

Morse Code Tutor - From the ground up.

Part 12: Builder's Guide for ESP32 (rev 2B)



Bruce E. Hall, [W8BH](#)

Part 9 of the [Morse Code Tutor](#) series is a set of build instructions for the project, using a circuit board that uses a 2.2" display. This article gives build instructions for an alternate board that supports the larger 2.8" and 3.2" displays. Here is a brief comparison between the two boards

	Original Board	New Board
Size	70 x 130 mm (approx.)	98 x 100 mm (approx.)
Cost	\$7.30 for 5	\$2.00 for 5
Supported Displays	2.2"	2.8", 3.2"
Power Options	7-15 VDC	3.7-5 VDC, 5 VDC, 7-15 VDC

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	Mouser	Prices/Other
2.8" TFT Display Module, ILI9341	0	n/a	n/a	Search Ebay (\$8-\$11)
3.2" Display Module, ILI9341	1	n/a	n/a	Search Ebay (\$10-\$15)
ESP32 Module	1	n/a	n/a	Amazon (\$11)
"Mini360" HW-187 Buck Converter Board	0	n/a	n/a	Search Ebay (\$1)
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	0	CP-102A-ND	490-PJ-102A	Digikey \$0.64 Mouser \$0.64
10K Potentiometer, Bourns PTV09A, logarithmic taper	1	PTV09A-4025F-A103-ND	652-PTV09A4025FA103	Digikey \$0.83 Mouser \$0.78
SPST slide switch, side mount (C&K OS102011MA1QS1)	0-2	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
JST-PH connector	0	455-1719-ND		Digikey \$0.17
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)				

The above table lists part numbers and/or links for a couple of my favorite suppliers. These parts should cost you about \$20 + shipping. There are many incompatible ESP32 modules out there – use only the HiLetGo version listed. Also, check the list of miscellaneous components. Do you have these already? If not add these to your list. Choose either the 2.8 or 3.2" display, not both. The other optional components, marked "Qty 0", depend on how you intend to power you board – see below.

Powering your Board.

This board includes at least 4 different ways to power your device. Do you plan on operating this device with batteries? Which kind? Or are you going to power it with an AC adapter? Here are a few choices and suggestions.



A) AC Power. The Morse Tutor accepts 7-15 VDC input via a 2.1mm DC barrel connector (center pin positive). You will need to provide a suitable AC power adapter. A 9V adapter is ideal.



B) 5V Power via micro-USB connector. Typical power supplies would be a computer or cellphone battery pack via a USB cable.



C) Lipo battery. The input power will be a fully charged single-cell 3.7V lithium-polymer battery terminated with a JST connector.



D) Alkaline battery pack. The input power will be three (3) 1.5V alkaline cells in series, yielding 4.5V total. The input voltage must be between 3.7 – 5.0V at all times.

Here is the number of optional components required according to the power supply methods above. Note that for construction purposes you can choose more than one option, but you should never simultaneously apply more than 1 source of power to the device.

Component	Option A: AC Power	Option B: Micro USB	Option C: 3.7V Lipo	Option D: 4.5V Alkaline
"Mini360" HW-187 Buck Converter	1	0	0	0
Barrel Jack PJ-102A	1	0	0	0
SPST slide switch, side mount	1	0	1	1
JST-PH connector	0	0	1	1

For example, if you want to use AC and/or Alkaline batteries, you will need 1 buck converter, 1 barrel jack, and 2 slide switches (option A + option D). I recommend installing the AC power option first, then consider Lipo and/or alkaline battery power as options. The following build instructions assume that you are powering with an AC power adapter. Battery power options are listed in the Appendix.

