QUAVERATO
MIDI MOD

Assembly Instructions

ZEPPELIN DESIGN LABS

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

QUAVERATO MIDI MOD

Assembly Instructions

WHAT YOU WILL NEED

Here's everything you will need to upgrade your Quaverato with the MIDI mod kit. In this assembly manual, we are assuming:

1. you have a Quaverato Harmonic Tremolo pedal, as a kit underway, or already assembled and in-service;
2. you know how to solder and how to use tools safely; but
3. you have no other particular experience with electronics.

SOFTWARE

Quaveratos shipped after November 16, 2018 come loaded with the latest MIDI-enabled software (version 2.3.6 or higher). If you are modding an older unit, you will need to update the software. Updates are easily obtained with the free ZDL Updater app for PC. You will also need a little piece of hardware called a USB-Tiny Programmer. See "UPDATING THE SOFTWARE" on page 17.

TOOLS

1. Soldering Iron (not a soldering gun, or a "cold heat" iron), good quality, 15-50 watt, with a good medium or small sized tip, conical or "screwdriver" shape. One with a temperature control and a stand is best. Also, make sure you have a way to clean the tip: a wet sponge or a dry solder cleaning pad.
2. #2 Philips Screw Driver
3. Flush cutters or small diagonal cutters
4. Needle-nose pliers
5. Clamp or vise
6. Solder sucker or solder braid – optional, but very handy if you have to remove / repair any components!
7. A center punch, awl, or nail-like sharp object
8. Drill press or other small power drill capable of drilling through the chassis
9. Drill bits: 1/8", 3/16, and 1/4" (3 mm, 4 or 5 mm, 6 mm)
10. Deburring tool or large drill bit around 7/16"
11. 4" length of 2x4 block of wood or similar, to drill into
12. Scissors

SUPPLIES

1. Solder, 60/40 rosin core, the smaller diameter the better (we prefer .032" diameter). Make sure it's good quality; we prefer Kester brand, but most brands will work fine.
2. Masking or painters tape

WHAT'S IN THE BOX

"Table 1: Quaverato MIDI Mod Kit Bill of Materials"" (BOM) is a complete parts list of everything that should be present in your kit, followed by photos of each part. Carefully go through the kit, identifying every part. Compare them 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 support@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. Some of the parts in this kit are fairly common and you might be able to find them at your local electronics or hobby shop, but they can also be ordered directly from us. Contact support@zeppelindesignlabs.com.

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 1: What's In The Box

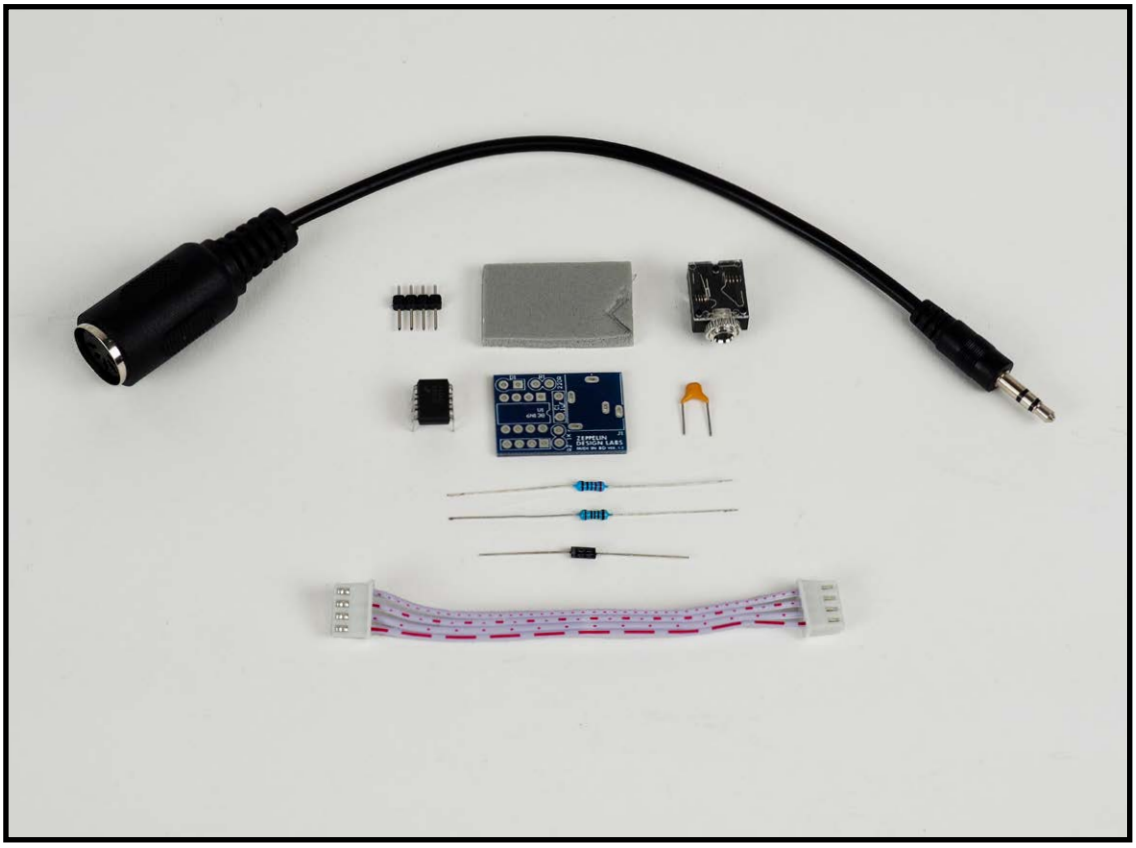


Table 1: Quaverato MIDI Mod Kit Bill of Materials

Part #	Description	Notes	Qty
RS-80-32	Resistor, 220R	R1	1
RS-80-40	Resistor, 1K	R2	1
CP-30-18	Ceramic Capacitor 1uF	C1	1
DI-20-01	1N4007 General Purpose Diode	D1	1
IC-40-13	Optocoupler 6N138 DIP 8	U1	1
HE-20-01	Header, Single Row, 4 Pins	P1	1
CB-06-10	4 Pin Header Cable	Connects to the Quaverato PCB	1
HD-40-41	1/8" TRS Audio Jack	J1	1
PC-10-03	MIDI Mod PCB		1
CB-30-10	Female MIDI to 1/8" Male TRS Adaptor		1
TP-20-10	Foam Tape		1

