CTR2-MIDI Assembly Manual

Version 1.0d

December 29, 2025

Contents

Introduction 3

Manual Revision Notes 3

December 29, 2025 – v1.0d 3

Tools You Will Need 3

Kit Bill of Material 4

PCB 5

Installing Components 5

Let the Fun Begin! 6

Step 1: Install SW1, SW2, S3 6

Step 2: Install the Encoder 6

Step 3: Install the Buzzer 7

Step 4: Install SW4, SW5, SW6 7

Step 5: Install the Volume Control – RV1 8

Step 6: Prepare the LED Spacers 8

Step 7: Install the LEDs 9

Step 8: Install the Paddle Jack 9

..... 9

Step 9: Install Headers on the Processor Board 10

Step 10: Prepare the WiFi/Bluetooth Antenna 11

Step 11: Install the Processor on the PCB 12

Step 12: Trim the Bottom Leads 13

Step 13: Routing the Antenna Coax 14

Step 14: Install the Spacers 14

Step 15: Install the Labels and Feet 15

Step 16: Install the PCB in the Enclosure 16

Step 17: Install the Knob 16

Step 18: Button Calibration 17

Finishing Up 17

Introduction

CTR2-MIDI is a Bluetooth-LE MIDI controller based on the Xiao ESP32-S3 from Seeed Studios. The **CTR2-MIDI** kit, herein referred to as the *MIDI*, contains everything you need to build the *MIDI*.

This manual will cover the construction of the *MIDI* kit.

Manual Revision Notes

December 29, 2025 – v1.0d

- Added RV1 volume control and instructions
- Changed antenna/foam assembly instructions

September 13, 2024 – v1.0c

- Changed Allen wrench size for knob from 1/16" to 1.5 mm

July 5, 2024 – v1.0b

- Added side label info and updated antenna coax routing info

April 22, 2024 – v1.0a

- Added a few notes to what you'll need to build the kit

April 13, 2024 – v1.0

- Initial release of the assembly manual

Tools You Will Need

To build this kit you will need basic hand tools like a flush cut diagonal cutters and a temperature controlled soldering iron with a chisel tip. You'll also need solder, 8" of ¾" electrical tape, and possibly solder braid. You should use a grounding wrist strap when handling the bare PCB. Where safety glasses especially when trimming the component leads. The boards are built with lead-free solder so you can use that if your country requires it.

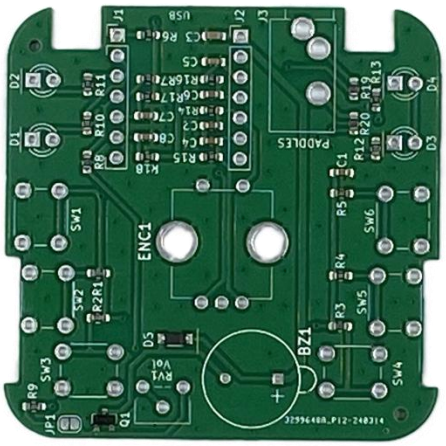
Kit Bill of Material

The kit for the *MIDI* contains these items. You should verify that each item is included in your kit by marking the *Rxd* column. Let me know if you are missing any items and I'll ship them to you.

CTR2-MIDI Kit			
ID	Quant	Rxd	Description
BZ1	1		CCG-1206 buzzer
D1, D2	2		3mm Yellow LED
D3	1		3mm Red LED
D4	1		3mm Green LED
ENC1	1		No-detent encoder
Enclosure	1		Hammond 1551SNAP3BK enclosure - CNC machined
J1, J2	2		7-pin single-row .254" header
J1	1		3.5mm (1/8") stereo jack
Microprocessor	1		Pre-programmed Xiao ESP32-S3 development board with WiFi antenna
PCB	1		CTR2-MIDI PCB with pre-installed SMT components
RV1	1		50KΩ Potentiometer
SW1, SW2, SW3, SW4, SW5, SW6	6		Pushbutton with 16mm actuator
Heat shrink tubing	4		1/16" x 1"
Foam Tape	1		2" of ¼" foam tape
Rubber Feet	4		For bottom of enclosure
Knob	1		32mmx13mm Aluminum Knob
Labels	1 set		Labels for enclosure

PCB

The PCB supplied with the kit comes with all of the SMT devices pre-installed. Because the ESP32-S3 is a CMOS device you must observe ESD (Electro-Static Discharge) procedures when handling the board. Use of a wrist strap and ground mat during construction is highly recommended.



