

Contents

Introduction	3
Manual Revision Notes	3
Tools You Will Need	3
Kit Bill of Material	4
PCB	5
Installing Components	5
Let the Fun Begin!	5
Step 1: Install the Pushbuttons	5
Step 2: Install the CAT Polarity Header/Switch.....	6
Step 3: Install the Encoder	7
Step 4: Install the Volume Control and Buzzer	8
Step 5: Install the LED	9
Step 6: Install the 3.5mm Stereo Jacks	10
Step 7: Install the Headers on the Microprocessor Board.....	11
Step 8: Prepare the WiFi Antenna	11
Step 9: Install the Xiao Processor Board	13
Step 10: Trim the Bottom Leads	14
Step 11: Routing the Antenna Coax.....	15
Step 12: Install the Labels	16
Step 13: Install the PCB in the Enclosure	17
Step 14: Install the Knob	19
Step 15: Button Calibration and Programming.....	19
Finishing Up.....	19

Introduction

CTR2-Micro is a tiny radio controller/keyer based on the [Xiao ESP32-C3](#) from Seeed Studios. CTR2-Micro, herein referred to as the Micro, can be ordered as a kit of parts as shown in the lead photo, or as complete, fully assembled unit

This manual will cover the construction of the Micro kit.

NOTE: I've had two ESP32-C3 processors fail to be recognized by the computer when they were first plugged in. For this reason, starting December 1, 2023 I am now downloading the latest Micro firmware onto the processor board in every kit I ship. When your kit arrives you'll still want to set up the [Espressif Flash Download Tool](#) and check my web site for updated firmware so you can keep your Micro's firmware current.

Manual Revision Notes

April 14, 2024

- More fine-tuning of construction steps

December 1, 2023

- Rearranged the construction steps to match the sequence I use when I build the assembled units.
- Added notes about pre-programmed ESP32-C3's in every kit.

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 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 Micro 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-Micro Kit			
ID	Quant	Rxd	Description
BZ1	1		CCG-1206 buzzer
D1	1		3mm red LED
ENC1	1		Detent –OR- no-detent encoder
Enclosure	1		Hammond 1551SNAP3BK enclosure - CNC machined
J1, J2	2		7-pin single-row .254" header
J4, J5, J6 (J3 was skipped)	3		3.5mm (1/8") stereo jack
Microprocessor	1		Xiao ESP32-C3 development board with WiFi antenna
PCB	1		CTR2-Micro PCB with pre-installed SMT components
RV1	1		50Kohm side mount potentiometer
S1	1		16-pin dual-row header –OR- 8-pos right-angle DIP switch
SW1, SW2, SW3	3		Pushbutton with 16mm actuator
Heat shrink tubing	2		1/16" x 1/2"
Foam Tape	2		2" of 1/4" foam tape
Rubber Feet	4		For bottom of enclosure
Knob	1		32mmx13mm Aluminum Knob
Labels	1 set		4 labels for enclosure

PCB

The PCB supplied with the kit comes with all of the SMT devices pre-installed. Because it has a CMOS device on it 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. **There are no components installed on the bottom of the board.**



Installing Components

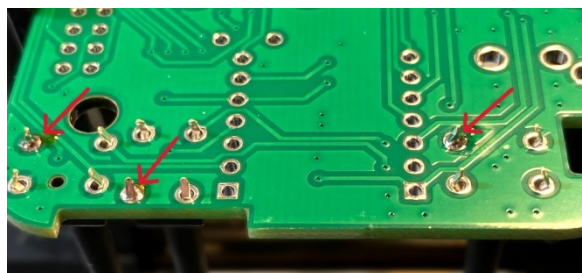
Every component you need to install is of the through-hole variety. I find it's much easier to install these components if the board is installed in a small vise with the bottom of the board facing up. I position the component in its holes from beneath the board. I hold the solder with my thumb and index 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.

Let the Fun Begin!

Step 1: Install the Pushbuttons

Place the three pushbuttons into the holes for SW1, SW2, and SW3 from the component (top) side of the board. Note that SW2 is positioned 90 degrees from the other two.

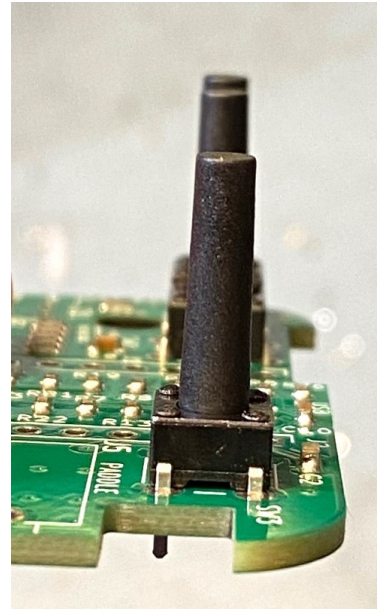
Hold them tight to the board as you solder one lead on each switch to hold it in place. I've added arrows pointing to the pins I soldered in this photo.



