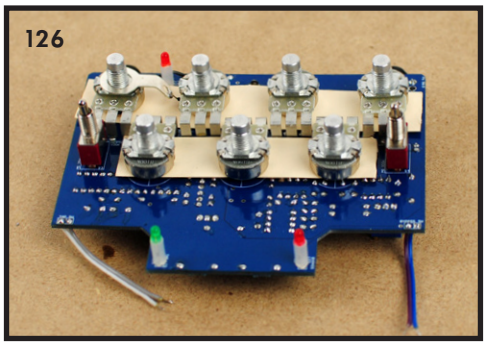
c. Place the solder lug on VR3 (the DEPTH pot) with the bend part of the lug facing down (123). Guide the wire around the power LED and into the ground hole in the corner of the PCB, near the input jack. Solder in place on the component side (124).





20. Toggle Switches (S3, S4): Remove all the washers and nuts from both toggle switches and set aside. Place the switches in their places at S3 and S4 (125,126). We'll solder them in once the PCB is in the chassis.





Congratulations!

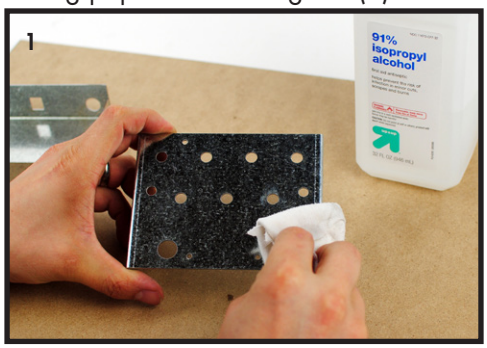
Your Quaverato PCB is all done. It's not quite ready to plug in, though. You still must mount it in its chassis and connect the footswitches before you can power it up and begin the calibration process. Be of good cheer! You are almost done with that soldering iron! Then you can go wash up and enjoy a cool drink.

PUTTING IT ALL TOGETHER

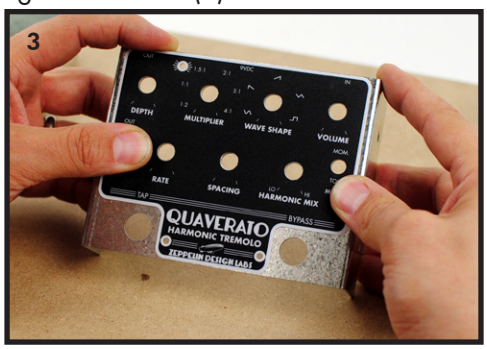
1. Labels:

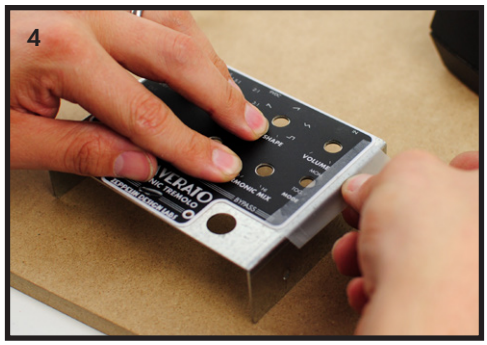
   a. Clean the outside of the chassis with some isopropyl alcohol and a rag (1).

   b. Carefully remove the perforated holes from the label. Make sure not to peel off the label's backing paper when doing this (2).

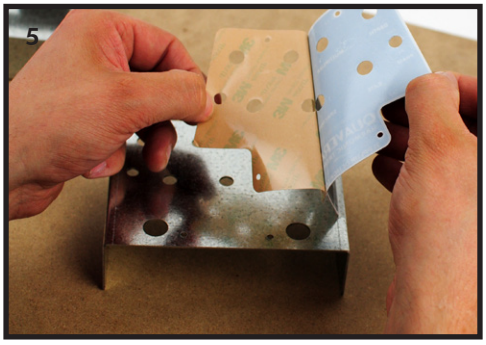


   c. Line up all of the label holes with all of the chassis holes (3). Securely tape down just one edge of the label (4).

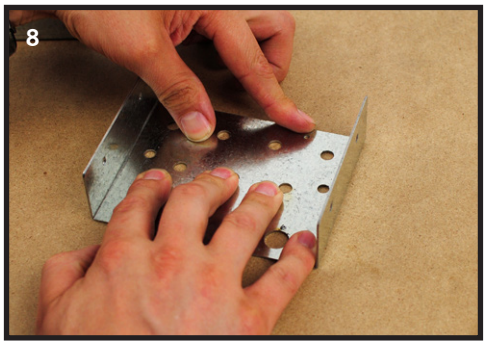




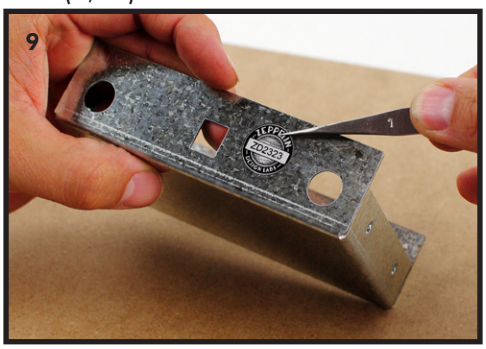
d. Lift up the free side of the label and carefully peel off the backing paper, making sure that none of the adhesive is removed with the paper (5). Gently and carefully roll the label back onto the chassis, starting from the taped side (6).