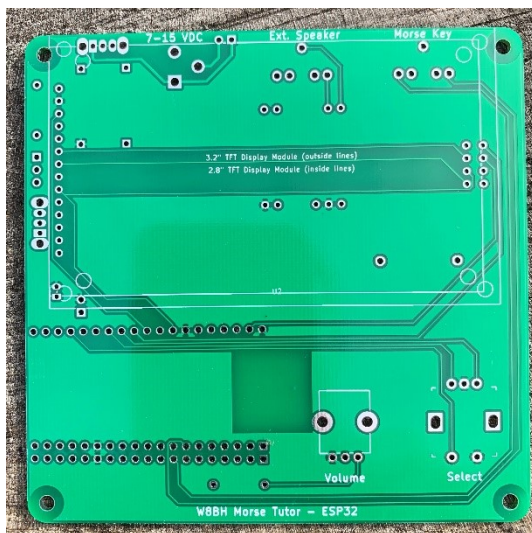
Step 0: Preparation.

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.

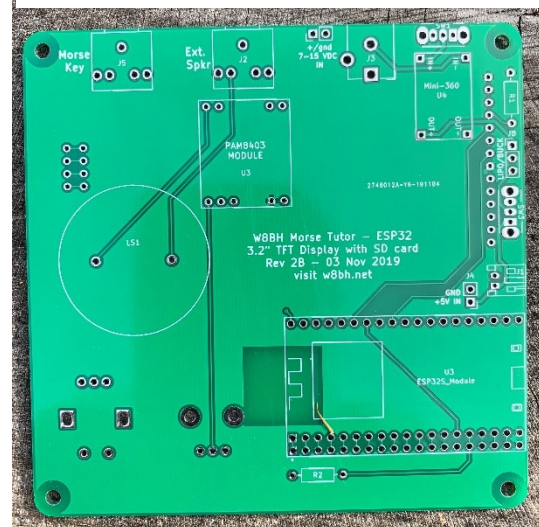
As I mentioned at the start of the series, I believe in starting small. Build the smallest something that you can and test it. Don't forge ahead until you are sure everything is working. In this build we will start with the power circuit on the left-hand side of the board. We will gradually work our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

- Follow the Morse Tutor series. Program your ESP32 microcontroller with a current copy of the sketch. Satisfy yourself that the microcontroller is programmed correctly.
- Obtain the PCB and necessary parts as outlined above.
- Familiarize yourself with the circuit board. Important: Components will be mounted on both sides of the board. Make sure you know which side is which! The front side has large rectangular outlines for the display, encoder, and volume. The back side has outlines for the microcontroller, jacks, speaker, and other components.

PCB Front



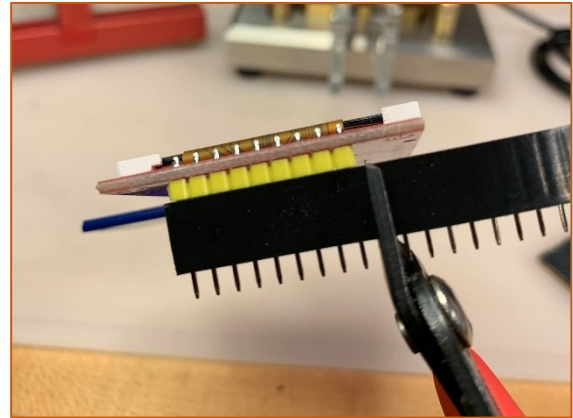
PCB Back



- Consider if you are going to enclose the board, and obtain the appropriate mounting hardware and enclosure material. The jacks 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.

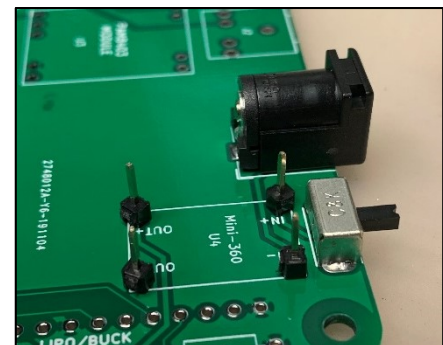
Step 1: Display Mount.

