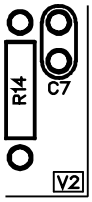
Version

These instructions are for Version 1 of the Thunderclap PCB (printed circuit board). You can tell which version of the PCB you have by looking in the lower right hand corner of the PCB. If you need the Version 2 instructions, you can download them here: http://delptronics.com/documents/ThunderclapKitInstructionsV2.pdf

Version 1



Version 2

Important Information

Thank you for your purchase of the Delptronics Thunderclap Kit!

Before you start, please read the Electronic Kit Soldering Tutorial. It contains important and useful information even for experienced kit builders. If this is your first electronic kit, the best piece of advice we can give is to take your time and be careful to put the right part in the right place. The tutorial can be downloaded here: http://delptronics.com/documents/SolderingTutorial.pdf

In the instructions that follow, each part type is followed by the PCB (printed circuit board) outline for that part. Refer to the enclosed photograph of the completed kit for assistance with part identification and placement.

The PCB is marked with the refdes (reference designator) of each part, not its value. For example R1 refers to resistor number one and C1 refers to capacitor number one. Once the part value is identified, it is easy to find the refdes on the PCB.

Some parts must be oriented in a particular way, that is, the correct lead goes in the appropriate hole in the PCB. For example, electrolytic capacitors are polarized (they have a positive lead and a negative lead). Ceramic capacitors are not polarized, so it does not matter which of the two leads goes in which of the two holes. In these instructions, the symbol **(±)** highlights parts that must be inserted with a particular orientation.

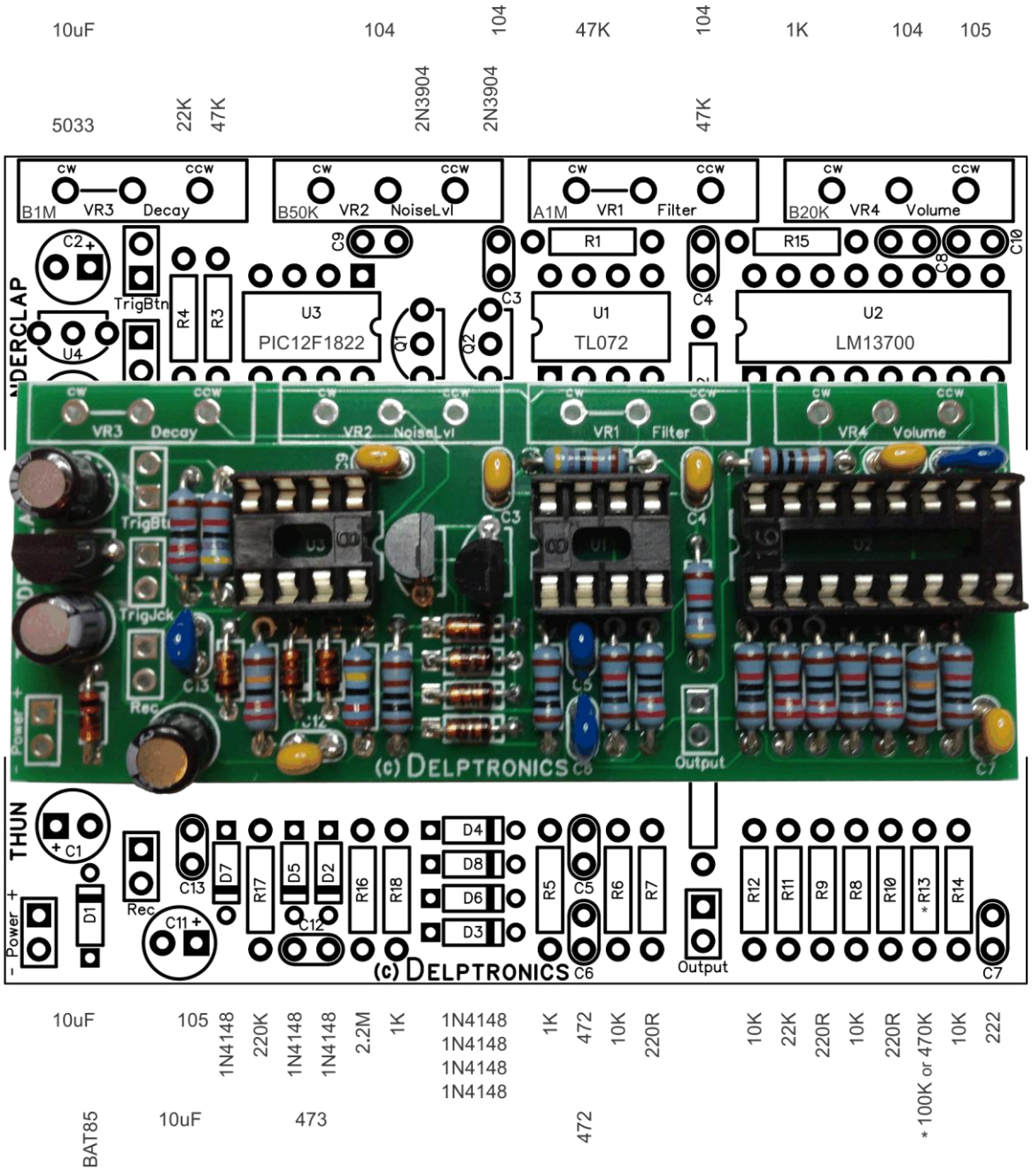
When you are ready to begin, separate the parts by type. Then, when you are ready to solder parts of a particular type, separate them by value. Compare your parts to the bill of materials, which can be found on the last page of this document. Make sure no parts are missing. These instructions list the parts in the recommended order of assembly. In general, the order is shortest to tallest.