All of the through-hole components will be installed on the top of the board – shown here. The silkscreen has component outlines and part numbers. No components are installed on the bottom of the board.

Installing Components

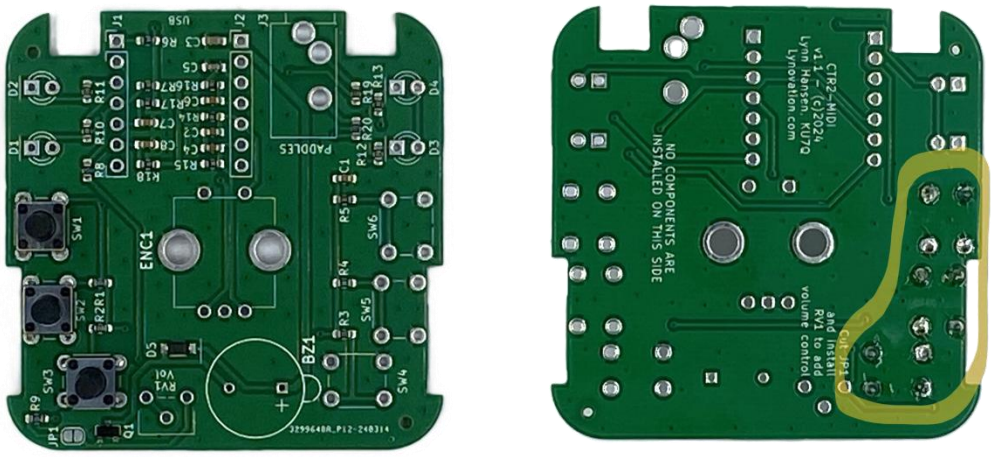
Every component you need to install is of the through-hole variety. I find it's much easier to install these components if the board is installed in a small vise with the bottom facing up. I position the component in its holes from under the board. I hold the solder with my thumb and index finger and use my middle finger to hold the part in position. Once I have soldered one or two pins, I remove the board from the vise and check to make sure the part is positioned properly and laying flat against the board. It's easier to reposition it if you only have a couple of the pins soldered. Once I'm happy with its placement I solder the rest of the pins.

To install components we will start with SW1-SW2-SW3 and work our way around the board counter-clockwise. In the steps below I'll show the part on the top of the board then show the bottom of the board with the pins you will solder highlighted in yellow.

Let the Fun Begin!

Step 1: Install SW1, SW2, S3

Insert three pushbuttons into the SW1, SW2, and SW3 holes from the top of the board.

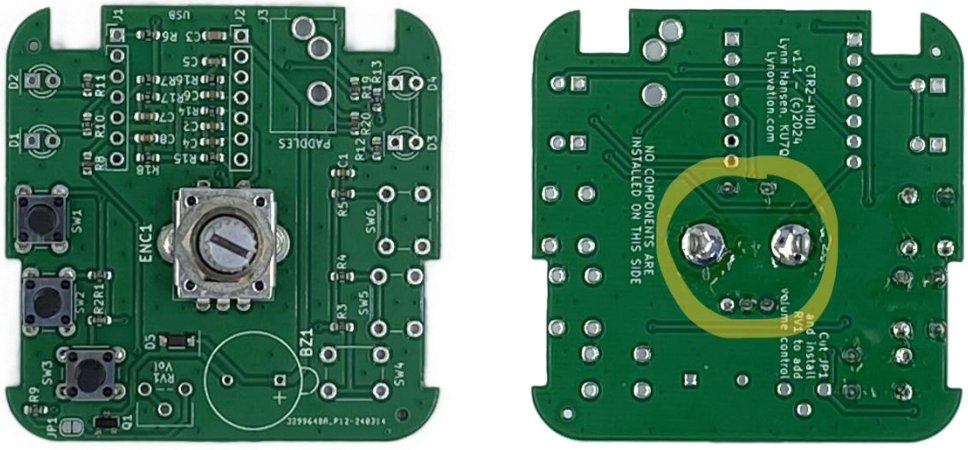


Solder one pin on each switch then turn the board over and verify that each switch is laying flat against the board and its shaft is vertical. When you're happy with their positions, solder the rest of the pins.

