Queen City Con 0x3 Badge Kit Instructions

Introduction

This kit contains the parts you need to build your Queen City Con 0x3 badge into a blinking object of envy and proof of your ability to wield hot tools and make molten metal obey your wishes.

You'll need basic component testing and soldering skills to install these parts. If you're building your kit at the con, the affable geeks in the hardware hacking village are happy to help, even (especially) if you have no experience. If you're building this at home, take your time and follow the instructions closely.

Kit Contents

- ☐ **B1** AA battery holder, Keystone 2462, with tape
- ☐ **C1, C2** 0.1 µF MLCC capacitor (yellow epoxy blob with two short leads, marked 104)
- ☐ **C3** 22 pF MLCC capacitor (yellow epoxy blob with two long lead, marked 22J)
- ☐ **C4, C5** 100 µF surface mount capacitors, 1206 package (brown rectangle with silver ends)
- ☐ **HEADPHONE** Black 1/8" headphone jack
- ☐ **L1** 100 nH axial inductor (green body with stripes, brown-black-silver-silver)
- ☐ **L2, L3** 1.5k@100MHz surface mount ferrite choke (small black rectangle with silver ends)
- ☐ **LED1** 10mm RGB LED (giant LED with 4 leads)
- ☐ **LED2-LED5** 3mm LEDs, assorted colors
- ☐ **R1** 10kΩ 1/8W 5% resistor (tan body, brown-black-orange-gold)
- ☐ **R2, R3, R4** 1.5k Ω 1/8W 5% resistor (tan body, brown-green-red-gold)
- ☐ **R5, R6, R7, R8** 470 Ω 1/8W 1% resistor (blue body, yellow-violet-black-black-brown)
- ☐ **SAOV2** 2x3 female header socket
- ☐ **SW1** SPDT slide switch
- ☐ **SW2, SW3, SW4** 6mm momentary pushbutton switch
- ☐ **TTL_SERIAL** 1x6 90 degree female header socket
- ☐ **U1** ATmega328PU microcontroller, pre-programmed (28-pin DIP IC)
- ☐ **U2** 3.3V buck-boost regulator (small red rectangular circuit board)
- ☐ **U3** RDA5807M FM radio receiver module
- ☐ **X1** 8MHz ceramic resonator (blue ceramic blob with 3 leads)
- ☐ (2) AA batteries, ear buds, USB programming adapter

Optional Components

- ☐ **OLED1*** 0.96 inch I2C OLED display

***OLED1 is not included in the base kit but is available for purchase in the QCC Hardware Hacking Village. All proceeds will support purchase of equipment, tools, and consumables used in the village.**

Tools Required

- ☐ Soldering iron, preferably temperature-controlled, with a fine tip
- ☐ Solder (Sn63/Pb37, flux core, .032" diameter recommended)
- ☐ Flush cutters for trimming component leads
- ☐ IC pin straightener (not required but HIGHLY recommended)
- ☐ Precision tweezers

The circuit board and schematics are freely available at https:/oshwlab.com/jeremy15/qcc-0x3-lite-badge. Source code is at https://github.com/jcourter/qcc-0x3-lite-standard-badge/. In the Arduino IDE, choose the Arduino Pro Mini at 8MHz/3.3v to upload via the serial programming interface. The microcontroller can also be programmed via the unlabeled ICSP header next to SW2.

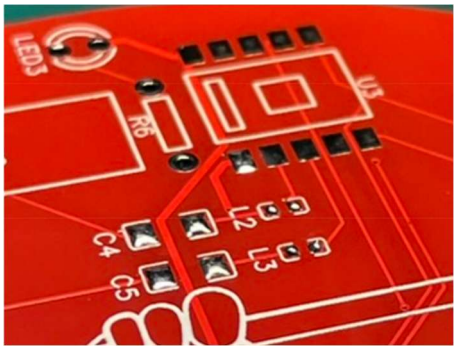
Instructions

We'll start building by installing the shortest components first and then install the rest ordered by their height from the circuit board. This will make keeping parts in place while they're soldered easier since they will be supported by your work surface.