This manual is separated into two sections: board-mounted parts, which are soldered directly to the PCB, and off-board parts which are connected to the PCB via wires.

Board-mounted Parts

All of the board-mounted parts are supplied in a little plastic bag to keep them together.

Refer to the diagram below when assembling the printed circuit board. The numbers along the top and bottom of the diagram are the part values. Experienced kit builders may find that this page is all they need to assemble the PCB.



***R13:** Use 100K for line level ≈ 2.5 VPP
Use 470K for synth level ≈ 9.4 VPP

Schottky Diode (±)

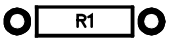
There are 8 diodes, but D1 is different. D1 is a Schottky diode and has “BAT85” printed on it in very small type. An easier way to tell D1 apart is that D1 is loose, while the other seven diodes are connected together on a piece of tape.

Diodes are red and black glass. When inserted into the PCB, the black stripe on the diode must line up with the stripe on the part outline on the PCB. The diode leads need to be bent close to the body of the diode. Hold the diode body and press down on each lead right at the body to make a U shape.

1N4148 Diodes (±)

The 7 1N4148 (non-Schottky) diodes are marked on the PCB as D2 through D8. Insert and solder them the same way you did for D1.

Resistors



The value of a resistor is indicated by colored stripes on its body. In all cases the fifth stripe is brown (indicating 1% tolerance), so that stripe has been omitted from the chart below.

The resistor leads need to be bent close to the body of the resistor. Hold the resistor body and press down on each lead right at the body to make a U shape.

*Note that R13 can be one of two different values depending on the desired output level. For a 9V powered pedal with line level output, use a 100K resistor. For a 12V or 15V powered synthesizer module, use a 470K resistor. With the volume at full and the other knobs at 11 o’clock (classic hand clap sound), line level output will be approximately 2.4 VPP, and synth level output will be approximately 9.4 VPP.