WARNING: The switches can be easily damaged by excessive heat. Leave the soldering iron on each pin only long enough to flow the solder.

Step 2: Install the Encoder

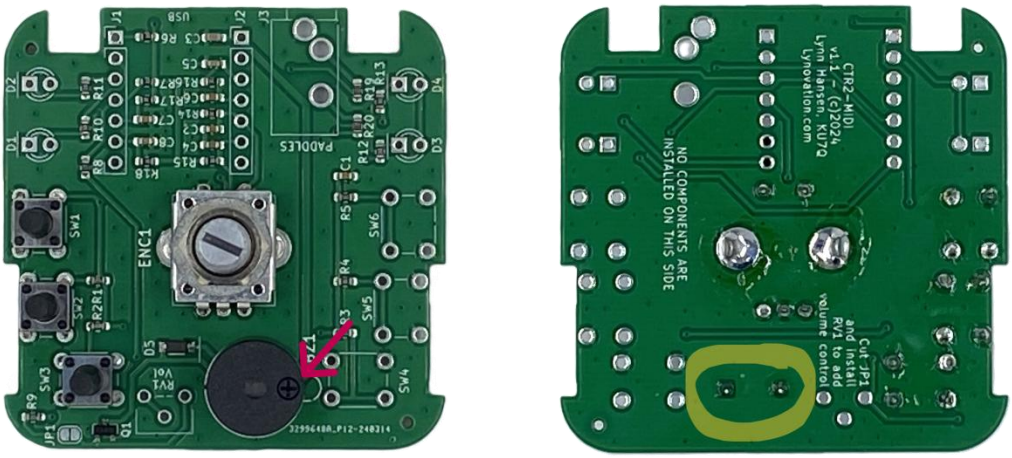
Next, insert the encoder from the top of the board.



Solder the 5 pins. Use a medium sized chisel tip on your soldering iron to solder the tabs on the encoder. Completely fill the holes with solder as shown above. This provides the mechanical strength needed for the encoder.

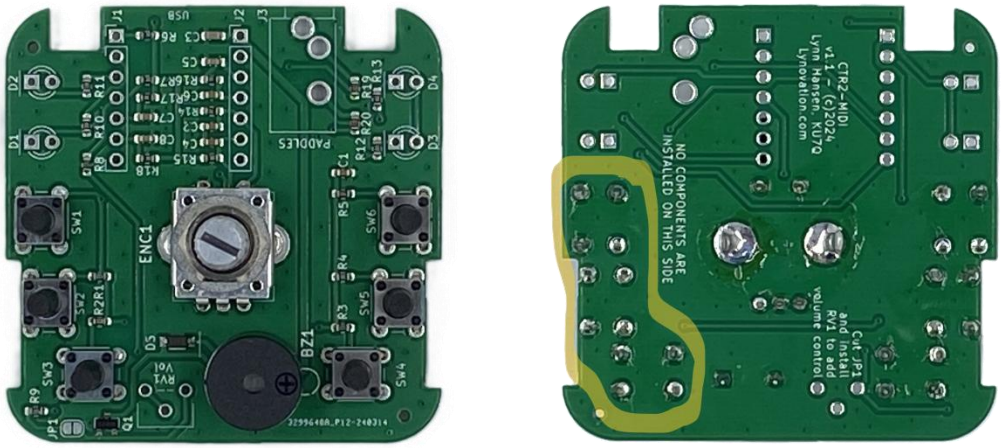
Step 3: Install the Buzzer

Before installing the buzzer remove the plastic seal over the top hole. This seal prevents cleaning solution from entering the buzzer when it's immersion washed. Insert the buzzer from the top of the board making sure the (+) pin is in the hole with the square pad as shown by the red arrow below.



Step 4: Install SW4, SW5, SW6

Next, insert SW4, SW5, and SW6 in their holes. As with the first three buttons, make sure the buttons are laying flat on the PCB and their shafts are vertical.