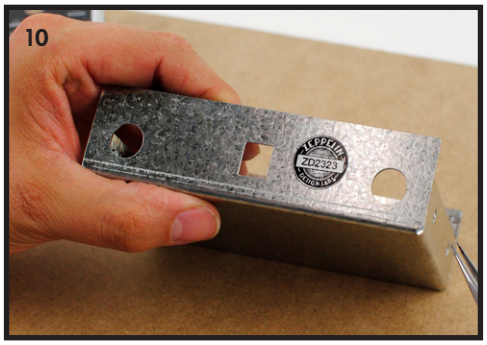


Press the label down as you go (7). Remove the tape, turn the chassis over and push the label down on the clean, flat surface of your table or bench (8).



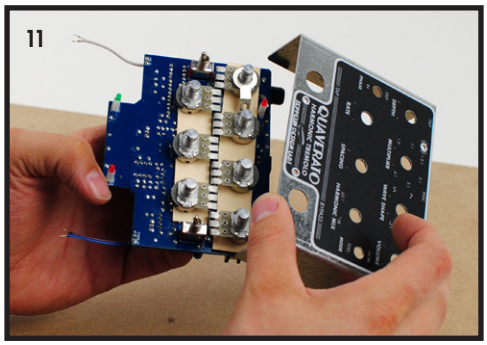
e. Stick the serial number label to the bottom part of the chassis to the right of the power jack cutout (9,10).

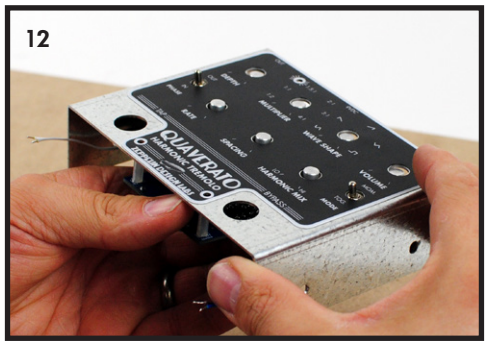




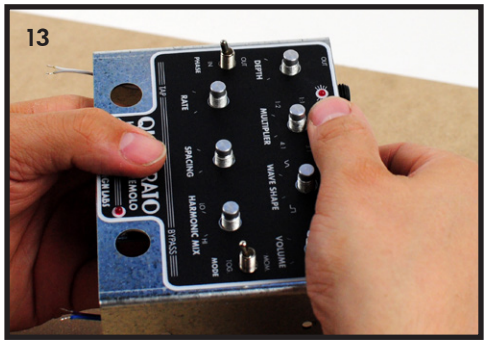
2. Install the PCB:

   a. With the toggle switches (S3, S4) not soldered in yet, place the chassis over the PCB (11,12) (continued).

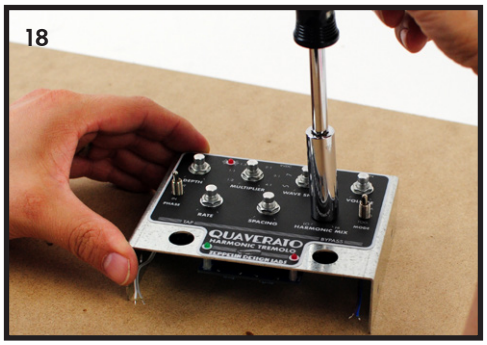




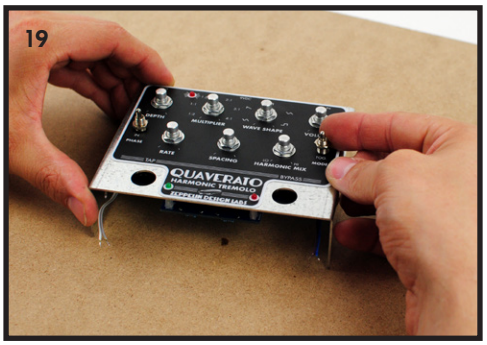
You may have to gently bend the pots and toggle switches a bit to get them all to slide in (13). Keep working it until all three LEDs are protruding through the chassis (14).

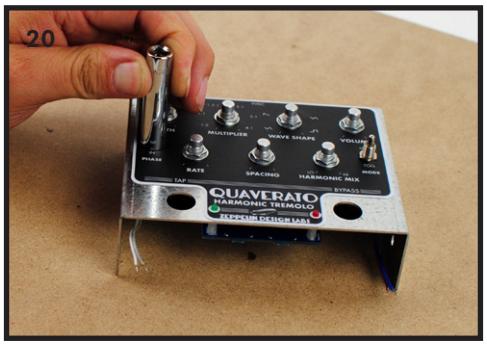


b. Place a washer and a nut on each of the seven pots (15,16,17). Tighten the nuts down with a 10mm socket, or a snub-nose pliers (18).



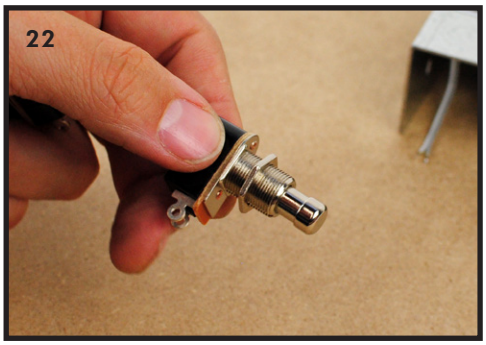
c. Thread a nut onto each of the toggle switches (19). Use an 8mm socket to very gently tighten these down – but no more than finger tight (20)! Do not over-tighten these; you could actually pull the threaded shaft out of the switch body!