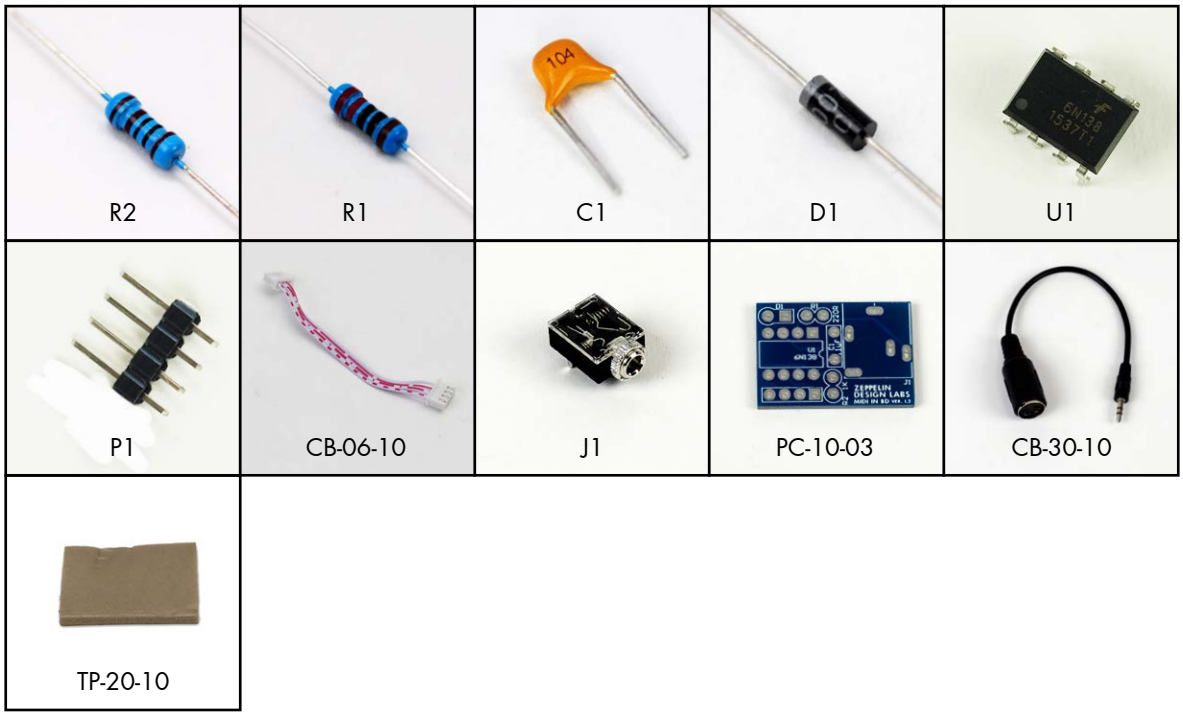
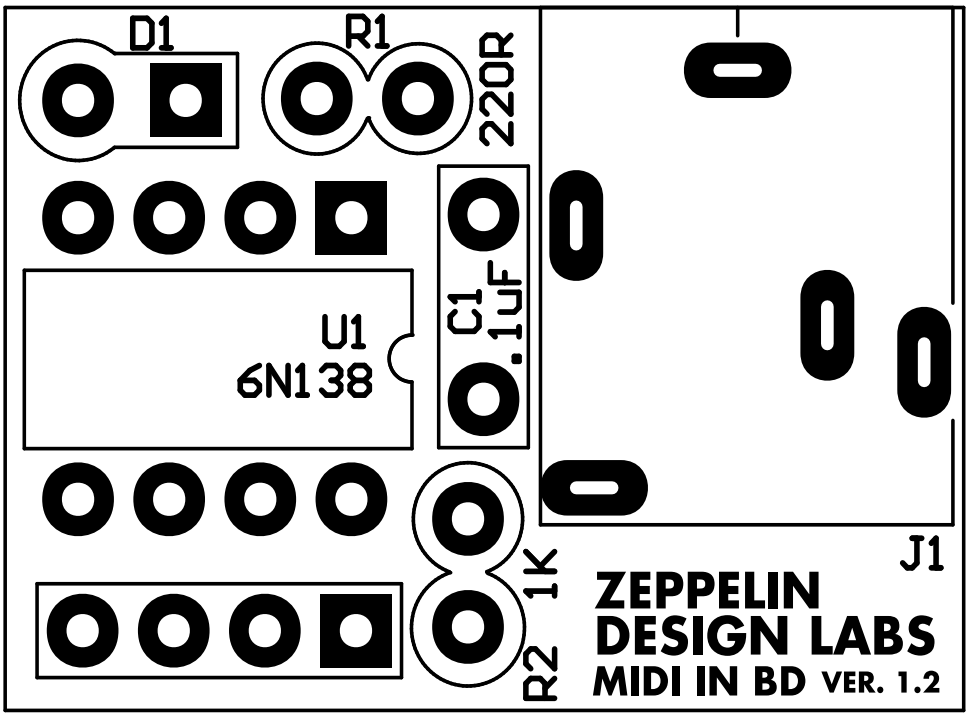


Figure 3: Component Values and Locations



POPULATING THE PRINTED CIRCUIT BOARD

In assembling this kit, 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 cover 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 onto your face.

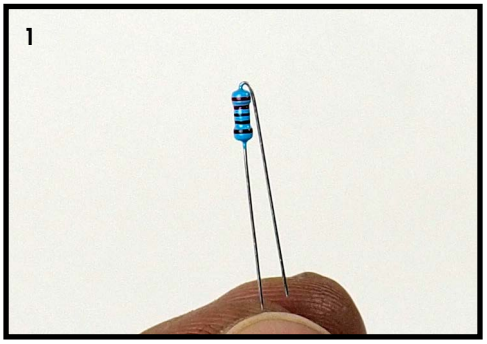
CAUTION: Solder residue usually contains lead, which is poisonous if you ingest it. Do not breathe the fumes, do not eat the supplies, wash your hands after you handle solder, and sweep and wipe up your work space after EVERY USE.