WARNING:

- The switches can be easily damaged by excessive temperature. Leave the soldering iron on each pin only long enough to flow the solder.
- The switches are not designed for washing (flux removal). If you choose to remove the solder flux from the PCB, be very careful not to get the flux removal agent on the switches

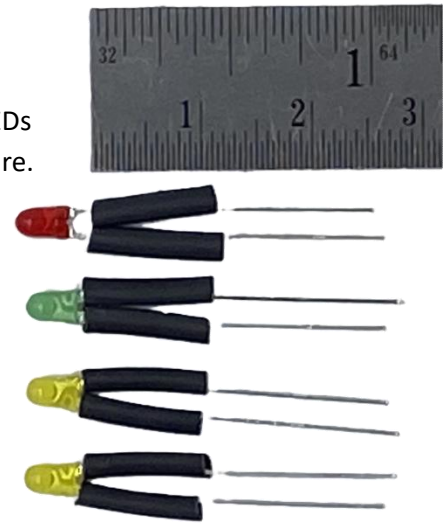
Step 5: Install the Volume Control – RV1

Starting in December 2025, RV1, a 50kΩ potentiometer has been included in the CTR2-MIDI kit. This control is used to adjust the volume of the CW keyer sidetone when CTR2-MIDI is used in Flex WiFi mode. There is no CW sidetone in MIDI mode as the CW keyer is handled by the 3rd party app.

Install RV1 as shown in the photo to the right. Make sure the control is turned fully clockwise. You can reduce the sidetone volume if it's too loud after construction.

Step 6: Prepare the LED Spacers

Prior to installing the LEDs, prepare eight 11mm spacers from the 1/16" heat shrink supplied in the kit. These spacers position the LEDs above the board so they just slightly protrude through the enclosure.



You can shrink the tubing to the leads but it's not necessary.

Step 7: Install the LEDs

The LEDs must be installed on the board with the short (cathode) lead in the square pad. The spacers will insure that all LEDs are at the same height.

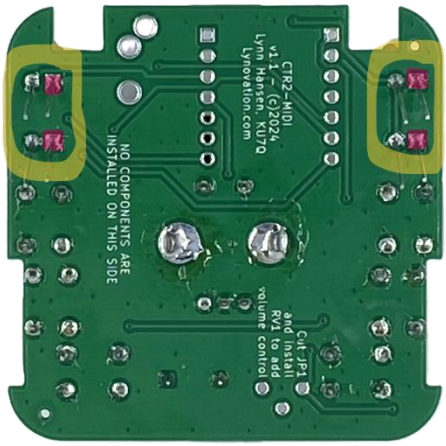
Insert the YELLOW LEDs in D1 and D2, GREEN LED in D3, and RED LED in D4. The square pads are marked in red in the right photo below.

D1

D2

D4

D3



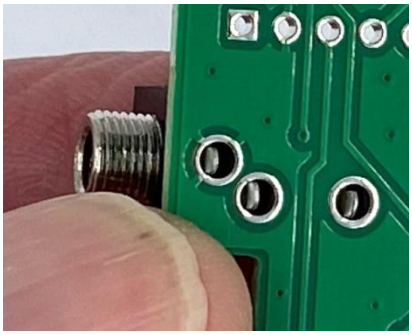
D1

D2

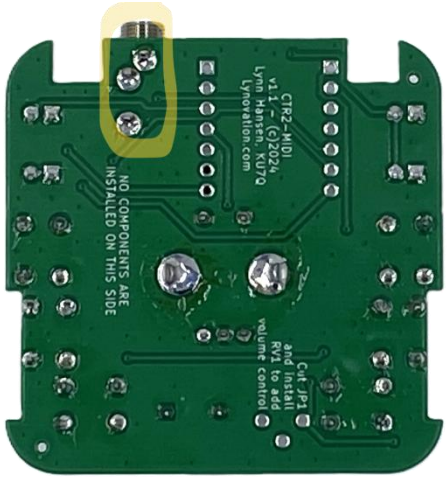
Step 8: Install the Paddle Jack

The paddle jack is installed next.