- Cut two lengths of 0.1" female header (one with 14 holes, one with 4) 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, place the display/headers on the front face of the PCB, and solder in place.
- Remove the display and set aside for later installation.

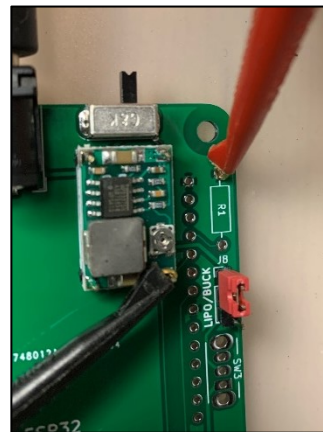


Step 2: (Optional) AC Power Supply.

- Remove the display from its headers and set it aside.
- Turn the PCB over so that you are looking at the back.
- Solder the barrel connector and on/off switch in their correct positions near the top-right 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.
- Solder the mini-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, component-side facing you, on the loose pins. Make sure that the word 'IN' on the converter matches the word 'IN' on the PCB. Solder the header pins to the converter, then flip the board over and solder the header pins to the PCB. Don't clip the header pins; we will use these to check the voltage output.

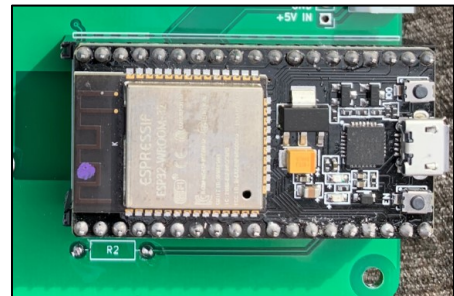


- **Test #1:** plug your DC power supply into the barrel connector. Briefly turn the switch to the ON position. Clip your voltmeter across its OUT- and OUT+ pins near as shown.
- Use a jeweler's screwdriver to adjust the small potentiometer until you get a reading of 4.9 to 5.0V. Rotate clockwise to decrease the voltage. Do not excessive force on the pot; it is quite fragile.
- Install a 3-pin header at J8 "Buck/Lipo" along the right-edge of the board, just below R1.
- Install a shorting jumper across the pins across the top two pins (the Buck side). IMPORTANT: the circuit board will not have power unless you install the jumper. Don't omit this jumper.
- **Test #2:** Keep the ground wire of your voltmeter on the OUT- of the buck converter and touch the red (+) lead to the upper R1 resistor hole as shown. Confirm a reading of 5.0V here.
- Turn the switch off and unplug your power adapter.



Step 3: Install the ESP32 module.

- Cut two lengths of 0.1" female header, each with 19 pins, as you did for the display. Mate them with the microcontroller module. Install on the back of the PCB with the USB connector along the right-edge of the board.
- **Test #3:** reconnect your power adapter and briefly turn the switch ON. The power LED on the microcontroller should light.
- After a brief pause, the LED on the microcontroller should flash short-long-short (Morse 'R'). 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.



Step 4: Install Resistors R1 and R2.

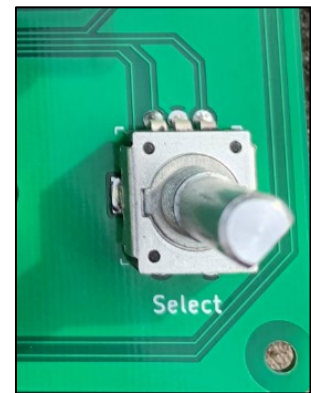
- **R1** is the 10K resistor (brown-black-orange) located beside the buck converter
- **R2** is the 20K resistor (red-black-orange) located beside the ESP32 module

Step 5: Install the Display

- Turn the board over so you are looking at the front. Install the display on the already-mounted female headers.
- **Test #4:** Plug in your power adapter and turn the switch on. After a brief pause, the display should show the start-up screen. If it does, stop and congratulate yourself! If it doesn't, recheck the programming and hardware connections for your ESP32.

