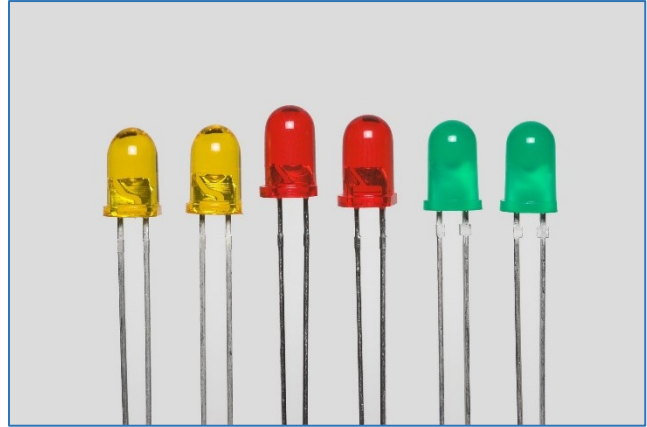


LED tester **Builder's Guide**

Bruce E. Hall, [W8BH](#)



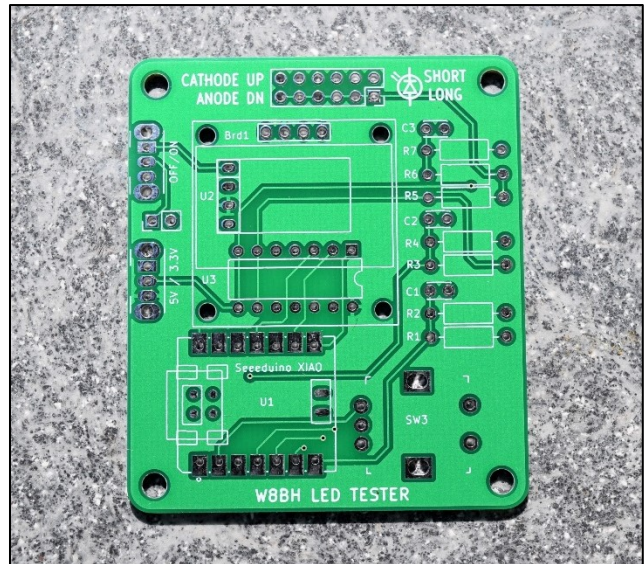
Not a Kit.

Forgive me for starting these notes with a disclaimer: this is not a kit, I am not selling anything, and there is no guarantee of success or suitability for any particular purpose. Still interested?

The PCB.

The first step is to order one or more of the printed circuit boards. To order a circuit board from a manufacturer you must provide a digital set of design files, called '[gerbers](#)'. You can obtain the gerbers on my [github account](#).

Almost all board fabricators' websites allow Gerber-file uploads. Choose the manufacturer you like. I've used [OSH Park](#) (USA, excellent quality) in the past and have been very satisfied. For this project I used [JLCPCB](#) (China, hobby-quality). Choose the default 2-layer options, get your quote, and choose whatever shipping option you want. At JLCPCB I paid \$2.00 + shipping for a set of 5 boards.

