Morse Code Tutor -
From the ground up.

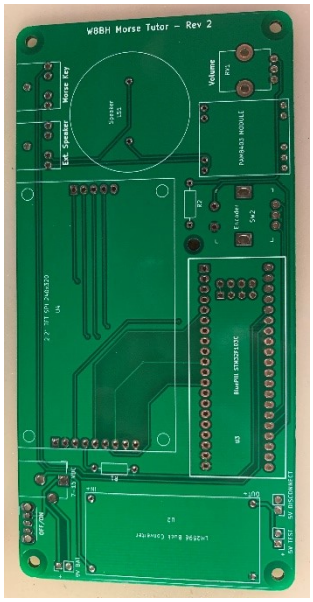
Part 9: Build it!

Bruce E. Hall, W8BH

After you've breadboarded your project, you may want to build it in a more permanent fashion. Here are a set of instructions for building the Morse Code Tutor using a circuit board designed here at W8BH. Read on if you are interested in building this inexpensive and useful 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 $7.30 + shipping for a set of 5 boards.

Step-by-Step building.

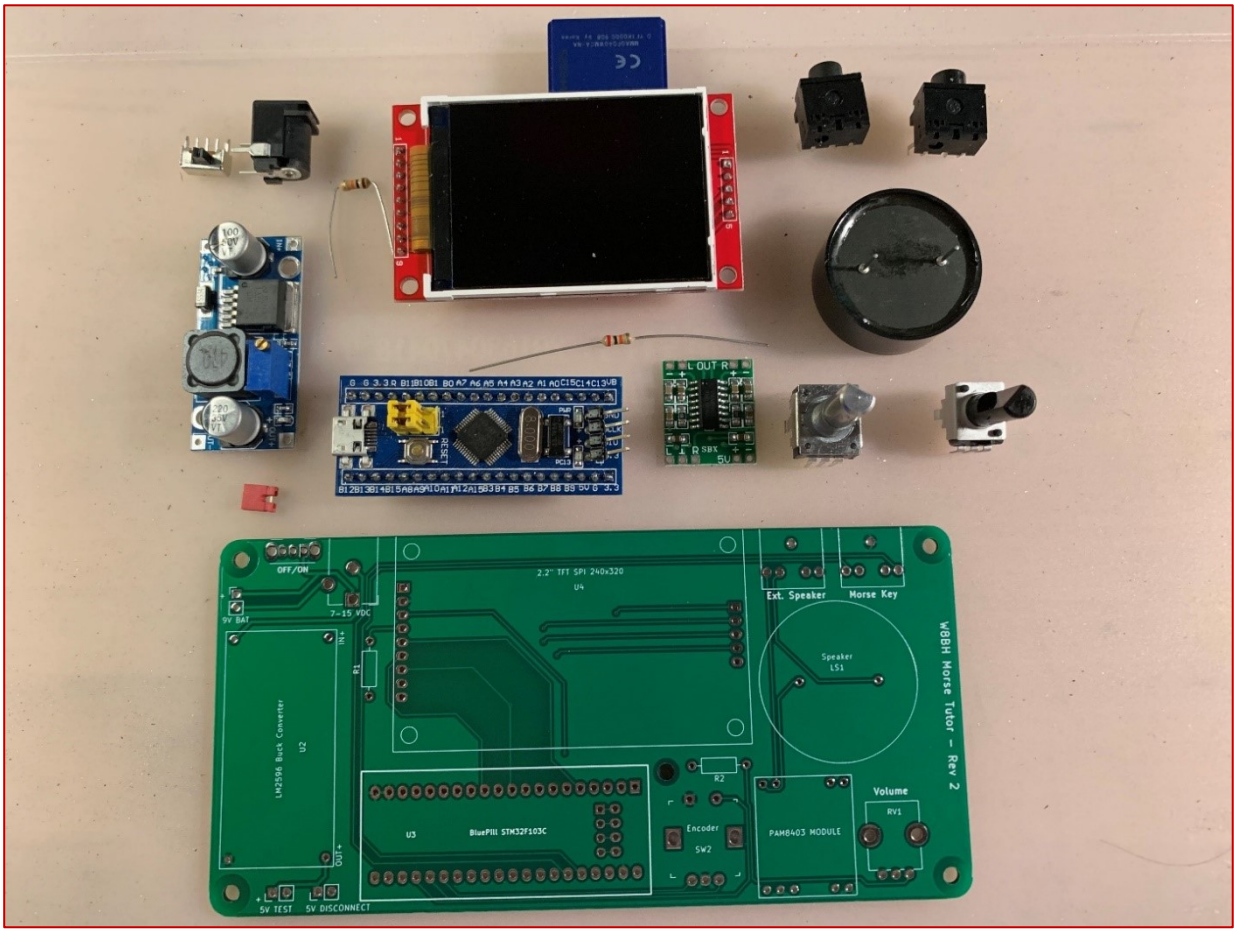
Before you start, make sure to obtain all the components you need. It is frustrating to get half-way through a build, only to realize that you are missing a key component!