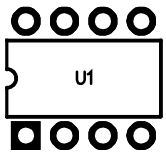
Quantity	Description	Marking				Refdes
3	220R Resistor	red	red	black	black	R7**, R9, R10
3	1K Resistor	Brown	black	black	brown	R5, R15, R18
4	10K Resistor	Brown	black	black	red	R6, R8, R12, R14
2	22K Resistor	Red	red	black	red	R4, R11
3	47K Resistor	Yellow	violet	black	red	R1, R2, R3
1	220K Resistor	Red	red	black	orange	R17
1	2.2M Resistor	Red	red	black	yellow	R16
1	100K Resistor	Brown	black	black	orange	R13***** (line level ≈ 2.5 VPP)
1	470K Resistor	Yellow	violet	black	orange	R13***** (synth level ≈ 9.4 VPP)

** In some kits R7 is 100R (brown black black black)

Ceramic Capacitors

Ceramic capacitors are very small tan or blue blobs. Their value is marked on them with a three digit code. The marking is rather small, so you may have to use a magnifying glass to read them.

Quantity	Description	Marking	Refdes
1	0.0022 µF Ceramic Capacitor	222	C7
2	0.0047 µF Ceramic Capacitor	472	C5, C6
1	0.047 µF Ceramic Capacitor	473	C12
4	0.1 µF Ceramic Capacitor	104	C3, C4, C8, C9
2	1 µF Ceramic Capacitor	105	C10, C13



Chip Sockets (±)

There are two 8-pin sockets and one 16-pin socket. Sockets are marked with a small U-shaped notch on one end that must line up with the outline on the PCB. Once the socket is soldered in place, the PCB outline will not be visible, so it is important that the sockets are oriented correctly in order to ensure that the chips are inserted correctly.

Electrolytic Capacitors (±)

Electrolytic capacitors look like little tin cans with two wires sticking out of the bottom. The value is clearly printed on the capacitor. All three electrolytic capacitors in this kit are the same value (10 µF).

Electrolytic capacitors are polarized, so which lead goes in which hole is important. The negative lead on the capacitor is the shorter one and it is marked with a gray stripe on its body. The positive lead is longer. The positive hole on the PCB has a square pad and is marked with a plus sign.

Voltage Regulator (±)

There is one voltage regulator chip in the kit. The voltage regulator has the same basic shape as a transistor. There are several ways to tell them apart. First and foremost, the voltage regulator is marked with the part number 5033. Second, the voltage regulator is loose and the three transistors are taped together. When in doubt, read the marking on the part.

The voltage regulator's outline on the PCB is labeled U4. Transistors have the same type of outline, so make sure you put the regulator in the right place. Make sure that the flat side of the voltage regulator lines up with the flat side of the outline on the PCB.

The regulator is more heat sensitive than most of the parts in this kit, so take care not to let the soldering iron linger too long. If you are unsure, then solder one lead at a time and let the part fully cool off before soldering the next lead.

Transistors (±)

There are two 2N3904 transistors in the kit. The outlines on the PCB are labeled Q1 and Q2. Note that the outline for the voltage regulator is similar to that of a transistor, but it is clearly labeled "U4" on the PCB.

Make sure that the flat side of the transistor lines up with the flat side of the outline on the PCB. The transistors will not sit flush against the PCB. Do not force them down any further than they will go with a little pressure.

Transistors are more heat sensitive than most of the parts in this kit, so take care not to let the soldering iron linger too long. If you are unsure, then solder one lead at a time and let the part fully cool off before soldering the next lead.

Chips (±)

There are three chips. Two of the chips are the same size, so refer to the photo to make sure you are inserting each chip into its appropriate socket. Note the direction of the chips before inserting them. The notch on the chip must line up with the notch on the socket.

The chip pins come from the factory a little bit splayed out, not pointing straight down. You may need to bend them inward a little before you insert them. Hold the body of the chip and rest all of the pins on one side against the table top and gently press down just a little bit. Then do the other side. If the pins do not line up well with the socket, repeat the straightening procedure.

Make sure not to bend any of the pins while inserting the chips. If a pin gets bent, remove the chip by inserting a small flat head screwdriver between the chip and the socket, then slowly and gently prying the chip out. Straighten the bent pin and reinsert.

PCB Inspection

At this stage, pause to inspect your work. Compare your PCB to the photo.

