Important Information

Thank you for your purchase of the Delptronics ThunderBell Mini Kit!

The most recent version of this document, and the instructions for the ThunderBell Eurorack Module Kit can be downloaded from: http://delptronics.com/thunderbell.php

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 from: http://delptronics.com/documents/SolderingTutorial.pdf

In the instructions that follow, each part type is followed by the PCB 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.

Using the ThunderBell

Power: The ThunderBell is powered by a 9V battery.

Trigger Button: Press the button to make the bell sound.

Pitch Knob: controls the pitch of the bell.

Decay Knob: controls how long it takes for the bell's volume to decay from its maximum level down to silence. The bell can be re-triggered even while the last bell sound is still decaying.

CV (control voltage) Jack: allows you to control the pitch of the bell with an external analog device, like a sequencer. It accepts zero to 5V, but has over-voltage protection. When a plug is inserted into the CV Jack, the pitch knob is ignored.

Trigger Jack: allows you to trigger the bell sound using an external analog device like a sequencer. It accepts a 5 to 15 volt positive trigger or gate signal. The trigger button can still be used even when a plug is inserted into the Trigger Jack.

Output Jack: Plug a ¼ mono cable into the output jack and plug the other end of the cable into your amplifier or mixer. When a mono plug is inserted, the bell is powered on. A stereo plug will not work.

Board-mounted Parts

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

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

VR1 10K

D4/D3 BAT85

R14 1K

C5 0.68µF(684) **C3** 0.1µF(104)

D2 1N4148

R11 1K

R13 47K

R12 22K

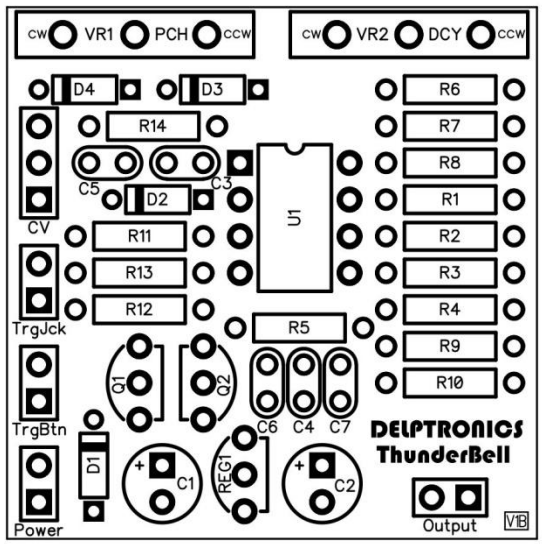
R5 470K

Q1/Q2 2N3904

D1 BAT85

C1/C2 10µF

REG1 LM78L05



VR2 1M

R6 1M

R7 2.2M

R8 2.2M

R1 100K

R2 220K

R3 470K

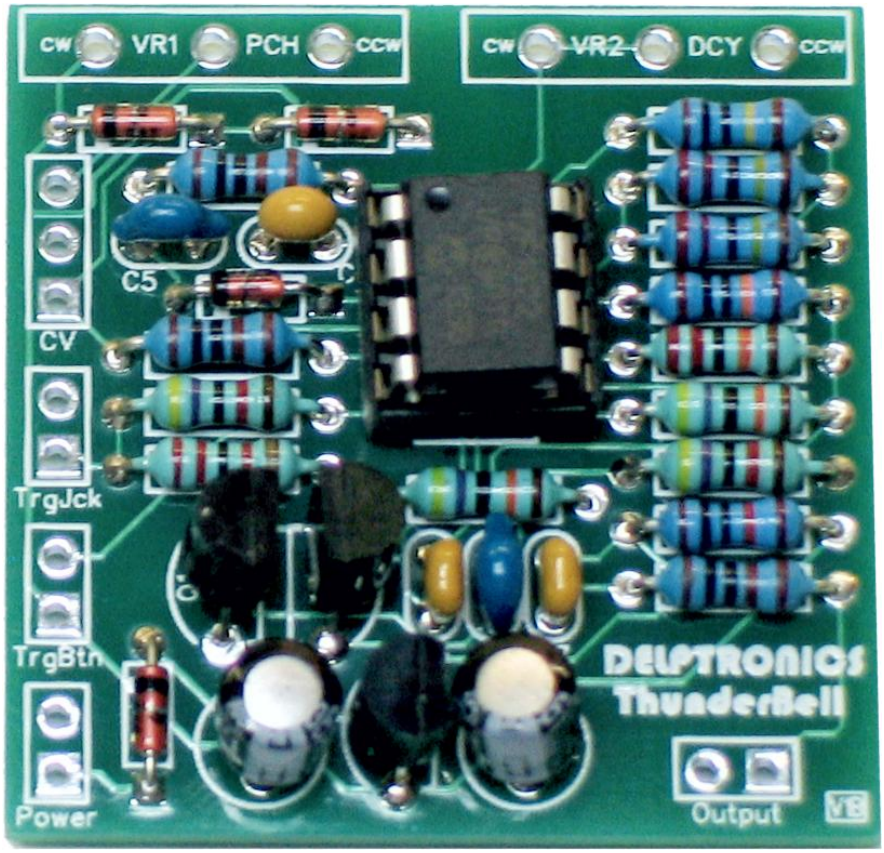
R4 47K

R9 10K

R10 1K

C6/C7 0.01µF(103)