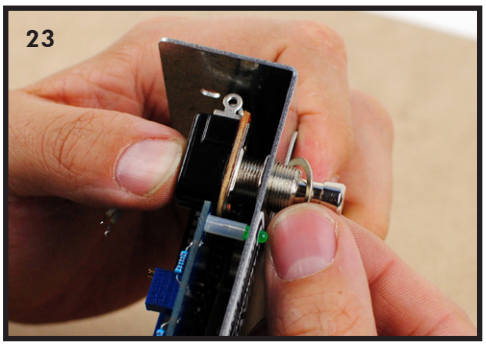


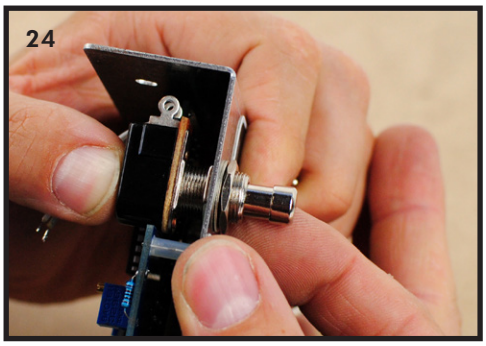
3. Footswitches:

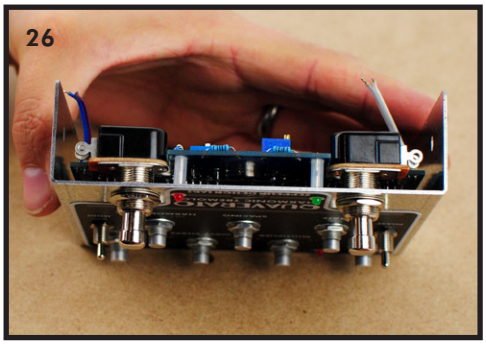
a. Remove the top nut and washer from each of the footswitches (21). Thread the bottom nut about half way up on the threaded shaft (22).



b. Install the footswitches in the chassis as in the picture, with the lugs facing out toward the sides (23). Add a washer and then a nut to each switch (24). Use a 9/16" (or 15mm if you must) socket to tighten the nuts down securely (25,26).

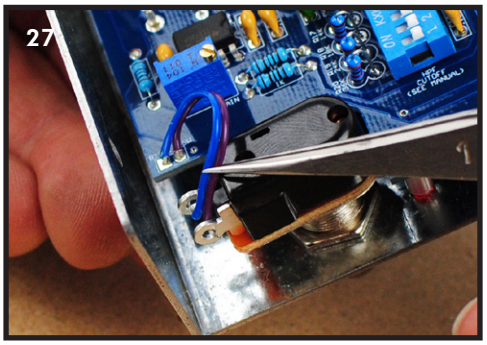


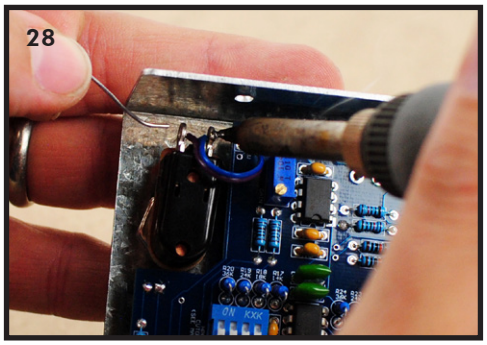




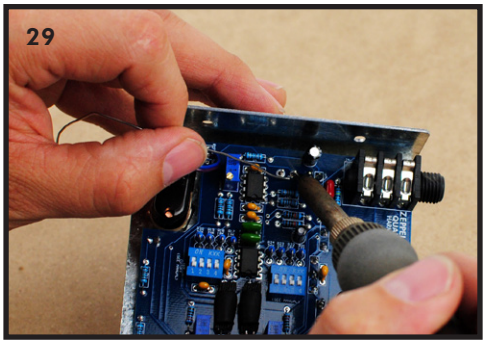
4. Solder everything in place:

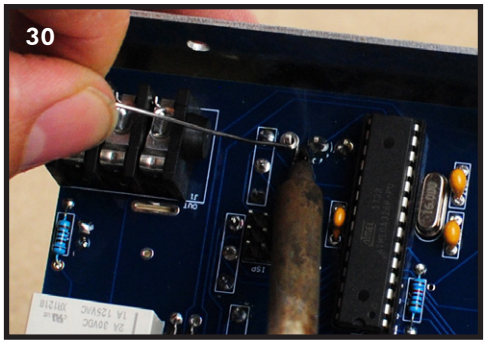
   a. Solder the wire pairs to the footswitches. Each wire goes to one of the footswitch lugs; it does not matter which one (27,28).



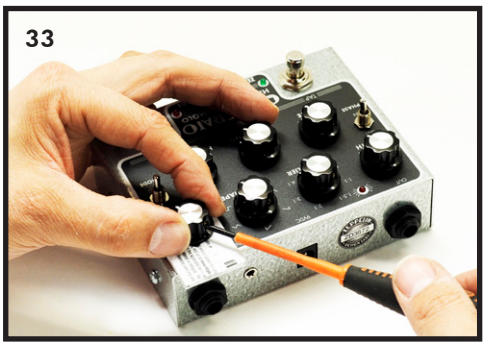


   b. Solder the toggle switches to the board (29,30).