Next, check the button alignment. Each button should lay flat on the PCB and the shafts should line up when viewed from the side. Also check that each shaft is standing vertically (not tipped to one side).

When you're satisfied that they are straight, solder the remaining pins.

NOTE: Do not leave the soldering iron on the switch pins any longer than necessary. Excessive heat will damage the switch mechanism.



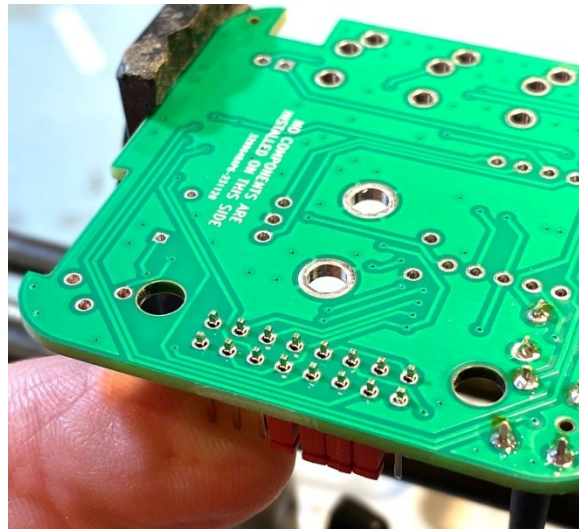
Step 2: Install the CAT Polarity Header/Switch

There are two options for the CAT polarity strapping.

OPTION1A: Dual-Row 16-pin Header Option

OPTION1A provides a 16-pin dual-row header and four header jumpers to configure the CAT polarity. This is the cheaper way to go and it provides a clean exterior. This is a good way to go if you only have one radio, plan on operating only a Flex radio or other SDR radio, or all of your radios use the same CAT configuration (like Icom CI-V or RS232).

Insert the dual-row 16-pin header from the component (top) side the short pins inserted in the holes. Make sure the header is sitting flat on the board and solder the pins.



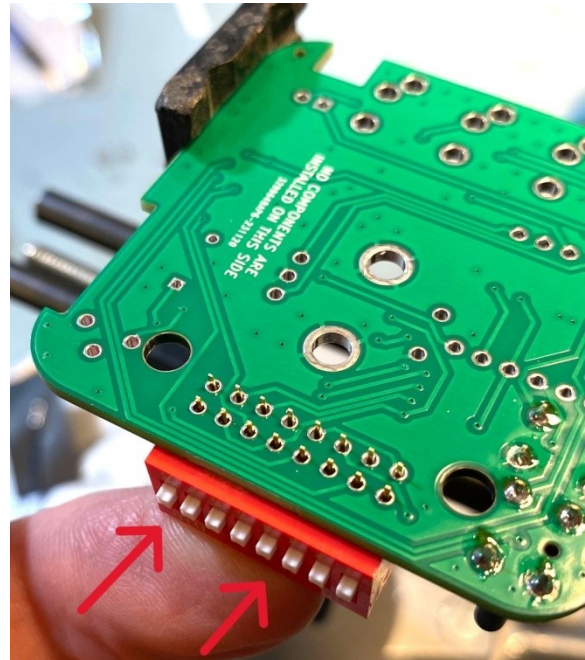
OPTION1B: DIP Switch Option

OPTION1B provides a right-angle 8-position DIP switch instead of the dual-row header.

The advantage of this option is that it allows you to change the CAT polarity without removing the board from the enclosure.

This is a good option if you have multiple radios that you're planning on using the Micro with and they have different CAT polarity requirements.

To install this switch, drop it into the CAT polarity header holes from the component (top) side of the board. Push it toward the inside of the board so the pins rest on the inside edge of each hole. This will recess the switch so the white switches are flush with the enclosure once assembled.

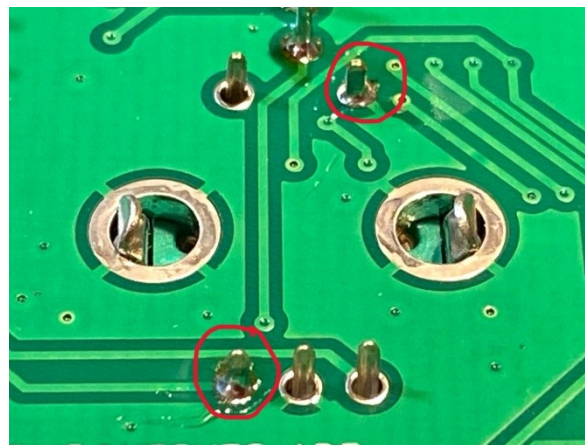


Solder a couple of the pins on the inside row then check the switch to make sure it is vertical. Once this has been verified solder the remaining pins.