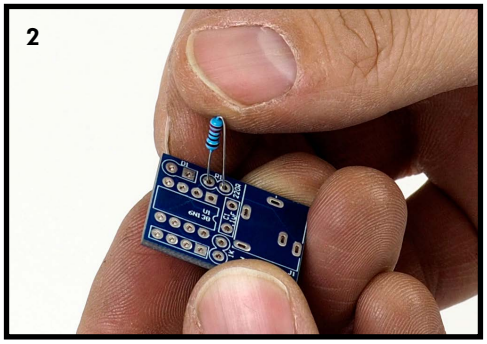
The printed circuit board (PCB) holds the components in this circuit. All of the components will be installed on the "component side" of the board, which is the side that has the part numbers on it. The other side of the board is called the "solder side", which, as the name implies, is the side on which the legs of the components are soldered to the board. 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 of its holes.
4. Flip the board over solder-side-up, and solder all of the components in one pass.
5. Clip the leads off with small diagonal cutters, right at the solder joint.

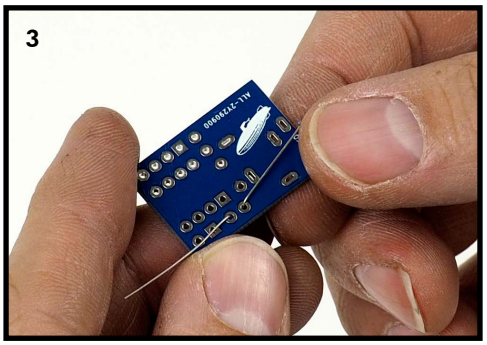
Let's begin!

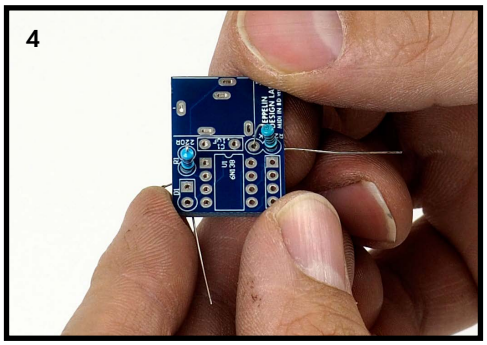
4. Resistors: The values of resistors are given by a series of colored stripes on their body. There are several tutorials online describing how to decode these stripes, but we will identify each resistor for you by simply naming the stripe colors, and giving you the value and the part number. Fortunately, there are only two resistors in this mod kit, so it shouldn't be too difficult to tell them apart. "Figure 3: Component Values and Locations" is a good reference for all the components. 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. Resistors are not polarized, meaning they can be installed in their holes in either direction. It doesn't matter which lead goes in what hole. The two resistors in this circuit are installed standing upright on the PCB so bend one lead nearly parallel with their bodies and install them standing up (1,2).





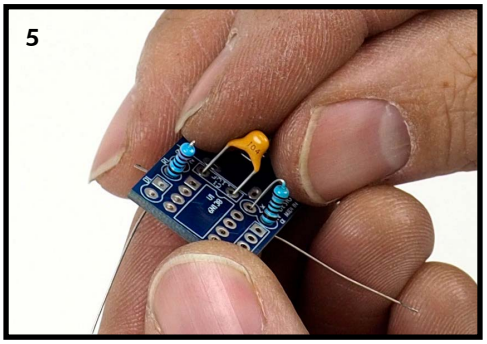
a. Start with the 220 ohm resistor (R1). This resistor is labeled RED, RED, BLACK, BLACK, BROWN. Bend the lead as described above and install it on the circuit board as in the picture (2). Bend the leads out on the back to keep the resistor from falling out of the board (3).



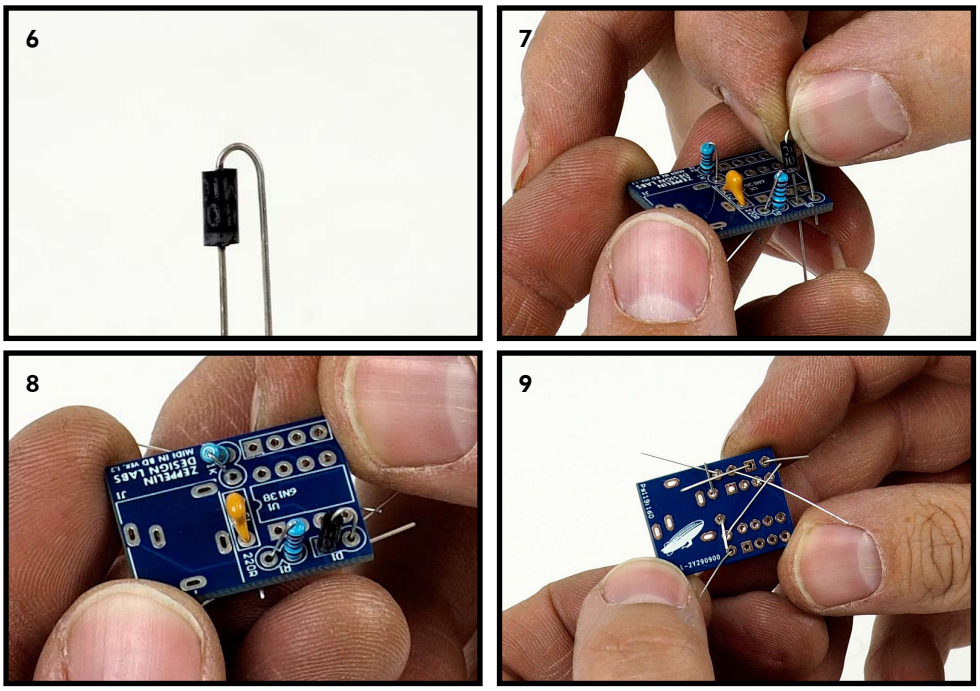


b. Next do the 1K resistor (R2), labeled BROWN, BLACK, BLACK, BROWN, BROWN. Compare to its picture in the BOM. Find it's location on the circuit board; install and bend the leads the same as the previous resistor (4).

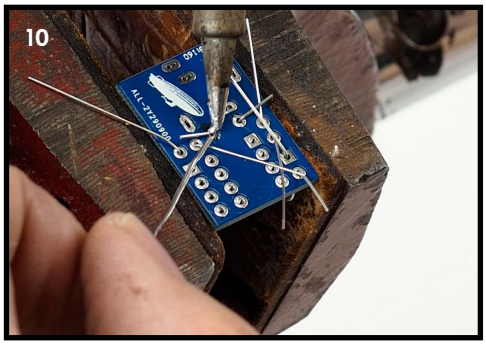
5. Capacitor (C1): This circuit only has one ceramic capacitor. It doesn't matter which direction the capacitor is installed. Place the cap in its holes and bend the leads out on the back (5).