5. Knobs: Before installing any of the knobs, file off any plastic burrs around the edge and loosen the set screw. Turn all the pot shafts fully counterclockwise. Place your knob setting jig (see Appendix A) on the chassis around the Rate pot. Slip the knob onto the RATE pot's shaft (31). Align the white stripe on the knob with the counterclockwise line on the label. Tighten the knob set screw. If you are not using a knob setting jig, hold the knob off the surface of the chassis about 1/8" (3mm) while you tighten the knob set screw. Test to see that the knob stripe lines up nicely with the label markings with the knob turned to both extremes. Continue with the SPACING and HARMONIC MIX pots (32). Then move on to the top row of pots (33,34).



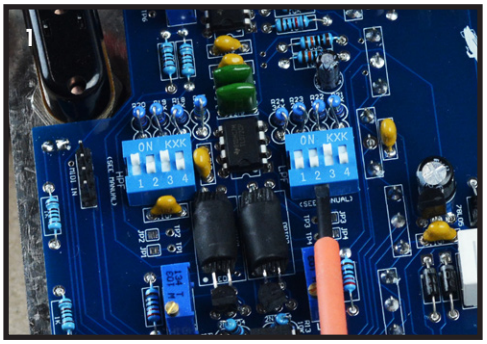
That's it!

You are done building your Quaverato Harmonic Tremolo Pedal. After you take a break (and I take a nap), we'll set up and calibrate the pedal, then finish assembling the chassis and start to jam.

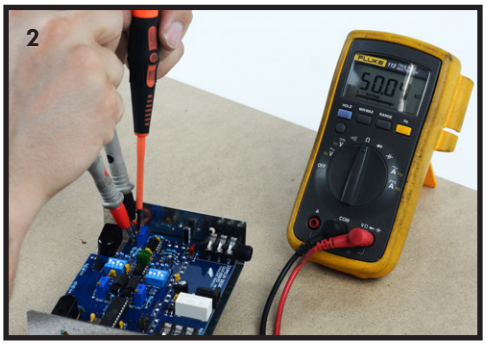
CALIBRATION AND SETUP

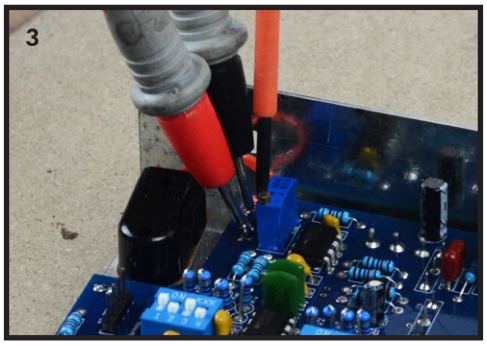
We will now set up the Quaverato to work properly. This process consists of setting the DIP switches and getting a starting point for the trim pot's optimal settings.

1. Set cutoff filters: The first step is to set the high pass and low pass filters to their desired cutoff frequencies. "Table 2: DIP Switch Crossover Frequencies" on page 49 shows the list of frequencies obtainable with the DIP switches. The same DIP switch settings apply to both the LPF and the HPF. To start out, we will set the LPF to 401Hz (1, 2, 4 off; 3 on) and the HPF to 600Hz (1, 3 on; 2, 4 off) (1). We encourage you to play around with these cutoff frequencies to see if any other settings suit your taste better, but first let's test the pedal with these cutoff frequencies.



2. Set GAIN trim pot: While measuring the resistance across TP5 and TP6 with your multimeter, use your small flat head screwdriver to turn the GAIN trim pot (VR10) until the meter reads 40K ohms (2,3). Turning the pot counterclockwise makes the resistance rise, clockwise makes the resistance fall. This is value is just a starting point and we expect you will want to adjust this once you are using the pedal.

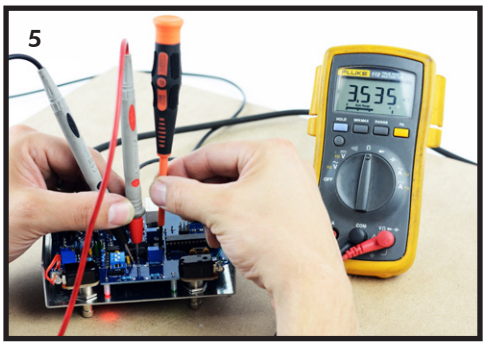


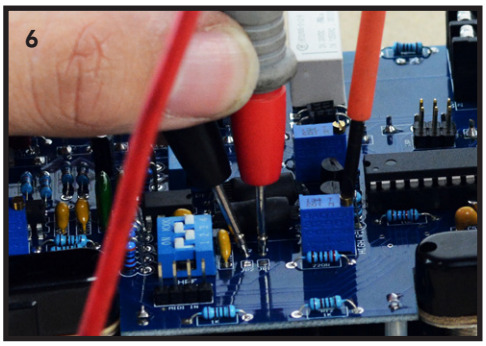


3. Set HIGH and LOW trim pots: In this step we will set the ratio of low and high frequencies.

   a. Plug in the DC power cable (center negative) and insert a 1/4" instrument cable into the IN jack. The red power LED should come on. (If not then something is installed backwards: check the power LED, D1 and C14.) Turn the DEPTH knob fully counterclockwise so the LFO is not pulsing. Press the BYPASS foot switch so the red bypass LED is on (4).