Make sure that:

- All parts have been soldered in place.
- Polarized parts are oriented correctly.
- All of your solder joints look good.
- There are no solder bridges (blobs of solder covering two leads/pins).
- The leads are clipped off short – right above the solder.

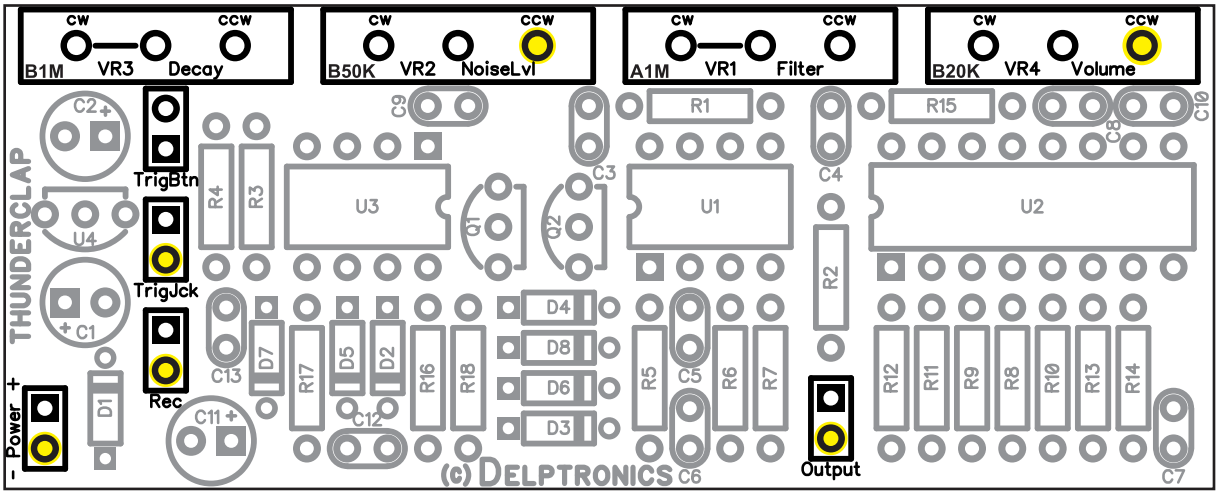
This is a good point to take a break and prepare yourself for assembling the off-board parts.

Off-board Parts

These parts are connected to the PCB via wires, as opposed to being soldered directly to the PCB. Off-board parts include pots (potentiometers), buttons, and jacks. The exact parts will vary depending on which version of the kit you ordered (stomp box or panel mount). How you install these parts will also vary, depending on your chosen drill pattern or panel layout.

It is a good idea to prepare your panel or enclosure before beginning the assembly the off-board parts. The placement of pots, jacks, and switches in the panel or enclosure determines how and when they are connected to the PCB as well as the length of wire needed.

In the below picture, the off-board parts are shown in bold. The ground pads are all highlighted. If you wish, you can simplify your wiring by connecting the grounds from part to part and using fewer ground wires to the PCB. If that last sentence was not entirely clear to you, then use a separate wire from each pin on each part to the appropriate hole in the PCB.



Connecting Off-Board Parts via Wires

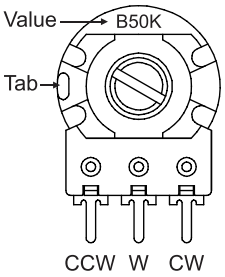
Cut a piece of wire to the appropriate length. Measure carefully, and give yourself a little extra wire to work with. Strip about ¼ inch of insulation off of each end of the wire. Twist the exposed strands of wire together. Insert the wire into the appropriate hole on the top side of the PCB, and solder it on the bottom, just like you did with the board-mounted parts. Wrap the other end of the wire around the appropriate pin on the potentiometer, jack, or switch, and solder it in place.

Potentiometers

The potentiometers (pots) can either be soldered directly to the PCB, or they can be connected to the PCB via wires. You can even do a combination where one or more pots are board-mounted and the others are connected with wires.