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.

As I mentioned at the start of the series, 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.

Bill of Materials.

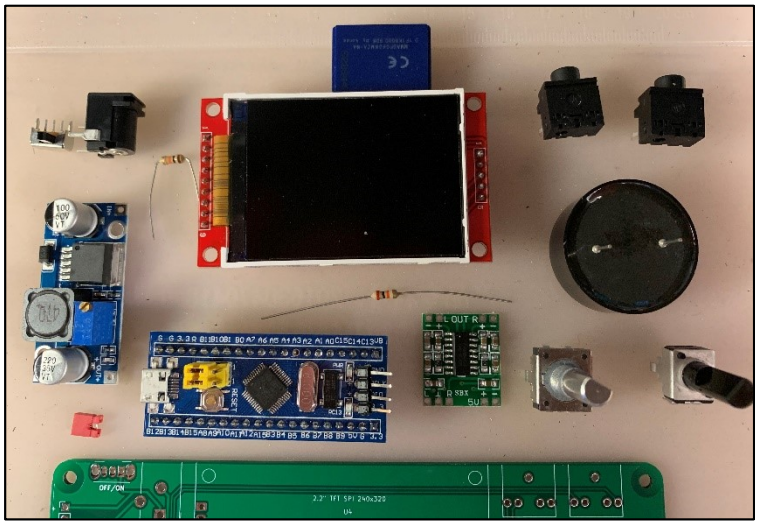
Next, order the parts that you need for the build. The following is a list of parts, as well as part numbers and/or links for a couple of my favorite suppliers. Please note any part can go out of production, or change in specification over its lifetime.



Part	Qty	DigiKey	Mouser	Prices/Other
2.2” TFT Display Module, ILI9341	1	n/a	n/a	Search Ebay ($6-$8)
Blue Pill Board	1	n/a	n/a	Search Ebay ($2-$5)
LM2596 Buck Converter Board “HW-411”	1	n/a	n/a	Search Ebay ($1-$2)
PAM8403 Audio Amplifier Board	1	n/a	n/a	Search Ebay ($1-$2)
1/8” Stereo Jack	2	2092-STX-3120-5B-ND	806-STX-3120-5B	Mouser $0.61 Adafruit $0.95 Digikey $0.73
Speaker, PCB Mount, 30mm	1	668-1136-ND	665-AST03008MRR	Mouser $3.26 Digikey $3.15 Alibaba $1.84 for 2 Sparkfun $1.95 Adafruit $1.85
Bourns PEC-11R Rotary Encoder	1	PEC11R-4220F-S0024-ND	52-PEC11R-4220F-S24	Digikey $1.68 Mouser $1.63 Adafruit $4.50
Barrel Jack PJ-102A	1	CP-102A-ND	490-PJ-102A	Digikey $0.64 Mouser $0.64
10K Potentiometer, Bournes PTV09A, logarithmic taper	1	PTV09A-4025F-A103-ND	652-PTV09A4025FA103	Digikey $0.83 Mouser $0.78
SPST slide switch, side mount (C&K 0S102011MA1QS1)	1	CKN9560-ND	611-OS102011MA1QS1	Mouser: n/a DigiKey: $0.39 Jameco $0.49
Shorting block	1	952-2169-ND		Jameco $0.15 Digikey $0.19
0.1” female header	2	HDR100IMP40F-G-V-TH-ND		Digikey $1.09
Misc: 0.1” male header pins, knobs, 10K resistor, 20K resistor, mounting hardware, 9VDC power adapter with 2.1mm DC jack, SD card (optional)				

These parts should cost you about $20 + shipping. Please check the list of miscellaneous components. Do you have these already? If not add these to your list.

Step 0: Preparation.



- Follow the Morse Tutor series. Program your Blue Pill microcontroller with a current copy of the sketch. Satisfy yourself that the Blue Pill is programmed correctly.

- Obtain the PCB and necessary parts as outlined above.

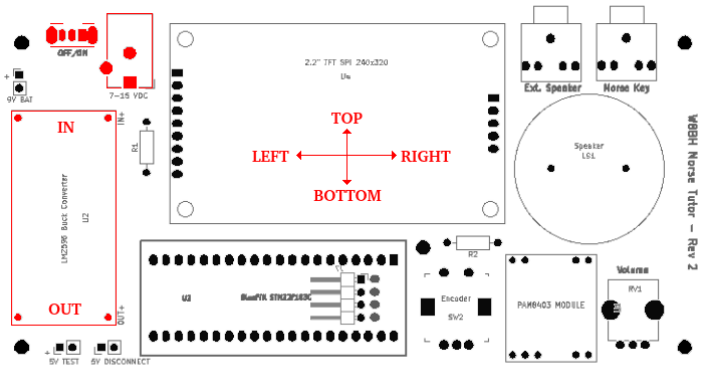
- Familiarize yourself with the circuit board. The components on the left side of the board are concerned with power regulation. The middle components are for processing and display. And the components on the right side are for audio output.

