Pocket Tutor

Build a handheld version Of the Morse Code Tutor

Part 6: Build it!

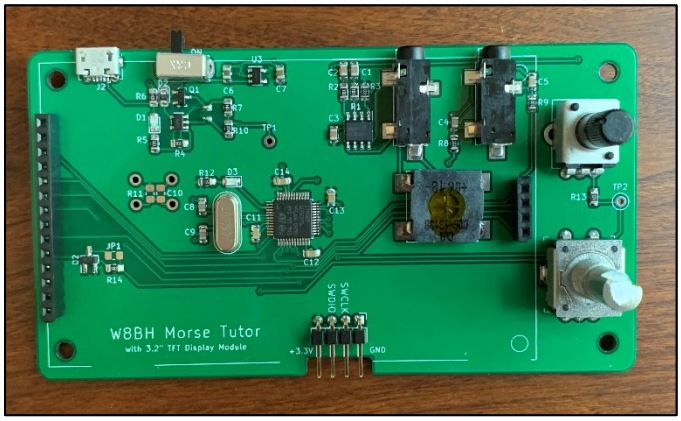
Bruce E. Hall, W8BH

Here is a set of instructions for building the Pocket Tutor, using a circuit board designed here at W8BH. Read on if you are interested in building this useful handheld device.

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 $5.90 + shipping for a set of 5 boards.

Step-by-Step building.

Before you start, make sure to obtain all the components you need. It is frustrating to get half-way through a build, only to realize that you are missing a key component! The Bill of Materials is below.

You should be comfortable with SMD soldering and have successfully assembled other projects using 0.5mm pitch components. You should have sufficient lighting, magnification, and workspace area. You will need a voltmeter for basic circuit checking. An oscilloscope may be helpful, but is not needed.

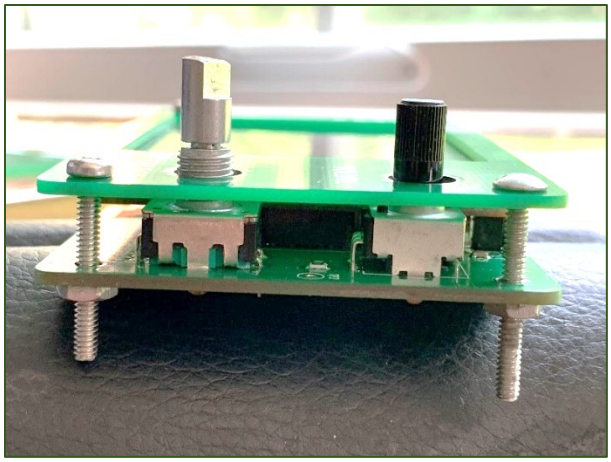
I believe in starting small. Build the smallest something that you can and test it. Don't forge ahead until you are sure everything is working. In this build we will start with the power circuit on the left-hand side of the board. We will gradually work our way across the board, testing as we go.

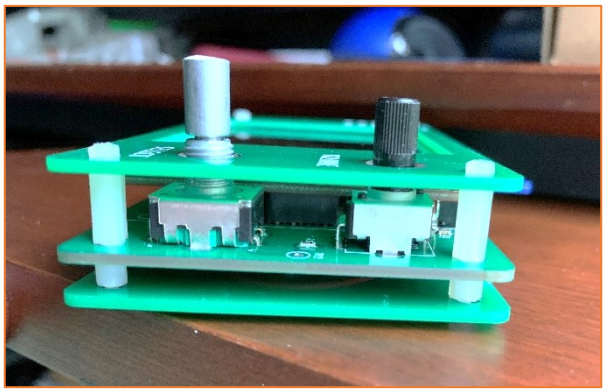
Enclosure.

Consider if you are going to enclose the board, and obtain the appropriate mounting hardware and enclosure material. The jacks and switches are placed on the top edge of the board to facilitate enclosure design, and the mounting holes are large enough (3.2 mm) to accommodate M3, M2.5, or 4-40 hardware. If you are facile with 3-D printing this might make a good project.

I hope that someone designs a nice enclosure for this project. Until then, I am using a "PCB sandwich": a front panel, main board, and back panel – all made of PCB material.

These side-views show the alignment of the top panel and main board. An optional bottom panel conceals the battery and reset switch.