Step 3: Install the Encoder

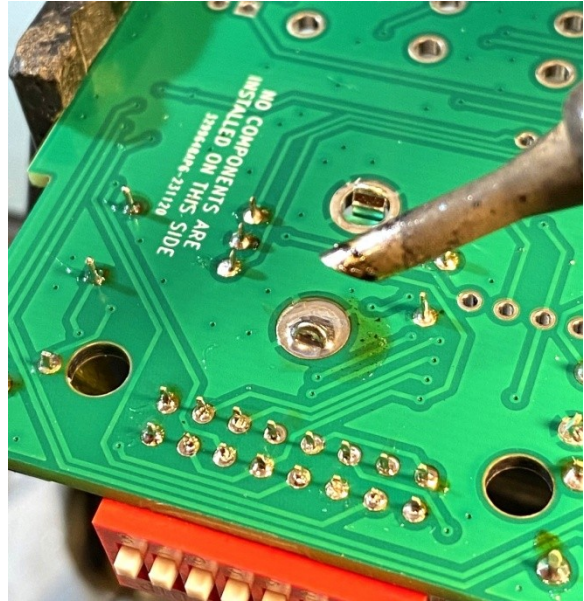
The encoder is installed next. Insert it into its holes from the component (top) side of the board. It will be a close fit next to the processor when it is installed but it will clear it.

Using the same process we used with the other components, hold it tight to the PCB then solder two of its pins to the PCB. Turn the board over and check to make sure it is sitting flat on the board and the shaft is vertical.



Once you're happy that it's positioned correctly, turn the board back over and solder the remaining pins and the two mounting tabs to the board.

Use a medium sized chisel tip on your soldering iron to apply heat and completely fill the mounting tab holes with solder. When you're done it should look like this. These tabs provide the mechanical support for the encoder.

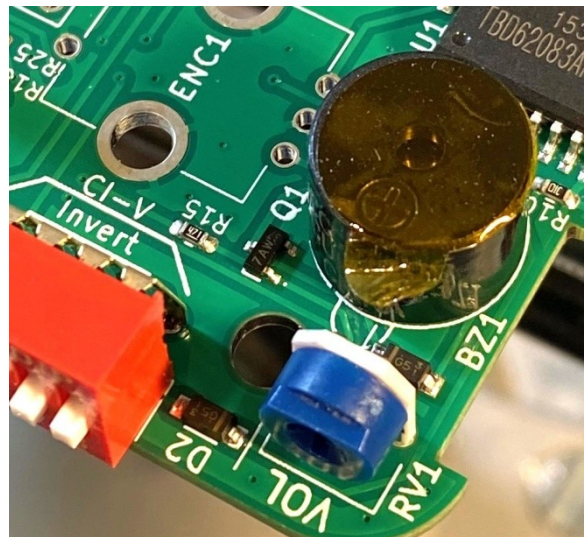


Step 4: Install the Volume Control and Buzzer

Mount the volume control next. Use the same procedure as you followed with the switches and DIP switch; solder one of the outside leads, check to make sure it's vertical, then solder the other two leads.

Install the buzzer next. The buzzer will have a protective cover over the top hole. You can remove this cover because it's only there for immersion cleaning. If you leave it on you won't be able to hear the buzzer.

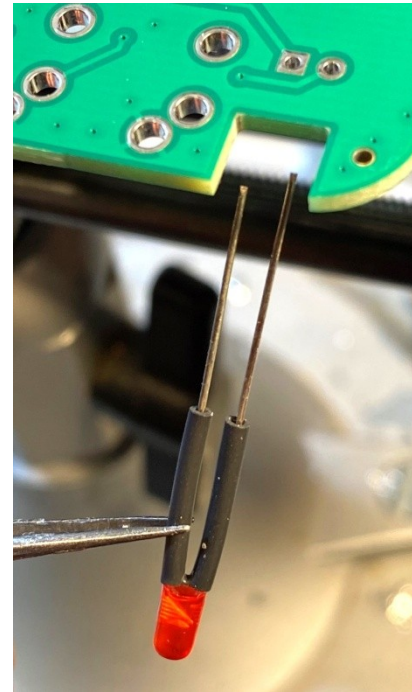
Insert the buzzer's (+) into the square pad. There's a + mark on top and side of the buzzer.