- Consider how you intend to power the board. The voltage input is 7-15 VDC with a current draw of about 150 mA. I recommend a 9 VDC power adapter. The power connector should be a 2.1mm barrel jack, center pin positive.

- 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 mm) to accommodate M2.5 or 4-40 hardware. If you are facile with 3-D printing this might make a good project.

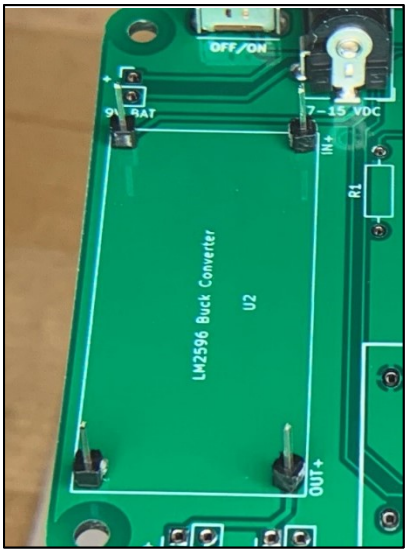
- Consider if you are going to socket the display and microcontroller, or permanently solder them in place. The instructions assume that you'll socket both components so that they can potentially be reused. The microcontroller is also easier to reprogram if socketed. However, keep in mind that female headers add 8.5mm height to each component, potentially reducing the amount of clearance that the encoder and volume shafts will have above an enclosure face. I provide alternate instructions if you want to directly mount the microcontroller (see Step 4 "the hard way").

Step 1: Power Supply.



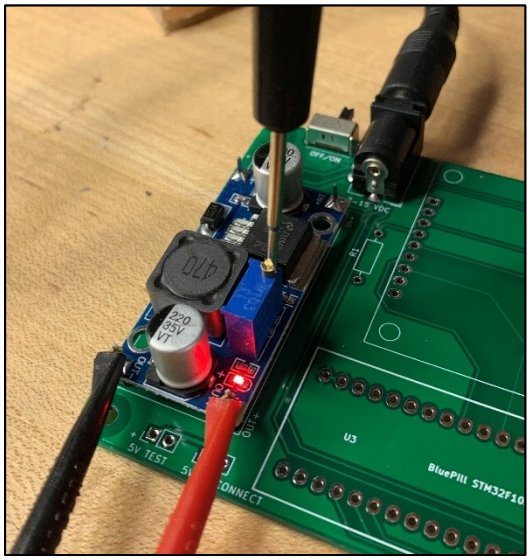
- Orientation: (almost) all of the parts in this project are soldered to the front face of the board – the face with component outlines and labels. As you look at the board facing you, I refer to the four edges as left and right, top and bottom.

- Solder the barrel connector and on/off switch in their correct positions near the top-left corner of the board. The switch is oriented such that the black-plastic slider is positioned above the top edge of the board, *not* facing towards the bottom edge. Put the slider in the OFF position, towards the left.



- Solder the buck converter module using 4 individual male header pins: Orient these pins with the longer end "up", as shown in the photograph. Place the converter on the loose pins. Make sure that the word 'IN' on the converter matches the word 'IN' on the PCB. Solder the top-side first, then flip the board over and solder the bottom-side. Don't clip the bottom two header pins; we will use these to check the voltage output.

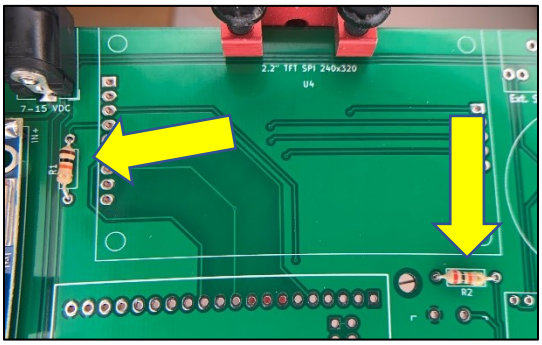
- **Test #1:** plug your DC power supply into the barrel connector. Briefly turn the switch to the ON position. The LED on the buck converter should light up. If not, turn the switch off and disconnect power. If the LED lights then the converter is getting power. Clip your voltmeter across its OUT- and OUT+ pins near the bottom-left of the board.



- Use a jeweler's screwdriver to adjust the blue square potentiometer until you get a reading of 5.0V. Turn the switch off and unplug your power adapter.

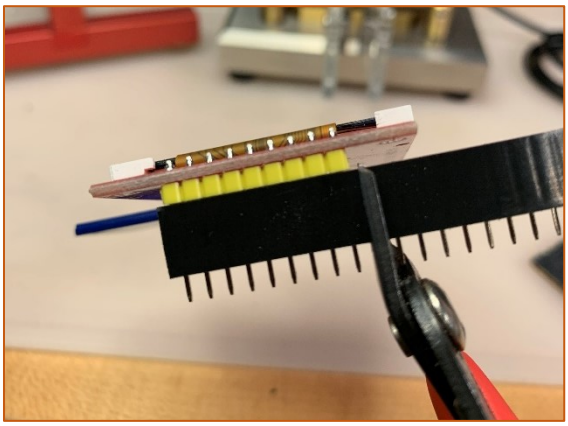
Step 2: Install Resistors R1 and R2

The 10K resistor goes to the left of the display. The 20K resistor (R2) goes below the display, near its bottom-right corner.