If you are building the Thunderclap into a panel, we recommend soldering all four pots directly to the circuit board. That way, they will line up nicely (0.75 inches center-to-center) and the pots themselves will hold the PCB in place without the need for a bracket of any kind.

When viewed with the shaft pointing toward you, as in the picture to the right, the pot's value is printed at the top. The pins are, from left to right, counterclockwise (anticlockwise), wiper, and clockwise. The holes on the PCB are labeled CW and CCW to indicate which pin on the pot gets connected to which hole on the PCB. If you are soldering one or more pots directly to the PCB, the shafts will point away from the PCB.



There is a tab on the pots that must be removed or it will get in the way when you try to mount them in the enclosure or panel. Hold the pot in one hand and grab the tab with a small pair of pliers. Rotate the pliers away from the pot, and the tab will snap right off.

The marking on the PCB for the Filter and Decay pots have a line drawn from the wiper hole to the CW hole. This indicates that the two pins are connected. You only need to connect a wire from the wiper pin from the pot to the wiper hole on the PCB. You can connect the wiper pin to the CW pin with a small piece of wire on the pot itself, or you can skip that connection entirely.

The pot values are as follows:

VR1	Filter	A1M	audio taper
VR2	Noise Lvl	B50K	linear taper
VR3	Decay	B1M	linear taper
VR4	Volume	B20K	linear taper

Record Button

In the Eurorack kit, the trigger and record buttons are the same. In the stompbox kit, the trigger button is the stomp button and the record button is a pushbutton. The buttons are not polarized, so it does not matter which pin on the button goes to which hole in the PCB.

Solder the wires to the button first, and then solder the other ends of the wires to the PCB. Check the style of the record button. If the retaining nut goes on the inside of the enclosure, then the button must be inserted through the hole in the enclosure before you solder the wires to the PCB.

Trigger Button

As stated above, in the Eurorack kit, the trigger and record buttons are the same. In the stompbox kit, the trigger button is a stomp button. The stomp button has the retaining nut on the outside, so the order of soldering is not critical.

Output Jack (±)

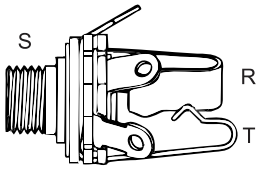
Your kit came with either ¼ inch or 3.5 mm jacks like the ones shown below. Refer to the diagram below. Make sure that the right pins on the jack are connected to the correct holes in the PCB. The round pad is the ground and connects to the shield lug on the jack. The square pad is the signal and connects to the tip lug.

If you are have ¼ inch open frame jacks, look carefully at which lug is connected to which contact, because not all open frame jacks are the same. If you have ¼ inch closed frame jacks that are switched, then there are six terminals on the jack, so make sure that it is oriented as shown. If you connect the wires to the switched terminals, then nothing will happen when you plug in the jack.

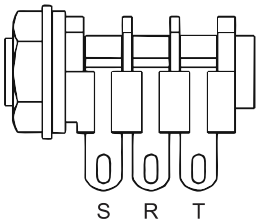
Trigger Jack (±)

Just as with the output jack, make sure that the right pins on the jack are connected to the correct holes in the PCB. Again, round pad is shield and square pad is tip.

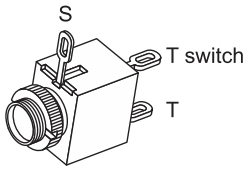
¼ in. Open Frame



¼ in. Closed Frame

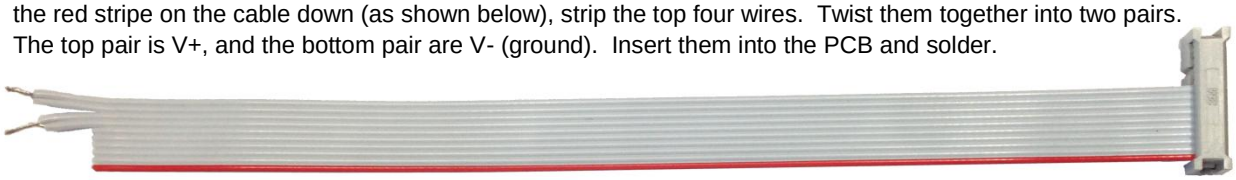


3.5 mm

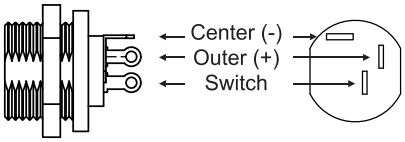


Power (±)