Step 5: Install the LED

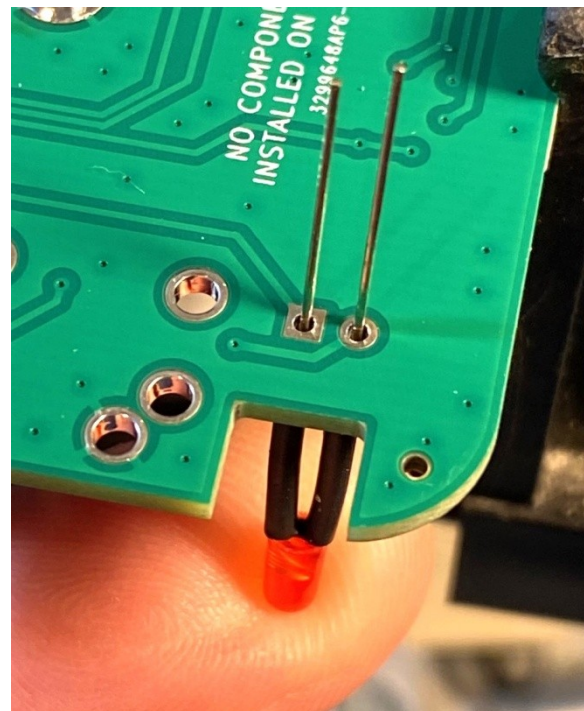
To install the LED, first cut the supplied piece of 1/16" heat shrink tubing to make two 11mm pieces (a little less than 1/2"). Slide these onto the LED's leads and use your soldering iron or a hot air workstation to shrink the tubing. The tubing provides a standoff so the LED will protrude through the enclosure's top after assembly.

Next, position the LED as shown then insert it into the two holes for D1. The short lead on the LED (cathode) must go into the hole with the square pad and the long lead on the LED (anode) must go into the hole with the round pad.



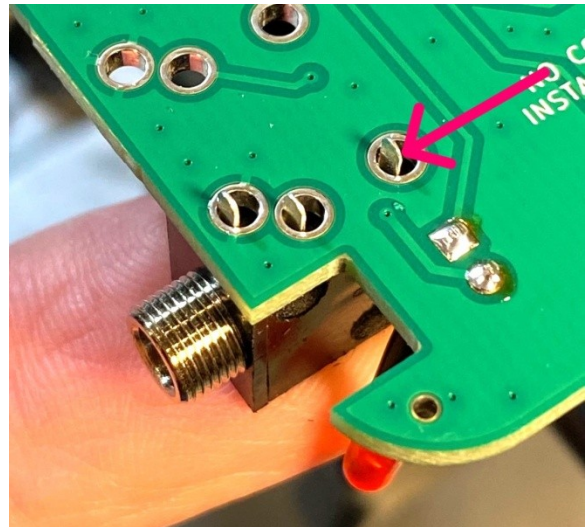
Next, position the LED so it is visible in the middle of the square notch on the PCB as shown here. This insures that the LED will align with the hole in the enclosure. Tilt it slightly toward the outside edge of the PCB and make sure the heat shrink tubing is touching the PCB. Finally, solder the two leads.

Use a pair of flush cut diagonal cutters to trim these leads flush with the board.



Step 6: Install the 3.5mm Stereo Jacks

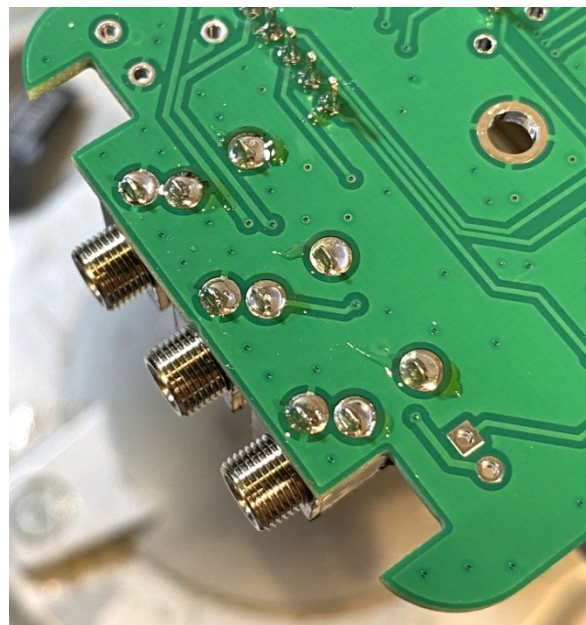
Insert the first jack into its holes the gently pull it toward the edge of the board so the solder tabs are on the outside edge of the holes, as shown where. This insures that all three jacks will be aligned the same across the back of the board. Make sure the jack is laying flat on the board.



Use a medium size chisel tip on your soldering iron and add enough solder to each hole to completely fill it, similar to how you soldered the encoder's tabs. Don't add too much solder or leave the soldering iron on the terminal too long as you might damage the jack.

When you're done the board will look like this.