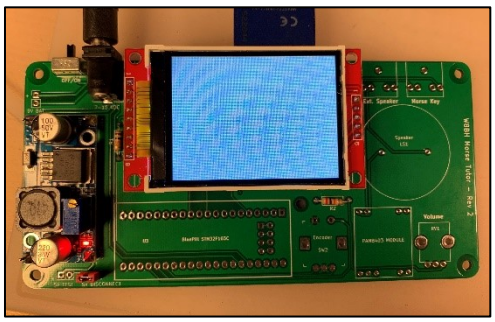


Step 3: Install the 2.2" Display

- Cut and install a 2-pin male header at the bottom of the board where it says "5V Disconnect".
- Install a shorting jumper across the pins you just installed. IMPORTANT: the circuit board will not have power unless you install the jumper. Don't omit this jumper.

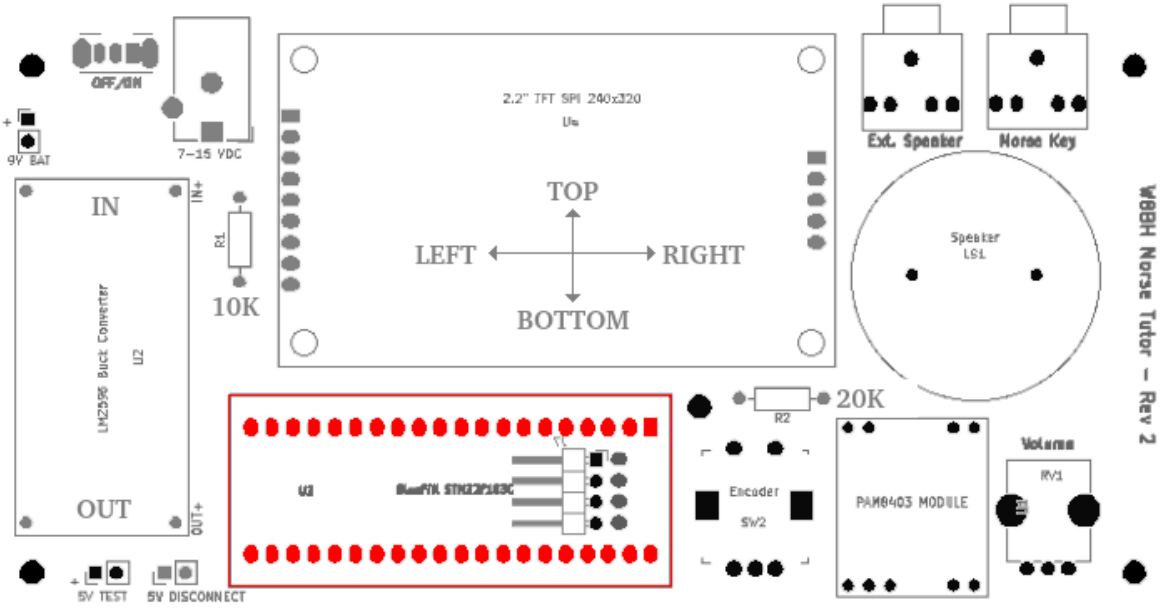


- Cut two lengths of 0.1" female header (one with 9 holes, one with 5) to match the display pins. It is difficult to make a perfect cut between two pins using diagonal cutters, so try this: cut directly over a pin, sacrificing it. The metal pin will fall out, leaving excess plastic on the side you just cut. You may file/sand this down, if you like, or leave it as is.
- Attach the headers on the display, which will hold them in square, and then solder the headers to the PCB.



- **Test 2:** reconnect your power adapter and briefly turn the switch ON. The display backlight should light. Does it? If not, see Troubleshooting below. Turn the switch off and unplug your power adapter.

Step 4: Install the BluePill microcontroller board (the easy way)



- There are two methods to install the blue pill board: the first way is easiest and allows you to easily swap out your board if it goes bad or needs re-programming. An alternate method (below) is more difficult, but allows in-circuit programming via a new 4-pin ST-LINK header

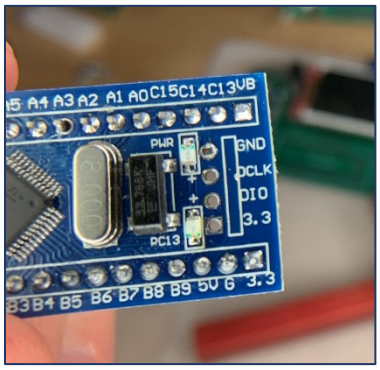
- Cut two lengths of 0.1" female header, each with 20 holes, as you did in the step above. Mate them with the microcontroller board, then solder to the board. The USB connector on the Blue Pill microcontroller board should be facing LEFT, toward buck converter. (*Note: if you are installing an ESP32 controller, it uses the opposite orientation. See Appendix A.)*

- **Test 3:** reconnect your power adapter and briefly turn the switch ON. The buck converter and LED and display backlight should still light. In addition, the power LED on the microcontroller should light.