If your Thunderclap is a module, then connect wires from the PCB to a power connector that is appropriate for your style of modular power bus. If you purchased the Eurorack kit, then a Eurorack power connector is supplied. With the red stripe on the cable down (as shown below), strip the top four wires. Twist them together into two pairs. The top pair is V+, and the bottom pair are V- (ground). Insert them into the PCB and solder.

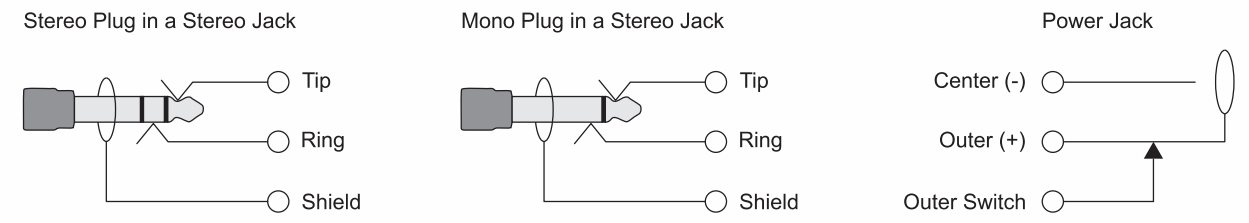


If your Thunderclap is a stomp box, the simplest way to connect the power is to connect wires straight from the power jack to the PCB. Then, the unit is powered on when the power supply is plugged in.



9V Battery Snap

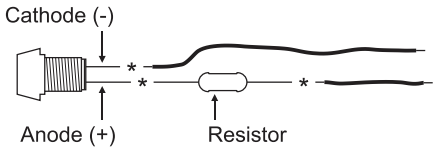
If you would like run your Thunderclap on battery power, you can connect a 9V battery snap (not included). Simply solder the red (positive) wire from the snap to the Outer Switch terminal of the power jack, and the black (negative) wire to the Ring terminal of the output jack. When connected as described, the positive power from the battery is only connected when the power adapter is not plugged into the power jack, and the negative power from the battery is only connected when a plug is inserted into the output jack. The diagram below makes it clear how this works.



Power LED (±)

If your Thunderclap is a module, then no power LED is needed, since the module is on when our rack is powered on.

Refer to the diagram below. The cathode (negative) lead of the LED is shorter than the anode (positive) lead. The points to be soldered are marked with an asterisk. The cathode of the LED is soldered to a wire, which is then connected to the negative power supply. The anode of the LED is soldered to a resistor. A 1K Ohm resistor will work well for most types of LEDs and voltages. The other end of the resistor is soldered to a wire, which is then connected to the positive power supply. After soldering the wire and resistor to the LED, use electrical tape or heat-shrink tubing on one or both sides to ensure that no bare wires touch.



If your kit came with an LED that is already in a holder with the resistor and wires already attached, then simply solder the wires to the power supply. The red wire is positive and the black wire is negative. The LED assembly is inserted into the enclosure from the outside, with the retaining nut on the inside. Therefore, it must be inserted through the hole in the enclosure before you solder the wires.