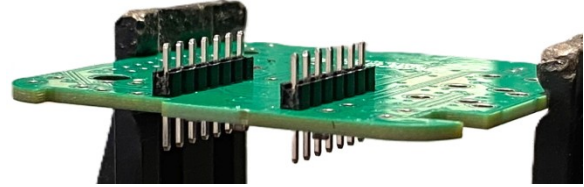
NOTE: If your soldering iron tip is too small you won't be able to get enough heat into the large pads to properly flow the solder.



Step 7: Install the Headers on the Microprocessor Board

Insert the two 7-pin single row headers into the holes labeled J1 and J2 on the PCB. Insert the long pins into the board. The short pins should extend upward as shown in this photo. We're using the PCB to hold the headers in alignment.

Do not solder the long headers to the board at this time.



Next, drop the microprocessor board onto the headers and solder it to the short header pins as shown in this photo.

Do not solder the long headers to the board at this time.

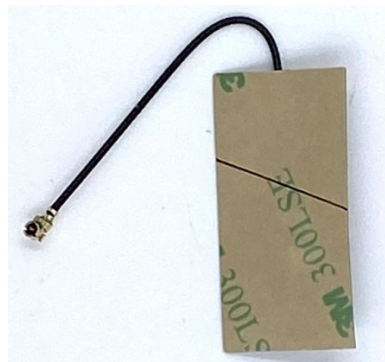


Step 8: Prepare the WiFi Antenna

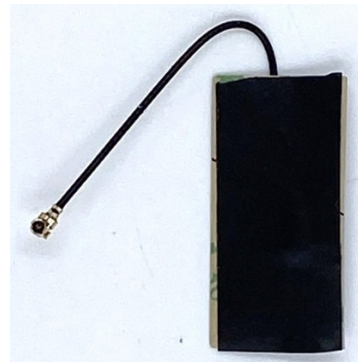
The Xiao ESP32-C3 processor board is supplied with a flexible WiFi antenna. This antenna uses an I-PEX MHF connector and plugs into the antenna socket on the Xiao board. The antenna's connector simply snaps into the socket on the Xiao. The antenna will lay under the PCB.

NOTE: This will reduce your WiFi range because the PCB will partially shield the antenna. If you need longer WiFi range consider installing a WiFi range extender on your router near your operating position or install an external antenna on the Micro. I can't give you any guidance here because the Xiao ESP32-C3 is FCC certified when used with the supplied antenna, so that's what I use.

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.



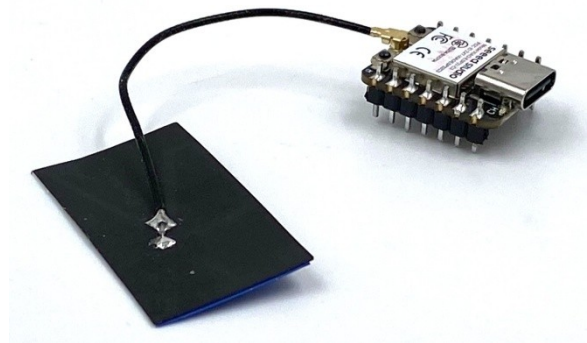
Antenna as supplied



Antenna with tape installed

Next, take the Xiao processor board in one hand and carefully insert the antenna's connector into the antenna jack on the board. Take your time. Rock the connector back and forth as you gently apply pressure to it. Once aligned it should snap into the socket.

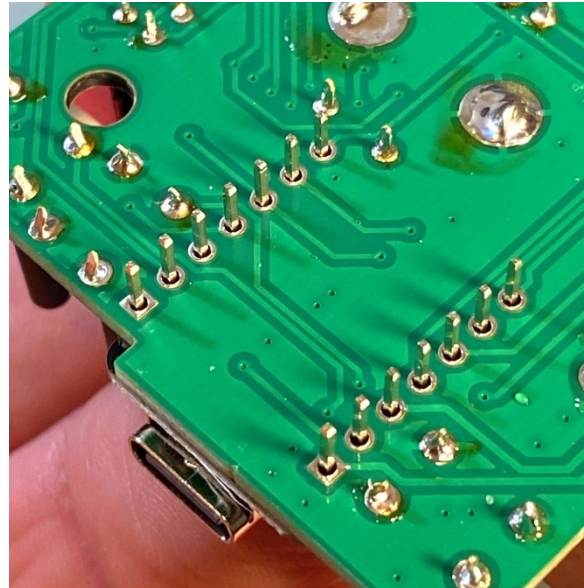
NOTE: Do not force the connector into the socket. It is fragile and can be easily damaged.



Step 9: Install the Xiao Processor Board

Take the processor board with attached antenna and insert the long header pins into the PCB from the component (top) side with its USB-C socket facing toward the outside of the board. It will just clear the encoder.