Place the jack in the solder holes then slide the jack toward the outside edge of the PCB as shown here. The pins should touch the left side of each hole. This will position the jack properly in the enclosure.



Fill the holes with solder like you did with the encoder tabs.

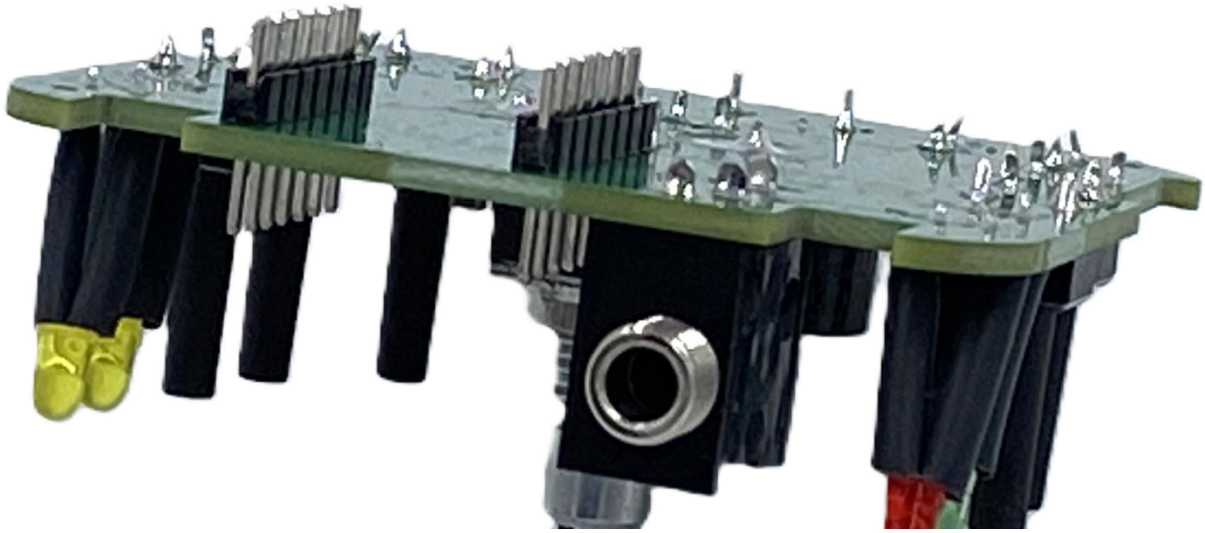


Step 9: Install Headers on the Processor Board

Two 7-pin single-row headers are supplied in the kit. These will be soldered to the processor board using the MIDI's PCB as an alignment jig before the board is installed on the PCB.

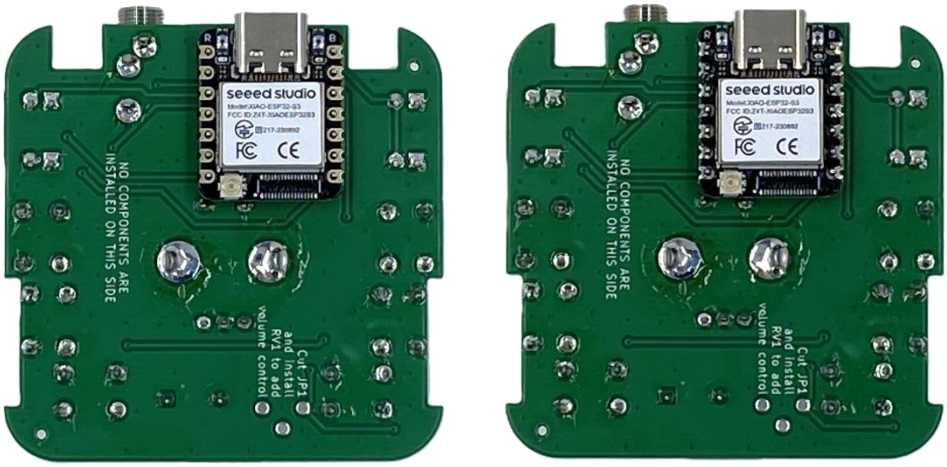
First, insert the long leads of the headers into the processor holes on the PCB from the bottom of the board as shown below. The PCB will serve to hold the pins in alignment while you solder them to the processor board.