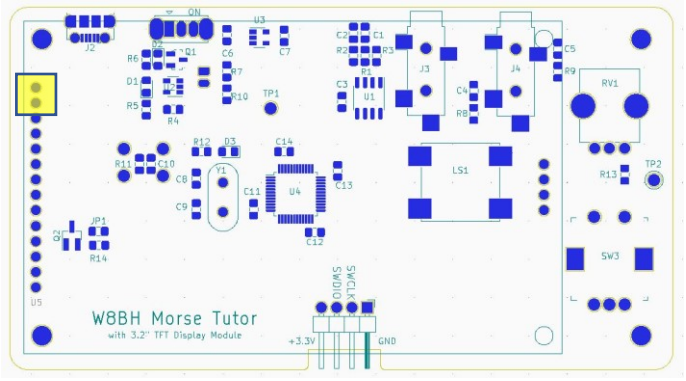


Bill of Materials.

Next, order the parts that you need for the build. Please note any part can go out of production, or change in specification over its lifetime. Here is a shared cart that contains most of the components listed below.

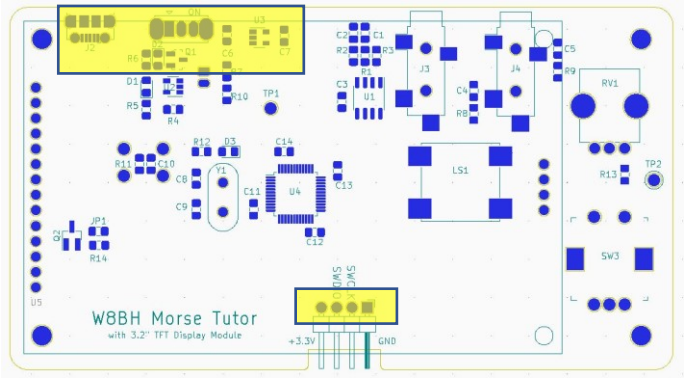
QTY	Description	Use	Digikey Part#
2	20pF capacitor, SMD 0805	C8, C9	311-4221-1-ND
4	10nF capacitor, SMD 0805	C4, C5	311-1136-1-ND
4	100nF capacitor, SMD 0805	C11, C12, C13, C14	399-1177-1-ND
4	1uF capacitor, SMD 0805	C1, C2, C3, C10	399-8011-1-ND
3	10uF capacitor, SMD 0805	C6, C7,Cextra	399-3138-1-ND
2	100 ohm resistor, SMD 0805	R1, R2	311-100CRCT-ND
2	680 ohm resistor, SMD 0805	R5, R12	311-680ARCT-ND
3	1 K resistor, SMD 0805	R8, R9, R14	311-1.0KARCT-ND
3	10K resistor, SMD 0805	R3, R4, R11	311-10.0KCRCT-ND
2	100K resistor, SMD 0805	R6, R13	311-100KCRCT-ND
2	200K resistor, SMD 0805	R7, R10	P200KDACT5-ND
1	Green LED, SMD 0805	D1	160-1423-1-ND
1	Red LED, SMD 0805	D3	160-1427-1-ND
1	Schottky Diode, SMD 0805	D2	478-7802-1-ND
1	DMP2054U-7 P-CH MOSFET, SOT23	Q1	DMP2045U-7DICT-ND
1	MMBT4403, PNP transistor, SOT23	Q2	MMBT4403LT1GOSCT-ND
1	PAM8302AAD audio amp, SO8	U1	PAM8302AADCRDICT-ND
1	MCP738310-2 charge controller, SOT23-5	U2	MCP73831T-2ACI/OTCT-ND
1	AP2112K-3.3 regulator, SOT23-5	U3	AP2112K-3.3TRG1
1	STM32F103CBT6 microcontroller, LQFP48	U4	497-17380-1-ND
1	8 MHz crystal, HC49-U	Y1	535-9062-ND
1	SPDT Slide Switch, CK_OS102011MA1QS1	SW1	CKN9560-ND
1	6mm Push Button Switch	SW2	450-1649-ND
1	10K potentiometer, Bourns PTV09A-1	RV1	PTV09A-4020U-B103-ND
1	Rotary Encoder, Bourns PEC11R	SW3	PEC11R-4225F-S0024-ND
2	Audio Jack, CUI SJ-3524	J3, J4	CP-3524SJCT-ND
1	Speaker, CUI #CDS-13138-SMT	LS1	102-3536-1-ND
1	MicroUSB Jack, Molex 105017	J2	WM11262CT-ND
1	JST-PH Rt angle, 2 Pos Jack	J1	455-1719-ND
1	Low-profile female header		Adafruit #3008, #3009
1	Lithium Polymer Battery (150-2000mAh)		Adafruit #258 or similar
1	Right-angle 0.1” male header	J5	Adafruit #1540 or similar
1	MSP3218 3.2” SPI TFT Display	U5	eBay or Alibaba