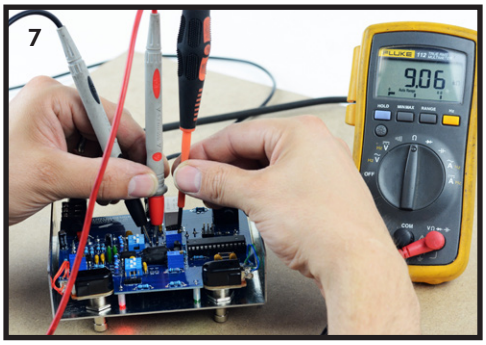
   b. While measuring the resistance across TP1 and TP2 turn the HIGH trim pot (VR8) until your meter reads around 3.25K ohms. Clockwise lowers the resistance and counterclockwise raises the resistance (5,6).

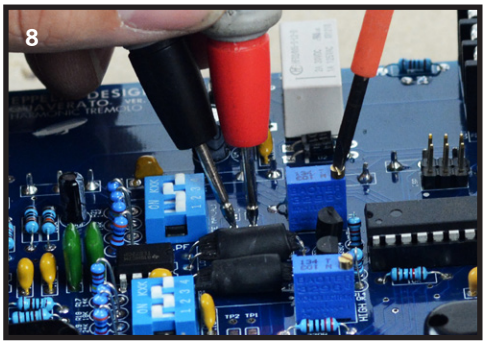




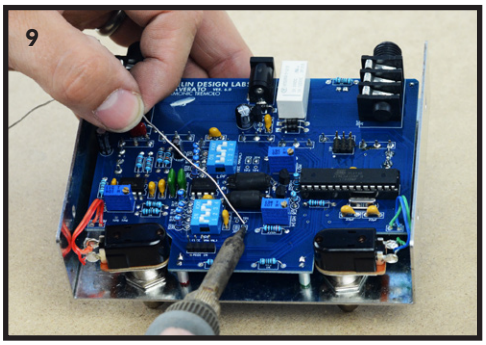
   c. While measuring the resistance across TP3 and TP4 turn the LOW trim pot (VR9) until your meter reads around 5K ohms (7,8).

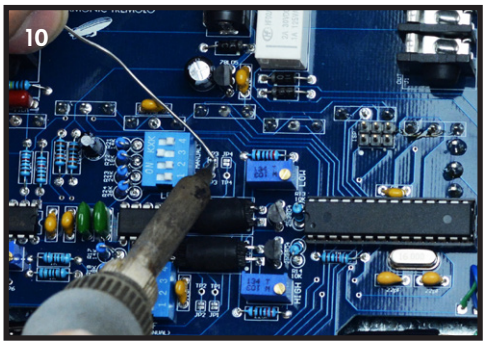
   Once again, these initial values are just starting points and we expect you to adjust these trim pots to get the tone you want once you start using the pedal.

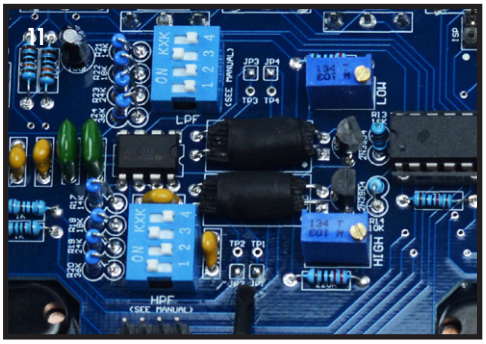


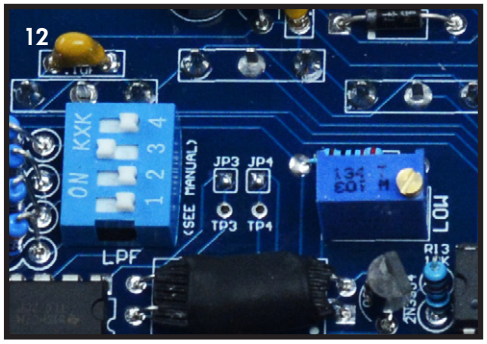


4. Solder Jumpers: Now that the low and high frequency ratio is set we can close the circuit to complete the signal path. Unplug the power. You will notice that JP1, JP2, JP3, and JP4 are made of two tiny little pads. Each of these pairs of pads needs to be bridged, so we will connect them with a little blob of solder. Get a little bit of solder on your iron and heat up both pads on the jumper simultaneously. Add a tiny bit more solder to help everything flow together (9,10,11,12). That's it, your jumpers should now be jumped!









5. Test the sound: Now you can plug your Quaverato into your amp and guitar to test it out and make any final adjustments. Set the VOLUME, SPACING, and HARMONIC MIX knobs to the 12:00 position; set the MODE switch to TOG, turn the DEPTH knob fully clockwise; adjust the RATE knob to a moderately fast LFO speed (13). Toggle the BYPASS footswitch on and off a few times. With these settings, the volume output of the pedal should be roughly the same when the effect is bypassed or engaged. If not, adjust the GAIN trim pot (VR10) until it is. Counterclockwise gives more gain, clockwise gives less gain.

If you would like to change your Quaverato's tone you can adjust the HIGH and LOW trim pots. The values to which we originally set these trim pots (see paragraph 3 above) give, in our opinion, a good balance of highs and lows. This being strictly a matter of taste, you should feel free to play with these pots. You will notice that the highs and lows need to be well balanced to hear the most dramatic effect in the harmonic tremolo mode (PHASE switch set to OUT).