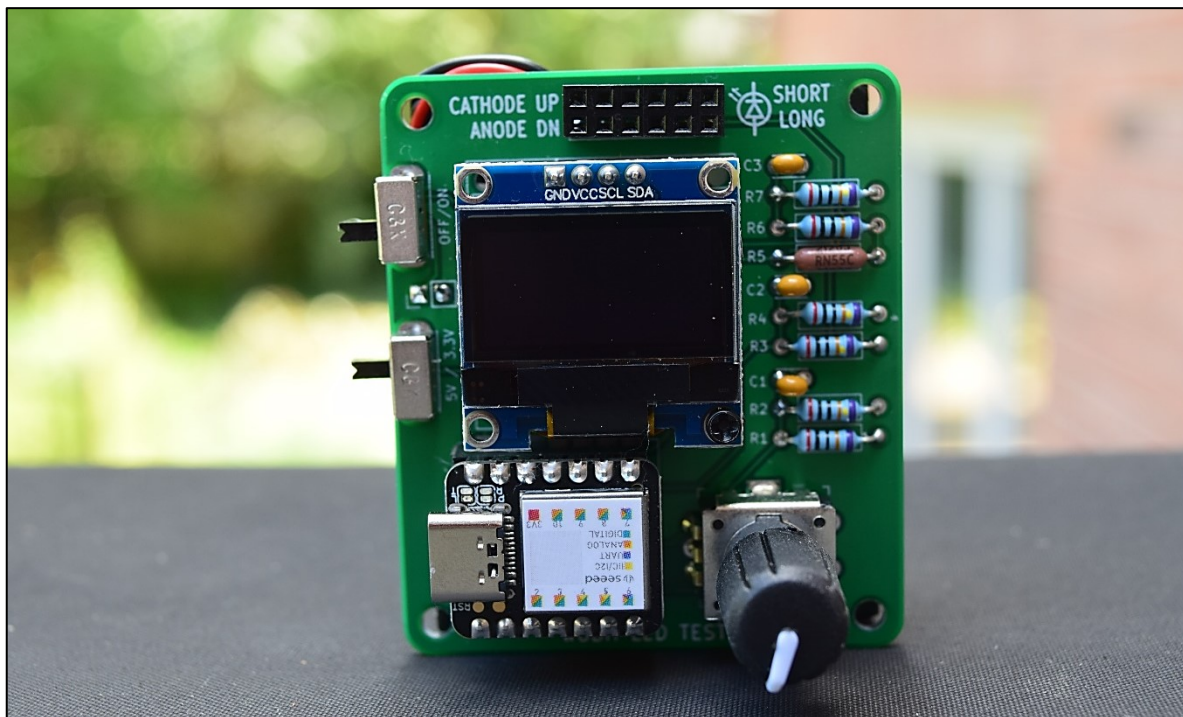


Bill of Materials

Part	Qty	DigiKey	Amazon	Prices/Other
Seeeduino XIAO	1	1597-102010328-ND	Seeed Studio	\$5.40
128x64 OLED Display	1	n/a	HiLetgo , Frienda	6.40
MCP4261 digipot	1	MCP4261-103E/P		1.58
Rotary Encoder	1	PEC11R-4220F-S0024		1.72
200K resistor, 0.1%	6	RN55C2003BRE6		0.60
100-ohm resistor, 0.1%	1	YR1B100RCC		0.60
100nF capacitor	3	C315C104M5U5TA7303		0.23
SPDT Switch OS102011MA1QN1	2	CKN9559-ND		0.48
2x6 Female Header	1	S7109-ND		0.82
Adafruit Miniboost 5V	1	1528-4654-ND		3.95
Misc: 0.1" male/female headers, LED, USB-C cable, M2 standoff hardware				

The above table lists part numbers and/or links for my favorite sources. These parts should cost you about \$20 + shipping. Please review the list of miscellaneous components. Do you have these already? If not, add them to your list. Several of the listed components are optional, so review the build instructions before purchasing.

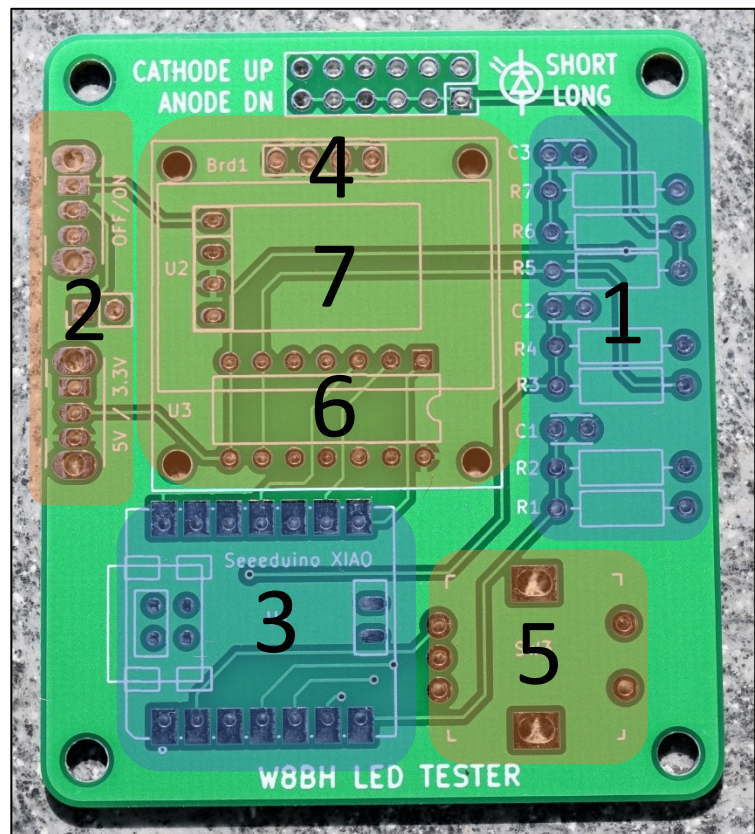
Here is a completed board, showing component layout.