1. **Prepare**
    a. Check display pins 1 and 2: no continuity
    b. Check board for display pins 1 and 4: continuity
    c. Be careful with U2 (charger) and U3 (regulator): both are SOT23-5 packages and are easily confused. My U2 is marked “KDPK” and U3 is marked “G3P”.
    d. Also be careful with Q1 (MOSFET) and Q2 (PNP): both are SOT23-3 packages



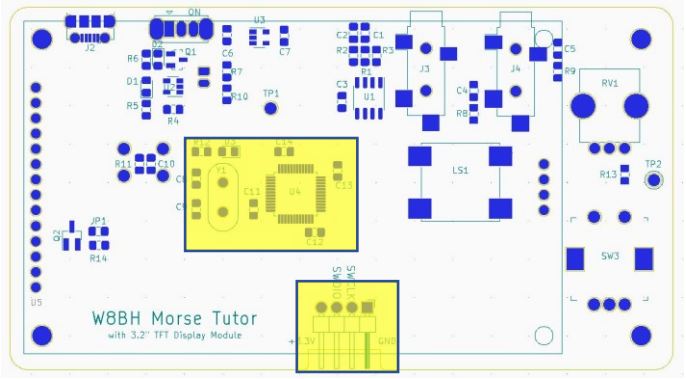
Display pins 1 & 2

2. **Build the USB Power Supply**
    a. Solder micro-USB connector
    b. Attach USB Power, check voltage across display pin 2 (gnd) and top pin of D1 (+5V)
    c. Solder R6 (100K), D2, Q1, C6 (10uF), U3, C7 (10uF).
    d. Solder SW1. Attach USB Power and turn switch ON
    e. Check voltage across display pins 1 and 2, and also between programming header pins 1 and 4. Both should read 3.3 volts. If not, refer to Troubleshooting Step 2 below before continuing.
    f. Your power supply works! Turn the switch off and remove USB power.



USB Power Supply; programming header pins

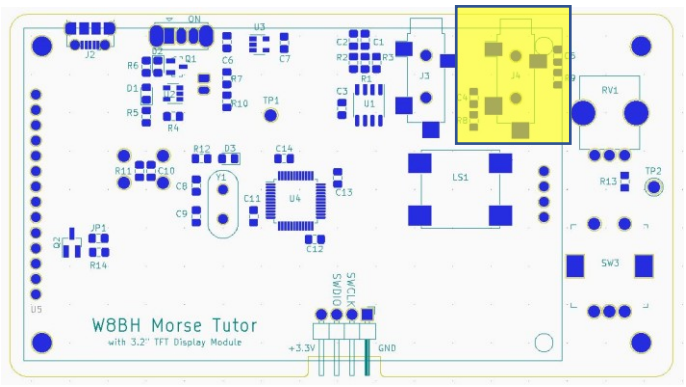
3. **Build Microcontroller circuit**
    a. Solder U4, the STM32 microcontroller.
    b. Check all pins for good connection, no solder bridges.
    c. Solder C8-C9 (20pF), R12 (680), D3(red), C11-C14 (100nF).
    d. Solder right-angle programming header J5



Microcontroller circuit; programming header

4. **Check Microcontroller**
    a. Reapply Power and turn switch ON
    b. Attach ST-LINK dongle to computer, then to programming header
    c. Run ST-LINK utility and connect to target. Does it see the microcontroller?
    d. Close utility and run Arduino IDE. Set board to generic STM32F103C, variant = 128K, upload method ST-link.
    e. Upload TUTOR_TEST sketch. If D3 double-blinks every 3 seconds, CONGRATULATIONS!! If not, refer to Troubleshooting Step 4.