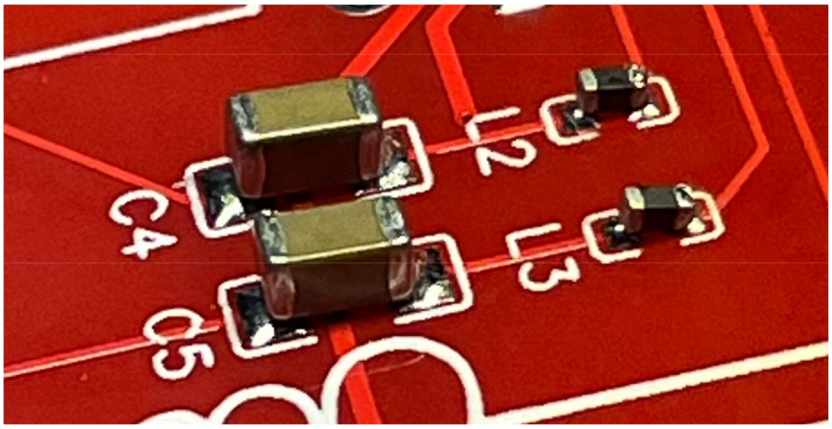
Surface Mount Parts

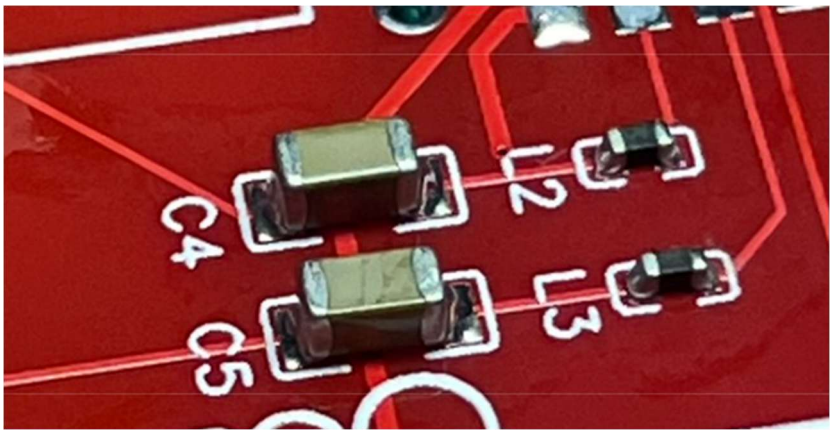
The kit contains 4 small surface-mount parts. There are two capacitors and two ferrite beads. The easiest way to install these is with solder paste, but we're going to do it with regular solder, because if you can master this method, you can easily do it with paste (also, paste is messy and has a limited shelf life). The parts are sealed in tape – leave them in there until you're ready to install them.

- ☐ Prepare the board by applying a **very small** amount of solder to the pads for **L2**, **L3**, **C4**, and **C5**. The smallest amount of solder you can manage to apply will be plenty. The easiest technique is to lay the end of the solder on the pad and quickly tap the end of the solder with the tip of the iron. The solder will almost instantly bond to the pad. Also do this for a single pad on **U3**, but only one pad.



- ☐ For each of the components, grab it with tweezers while you use the iron to melt the solder on one of the pads, then carefully place the part onto the pad. Remove the iron and hold the part in place with the tweezers long enough for the solder to solidify. The part should now be tacked to the board at one end. Repeat for each of the surface mount parts.
- ☐ Once you have placed all 4 parts, take your board over to the hot air station.
- ☐ Put a couple of drops of liquid flux on the parts (don't drown them, but all 4 parts should be wet with a small amount of flux).
- ☐ Use the hot air station to re-melt the solder on the parts. When the solder melts, the parts will sink and should self-center on the solder pads. Have your tweezers handy in case they stand on one end so you can fix this.

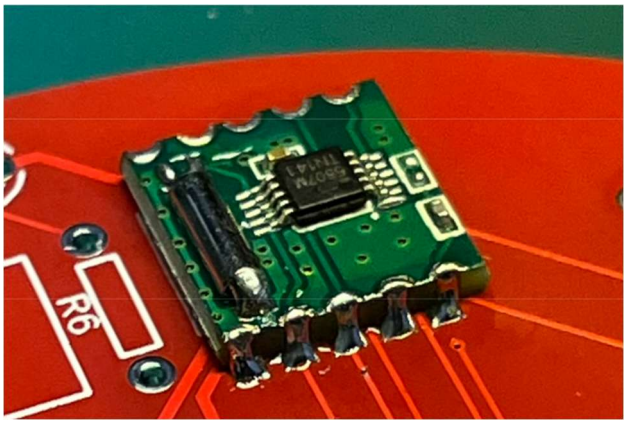




FM Radio Module

The next piece is the FM Radio module. Pay close attention to orientation – if you install this backward, it's not easy to remove.