WARNING: DO NOT solder the headers to the PCB at this time!



Next, place the processor board on the headers as shown in the left photo then solder the processor board to the header pins as shown on the right photo. Once this has been done, remove the processor board from the PCB.

AGAIN, DO NOT SOLDER THE HEADERS TO THE *MIDI's* PCB!



Step 10: Prepare the WiFi/Bluetooth Antenna

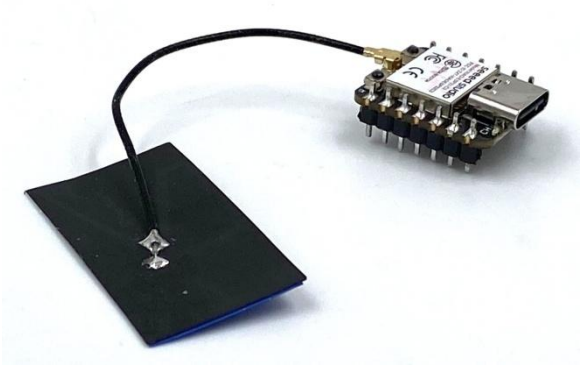
The Xiao ESP32-S3 processor board is supplied with a flexible WiFi/Bluetooth antenna. This antenna uses an I-PEX MHF connector and plugs into the antenna socket on the Xiao board. The antenna's connector simply snaps into the socket on the Xiao. The antenna will lay under the PCB.

Before you attach the antenna, apply two layers of electrical tape to the back side (the side with the paper backing). This will help insulate it from the trimmed leads on the bottom of the board.

Antenna as supplied

Antenna with tape installed

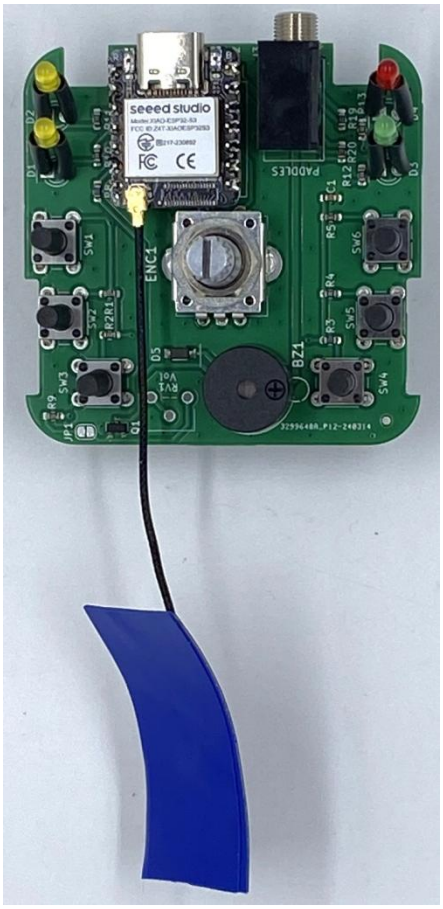
Next, take the Xiao processor board in one hand and carefully insert the antenna connector into the antenna jack on the board. Take your time. Start with the connector at an angle to the socket then gently rock the connector back and forth as you gently apply pressure to it. Once aligned it should snap into the socket.