- After a brief pause, the PC13 LED on the microcontroller should flash short-long-short (Morse 'R') and the display should show the start-up screen. If it does, stop and congratulate yourself! If it doesn't, repeat step 0 above and make sure the Blue Pill is programmed correctly.

- Turn the switch off and unplug your power adapter. Go to Step 5, Installing the Encoder.

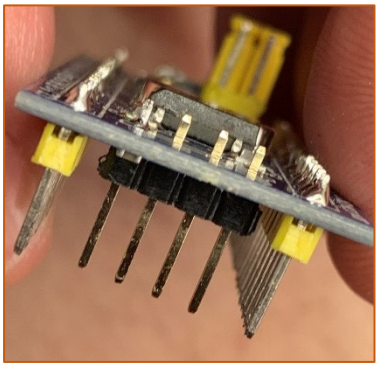
Step 4: Install the BluePill microcontroller board (the hard way)



- Do not attempt this unless you are comfortable soldering and desoldering in tight places!

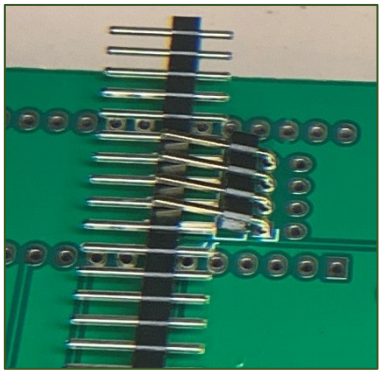
- Remove the existing 4-pin right-angle STLINK header on the Blue Pill. A suggested method:

    1. Carefully bend the pins into a vertical position.
    2. Remove the breakaway plastic spacer with diagonal cutters.
    3. Desolder each pin separately, applying a little extra solder at the base of each pin to free it.
    4. Clean up each hole with flux/desoldering braid.



- Install a new 4-pin vertical male header into the clean holes you just created. Solder only 1 pin to tack it into place. When the microcontroller board is fitted to the PCB you will solder the remaining pins.

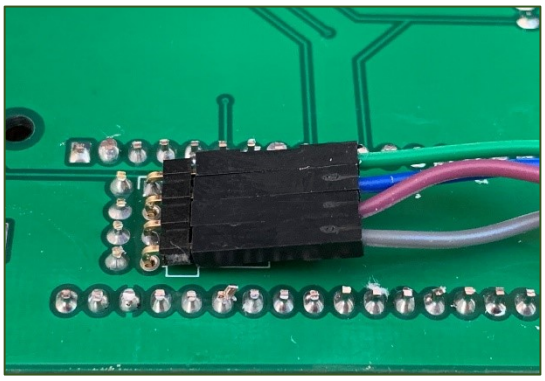
- Solder a 4-pin right angle header to the back face of the PCB as shown in the third photo. These pins allow you to reprogram the Blue Pill using an ST-LINK device. Use another header strip to slightly angle these pins from the board, making it easier to attach a cable to them.



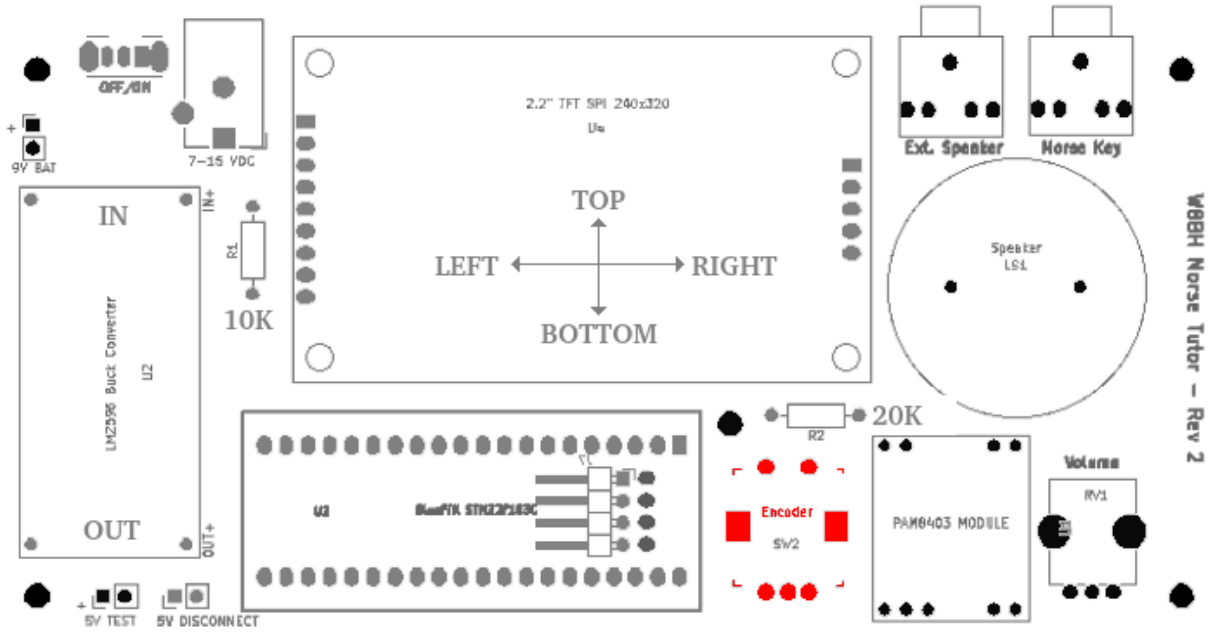
- Fit the Blue Pill directly onto the PCB. Solder in place, and finish soldering the new 4-pin header above and below.