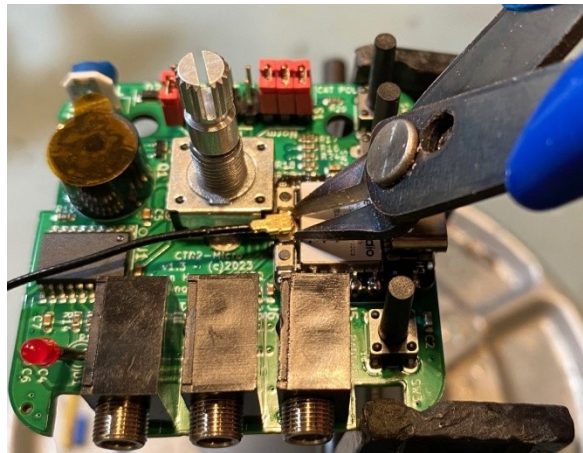
Solder the header pins to the PCB.



HINT: Removing the Antenna Connector

If you ever need to remove the antenna connector, *use extreme caution*. It's VERY EASY to snap the socket off the Xiao board if you apply uneven pressure in the removal process. Yes, firsthand experience talking here!

I've found that if you lay your open flush cut diagonal cutters on the metal case of the Xiao with the tips of the cutters just under the edge of the connector, then place your finger under the coax and gently pry up with the cutters and your finger at the same time, the connector will pop right out – and leave the socket on the board (hopefully!).



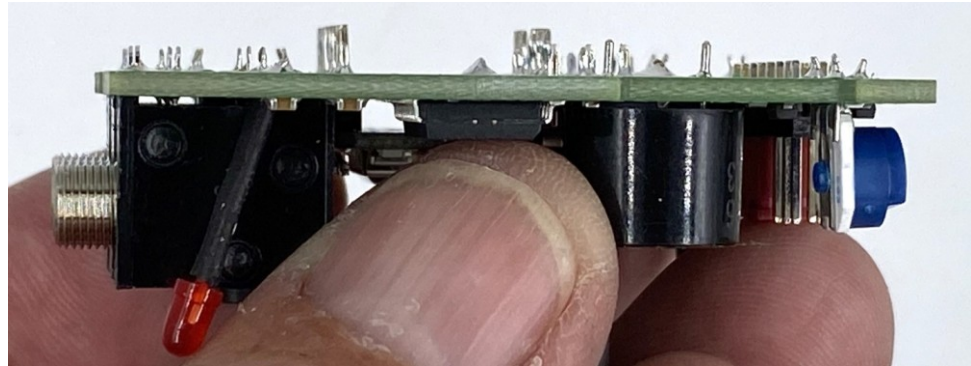
Step 10: Trim the Bottom Leads

The Micro's PCB fits snugly 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.

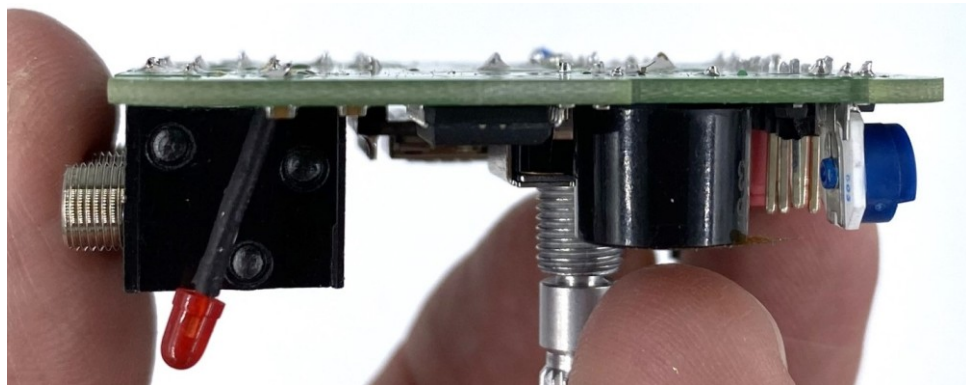
CAUTION! Use safety glasses when trimming these leads. They become tiny rockets when you clip them off!

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

Before trimming



After trimming



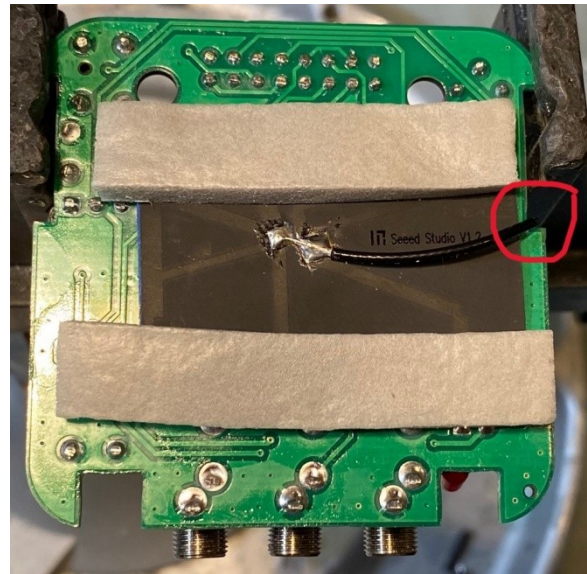
This completes the assembly of the PCB.