- ☐ Observing the orientation shown on the silkscreen, align **U3** with the pads. The white silkscreen line should be exactly even with the bottom edge of the board.
- ☐ While holding **U3** in place with your fingertips, use the soldering iron to melt the blob of solder you placed on one of the pads earlier. Hold **U3** still until the solder cools. Carefully check that all the castellated holes along the edge of the board line up with the solder pads. If they don't, re-melt the solder blob and carefully adjust. Once you are happy with the alignment, proceed to the next step.
- ☐ Solder the remaining pads. Make sure you apply heat to the module and to the solder pad. The solder should form a smooth shape that flows across both parts, as shown in the picture. If the solder doesn't look smooth, re-heat it and try again.

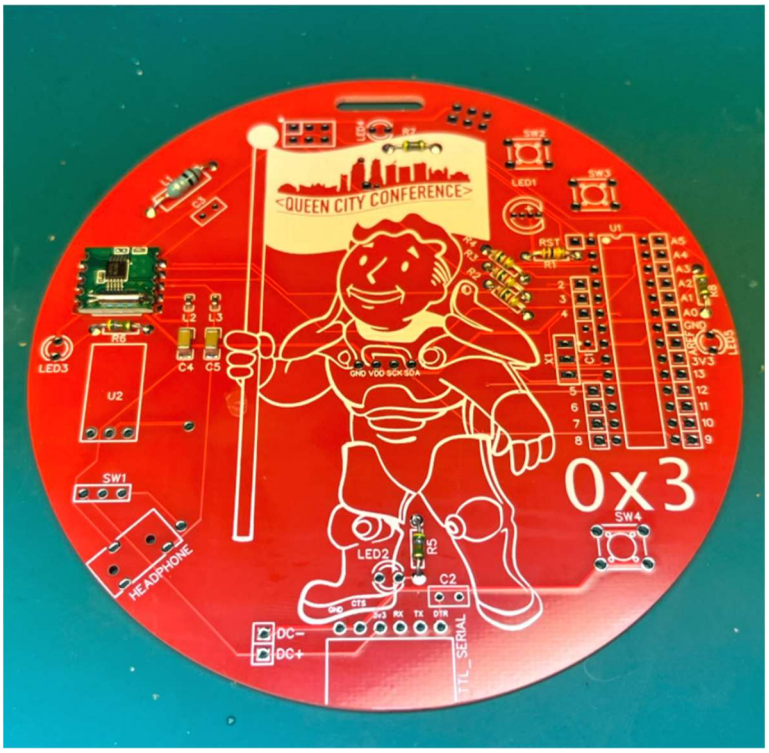


Resistors

Resistors don't have to be oriented in any particular direction, though some people do experience pain and an involuntary eye twitch if you don't orient them all in the same direction. Your choice here.

To install a resistor, gently bend the leads with your fingertip until you have a 90 degree bend, then insert the leads into the correct holes. Once the resistor is seated against the circuit board, bend the leads slightly outward so the resistor doesn't fall out when you turn the board over for soldering. It's easiest to install several at once and then flip the board over to solder them in groups.

- ☐ First, install the 10k Ω resistor, **R1**. This can be identified its stripes, which are brown, black, orange, and then gold.
- ☐ Install the three 1.5k Ω resistors, **R2**, **R3**, and **R4** (brown, green, red, gold). They are installed diagonally, to the right of the Pip-Boy logo's head.
- ☐ Install the four 470 Ω resistors, **R5**, **R6**, **R7**, and **R8** (blue body, yellow, violet, black, black, brown)**.**