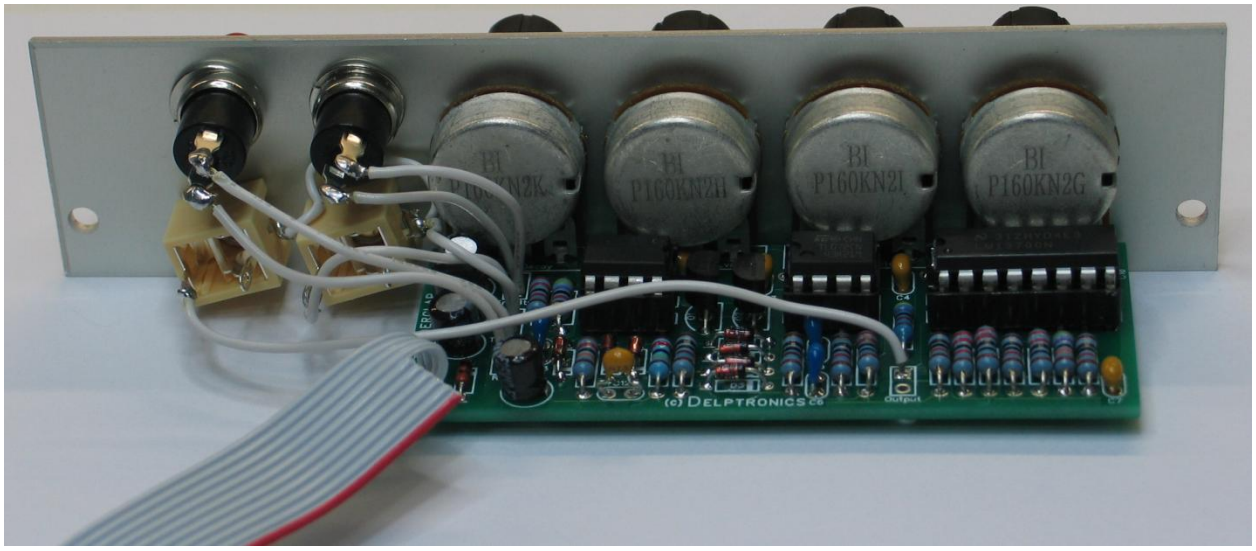
If you are using a power switch (either an actual switch or using the output jack as a switch), make sure you connect the LED at the same point as the wires that go to the PCB. Otherwise, the LED will be on and will drain power even when the unit is off.

Finished

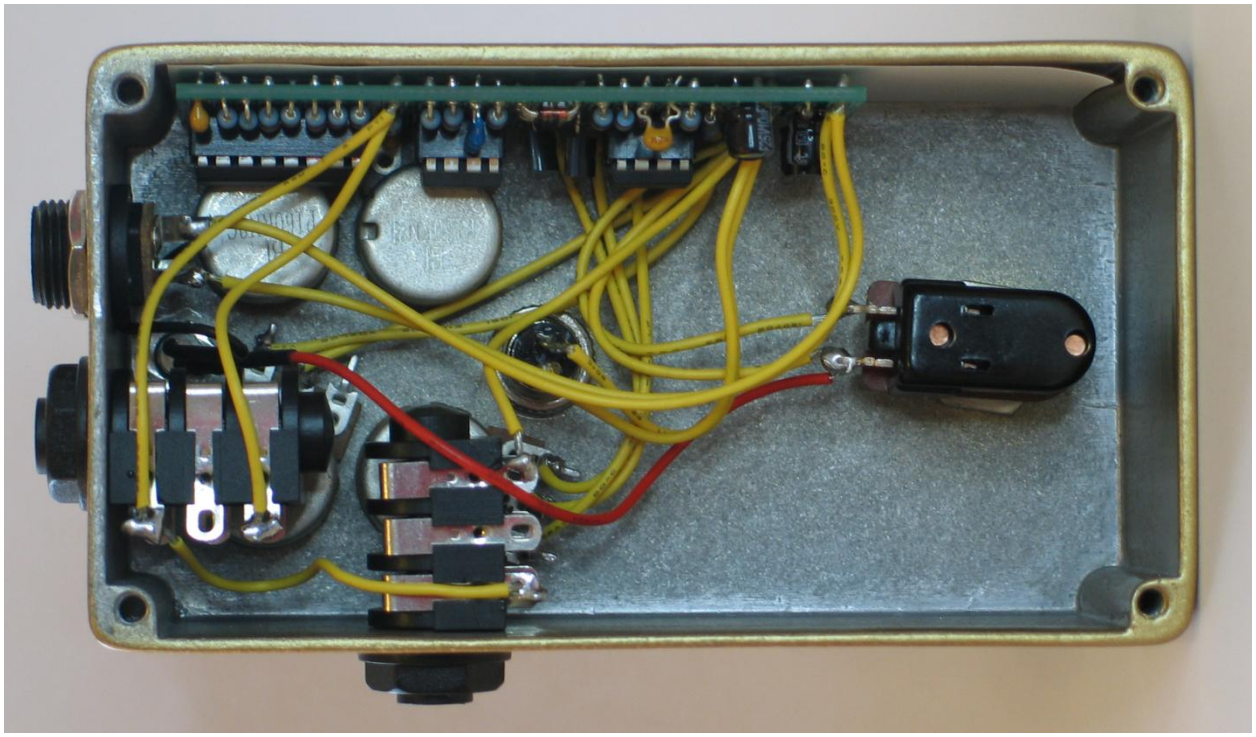
That's it. Your Thunderclap is complete. Congratulations!