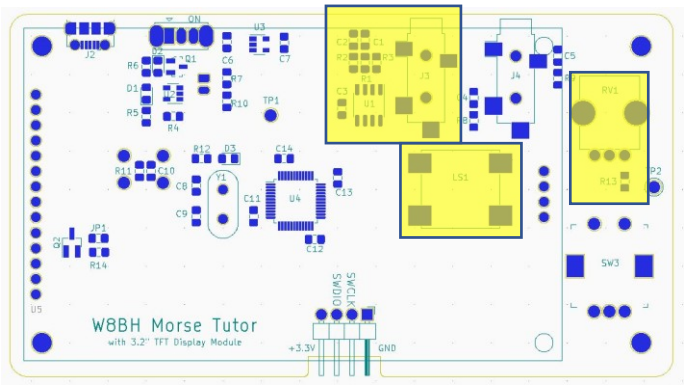
5. **Build Morse Key circuit**
    a. Solder R8, R9 (both 1K), C4, C5 (both 10 nF) and jack J4. NOTE: on rev2 boards, the jack is slightly recessed, and may not make good contact with some plugs. Try to position the jack as close as possible to the top edge of the board.



Morse Key Circuit

    b. Turn on, plug in key, and check result. Holding down left paddle (dit) changes flash pattern from 2 short flashes to 1 short flash. Holding down right paddle (dah) changes flash pattern to 2 long flashes. Holding down both paddles changes flashing pattern to 1 long flash. If it does not, go to Troubleshooting Step 5.
    c. Turn off and remove power.

6. **Build Audio circuit**
    a. Solder R1, R2 (both 100), R3 (10K), C1, C2, C3 (all 1uF), U1 (PAM8302).
    b. Solder R13 (100K), jack J4, and speaker LS1. NOTE: on rev2 boards, the jack is slightly recessed, and may not make good contact with some plugs. Try to position the jack as close as possible to the top edge of the board.

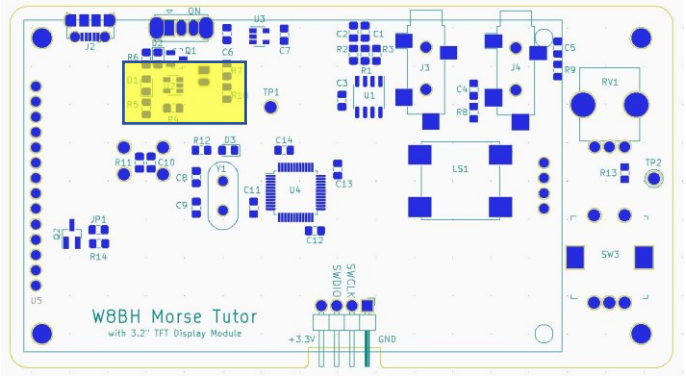


Audio Circuit

    c. Solder volume control RV1.
    d. Turn volume control counterclockwise all the way (mute)
    e. Attach power and turn on. Slowly turn volume clockwise to hear double-beep. Refer to Troubleshooting Step 6 if necessary.

f. Turn off and remove power.

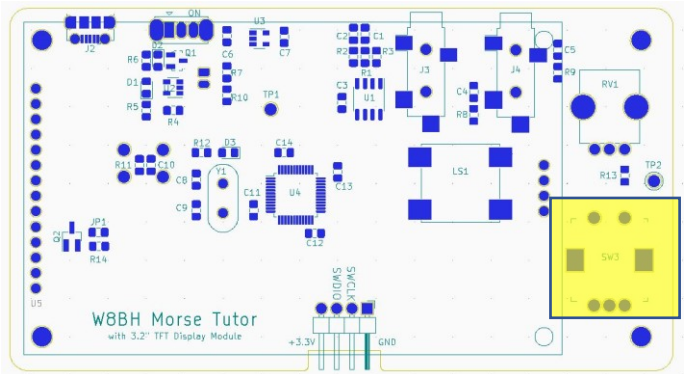
7. Build Lipo charger circuit



Lipo Charger

a. Solder R4 (10K), R5 (680), R7 & R10 (200K), D1(green), U2
b. Solder lipo battery jack J1 on rear of board.
c. On rev2 boards, install an additional 10uF capacitor between the pins of the lipo battery jack.
d. With no USB and switch turned off, connect Lipo battery. Make sure red wire of Lipo battery aligns with (+) label and black wire aligns with (-).
e. Turn switch on. LED should flash and speaker should beep.
f. Check voltage at TP1. It should register ½ the battery voltage. For example, a fully charged Lipo battery at 4.0V should give a TP1 reading of 2.0V. A discharged battery at 3.4V should give a TP1 reading of 1.7V.
g. Apply USB power. If the battery is not fully charged, D1 charging LED should light.
h. D1 will extinguish when battery is fully charged. See Troubleshooting Step 7 if needed.
i. Turn off and disconnect battery.