You will need a decent soldering iron, solder, and a voltmeter. You should be familiar with soldering and have successfully kitted other projects using ICs and other through-hole components. You should have sufficient lighting, magnification, and workspace area.

The following instructions apply only to Rev 1 of the board. If your board does not say “BN42” on the back, these aren’t the correct instructions for you.

- Consider if you are going to enclose the board. The 3 mm mounting holes are large enough to accommodate M2.5 or 4-40 hardware. If you are facile with 3-D printing this might make a good enclosure project.

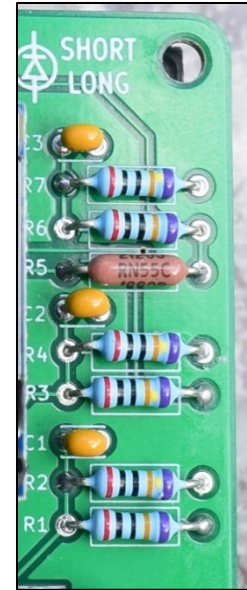


Step1: Install the resistors and capacitors

These passive components are located along the right edge of the PCB, and are easiest to place before the taller components are installed.

- Install precision 100-ohm resistor R5. This is a very important component! When I ordered this part, the vendor supplied me with the wrong value. Confirm its value with an ohm-meter, and make sure you install it as R5. Color codes: Brown-Black-Black-Black-Purple
- Install six 200K resistors, R1-R4, R6-R7. Color codes: Red-Black-Black-Orange-Purple
- Install the three 100nF capacitors, C1-C3. Marking “104”

Test 1: After installing all of the resistors and capacitors, check the resistance between the right side of R6 and right side of R7. You should see about 200K ohms. If not, refer to the troubleshooting guide.



Step 2: Install the switches

The switches are located along the left edge of the PCB. The upper switch controls battery power for the device, and the lower switch controls the LED testing voltage (3.3 vs 5.0 volt).

- Install both switches, taking care to mount them with the actuators facing left, away from the PCB. Solder one pin first, then check the position of the switch body. It should be flush with the PCB, and parallel with the PCB edge. Adjust, if necessary, then solder the remaining pins.

Test 2: Turn the board over and inspect all of your solder joints. You’ve made 30 solder connections so far, which is almost half the total. Take a moment to clean your solder joints. Isopropyl alcohol and an old toothbrush work very well for this.



Step 3: Install the microcontroller

Decide how you will mount the microcontroller. You can a) solder the microcontroller directly to the board using the castellated pads; b) permanently mount it using male header pins; or c) use mating male/female headers. I do not recommend direct mounting for this project, since it places the USB-C jack too close to the PCB, inhibiting future programming. Permanent mounting with male headers makes for a sturdy and compact installation, but it’s hard to undo if you want to repurpose the MCU for another project. Male/female header mounting is helpful if you want to swap out the microcontroller at a later date. Its only disadvantage is making the MCU slightly taller than the display. I chose the last option for my board.

Method A: direct mounting (not recommended).

- Watch this [Sparkfun video](#) before soldering castellated pads. It is a good overview of the process.
- Align the XIAO over its board outline with the USB jack facing left. Ensure that the PCB pads extend beyond the XIAO pads on both sides.
- Tack one set of pads together with solder.
- Inspect the alignment. Reheat the soldered pad and realign if necessary.
- Solder the remaining pads.

Method B: mount using male headers only.

- Solder male header pins to your microcontroller.
- Place the microcontroller into the PCB, with the USB jack facing left.
- Solder a single pin. Check the height and alignment of the MCU. Adjust as necessary
- Solder the remaining header pins.



Method B: male headers

Method C: mount using male/female headers.

- Solder male header pins to your microcontroller.
- Prepare two 7-pin lengths of female header.
- Temporarily mate the male and female headers
- Solder a single pin of both female headers to the PCB. Check header alignment.
- Remove the microcontroller and solder the remaining female header pins.