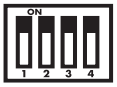
If you would like the tone to be a little brighter then turn up the HIGH trim pot (clockwise) and turn down the LOW trim pot (counterclockwise). If you would like the tone to be a bit darker do the opposite: turn down the HIGH trim pot (counterclockwise) and turn up the LOW trim pot (clockwise). Turn the trim pots just one rotation at a time so you can keep track of how much you've adjusted them. You can always adjust the trim pots back to their original values if you don't like the results. If you want to measure across the test points to re-calibrate the trim pots, you will need to use some solder wick to remove the solder jumpers.

Also, feel free to play with the DIP switch settings to explore the sonic possibilities of adjusting the crossover frequencies. Table 2 shows the DIP switch settings for various cutoff frequencies. You will get the most dramatic whooshy / phasey harmonic tremolo sounds when the HPF is set to a very high setting and the LPF is set to a very low setting. The further apart they are set, the more dramatic the effect will be. You will also notice that a lot of the high- and mid-range frequencies are more muted in these extreme settings, causing a lot of the sonic clarity to be lost.

With the HPF set to a very low setting and the LPF set to a very high setting you will notice that much of the dramatic phasing harmonic tremolo sounds are much more subdued, but the high-mid clarity is back. We set these filters to 401Hz (for the LPF) and 600Hz (for the HPF) because it gives a good mix of the phasey sound but still lets most of the clarity of the high-mids through.

NOTE: For circuit stability make sure at least one of the switches on each DIP switch is in the ON position at all times.

Table 2: DIP Switch Crossover Frequencies

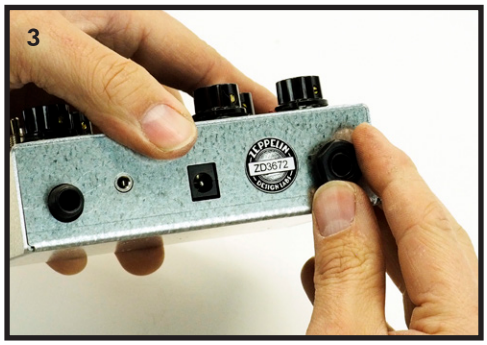
	200Hz
	301Hz
	401Hz
	500Hz
	517Hz
	600Hz
	701Hz
	722Hz
	822Hz
	904Hz
	919Hz
	1000Hz
	1120Hz
	1220Hz
	1420Hz

FINAL ASSEMBLY

1. Slide the top chassis (with the PCB installed) into the bottom chassis (1). Place the 1/4" jacks through the holes in the bottom; also align the DC jack to fit through its square hole, not behind it. You will know the chassis is properly closed when the DC jack is flush with the outside of the bottom chassis (2).

2. Place the plastic nuts onto the jacks, and finger-tighten (3).

3. Screw the chassis together with the four machine screws. Drive them tight with your #2 Phillips driver (4,5).



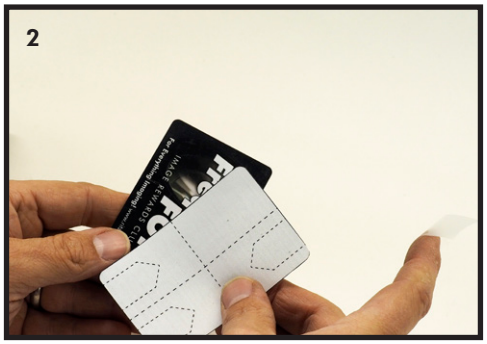
Are you ready to rock? Yes you are!

Plug in a 9V DC center-negative power supply (100 mA or more), stick your Quaverato between your guitar and amp, and prepare to quaver like a leaf in a breeze; flutter like my heart when I see my wife; throb like my head after a late-night Monopoly loss to Colleen. For complete operating instructions and an in-depth look at all of the Quaverato's features, please see the Quaverato Owner's Manual, available for free download at www.zeppelindesignlabs.com.

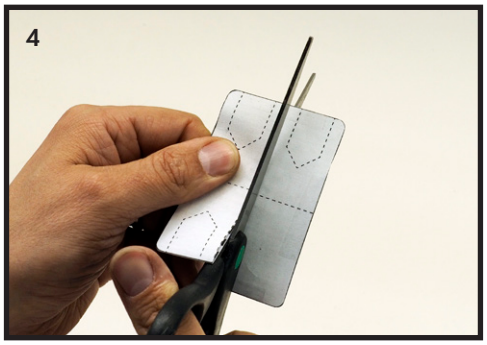
APPENDIX A: KNOB-SETTING TOOL

Using a plastic hotel key card or gift card, you can make a handy jig to help you set your Quaverato's knobs at the proper height. The bottom of the knobs should be set about 2.3 mm above the top of the pedal, and sometimes it's hard to hold the knobs at the proper height while you also tighten the set screws. This jig will help with that. You'll need a plastic gift card that is about 0.8 mm thick.