- Perform Test #3 as above.

The new header allows you to attach an ST-LINK cable for in-circuit programming.



Step 5: Install the Encoder



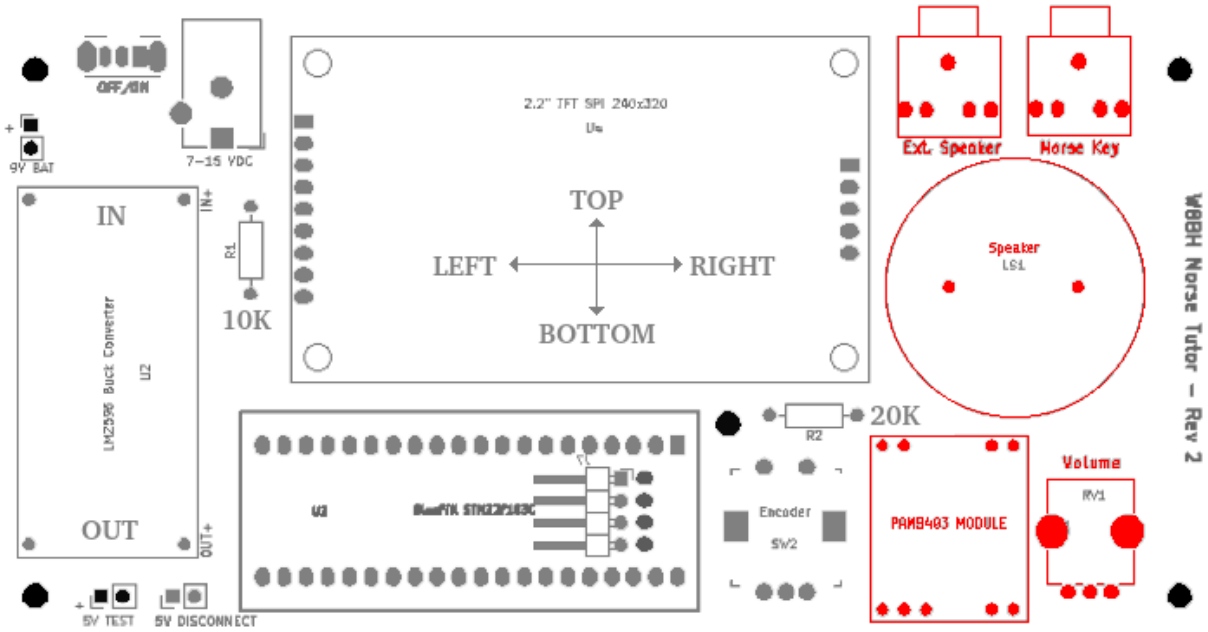
- The encoder pins are a tight fit: slightly bend the retaining clips and the encoder pins so that they are perpendicular to and align with the PCB holes. They do fit. Take care to apply enough pressure to seat the encoder but not bend the pins. Do not rush.

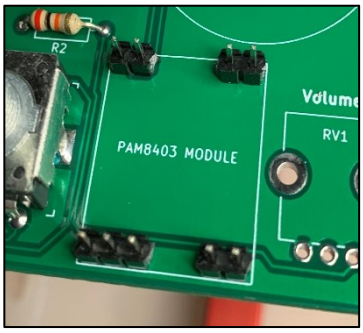
- Solder the encoder and apply power as usual. The startup screen should appear as before.

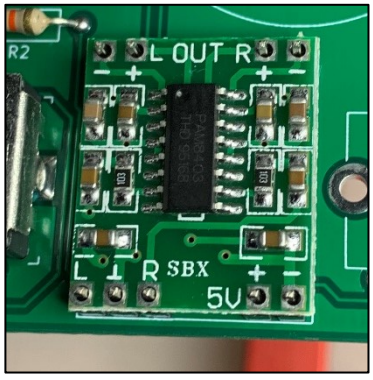
- **Test 4:** Turn the encoder knob, and the menu selection should scroll. Push the encoder button, and the menu should change to reflect your choice.