Step 6: 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 #5:** 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.

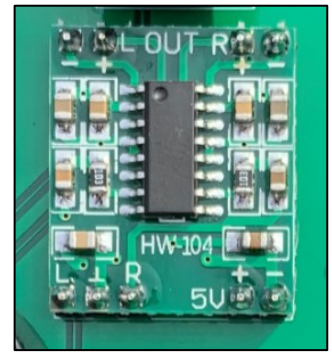


Step 7: Install the Audio Circuit

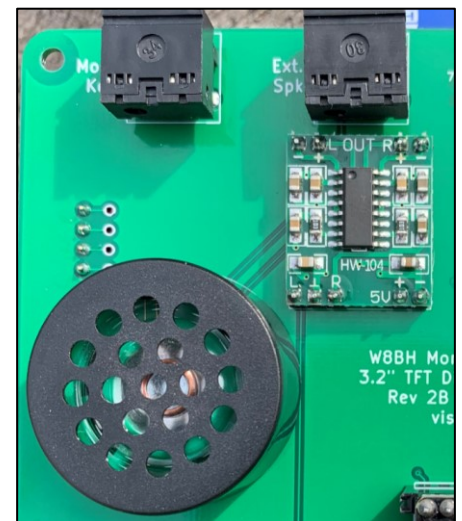
- Install the potentiometer & solder. Like the encoder, the fit is tight and you may need to slightly bend the leads before inserting them into the board. Do not rush.
- Turn over the PCB to install the remaining components on the back.



- 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 to the PCB.



- 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 #6:** 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 8: 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.
- Find suitable knobs for the encoder and volume control. The shafts are 6 mm.
- 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: No Buck Converter output voltage	<ul style="list-style-type: none">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 potentiometer on the buck converter. If you have a normal input voltage above but no output voltage, your converter may be faulty.
Test 2: No voltage on R2 pin	<ul style="list-style-type: none">a) Make sure that you have installed the shorting jumper across the pins marked "Buck". Unless these pins are shorted, the 5V output from your buck converter will not pass to the rest of the circuitb) Remove the shorting jumper and redo Test #1. Make sure the output voltage is 5.0V.c) Ensure good connection to the R2 pad shown in the photo. You may install R2 if this helps to ensure a good connection.d) Lack of power may indicate a short between power and ground. Disconnect power, and do continuity testing between +5 and GND points on the board.
Test 3a: The microcontroller LED doesn't light	<ul style="list-style-type: none">a) Remove and reseal 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'	<ul style="list-style-type: none">a) The microcontroller board is getting power but not executing the sketch.b) Reprogram the microcontroller.

Test 4: The LED flashes ‘R’ on startup but the screen does not display anything meaningful	<ul style="list-style-type: none"> 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 5: The encoder isn’t working	<ul style="list-style-type: none"> a) Check encoder soldering connections for a poor solder joint or solder bridge b) Check microcontroller connections for poor solder joints or solder bridges. c) Reseat microcontroller and cycle power.
Test 6: The display works but there is no audio	<ul style="list-style-type: none"> d) This is fun to troubleshoot! Do you have any test equipment, like an oscilloscope, to trace an audio signal? e) Put the device into a mode that creates a long stream of Morse, like any of the sending exercises. f) 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. g) Follow the audio across resistor R2. See it? Continue following the signal across each step below. h) Follow the audio to pin 3 of the volume control – the pin on the right. i) Follow the audio to the center pin of the volume control j) Follow the audio to the PAM8403 module pin “L” k) 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. l) 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).

Appendix A: USB and Battery Power Options.

If you take your Morse Tutor to field day, a hamfest, or some other show-n-tell event, AC power may not be available. Batteries to the rescue! A quick and easy method is to connect a 9V battery to the barrel connector using an adapter like the one shown on the right. It works, but a fresh 9V battery lasts less than an hour. Fortunately, there are other options.