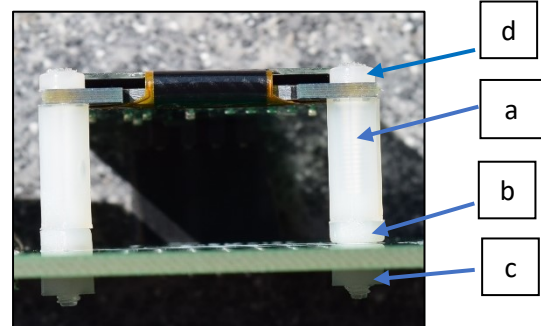
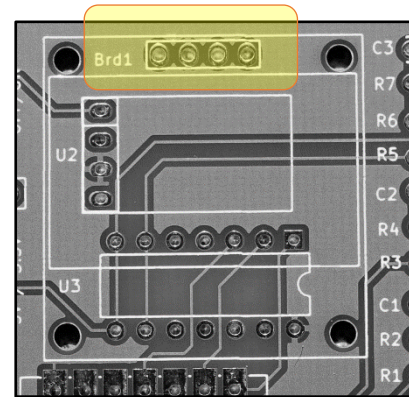
Method C: male/female headers

Test #3a: Connect the USB-C cable to your microcontroller and briefly apply power. The green power LED should light. After a few seconds, the orange LED should flash 'L': dit-dah-dit-dit. If not, refer to the troubleshooting guide.

Test #3b: Confirm testing voltages: Put the lower switch into its 3.3V (up) position. With the power still applied to the microcontroller, use your DMM to measure the voltage from the right side of R1 (+V) to the right side of R2 (GND). It should read about 3.3V. Put the switch into its 5V (down) position, and check the voltage again. It should read about 5V.

Step 4: Install the OLED display.

- Cut a 4-pin length of female header. Solder the female header at the horizontal row of 4 holes marked “Brd1”. Solder one pin, ensure the header is firmly seated and perpendicular to the board, then solder the remaining pins.
- Cut a 4-pin length of male header, and solder it to the OLED display.
- Install the OLED display, mating the male & female headers.
- **Test #4:** Connect the USB-C cable and apply power. After a brief pause, the display should come to life. The top line should say “Testing LED at ...”, with the testing voltage indicated.
- Secure the display with M2 nylon standoffs using the bottom/left and bottom/right display mounting holes. The side-view photo below shows a suggested arrangement:
 - a. Use a 10mm male-female standoff above the PCB. The male end faces the PCB and the female end faces the display.
 - b. Place an M2 nut on the male end of standoff, just above PCB, to add 1mm of height to the standoff.
 - c. Place an M2 nut below PCB to secure the standoff to the PCB
 - d. Use an M2 screw to secure the OLED display to the standoff.
- Congratulate yourself and take a break.



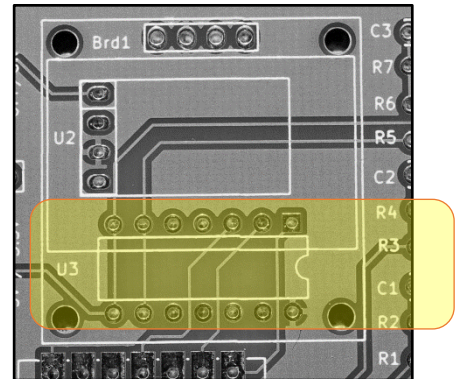
Step 5: Install the rotary encoder

- Carefully insert the encoder into the PCB near the bottom/right corner. It is a tight fit. Take your time, gently rocking the encoder while applying enough pressure to seat the encoder. Solder in place.
- **Test #5:** Connect the USB-C cable and apply power. After a brief pause, the display should appear as before. Rotate the encoder knob, and watch the resistance value change. If not, refer to troubleshooting guide. Remove power.



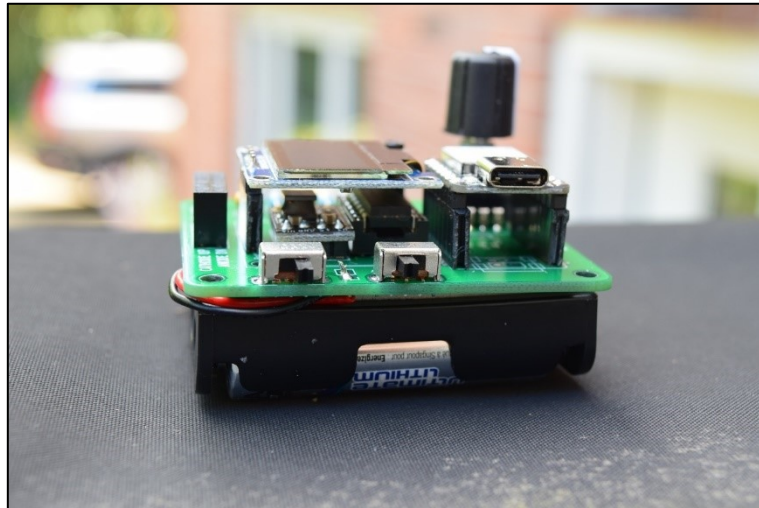
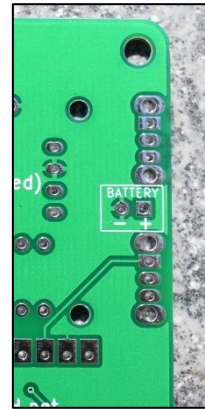
Step 6: Install the digital potentiometer and testing socket