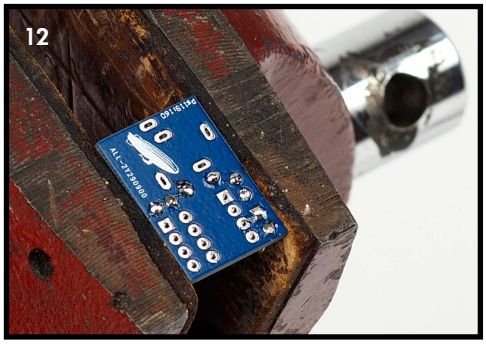


6. Diode (D1): The diode is installed standing up like the resistors, but it is polarized, meaning it matters which direction it is installed. Notice the white stripe on one end of the body of the diode (6). The lead closest to this white stripe goes into the hole with the square pad. Bend the other lead nearly parallel with the body and install the diode as in the picture (7,8). Bend the leads out on the back (9).



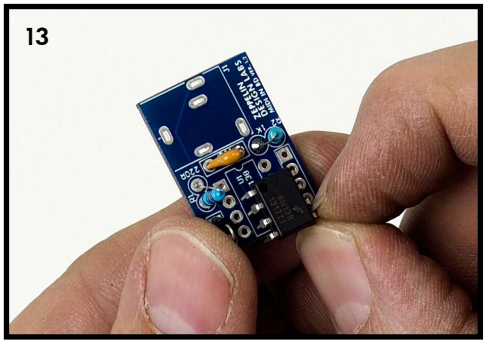
7. Solder all the leads of the resistors, capacitor, and diode (10). Use your flush cutters to snip off the leads at the solder joint (11,12).

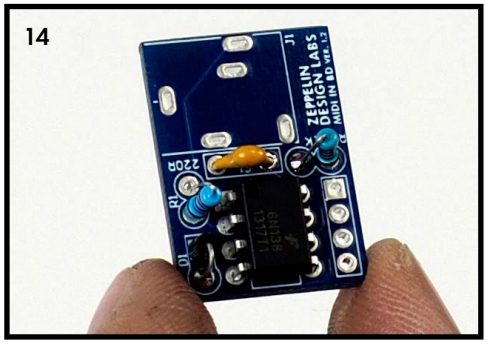




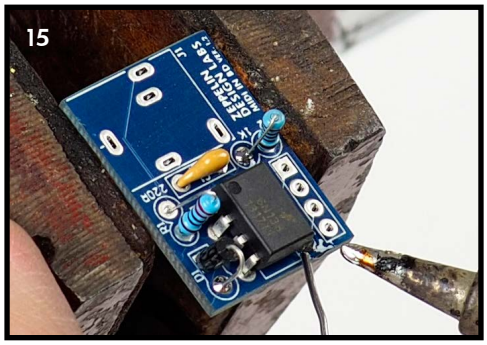
8. Optocoupler (U1): The optocoupler is what we call an IC or integrated circuit chip. When soldering this IC, try to keep it from getting too hot. Most chips have a temperature threshold that shouldn't be exceeded. As a rule of thumb don't keep your iron on any leg longer than two seconds, and make sure the chip stays cool enough to touch. If you find you have a hard time keeping the IC from getting too hot, just solder one leg at time and let the chip cool off before proceeding to the next leg.

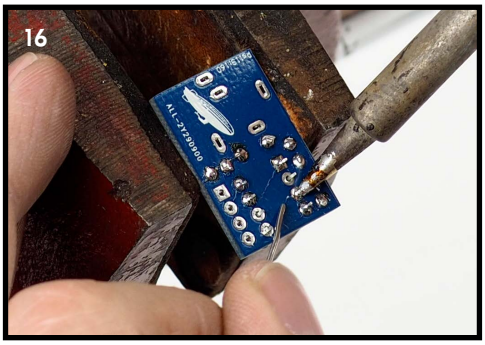
   a. The IC has a specific orientation. If you install it wrong, the MIDI mod will not work. These optocouplers have a dot near one corner. The pin nearest this dot is pin 1, and it goes into the hole with the square pad. Make sure it is installed as in the pictures (13,14).



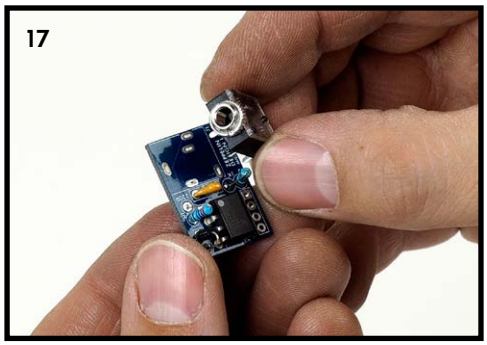


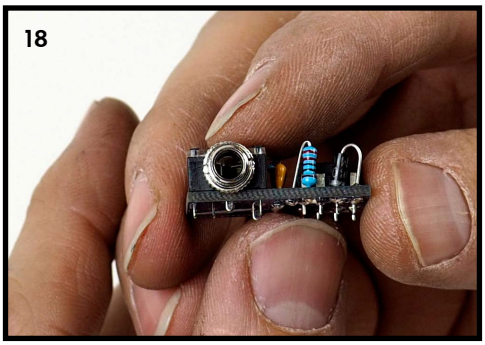
b. The leads on this IC are too short to bend out to hold the chip in place. A good way to install this component is to tack in one pin from the top of the PCB and then turn the board over to solder the rest of the pins (15,16). Don't forget to re-solder the tacked pin from the bottom. The pins are too short to snip off so you can leave them.