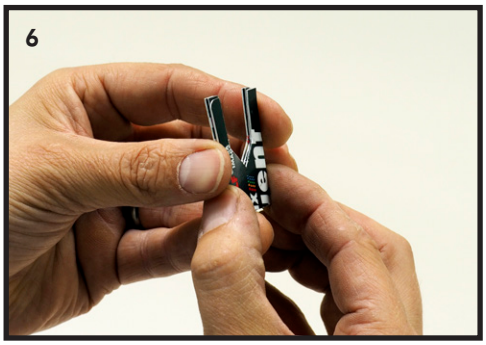
1. Print out the template at full scale and then cut it out on the outside of the black line (1,2).

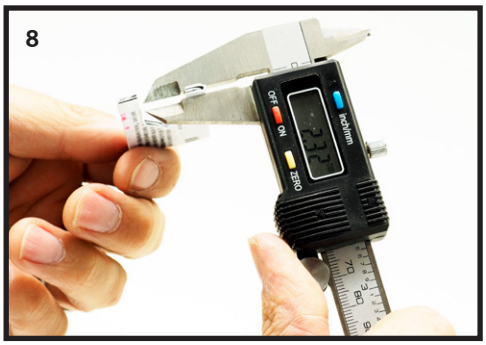


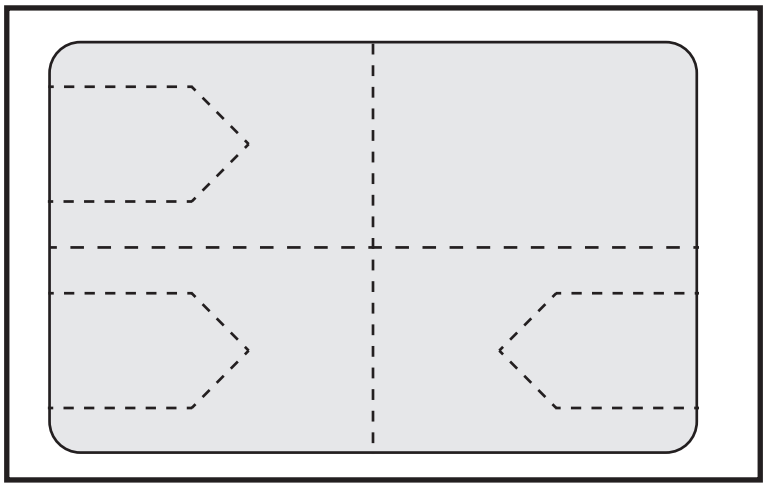
2. Use some transparent tape to hold the template to the face of a gift card (3).
3. Use a heavy, sharp scissors to cut the card on the dotted lines (4).



4. You should end up with 3 identical shapes (5). Discard the piece not shaped like the others, as well as the template paper and tape. Make sure you don't have any burrs or rough edges on those pieces and tape them together with transparent tape (6,7). The final piece should be very close to 2.3mm thick in order to set the knobs at the proper height (8).