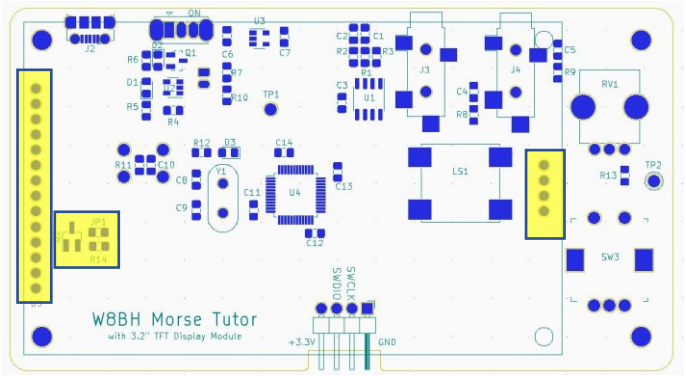
8. Rotary Encoder



Rotary Encoder

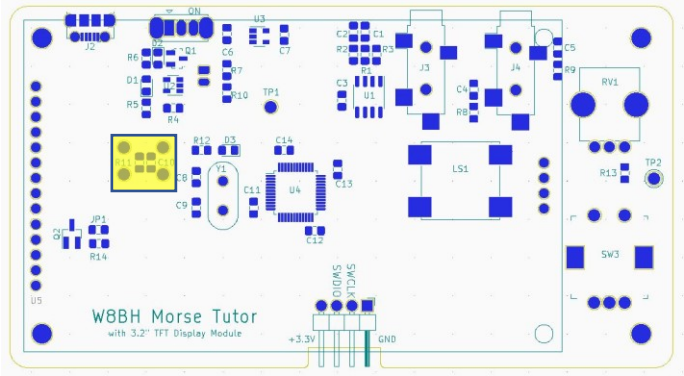
a. Solder the rotary encoder.
b. Apply power and turn on switch
c. Rotate the encoder. The pitch of the beep should increase/decrease with encoder rotation.
d. Push the encoder button. The pitch should return to 700 Hz.
e. Turn switch off and remove power.

9. **Build the Display circuit**
    a. Solder R14 and Q2 (Alt: if display has on-board backlight control, solder across JP1 instead)
    b. Solder low-profile 14 pin and 4 pin female headers.
    c. Install LCD display on headers.
    d. Apply power and turn switch on.
    e. The display will show brightness level, pitch, paddle status, and battery voltage.



Display circuit

10. **(Optional) Build the reset circuit**
    a. Temporarily remove the display
    b. Solder R11 (10K) and C10 (1uF).
    c. Solder pushbutton SW2 to the rear face of board.



Reset Circuit

Finishing touches

The main PCB is now complete. To finish this project, download the latest software from my github account and install. At a minimum, consider applying rubber feet to the rear face of the board. This will protect any surface it rests on, and help prevent shorted connections. You could also use standoffs to elevate the board. Use taller standoffs along the top of the board to tilt the display. Finally, consider an enclosure to protect and enhance your handiwork.