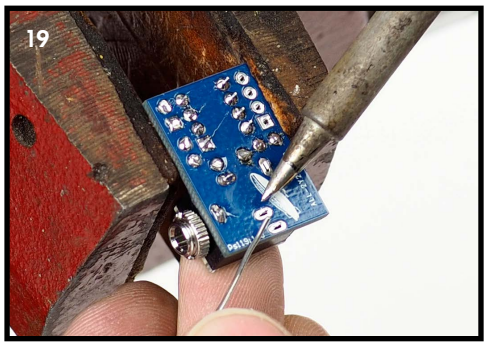




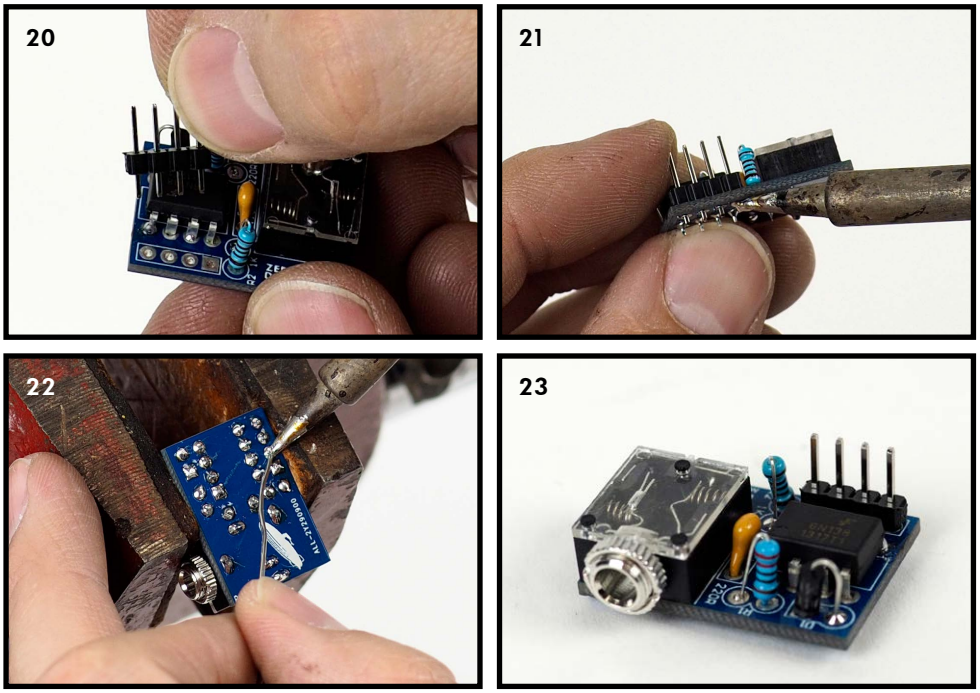
9. 1/8″ TRS Audio Jack (J1): Install the 1/8″ jack at J1. Make sure all the pins on this jack are straight. Line up the pins to the slots in the PCB and press the jack snug to the board (17,18). While holding the jack in place, flip the board over and solder the pins (19).







10. 4-Pin Header: The short pins go through the board; the long pins point up (20). Make sure the bottom of the header is flat against the circuit board. Tack one pin on the end of the row down with solder while you hold the header in from the top (21). Once one pin has been tacked on, you can solder the other pins in place (22). Remember to properly re-solder the “tacked on” pin.

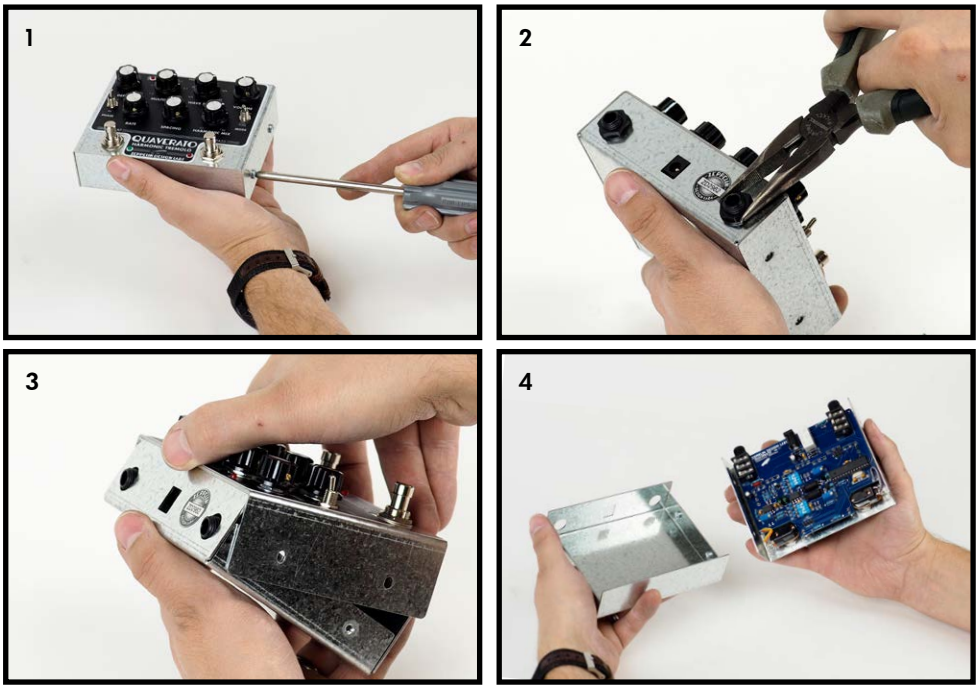


The PCB is now done. Put it aside while we modify the chassis of the Quaverato.

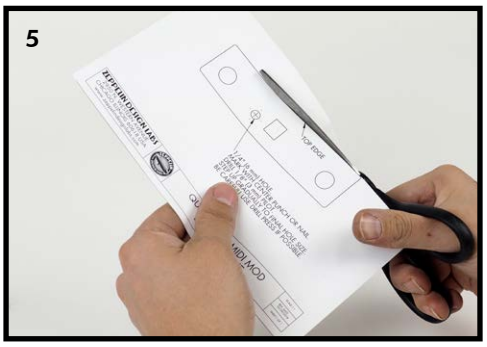
THE CHASSIS

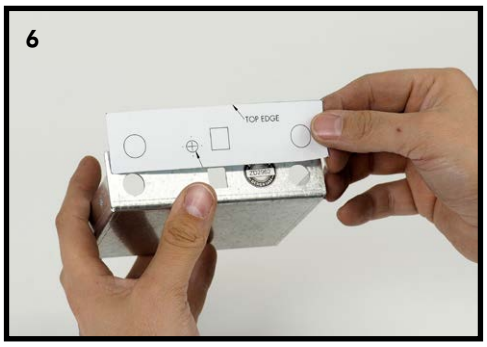
If you are modding a complete Quaverato that is already in service, you will have to open it up first. If you are working on the Quaverato kit, your pedal is still in pieces; skip to 2.