C4 1µF(105)



1N4148 Diode (±)

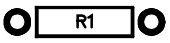
Diodes are red and black glass cylinders. There are four diodes, and they all look similar, but D2 is different. D2 is a switching diode and has “1N4148” printed on it in very small type. An easier way to tell D2 apart is that D2 is loose, while the other three diodes are connected together on a piece of tape.

Diodes are polarized, so it is important that the right lead goes in the right hole. 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 into a U shape in order to insert them into the PCB. Bend the leads by holding the diode body and pressing down on each lead close to the body.

Schottky Diodes (±)

D1, D3, and D4 are Schottky diodes and have “BAT85” printed on it in very small type. When inserted into the PCB, the black stripe on the diode must line up with the stripe on the part outline on the PCB.

Resistors



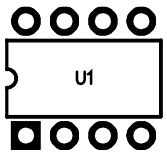
Resistors are cylindrical parts with colored stripes indicating their value. The resistor leads need to be bent into a U shape in order to insert them into the PCB. Bend the leads by holding the resistor body and pressing down on each lead close to the body.

Quantity	Description	Marking					Refdes
3	1K Resistor	brown	black	black	brown	brown	R10, R11, R14
1	10K Resistor	brown	black	black	red	brown	R9
1	22K Resistor	red	red	black	red	brown	R12
2	47K Resistor	yellow	violet	black	red	brown	R4, R13
1	100K Resistor	brown	black	black	orange	brown	R1
1	220K Resistor	red	red	black	orange	brown	R2
2	470K Resistor	yellow	violet	black	orange	brown	R3, R5
1	1M Resistor	brown	black	black	yellow	brown	R6
2	2.2M Resistor	red	red	black	yellow	brown	R7, R8

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
2	0.01 µF Ceramic Capacitor	103	C6, C7
1	0.1 µF Ceramic Capacitor	104	C3
1	0.68 µF Ceramic Capacitor	684	C5
1	1 µF Ceramic Capacitor	105	C4



Chip Socket (±)

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

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. Both electrolytic capacitors in this kit are the same value (10 µF).

Electrolytic capacitors are polarized, so it is important that the right lead goes in the right hole. 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 78L05. Second, the voltage regulator is loose and the two transistors are taped together. When in doubt, read the marking on the part.

The voltage regulator's PCB outline is labeled REG1. The outline is similar to that of a transistor, 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 will not sit flush against the PCB. Do not force it down any further than it will go with a gentle pressure.

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 transistor PCB outlines are labeled Q1 and Q2. The outline is similar to that of the voltage regulator, so make sure you put everything in the right place. 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 gentle 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.

Chip (±)

There is one chip in the kit. Note the direction of the chip before inserting it. 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.

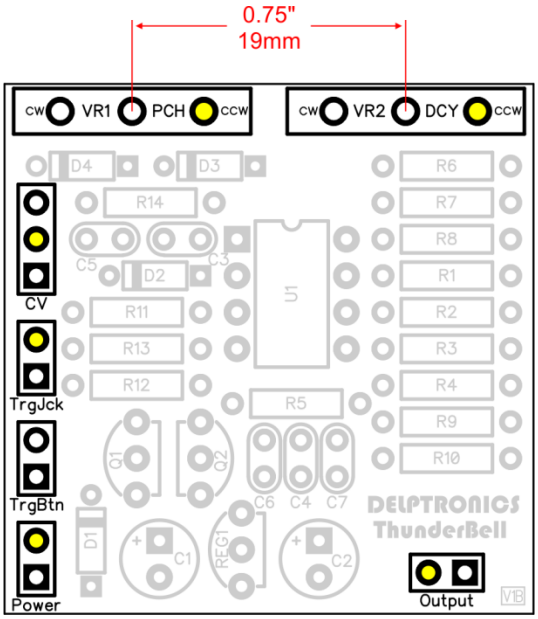
This is a good point to take a break before assembling the off-board parts.

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. 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 connect 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.