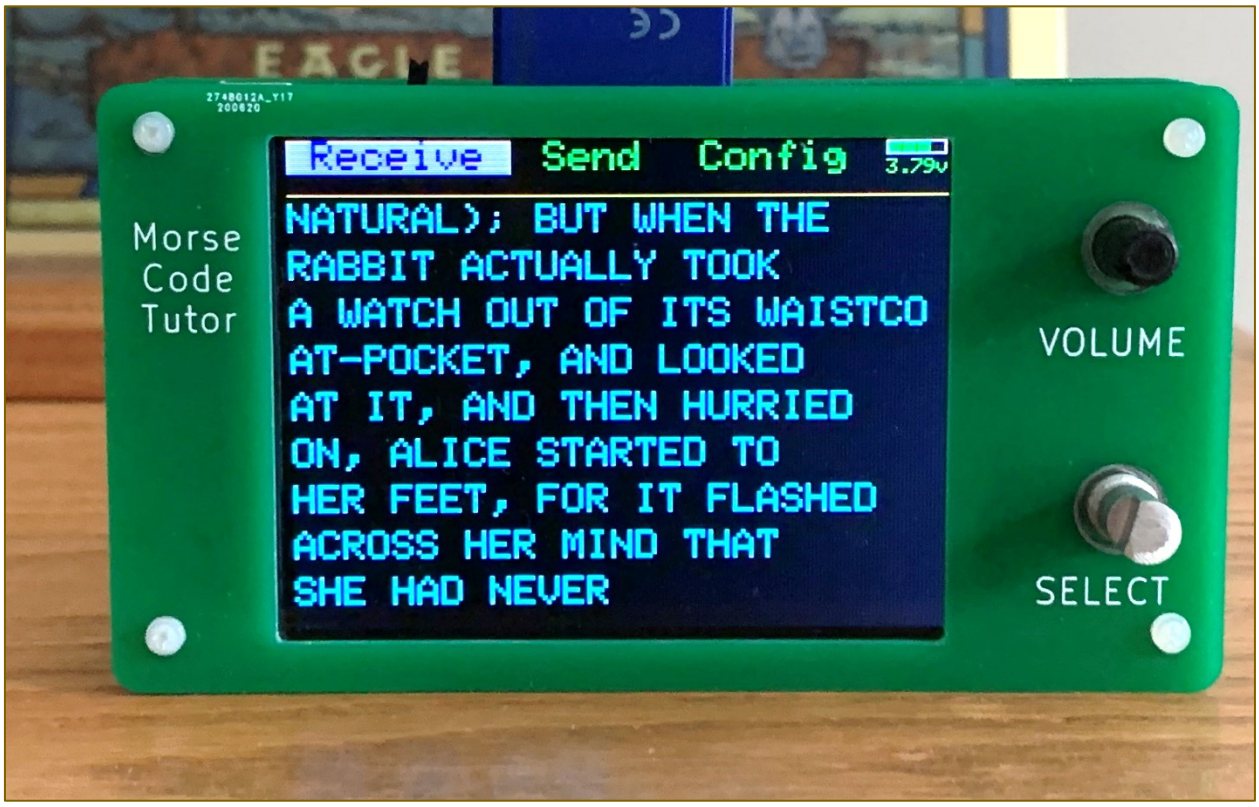
Troubleshooting

I find most troubleshooting sections lacking. They never seem to include the problem I am experiencing. If you followed the step-by-step approach above you will have a good idea of WHERE the problem is. Here are a few follow-up suggestions to isolate the problem further.

Problem	Suggestions
Step 1: The board fails its continuity tests	a) Make sure your continuity tester is working correctly. If pins 1 and 4 are not shorted, your tester may be faulty. If pins 1 and 2 are truly shorted, the board is faulty and you should not use it.
Step 2: The voltage does not read 3.3V	a) First, make sure that your USB power supply is plugged in and you are using a known-good micro-USB cable. Put the switch in the off position. b) Check the voltage across resistor R6. Make sure it is ~5.0V on the lower edge and 0V on the upper edge. If it is not, then your USB connector is not installed correctly. Remove and reinstall. For me, the USB connector is the most difficult part to solder. Take your time and avoid solder bridges. c) Check the voltage across Schottky diode D2. The lower edge should read 5.0V. The voltage at the upper edge should be about 4.7V. If it is less than 4.5V, the diode is faulty. If it reads 4.9V or higher, check for shorts across D2 or Q1. d) Turn the switch ON. Voltages in b)-c) above should remain the same. If they do not, check for shorts around U3. Voltages at the bottom edge of C6 and C7 should be ~4.7V and 3.3V, respectively. If necessary, remove and replace U3.
Step 4: No double-blink	a) First, double check the microcontroller hardware. Use magnification and good light to exclude solder bridges or unsoldered pins on the microcontroller. b) With power ON, confirm 3.3V across the pins 1 and 4 of the programming header. If less than 3.3V, something in the microcontroller circuit is shorting. c) If you have an oscilloscope and good 10x probe, set the timebase to 50nS and probe the lower crystal lead with respect to ground. You should detect 8MHz oscillation with >1V peak to peak. d) Re-run ST-LINK. With dongle connected to computer but not to Pocket Tutor, Select Target->Settings. Verify that the utility sees the ST-LINK dongle. If not, reconnect and refresh. e) Now, in the following order, disconnect dongle from computer, connect ST-LINK dongle to programming header, and reconnect dongle to computer.