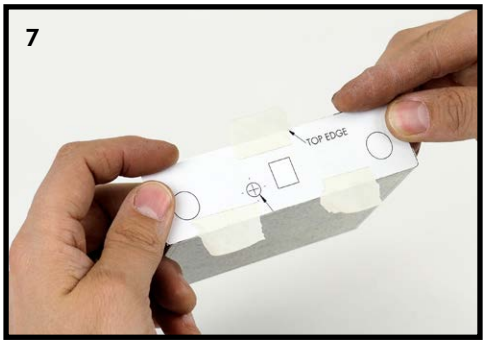
1. Use your #2 Philips screwdriver to remove the two screws in either side of the pedal (1). Use your pliers to remove the two plastic nuts from the 1/4" jacks (2). The bottom half of the chassis is now loose and you should be able to slide the two halves of the chassis apart (3,4).

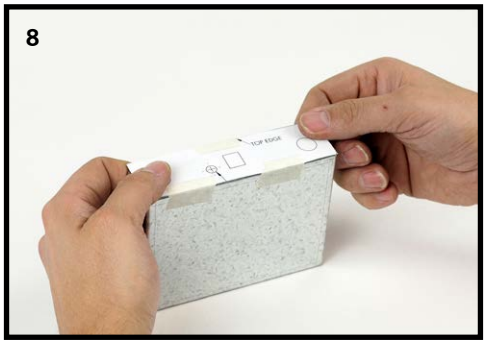


2. Cut out the paper drilling template by cutting around the solid line (5). Align the template to the front of the bottom half of the chassis (6). As you hold the template in place, hold it up to the light to align the holes in the chassis to the hole markings on the paper. The edge of the template should line up with the edge of the chassis. While making sure the template doesn't move from that position, use a few pieces of tape to hold it in place (7,8).









3. With a center punch or a sharp, nail-like object (such as a nail), punch a divot in the chassis at the location indicated on the template (9). If you use a hammer and nail, do not stand the chassis up on top of the table as we did here. You might badly bend the chassis. Instead, hang the chassis over the edge of your work bench, so that the chassis face is supported directly by the table. Once you have made your mark, remove the template (10).

4. In this step we will drill a 1/4″ (6 mm) hole in the chassis. For this process you need to be very careful. This is by far the most dangerous part of this whole operation. Use work gloves and safety goggles. If you have a drill press, use it. If you are not qualified to drill a hole in the chassis, do not attempt it. Get a qualified friend to help you.

   In this process, we will not try to drill out the 1/4″ hole all at once. We will first drill a small hole through the chassis and then use larger bits to increase the hole size.

   a. Place the block of wood behind the face of the chassis that will be drilled, as in the picture. Mount the chassis and piece of wood in a vice to hold it steady, with the surface to be drilled facing up (11).

   b. Start by drilling a 1/8″ (3 mm) hole. Put the bit in the chuck as far deep as it will go. Align the bit to the divot you made and start drilling on low speed (12). Go slow and use constant force against the metal. Do not let the drill bit slip away from the divot. If it slips away, re-align and start again.

   c. Now re-drill with a 3/16″ (4, 4.5 or 5 mm) bit. Drill very slowly.

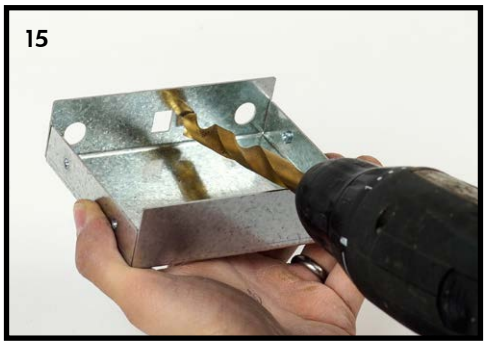
   d. Finally, drill out the 1/4″ (6 mm) hole in the same way (13).


11


12


13

e. Level out the lip around the hole with a deburring tool or a large drill bit – about 7/16″ (12 mm) (14). Deburr the chassis on both sides of the hole to allow the jack and circuit board to sit flush against the inside of the chassis (15).



We are now ready to flash the MIDI-compatible software onto your Quaverato.

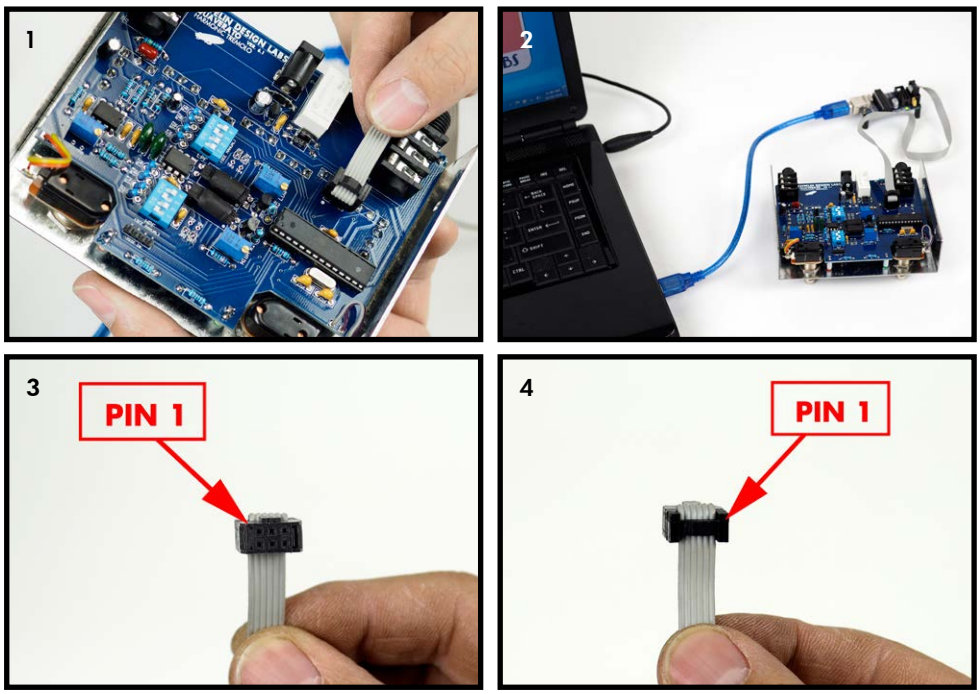
UPDATING THE SOFTWARE

Quaveratos shipped after November 16, 2018 should be loaded with the latest MIDI-enabled software, version 2.3.6 or higher. If you are modding an older unit, you will need to update the software. To verify the software version on your pedal, do the following:

1. Power off the Quaverato.
2. Press and hold down the TAP switch. Power it back up.
3. Continue to hold the TAP switch for a few seconds. The TAP LED will blink out the three-digit version number.