Micro-USB Option: The micro-USB jack on the ESP32 module provides an alternate method to power your Morse tutor. First, remove the “Buck/Lipo” shorting-jumper. This will disable AC power and battery power inputs. Then, plug a micro-USB cable into the jack. Plug the other end of the cable into your computer, battery stick, or cellphone battery pack:

Note that there is no switch when using the micro-USB input.



3.7V Lipo Battery Option: To use this option, do the following:

- Install the shorting-block between the two “Lipo” pins as shown.
- Solder the side-mount switch and Lipo battery connector as shown

A few notes are in order. 3.7V Lipo batteries are 4.2V when fully charged, and about 3V when depleted. Since the Morse Tutor can function down to about 3.65 volts, it will use about 50% of the battery’s potential without any voltage conversion circuitry.

I recommend using only “protected” cells that have built-in over/under-voltage and overcurrent protection.

Do not combine Lipo cell use with any other power method. It is not safe! There is no protection circuitry on the PCB to stop current flowing into the battery from other 5V sources.

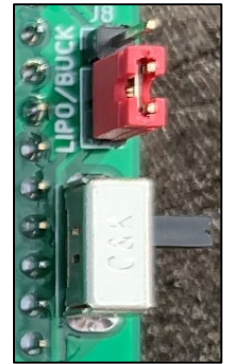
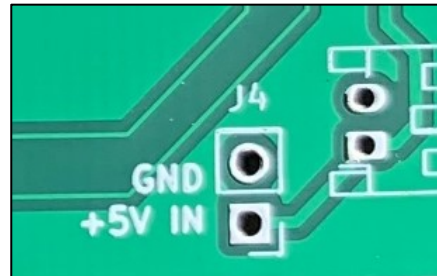


Finally, there is no universal standard for LiPo battery lead polarity! The white JST connector you have just installed assumes that your battery red (+) lead will be on the side closer to the ESP32 module and the black (GND) lead will be on the side closer to the slide switch. All SparkFun and Adafruit batteries use this configuration. Please check your battery before using.



Alkaline Battery Option: To use this option, do the following:

- Install the shorting-block between the two “Lipo” pins
- Solder the side-mount switch
- Connect your 3 AA battery holder to the holes marked J4 “GND/+5V in” as shown



Do not combine alkaline battery use with any other power method. It is not safe! There is no protection circuitry on the PCB to stop current flowing into the battery from other 5V sources.

See [Battery Operation](http://w8bh.net/Battery%20Operation) at w8bh.net for a few notes on using batteries with the Morse Tutor.

Appendix B: Improve your Mini-360

The “Mini-360” buck regulator is an inexpensive part with cheap components. A particularly troublesome component is the tiny trimmer potentiometer. There are two common points of failure:

1. The trimpot is designed for only a handful of rotations before it fails. You should minimize the amount of turning when setting the output voltage.
2. The trimpot wiper can unexpectedly change position, causing the output voltage to rise or fall. I recommend periodically checking the output voltage. Some have advocated gluing or soldering the wiper in place. But both methods are destructive and prone to change its resistance and therefore voltage.

To address both issues, consider replacing the onboard potentiometer with a fixed-value resistor. If you are comfortable with SMD repair, read on! The following fix will significantly improve the reliability of your regulator.

- Rotate the trimpot until you reach the desired output voltage
- Measure the resistance across the trimpot. For example, I measure 33K at 4.6V output and 37.5K at 5.0V output.
- Remove the trimpot. There are several ways to accomplish this, but I find a hot-air rework tool is the easiest.
- Clean the trimpot pads with flux and solder-wick, wipe with alcohol, and dry.
- Install a 0805 resistor across the two closest pads, as shown. The value of the resistor should approximate what you measured above. For this step I use solder paste and a hot-air gun, but you may use any method you are comfortable with.
- Clean with alcohol and dry. Check your output voltage. For this circuit, 4.5 – 5.0 volts will work perfectly fine.