Pictures

This is what the assembled Eurorack module looks like. Note that the ground of the output jack is wired to the ground of the trigger jack, which is then wired to the PCB. That way, we do not need to run a wire from the output ground hole on the PCB.



This is what an assembled stompbox looks like inside. Yours might look different depending on how you connect the pots. Notice that we have inserted a piece of cardboard between the PCB and the side of the enclosure so that the bottom of the PCB does not touch the metal enclosure, which would short the circuit.



Bill of Materials

Board-mounted parts:

Quantity	Description	Marking on part				Refdes
1	BAT85 Schottky Diode	BAT85				D1
7	1N4148 Diode	1N4148				D2, D3, D4, D5, D6, D7, D8
3	220R Resistor	red	red	black	black	R7, R9, R10
3	1K Resistor	brown	black	black	brown	R5, R15, R18
4	10K Resistor	brown	black	black	red	R6, R8, R12, R14
2	22K Resistor	red	red	black	red	R4, R11
3	47K Resistor	yellow	violet	black	red	R1, R2, R3
1	220K Resistor	red	red	black	orange	R17
1	2.2M Resistor	red	red	black	yellow	R16
1	100K Resistor	brown	black	black	orange	R13 (line level)
1	470K Resistor	yellow	violet	black	orange	R13 (synth level)
1	0.0022 µF Ceramic Capacitor	222				C7
2	0.0047 µF Ceramic Capacitor	472				C5, C6
1	0.047 µF Ceramic Capacitor	473				C12
4	0.1 µF Ceramic Capacitor	104				C3, C4, C8, C9
2	1 µF Ceramic Capacitor	105				C10, C13
3	10 µF Electrolytic Capacitor	10 µF				C1, C2, C11
2	2N3904Transistor	2N3904				Q1, Q2
1	3.3V Voltage Regulator	5033				U4
1	TL072 Op Amp Chip	TL072				U1
1	LM13700 OTA Chip	LM13700				U2
1	PIC12F1822 MCU Chip	PIC12F1822				U3
2	8 Pin DIP Socket					U1, U3
1	16 Pin DIP Socket					U2

Off-board parts:

Quantity	Description	Marking on part	PCB Legend
1	1M Audio Taper Potentiometer	A1M	VR1 Filter
1	50K Linear Taper Potentiometer	B50K	VR2 Noise Lvl
1	1M Linear Potentiometer	B1M	VR3 Decay
1	20K Linear Potentiometer	B20K	VR4 Volume
4	Knobs		
2	Jacks ¼ inch or 3.5mm		Trig Jck, Output
1	Stomp Button		Trg Btn
1	Pushbutton		Rec
1	Hookup Wire		
1	Power Jack (pedal kit only)		Power
1	Power LED (pedal kit only)		
1	Panel (Eurorack kit only)		
1	Eurorack Power Connector (Eurorack kit only)		