If it turns out you have to update the software, do the following:

1. Download the ZDL Updater program from our website (www.zeppelindesignlabs.com) and install it on your PC (sorry, neither MacOS nor Linux is supported at this time). Run the Updater and click the "?" to open the Help file. Read this carefully to complete the installation.
2. Plug in the 6-pin ribbon cable header from the USBTinyISP into your Quaverato as in the picture (1). Please note the orientation of the ribbon cable on the Quaverato's ISP header: pin 1 is closest to the white dot on the PCB (3,4).



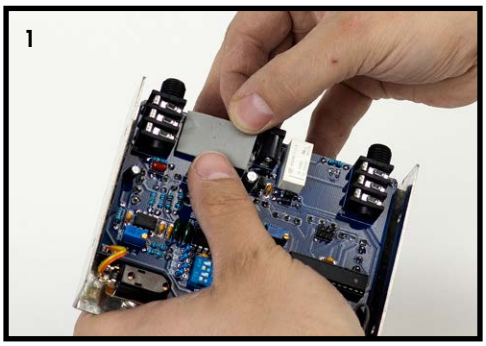
3. Plug the programmer's USB cable into your computer and start the ZDL Updater software (2).
4. Choose "Quaverato" from the Products drop down menu. Choose the latest version from the software drop down menu (version 2.3.6 or higher). Press the FLASH button. If everything is

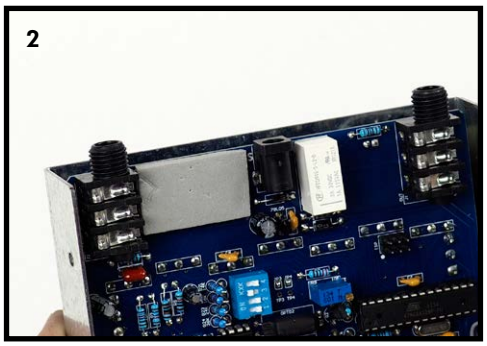
connected correctly then you should see a status bar indicating that the Quaverato is being programmed.

5. If you got the message that the flash was a success (the status bar should reach 100%), then unhook the ribbon cable from your Quaverato and continue with the next step. If you got any other message then consult the Help file by pressing the "?" button. Review the troubleshooting procedures.

PUTTING IT ALL TOGETHER

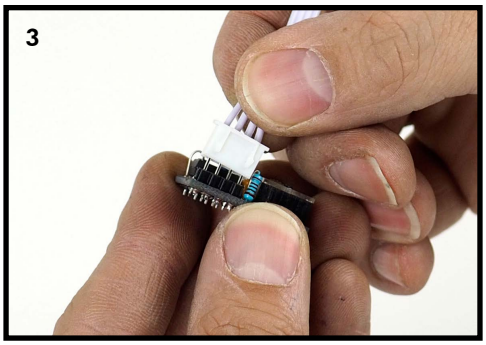
1. Apply the foam tape to the edge of the Quaverato's circuit board as shown in the picture. Don't let it overhang the board (1,2).

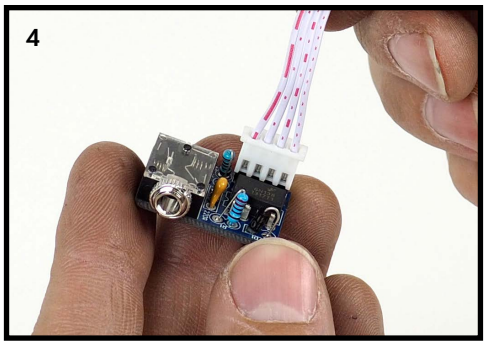


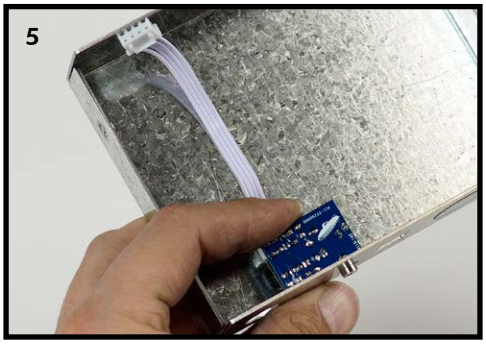


2. Attach the 4-pin ribbon cable to the header on the MIDI board (3,4).

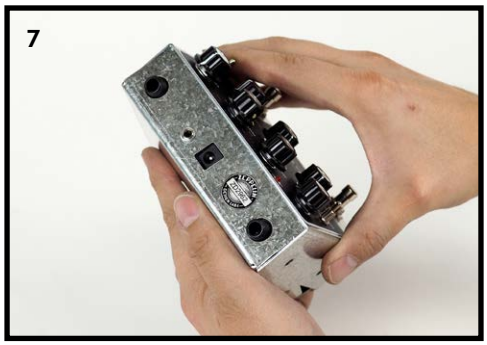
3. Attach the MIDI board to the chassis with the 1/8" jack nut (5,6). The board should be parallel to the bottom of the chassis, with the solder side facing up. The ribbon cable should be laying against the bottom of the chassis (5). Screw the nut onto the jack, but don't over tighten it (6).

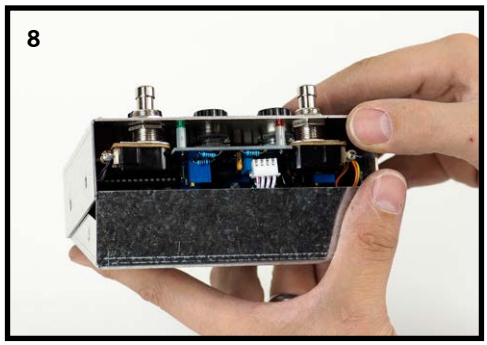


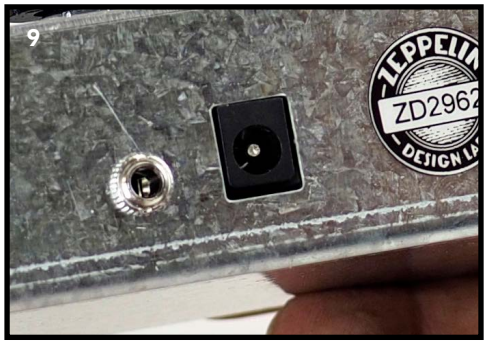




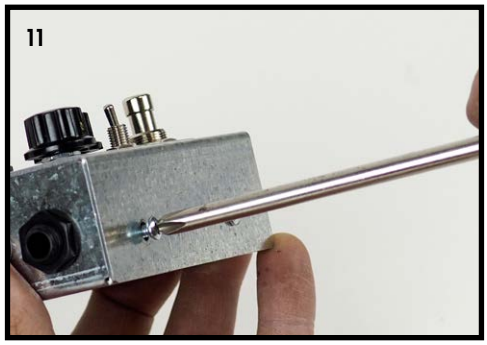
4. Slide the two parts of the chassis together by lining up the 1/4″ jacks to the holes on the bottom chassis (7). While the chassis is still open, plug the other end of the 4-pin ribbon cable into the header on the Quaverato's PCB (8). It is important to make sure the wire is not twisted. Close the chassis. Make sure the power jack is properly lined up with its square hole on the chassis and that it didn't end up behind the hole (9).