- The digital potentiometer is located beneath the display at the position labeled “U3”. You may solder it directly to the PCB, or use a low-profile IC socket. Temporarily remove the display, install the MCP4261 IC with its notch facing right, and replace the display.
- If you decide to socket the chip, make sure there is enough space for the socket you are using. Check the space between the socket and the lower mounting holes of the display, as well as the boost module U2.
- Install the LED testing socket: use a 2x6 female socket, or prepare two 6-pin strips of female header. Solder one pin. Check alignment, adjust if necessary, and solder in the remaining pins.
- **Test #6:** Do this test only if you are comfortable measuring current. Connect the USB-C cable and apply power. Set the resistance to 1000. Set your DMM to measure current (mA). Connect the DMM leads to the testing socket, with the red lead in the anode row, and the black lead in the cathode row. The OLED display will read “Shrt”. You should read a few mA on the DMM, depending on the testing voltage. Increase the resistance to 2200 ohms and the current should decrease. Decrease the resistance to 680 ohms, and the current should increase. Do not let the current exceed 10 mA. Remove power.



Step 7: Install batteries & power boost module

- Temporarily remove the display. The power boost module is located beneath the display, at the position labeled “U2”. The module is slightly shorter and wider than the PCB outline.
- Cut a 4-pin length of male header, and solder to the boost module.
- Solder one pin of the boost module, check alignment, then solder the remaining 3 pins. Replace the display.
- Turn the power switch off (down position).
- Review the battery connections. On the back side of the PCB, holes for the battery leads are situated between the two switches, and labeled “Battery”.
- Consider how long your battery leads should be. In my case, I wanted a hand-held device, so I mounted the PCB on the back of my 3AA battery holder with double-sided tape.
- Cut the battery leads to length. Solder the red lead of your battery holder to “+” and black lead to “-”.
- **Test #7:** Make sure the USB-C cable is disconnected. Install fresh batteries and turn the power switch on. The green MCU power led should light, followed by the display. If not, refer to the troubleshooting guide. Check the voltage between the Vcc and ground pins on the display; you should see 5V here.
- Turn off power.



Done. Your LED tester is now complete!

LED tester – Final check outs

The following checkouts will require an accurate voltmeter and a known-good red LED

Power checkout

- Turn the unit on. The green MCU led indicates that power is present.
- On startup, the display should indicate 1000 ohms resistance and “NO LED”
- Verify that the top line of the display indicates the testing voltage. Toggle the voltage switch between its 5V and 3.3V settings. The display should update within 1 second of the change.
- Using a voltmeter, verify that the testing voltage, between the right leads of R2 and R1, corresponds to the voltage shown on the display. Please note that the 5V setting is unregulated, and may vary between 4.90 and 5.25 volts. This is normal.

Digipot checkout

- Create a short by inserting a jumper wire in the testing socket between LED anode and LED cathode. Verify that the display shows a V_f of “----” and mA of “Shrt”. Remove the jumper.
- Insert a known-good red LED into the testing socket, oriented with the cathode/short lead up and anode/long lead down. The LED should illuminate.
- Turning the encoder clockwise should dim the LED and increase the resistance towards 4700 ohms. Turning the encoder counter-clockwise should brighten the LED and decrease the resistance towards 200 ohms.

Forward voltage (V_f) checkout

- Return the resistance to 1000 ohms and check the displayed V_f . Common red LEDs have a V_f of approximately 1.8 – 2.0 volts
- Connect your voltmeter to the inserted LED leads, red to anode and black to cathode. Your voltmeter reading should agree with the V_f on the display.