NOTE: I don't recommend cleaning the flux off the board. I've had several instances where the flux remover (alcohol)/flux solution has seeped into the pushbutton switches and cause the switch to perform erratically.

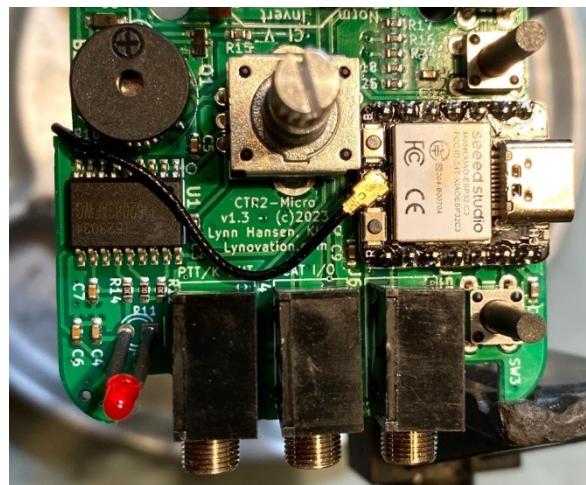
Step 11: Routing the Antenna Coax

The antenna is installed under the PCB. Route the coax over to the slot on the side of the board as highlighted in the photo here then lay the antenna on the board with the electrical tape facing the board and the bare coax connection facing away from the board. The coax connection should lie approximately in the middle of the board. This allows it to sit in the recess in the bottom cover.

Next, take the two strips of 1/4" foam supplied with the kit and place them on either side of the antenna to hold it in place. These will cushion the board and lock everything into place when you install the bottom cover.



Route the coax as shown on the top of the board.



Step 12: Install the Labels

The Micro kit is supplied with four labels for the enclosure. Although these photos show installing the labels after the PCB and knob have been installed in the enclosure, it's easier to install the labels and feet first.

Trim the labels with a sharp pair of scissors. Install the large label on the top, just above the encoder knob. I find it's easier to hold the labels with a pair of tweezers while I align them.

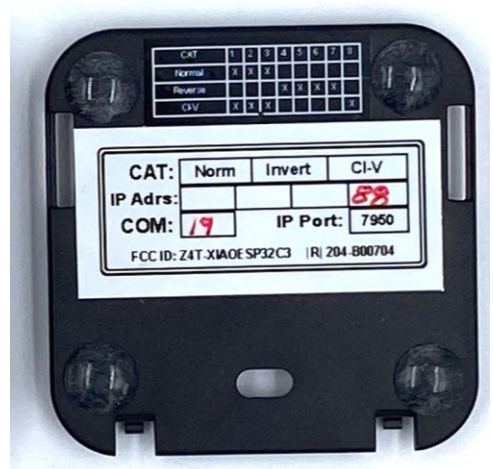
The Paddle, CAT, Key/PTT label is installed under the three 1/8" stereo jacks.



Install the CAT polarity strapping label and the white label on the bottom as shown in the photo below.

Use the white label to record the internal CAT jumper settings (not needed if you ordered the external DIP switch option), the COM port #, and IP address of your Micro. Use a fine-point permanent marker to write on the white label. The ink will rub off so place a piece of transparent tape over the label to keep them from smudging.

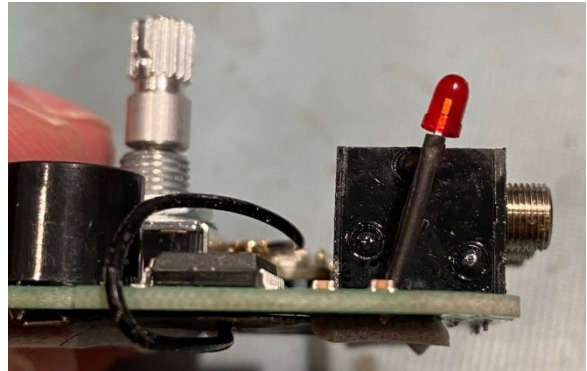
Remove the rubber feet from their carrier and install one on each corner on the bottom case.