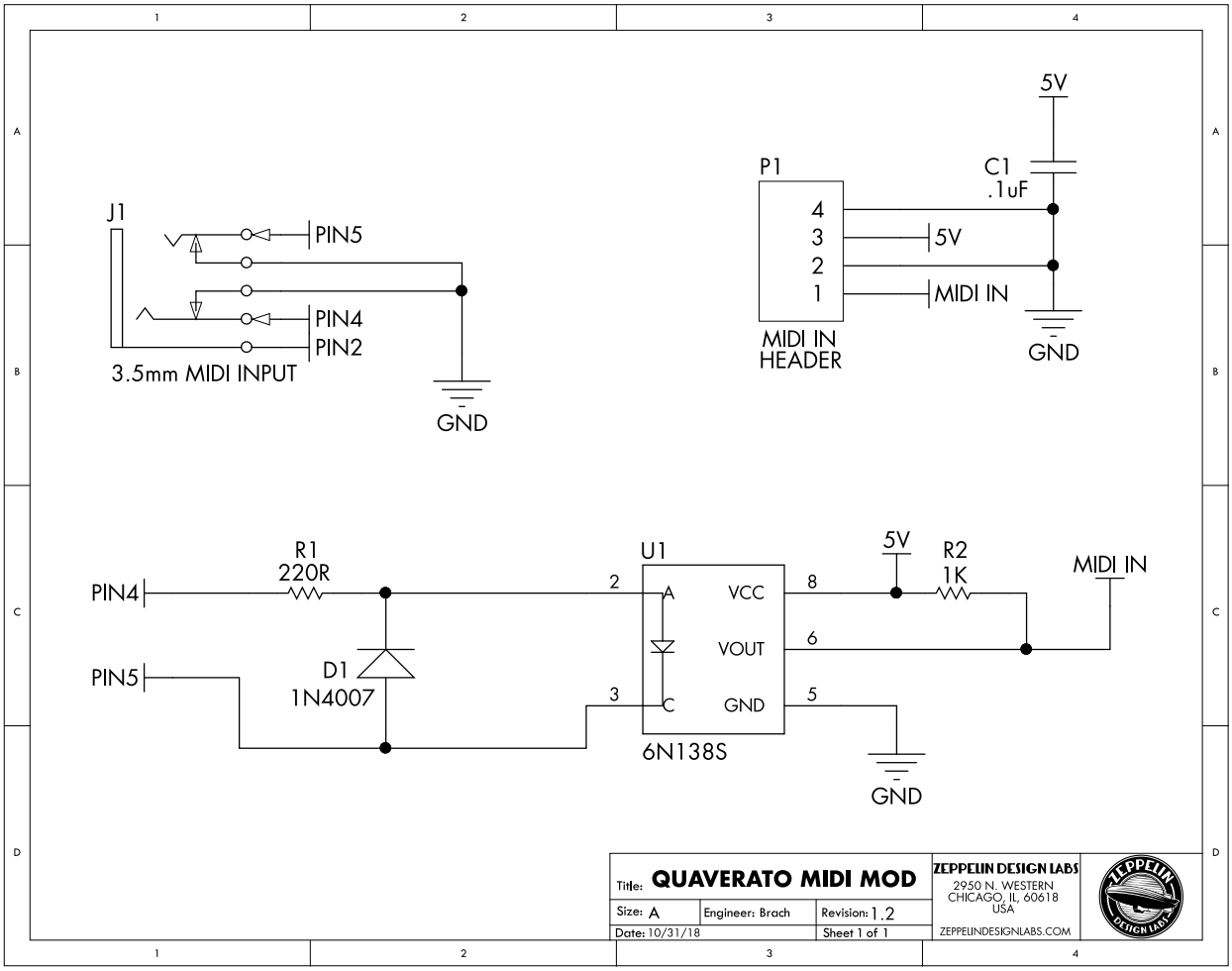


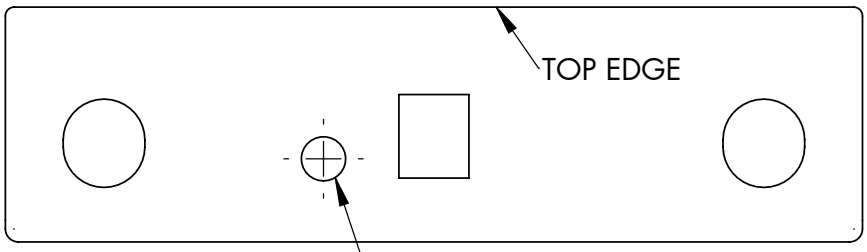
5. Replace the 4 screws and two nuts you removed earlier (10,11).



That's it!

Your Quaverato is ready to be controlled by your favorite MIDI controller or sequencer or DAW. Please see the Quaverato MIDI User Manual for the details of how MIDI works on the Quaverato.





1/4" (6 mm) HOLE.
MARK WITH CENTER PUNCH OR NAIL.
DRILL 1/8" (3 mm) PILOT.
STEP UP GRADUALLY TO FINAL HOLE SIZE.
BE CAREFUL! USE DRILL PRESS IF POSSIBLE.

ZEPPELIN DESIGN LABS
2950 N. WESTERN AVENUE
CHICAGO ILLINOIS 60618 USA
www.zeppelindesignlabs.com

QUAVERATO MIDI MOD DRILL TEMPLATE

SCALE:1:1
REV. DATE 10/29/2018
SHEET 1 OF 1

ZEPPELIN

DESIGN LABS