- Turn the switch off and unplug your power adapter.

Step 6: Install the Audio Circuit







- Install the PAM8403 amplifier module using 0.1” male header pins: First, fit the header pins into the PCB, short-side up.
- Next, fit the PAM8403 module over those loose pins and solder on top.
- Turn the board over and solder the pins on the bottom.
- Install the potentiometer & solder. Like the encoder, the fit is tight and you may need to slightly bend the leads before inserting them into the board. Do not rush.
- Turn the potentiometer knob fully counter-clockwise (mute).
- Install the speaker & solder. Please ignore any (+) or (-) polarity markers on the speaker pins, if present.
- Install the two stereo jacks at the top right corner of the board.
- **Test #5:** Apply power as above. Each time you start, advance the potentiometer slowly clockwise until you can hear the startup Morse ‘R’. You should hear ample volume by the time the shaft has been turned 60 degrees (bevel-side down). Note: the board-mounted speaker is not suitable for loud-volume; use an external speaker for this instead.

Step 7: Finishing touches

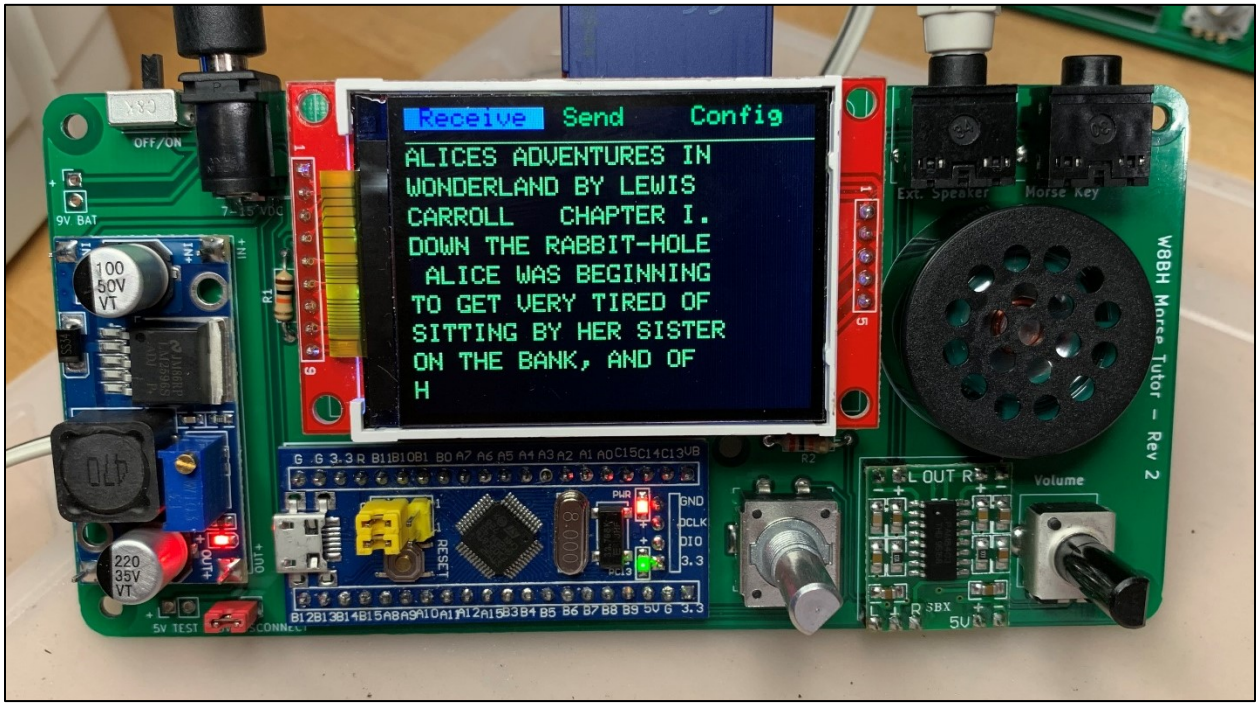
- Clip off excess leads on both faces of the board.
- At a minimum, consider applying rubber feet to the rear face of the board. This will protect any surface it rests on, and help prevent shorted connections.
- You could also use standoffs to elevate the board. Use taller standoffs along the top of the board to tilt the display.
- A bottom-half enclosure would protect the board and allow you to see 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 an idea of WHERE the problem is. Here are a few follow-up suggestions to isolate the problem further.