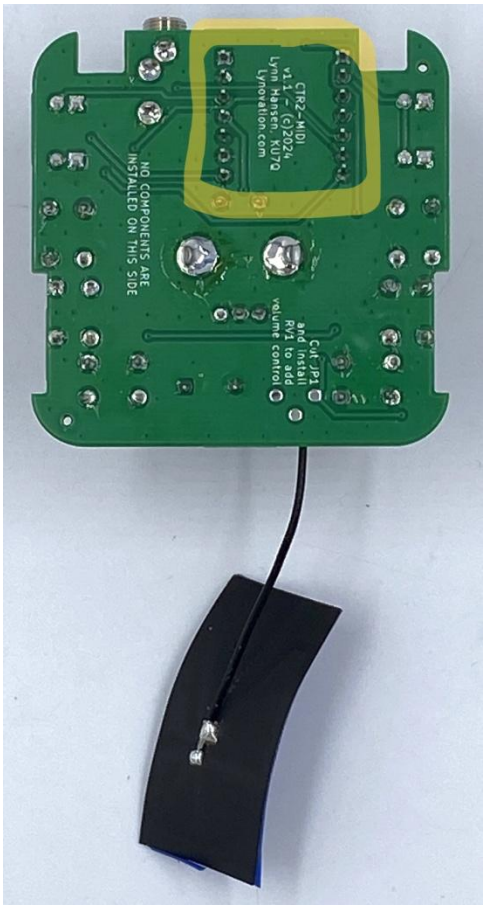


NOTE: Do not force the connector into the socket. It is fragile and can be easily damaged.

Step 11: Install the Processor on the PCB

Insert the headers on the processor board into the processor holes in the top of PCB. Solder the header pins from the bottom of the PCB as shown below.





Step 12: Trim the Bottom Leads

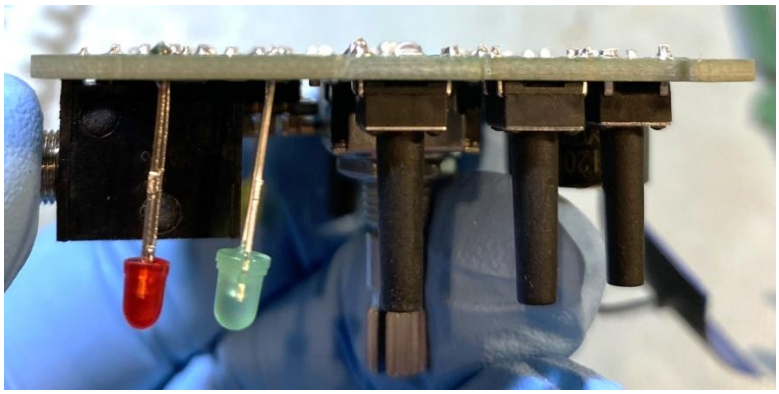
The *MIDI's* PCB fits snuggly in the enclosure. So snug in fact that it won't fit if you don't trim every lead flush (less than 1/16" high) on the bottom of the PCB. Make sure the ends aren't pointed or they can puncture the tape applied to the back of the antenna.

CAUTION! Use safety glasses when trimming these leads. They become tiny rockets when you clip them off!

Here are the before trimming and after trimming photos so you get the idea.

Before trimming

After trimming



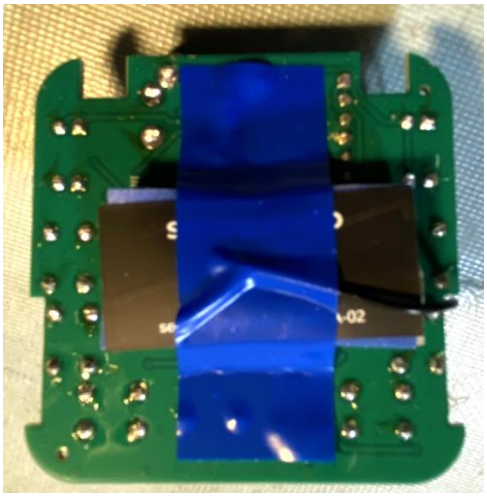
The switches provided are not designed for immersion cleaning. Therefore, I don't recommend cleaning the flux off the bottom of the board. It won't hurt anything to leave it. I've had several pushbutton switches fail after cleaning because traces of flux got washed into the contacts of the switches. If you decide to clean your board off try to keep any residue away from the switches.

Step 13: Routing the Antenna Coax

The antenna is installed under the PCB. Route the coax into slot on the side of the board as shown on the left photo below.

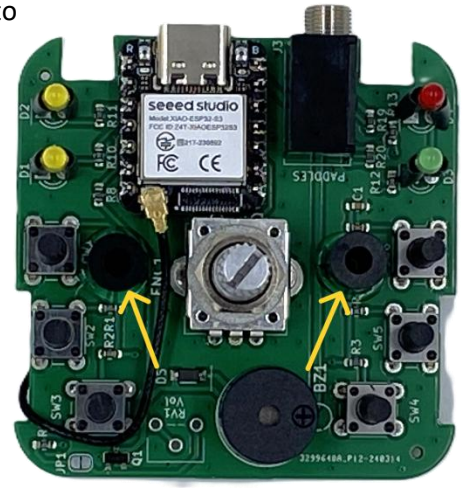
NOTE: The coax on newer ESP32-S3 boards is shorter than shown here. Route it between SW1 and SW2 instead.