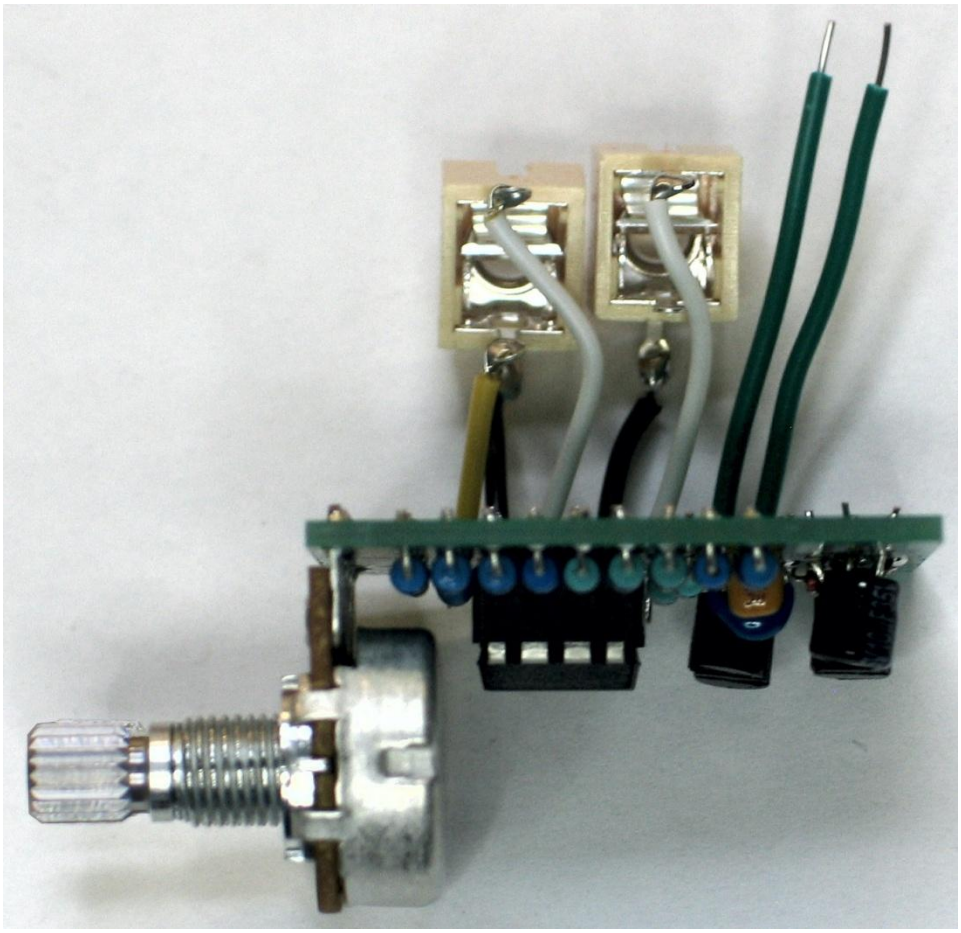


Connecting Off-Board Parts via Wires

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 (6mm) 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 button, and solder it in place.

Pots, Jacks and Trigger Button Wires

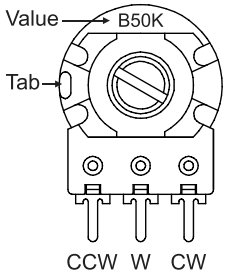
Refer to the photo below when installing the pots, 3.5mm jacks and trigger button wires Note that the very short wires are necessary for installing the PCB into a tight space, like a cowbell. If you are using a larger enclosure, you should use longer wires.



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 pot is board-mounted and the others are connected with wires. Board-mounting the pots is most convenient because it means that the PCB will be supported inside your enclosure.

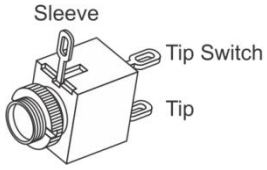
When viewed with the shaft pointing toward you, as in the picture to the right, the pot's value is printed at the top. The pins are, from left to right, counterclockwise (anticlockwise), wiper, and clockwise. The holes on the PCB are labeled CW and CCW to indicate which pin on the pot gets connected 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.

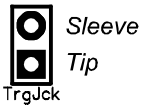


There is a tab on the pots that must be removed or it will get in the way when you try to mount them in the enclosure or panel. Hold the pot in one hand and grab the tab with a small pair of pliers. Rotate the pliers away from the pot, and the tab will snap right off.

The pot values are as follows:
VR1 (Pitch) 10K linear taper potentiometer
VR2 (Decay) 1M linear taper potentiometer

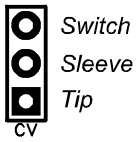
3.5 mm Switched Mono Jack





Trigger Jack (±)

The Trigger jack is a 3.5 mm mono jack. Only two of the terminals of the Trigger Jack are connected to the PCB (the switch terminal is not used). It is very important that the right terminal connects to the right hole in the PCB. Refer to the above jack diagram and PCB outline, as well as the photo on the previous page. Connect the tip terminal of the jack to the square hole in the PCB, and the sleeve terminal to the round hole.