Problem	Suggestions
Test 1: The Buck Converter LED doesn't light	a) Make sure that your power adapter is in working condition and the wall plug has power! b) Use a voltmeter to test across the front and back barrel connector pins. You should see your input voltage. If not, check your adapter again. c) Turn the switch to the ON position and test the voltage across the buck converter's IN+ and IN- pins. You should see the same input voltage as in (b). d) Check the converter output voltage by putting your voltmeter across the OUT+ and OUT- pins. You should see 5V here. If you see some other voltage, try adjusting the blue potentiometer on the buck converter. If you have a normal input voltage above but no output voltage, your converter may be faulty.
Test 2: The display backlight doesn't light	a) Make sure that you have installed the shorting jumper across the "5V disconnect" pins. Unless these pins are shorted, the 5V output from your buck converter will not pass to the rest of the circuit b) Remove the display and redo Test #1. If the buck converter LED lights then you have power. Make sure it is 5.0V. c) Insert a small resistor lead or male header pin into the Vcc and GND pins of the display (these are pins 1 & 2 – the top pins on the left row of 9). Connect your voltmeter to these pins - you should see 5V here.

	d) Now move the lead on Vcc to the display pin 8 on the left. You should see power here as well. If the power checks out, reinsert the display. The backlight should light if is there is 5V between the LED and GND pins. If not, you may have a faulty display.
Test 3a: The microcontroller LED doesn't light	a) It is unlikely that the display is powered but the microcontroller isn't. Remove and reseat the controller board to see if it is poor connection. b) Check all of the controller solder joints with good light and magnification. c) Carefully check the voltage between the controller pin marked "5V" (bottom row, 3rd from right) and ground.
Test 3b: The microcontroller power LED is OK but the other LED never flashes 'R'	a) The microcontroller board is getting power but not executing the sketch. b) Reprogram the microcontroller.
Test 3c: The LED flashes 'R' on startup but the screen does not display anything meaningful	a) This means that the sketch is executing, but there is a communication issue between the controller and the display. b) Check all display and microcontroller solder joints. c) Reseat the microcontroller and display. Cycle the power.
Test 4: The display works but there is no audio	a) This is fun to troubleshoot! Do you have any test equipment, like an oscilloscope, to trace an audio signal? b) Put the device into a mode that creates a long stream of Morse, like any of the sending exercises. c) Before trying anything else, repeat Test 5. d) Place your audio probe on microcontroller pin 7 (top row, seventh from the right) and ground. You should see audio output here if the sketch is programmed correctly. e) Follow the audio across resistor R2. See it? Continue following the signal across each step below. f) Follow the audio to pin 3 of the volume control – the pin on the right, near the bottom-right corner of the board. g) Follow the audio to the center pin of the volume control h) Follow the audio to the PAM8403 module pin "L" i) Connect your audio probe to the PAM8403 module outputs "L+" and "L-" (not ground). If the audio stops here you may have a faulty module. j) Connect your probe to the speaker pins. If no audio is here, the ext. speaker jack may be faulty. If audio is here, the speaker may be faulty (try an external speaker through the jack instead).



Resources.

Here are links to this project's documentation:

- Part 1: Introduction
- Part 2: Simple Morse
- Part 3: More Morse
- Part 4: Add a Display
- Part 5: Add a Rotary Encoder
- Part 6: Menu System
- Part 7: Simple Matter of Programming
- Part 8: Add an SD card
- Part 9: Build it!
- Latest Source Code
- Photo: PCB front face
- Photo: PCB rear face
- Photo: PCB populated
- PCB Layout
- Schematic
- Gerbers

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

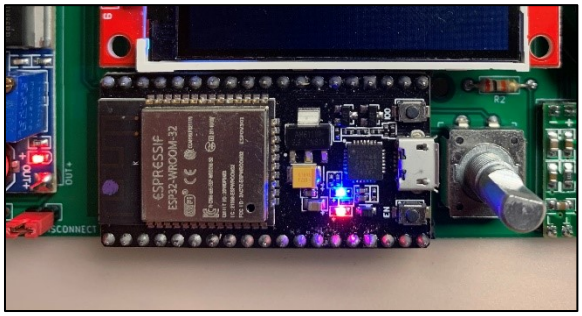
73, Bruce.

Appendix A: **Installing the ESP32 microcontroller board**

As of this writing there are two PCB designs, one for the Blue Pill and one for the ESP32. The ESP32 board is nearly exactly the same as the Blue Pill board. If you are building the ESP32 project, replace Step 4 above with the following instructions:

- Obtain the ESP32 module. This board is compatible with the ESP-WROOM-32 module from HiLetGo, which is available from Amazon here.

- Cut two lengths of 0.1" female header, each with 19 pins, as you did in the preceding step. Mate them with the microcontroller board, then solder to the board. The USB connector on the microcontroller board should be facing RIGHT, and the square silver can should be on the LEFT. The USB connector will become inaccessible when the encoder is installed.



- **Test 3:** reconnect your power adapter and briefly turn the switch ON. The buck converter and LED and display backlight should still light. In addition, the power LED on the microcontroller should light.

- After a brief pause, the LED on the microcontroller should flash short-long-short (Morse 'R') and the display should show the start-up screen. If it does, stop and congratulate yourself! If it doesn't, repeat step 0 above and make sure the ESP32 is programmed correctly.

- Turn the switch off and unplug your power adapter. Go to Step 5, Installing the Encoder.

ERRATA

On the Rev 2 PCB (prior to 10/03/19), the speaker holes are too small for speakers from Digikey and others. Either use the speaker from Adafruit, or use the Rev 2A PCB.