Next, take the two strips of 3/8” foam supplied with the kit and apply them to the tape on the antenna. These strips will insulate the antenna from the PCB and distance it slightly away from the PCB. They also cushion the board and lock everything into place when you install the bottom cover. Position the antenna/foam assembly so the center of the antenna is in the center of the board and use a 2-1/2” piece of tape to secure it to the board.



Step 14: Install the Spacers

Two 12mm spacers are supplied with the kit. These spacers maintain an equal distance from the PCB to the inside edge of the top cover so the pushbuttons align at the same height when the PCB is installed in the enclosure. The spacers must be glued onto the PCB next to SW1 and SW6 as shown by the yellow arrows in this photo. You can use epoxy, silicone, or super glue to install them since there won’t be any side pressure on them once the PCB is installed in the enclosure.



Step 15: Install the Labels and Feet

Three labels are supplied in the kit. You'll need to trim the black label for the top cover and cut the side label in half. It helps to hold them with a small set of tweezers while moving them into position. Peel the rubber feet off their carrier and stick them on the bottom section next to each corner as shown below.

The side labels reflect the default button mapping for SmartSDR for iOS and MacOS. You'll need to create your own labels if you change the mapping. I use .47" white on black label tape and 6 pt Ariel Bold font.

The white bottom label has areas where you can record your unit's Bluetooth ID# (CTR2_####) and COM port #.

Step 16: Install the PCB in the Enclosure

Installing the PCB into the enclosure is the trickiest part of the assembly. Take your time and don't force it together or you'll probably break something!

Start by tilting the LEDs about 15 degrees toward the outside edge of the board. This makes it easier to align them with their holes in the top case.

Next, take the PCB and carefully align the phone jack and the USB-C connector with their holes in the top cover. As you guide them into these holes watch the LED holes and make sure the LEDs are lined up. Once you have the jacks in their holes the LED should be visible in their holes and the encoder shaft should just fit inside its elongated hole on the top case.

While gently applying pressure on the PCB take a sharp pair of tweezers or a pin and gently pull each pushbutton's shaft into its hole. Start with SW1 and SW6 and work your way up to SW3 and SW4. Once they are lined up the PCB will snap into the top cover.

Now take the bottom cover and positioning it as shown, insert the two locking tabs into the slots on the PCB on the side with the connectors. These tabs lock into ledges on the top case.

Rotate the bottom cover until it interlocks into the two tabs on the side of the top cover. They should firmly lock into place.

Step 17: Install the Knob

The *MIDI* is supplied with a nice 32mm aluminum knob with a finger dimple. To install the knob, align the set screw in line with the slot in the encoder shaft. This will prevent the set screw from compressing the shaft when it's tightened. Tighten the set screw using a 1.5mm Allen head screwdriver or wrench (a 1/16" wrench is tight but will work too).

Step 18: Button Calibration

All *MIDI* kits come with the ESP32-S3 processor pre-programmed. Unfortunately the pushbutton resistance ladder can't be calibrated until the kit is built. Instructions for button calibration can be found in the Operations Manual under **Configuring the MIDI Firmware for First Use**. You'll also need to map the *MIDI's* controls into your app. Instructions to do this are also found in the operation manual under **Connecting the App to CTR2-MIDI.**

Finishing Up

Thank you for choosing the **CTR2-MIDI** for your station. The CTR2 system has been evolving over the last 4 years and the *MIDI* is an offshoot from that effort. I enjoy everything about the process of bringing a new device into the ham community and I hope the *MIDI* makes you smile!

If you have any questions, are confused by something in this manual, or have suggestions for improvement, please let me know.

You can reach me through my blog, https://ctr2.lynovation.com/contact-me, or through my email address on QRZ.com.

73, Lynn, KU7Q