CV Jack (±)

The CV (control voltage) jack is a 3.5 mm switched mono jack. All three terminals of the CV jack are connected to the PCB. It is very important that the right terminal connects to the right hole in the PCB. Refer to the above jack diagram and PCB outline, as well as the photo on the previous page. Connect the tip terminal of the jack to the square hole in the PCB, the sleeve terminal to the middle round hole, and the switch terminal to the last round hole.

Trigger Button

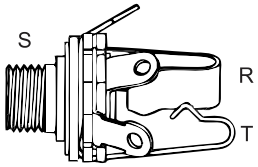
Buttons are not polarized, so it does not matter which terminal on the button goes to which hole in the PCB. Make a note of where the retaining nut is on your button. If the nut goes on the inside of the enclosure (most likely), then the button must be inserted into the enclosure before it can be connected to the PCB. It is easiest if you do it in this order: solder the button wires onto the PCB first, then put the button through its hole in the enclosure, then tighten the button retaining nut, then solder the wires to the button.

Output Jack (±)

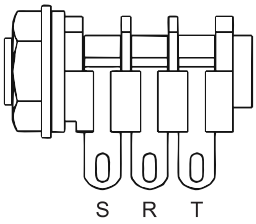
The output jack is a ¼ inch stereo jack, also called a TRS (tip, ring, sleeve) jack. Refer to the PCB outline above and the diagrams below. Make sure that the right pins on the jack are connected to the correct holes in the PCB. The round pad is the ground and connects to the sleeve lug of the jack. The square pad is the signal and connects to the tip lug.

If you are using an open frame jack, look carefully at which lug is connected to which contact, because not all open frame jacks are the same. If you are using a closed frame jack that is 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.

¼ in. Open Frame



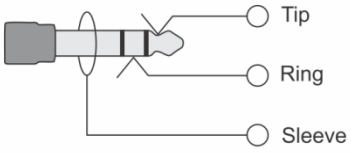
¼ in. Closed Frame



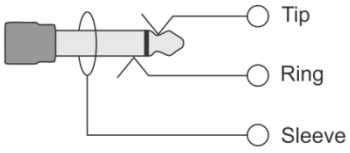
9V Battery Snap (±)

A 9V battery snap with wires is included with your kit. Solder the red (positive) wire from the snap to the square hole on the PCB power connector. Solder the black (negative/ground) wire to the ring (middle) terminal of the output jack. When connected as described, the battery ground is only connected to the circuit when a plug is inserted into the output jack. So, the device is on when a jack is inserted, and it is off when no jack is inserted. The diagram below illustrates how this works.

Stereo Plug in a Stereo Jack



Mono Plug in a Stereo Jack



Bill of Materials

Basic Kit Board-mounted Parts:

Quantity	Description	Marking on part				Refdes
3	BAT85 Schottky Diode	BAT85				D1, D3, D4
1	1N4148 Diode	1N4148				D2
3	1K Resistor	brown	black	black	brown	R10, R11, R14
1	10K Resistor	brown	black	black	red	R9
1	22K Resistor	red	red	black	red	R12
2	47K Resistor	yellow	violet	black	red	R4, R13
1	100K Resistor	brown	black	black	orange	R1
1	220K Resistor	red	red	black	orange	R2
2	470K Resistor	yellow	violet	black	orange	R3, R5
1	1M Resistor	brown	black	black	yellow	R6
2	2.2M Resistor	red	red	black	yellow	R7, R8
2	0.01 µF Ceramic Capacitor	103				C6, C7
1	0.1 µF Ceramic Capacitor	104				C3
1	0.68 µF Ceramic Capacitor	684				C5
1	1 µF Ceramic Capacitor	105				C4
2	10 µF Electrolytic Capacitor	10 µF				C1, C2
2	2N3904Transistor	2N3904				Q1, Q2
1	5V Voltage Regulator	78L05				REG1
1	PIC12F1822 MCU Chip	PIC12F1822				U1
1	8 Pin DIP Socket					U1

Basic Kit Off-board Parts:

Quantity	Description	Marking on part	Marking on PCB
1	10K Linear Taper Potentiometer	B10K	VR1 PCH
1	1M Linear Taper Potentiometer	B1M	VR2 DCY
2	Knobs		
1	1/4 inch Jack		Output
2	3.5mm (1/8") Jacks		TrgJck, CV
1	Pushbutton		TrgBtn
1	9V Battery Snap		Power
1'	Hookup Wire		

Jack+Amp+Speaker add-on Parts:

Quantity	Description
1	JAS Module Kit
1	1SPST Toggle Switch

Cowbell Enclosure Parts:

Quantity	Description
1	Cowbell
1	Battery holder
2	Screws
2	Nuts