LED current (mA) checkout

- Return the resistance to 1000 ohms, and connect your voltmeter across the leads of the 100-ohm resistor, R5 (red on left, black on right). The voltmeter should display under 1 volt. This value, multiplied by 10, should agree with the mA shown on the OLED display.
- Change the current by varying the resistance, and again verify that the voltage across R5 (multiplied by 10) agrees with the displayed mA.

How accurate is this device? The displayed V_f should be accurate to within 0.05 volts. The displayed current should be accurate to within 0.2 mA.

Troubleshooting

If you followed the step-by-step approach above you will have an idea of WHERE any problem is. Here are a few follow-up suggestions to isolate the problem further. When measuring voltages, the cathode side of the LED testing socket is a good location for ground: insert a jumper wire and attach this to the black lead of your DMM.

Problem	Suggestions
Test 1: The resistance measured across R6/R7 is not 200K	<ul style="list-style-type: none">a) Why 200K? The circuit between the test points is equivalent to a 400K resistor and a 400.1K resistor in parallel, for a combined resistance of about 200K.b) If no resistance can be measured, make sure there is good contact between the DMM and the testing points. If still no resistance, turn off your DMM's auto-ranging feature and select the mega-ohm scale. You should read 0.2 M-ohm.c) If you read 400K, then one of the resistor joints is bad. Reflow solder joints and recheck.d) If you see some other value, then the resistors are either in the wrong locations or have the wrong values. Confirm that R5 is the 100-ohm resistor, and that all other resistors are 200K.
Test 2: Inspection	<ul style="list-style-type: none">a) Nothing to go wrong here. Use magnification and good lighting. Adjusting the lighting to see the solder joints at different angles.
Test 3a: No green/orange MCU lights	<ul style="list-style-type: none">a) Green LED off: no power. Confirm that your USB-C cable is providing power. Remove the MCU from the board and reconnect the USB-C cable. If the LED illuminates now, it might mean a short on the PCB between power and ground. Inspect the PCB. Check for infinite resistance between the 3.3V MCU pin and ground.b) No orange LED flashes: your MCU has not been programmed correctly. Try reloading the sketch.
Test 3b: Incorrect Testing voltages	<ul style="list-style-type: none">a) Measure the voltage across the right leads of R1 and R2. This is the testing voltage. Your DMM should show 3.3V/5V depending on the voltage switch selection. If not, check the solder joints on the voltage switch.b) If your DMM does show 3.3V/5V, then the basic circuit is intact. Check the voltage at MCU pin 7. This voltage should be half of testing voltage. If there is no voltage here, check/reflow the solder joints on MCU pin 7, R1, and R2.
Test 4: No OLED display	<ul style="list-style-type: none">a) Remove and reseat the display to see if it is a poor connection.b) Check all of the microcontroller and display solder joints with good light and magnification.

	<ul style="list-style-type: none"> c) Carefully check the voltage between the first and second display pins (marked “GND” and “VCC” on the display. You should see about 5V d) With the device unpowered, use a voltmeter to check continuity between pin 3 of display (marked “SCL”) and microcontroller pad D5. Also, display pin 4 “SDA” and pad D4.
Test 5: No rotary encoder response.	<ul style="list-style-type: none"> a) Check/reflow the encoder solder joints b) Check/reflow MCU pins 1 and 2
Test 6: No/incorrect current readings on DMM.	<ul style="list-style-type: none"> a) If the display does not read “Shrt” and/or the DMM reading is incorrect, double check your DMM settings. On most DMMs, you measure current using a different jack than you use for measuring voltage or resistance. Set the meter for mA, not A or uA. b) Confirm that your MCP4261 IC is installed correctly, with no bent/missing pins, and the notch facing right. c) If the current does not change when you vary the resistance, it indicates that commands are not being passed from the MCU to the digipot. Check the SPI bus connections: MCU pin 11 to digipot pin 3, MCU pin 9 to digipot pin 2. With power off, check these lines for continuity. Inspect and reflow solder joints.
Test 7: No battery power	<ul style="list-style-type: none"> a) Replace the batteries. b) With the device off, measure voltage across the battery leads: 4.5V nominal for 3AA batteries, 3.0 for 2AA batteries, etc. c) Turn on device, and remeasure the battery voltage. It may decrease slightly, but not more than a few tenths of a volt. d) Measure voltage output of boost converter. The easiest place to measure is between VCC and ground pins on the display. This should read 4.9 to 5.25 volts.