	f) Re-run ST-LINK and select Target->Settings. You should see the ST-LINK serial number, as before, and the STM32 Target should indicate "STM32F10xx Medium-density". If not, consider the following, in order of likelihood: programming cable from dongle to header is incorrectly installed, microcontroller is incorrectly soldered, programming header is improperly soldered, or microcontroller is faulty. g) Run ST-LINK and select "Connect to Target". Check for errors. The device ID should be 0x410. If you get this far, your microcontroller is working and is programmable.
Step 5: Morse key input does not change LED flash	a) Check the following, in order of likelihood: the key plug is not fully seated in the key jack, the key wires are disconnected/shorted, or there is a short or disconnect involving microcontroller pins 43 and/or 45. The latter actually happened to me. b) Remove the key plug and carefully and briefly short the top of R8 to ground. If the LED does not change from 2 short flashes to 1 short flash, the DIT circuit is faulty: check C4, R8, uC pin 45. c) Carefully and briefly short the top of C9 to ground. If the LED does not change from 2 short dits to 2 long dits, the DAH circuit is faulty: check C5, C9, uC pin 43.
Step 6: No Audio	a) This is fun to troubleshoot! Do you have any test equipment, like an oscilloscope, to trace an audio signal? b) Before trying anything else, check the board to make sure your diagnostic LED is still double-blinking. This verifies that the diagnostic sketch is still running and an audio signal is also being produced. c) Place your scope/audio probe on TP2 (located between the potentiometer and encoder). You should see/hear 700 Hz beeps if microcontroller pin 12 is correctly soldered and the audio circuit is not shorted. If removing R13 results restores the signal, there is a short in the audio circuit. d) Follow the audio across resistor R13. Continue following the signal across each step below. e) Follow the audio to the right pin of the volume control. f) Follow the audio to the center pin of the volume control g) Follow the audio to the PAM8302 pin 3. h) Connect your audio probe to the PAM8403 output pins 5 and 8. If the audio stops here you may have a faulty IC. i) Connect your probe across the upper two speaker pins. If no audio is here, the ext. speaker jack may be faulty. If audio is here, the speaker itself may be faulty (try an external speaker through the jack instead).

Step 7: Lipo Battery issues	a) Remove any USB cable. Connect a fully charged Lipo battery and carefully check the voltage across its pins (J1). A fully charged battery should read 4.1-4.2V. b) With the switch turned off, the voltage on the center pin the switch should read close to the battery voltage measured in a). If not check Q1 connections. Both the source and drain pins of Q1 should be at battery voltage levels. c) Verify that the voltage at base of R6 is less than 0.1V. If not, your U2 Lipo Charger IC is faulty. A shorted D2 could cause the same probable, but less likely. d) If all voltages seem correct but the LED does not light, check its orientation. The band or colored dot(s) should be toward the bottom side of the LED.
Step 9: Display not working	a) Does the display backlight work? If not, remove power and check voltage on the right side of R14 (backlight control). Even better, use an oscilloscope. Set the timescale to 20uS and 1V/div. You should see a 10 kHz square wave with 3V amplitude and 50% duty cycle. If not, check the following: uC pin 19, R14, Q2, display pin 8. b) With the display plugged in, you should see the same signal at display pin 8. If not, recheck R14 and Q2. c) If the backlight works but there is a blank screen, then either the display is faulty, or the data signals are not making it to the display. Check the following connections: uC pins 10, 11, 15-17; display pins 3-7.



Resources.

Here are links to this project's documentation:

- Part 1: Overview
- Part 2: Power Supply
- Part 3: Microcontroller
- Part 4: Audio
- Part 5: PCB Design
- Part 6: Build Instructions
- Latest Source Code
- Schematic
- Gerbers
- Morse Code Tutor series

I hope you have enjoyed this project as much as I have. Please send me a photo of your work!

73, Bruce.