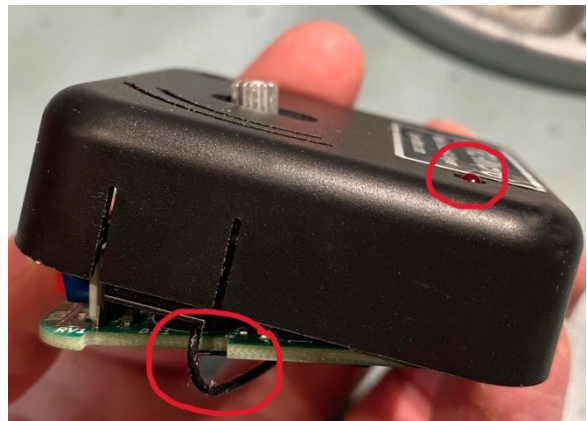
Step 13: Install the PCB in the Enclosure

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 LED about 15 degrees toward the outside edge of the board. This makes it easier to align it with its hole in the top case.



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



Make sure the coax for the antenna is in the slot on the PCB as shown in this photo.

While gently applying pressure on the PCB take a small flat blade screwdriver and insert it into the USB cutout. Use this screwdriver to push the button next to the label, MFB3, into its hole. Once it's in its hole the PCB will drop a little more into the top case.



Next, you should see the edge of the second button, MFB2, close to its hole. Insert the same screwdriver or a sharp pointed set of tweezers, into the hole and pull the second button into its hole. The PCB should again drop a little more into the top case.

NOTE: If you can't see the button in the hole, insert the small screwdriver through the slot on the PCB and push the button toward the hole.

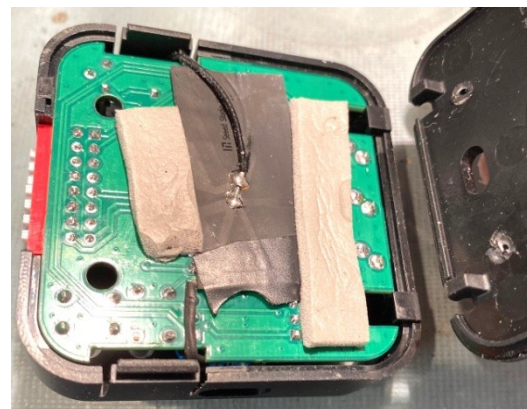


Finally, you should see the edge of the third button, MFB1, close to its hole. Using the same process as used with the second button, pull the third button into its hole. The PCB should now drop completely into the top case.



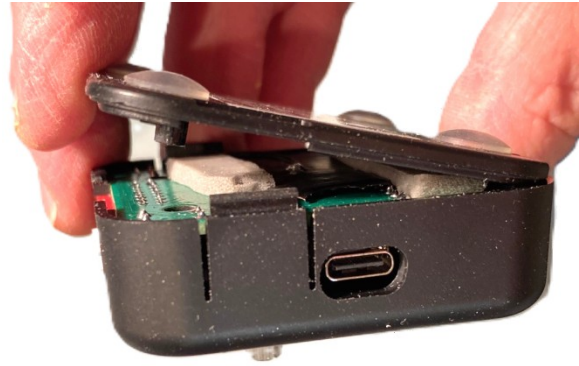
Now take the bottom cover and holding it as shown, insert the two locking tabs into the slots of the PCB on either side of the phone jacks. These tabs lock into ledges on the top case.

NOTE: All of the mounting bosses on the bottom cover must be cut off flush.



Lower the bottom panel onto the top case until the side tabs lock into the bottom panel.

This completes the assembly of the enclosure.



Step 14: Install the Knob

The Micro is supplied with a nice 32mm aluminum knob with a finger dimple. To install the knob, align the set screw 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 1/16" Allen head screwdriver or wrench.

Step 15: Button Calibration and Programming

As of December 1, 2023 all Micro kits come with the ESP32-C3 processor pre-programmed. I decided to do this because I've had two new ESP32-C3s that refused to connect to the PC so they couldn't be programmed. By pre-programming it this will insure that your Micro kit will work as designed.

The pushbutton resistance ladder can't be calibrated until the PCB is assembled. The procedure to do this is in the [Operations Manual](#) under **Appendix C: First Time Configuration, Calibrate MFBs**.

I load the current version of the firmware before I ship your kit, but you will want to check my web site to see if a newer version is available. I update the firmware frequently to add new features or fix bugs. Instructions for loading the firmware on the Xiao processor can be found in the [Operations Manual](#) under **Appendix B: Loading and Updating the Firmware**.

Finishing Up

Thank you for choosing the CTR2-Micro for your station. The [CTR2 system](#) has been evolving over the last 4 years. I enjoy everything about the process of bringing a new device into the ham community and I hope the Micro makes you smile!

If you have any questions, are confused by something in this manual, or have suggestions for improvement, please let me know. Many of the features in the Micro today came from users of the Mini and Micro asking for them.

You can reach me through my blog, <https://ctr2.lynovation.com/contact-me>, or through my email address on [QRZ.com](#).

73, Lynn, KU7Q