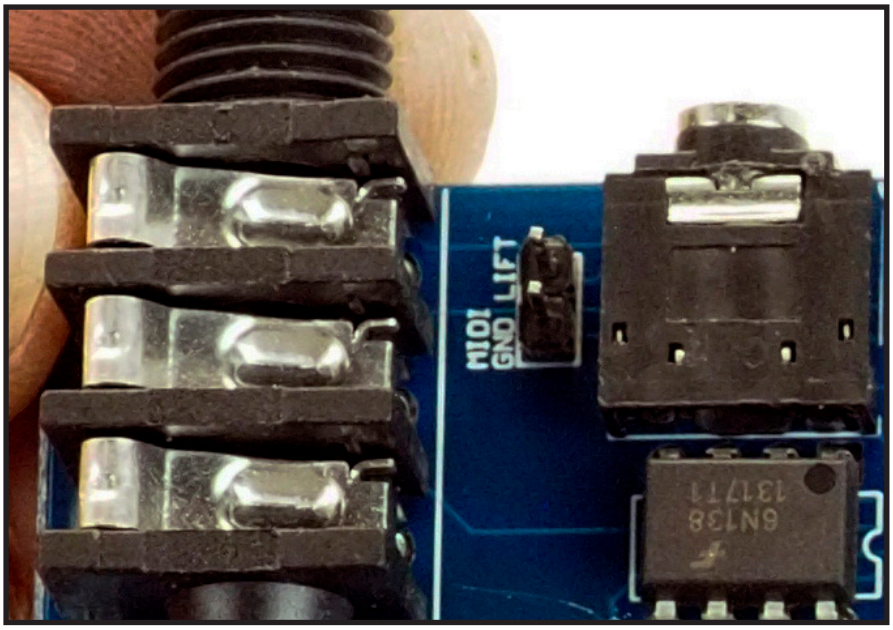


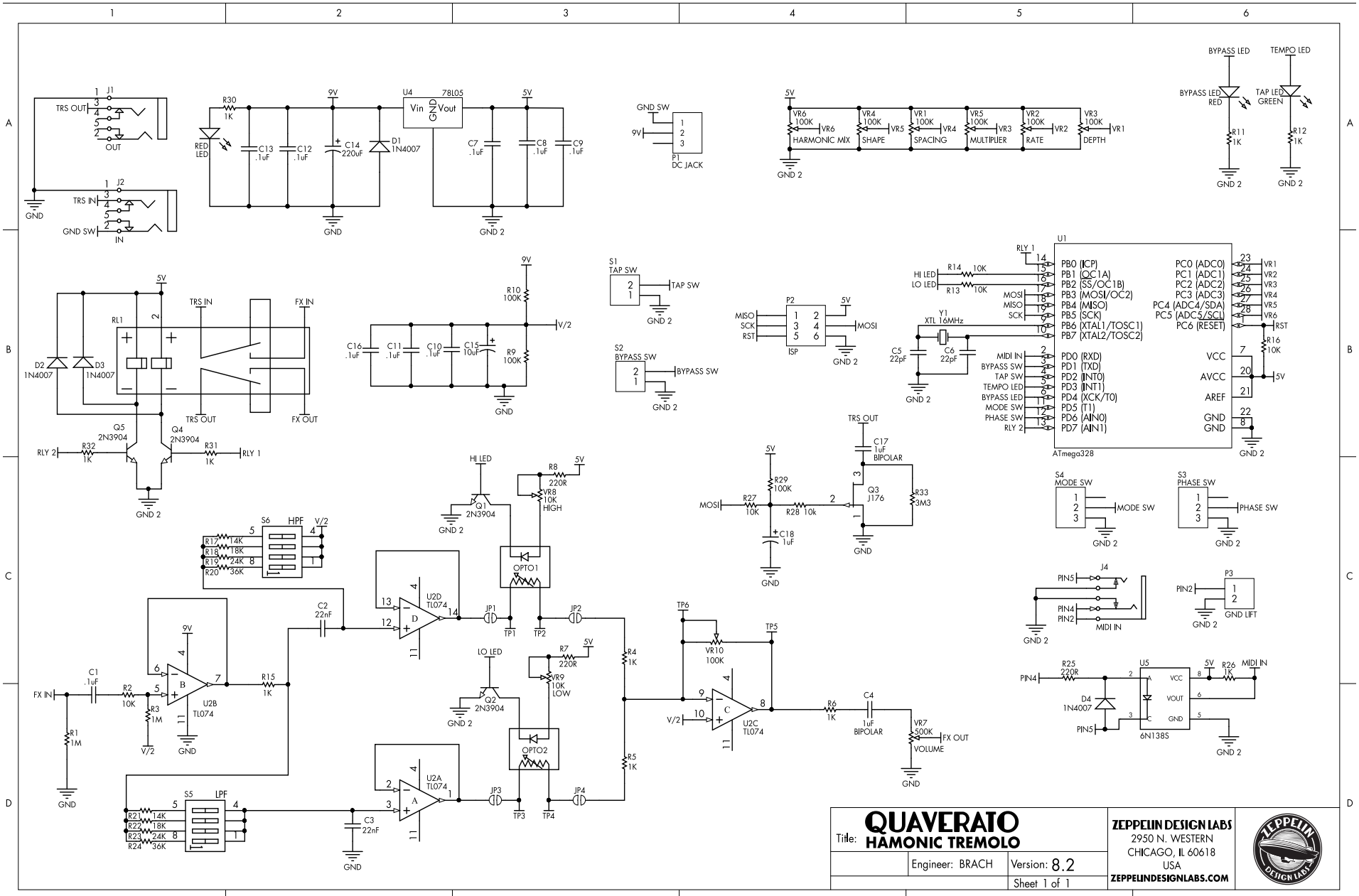
Print this page at 100% with NO SCALING

APPENDIX B: MIDI GROUND LIFT

The Midi ground lift jumper gives you the option of disconnecting to your midi controller's ground circuit from the Quaverato's ground circuit. Depending on the type of midi controller you are using and how it is powered ground loops can occur. This can happen when your Quaverato (or amplifier) and your midi controller are powered using a different circuits. A small current (often times consisting of digital switching noise) can travel through the ground path from one device to the other, causing an audible noise in the signal path. When this happens, removing (lifting) the ground of the midi controller can really help to clean up this noise.

In general when you use midi with your Quaverato, we suggest keeping the ground lifted, as in keeping the jumper off the header. This is usually the best position to keep noise out of the audio path. On some rare occasions when using midi your audio signal may be noisy, even when the ground is lifted. In which case, try connecting the jumper to see if that helps, but keep it off normally.




QUAVERATO
HAMONIC TREMOLO
Engineer: BRACH
Version: 8.2
Sheet 1 of 1
ZEPPELIN DESIGN LABS
2950 N. WESTERN
CHICAGO, IL 60618
USA
ZEPPELINDESIGNLABS.COM
U1
ATmega328
U2A TL074
U2B TL074
U2C TL074
U2D TL074
U4 78L05
U5 6N138S
Q5 2N3904
Q4 2N3904
Q1 2N3904
Q2 2N3904
Q3 J176
RL1
D1 1N4007
D2 1N4007
D3 1N4007
D4 1N4007
OPTO1
OPTO2
S5 LPF
S6 HPF
S1 TAP SW
S2 BYPASS SW
S3 PHASE SW
S4 MODE SW
P1 DC JACK
P2 ISP
P3 GND LIFT
J1 OUT
J2 IN
J4 MIDI IN
Y1 XTL 16MHz
VR1 DEPTH
VR2 RATE
VR3 MULTIPLIER
VR4 SHAPE
VR5 SPACING
VR6 HARMONIC MIX
VR7 500K VOLUME
VR8 10K HIGH
VR9 10K LOW
VR10 100K
BYPASS LED RED
TAP LED GREEN
TEMPO LED
HI LED
LO LED
RED LED
TRS OUT
TRS IN
GND SW
FX IN
FX OUT
MIDI IN
GND
GND 2
9V
5V
V/2

ZEPPELIN

DESIGN LABS