Inductor

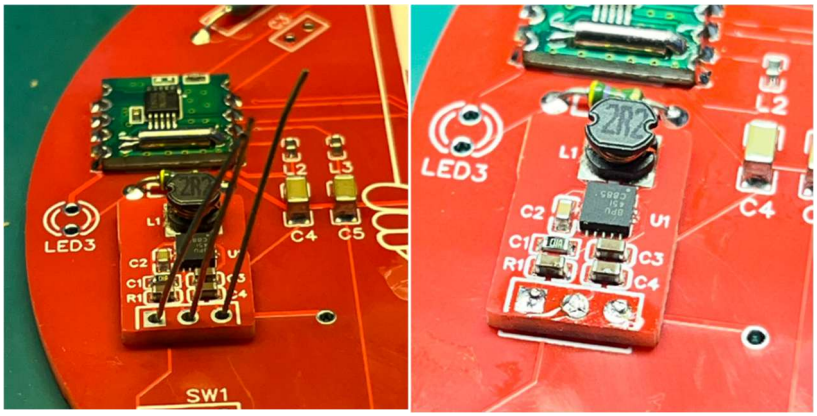
- ☐ Next, install the 100 nH inductor, **L1**. It looks like a green resistor. Your badge should look like the one to the right.

Programming Header

- ☐ Install the **TTL_SERIAL** header so the header sits on the front side of the board. Solder the connections on the rear side of the board.

Buck-boost Module

- ☐ Align the holes on the buck-boost regulator **U2** with the holes on the board. Grab 3 trimmed leads from previous steps and insert them into the holes, as shown here.
- ☐ From the top side, carefully solder the three leads, while maintaining alignment of the module.
- ☐ Trim the leads on the top.
- ☐ Turn the board over and solder the bottom side of the leads.



IC Socket

- ☐ Install the 28 pin IC socket, lining up the notch with the notch on the silkscreen outline. It's not the end of the world if your socket is installed backward, but it could lead to you inserting your microcontroller backward, and that would be bad. Bend two pins at opposite corners to keep the socket in place when you flip the board over to solder it. Make sure you apply heat to each pin until you see the solder pulled into the plated hole on the board.

Power Switch

The slide switch, **SW1**, can go in either orientation – it will work the same either way.

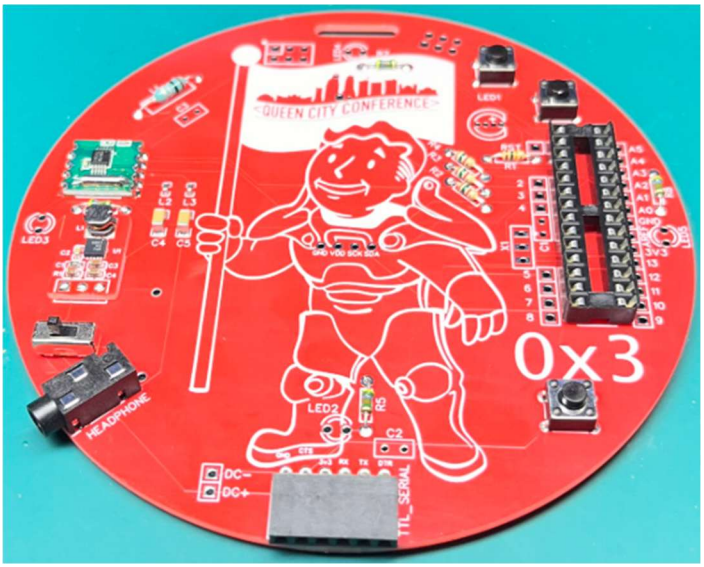
- ☐ Insert the power switch into the **SW1** location, immediately below the buck-boost module on the board. I find it easiest to solder the center lead first, then stop to verify the switch is seated fully and not crooked before soldering the other two leads. Once you make any needed adjustments, solder the final two connections and trim the leads.

Pushbutton Switches

- ☐ Install **SW2**, **SW3**, and **SW4**. The switches will snap into place with light pressure when inserted straight into the board. Flip the board over and solder all 4 pins on each switch.

Headphone Jack

- ☐ Install the headphone jack into the **HEADPHONE** position on the board. The jack has two plastic posts that will snap into the small holes on the board. Flip the board over and solder the 3 connections.



Capacitors

- ☐ Install C1 and C2, the two capacitors marked 104 with short leads.
- ☐ Install **C3**, the capacitor marked 22J with long leads. **C3** is located next to **L1**, the green inductor.

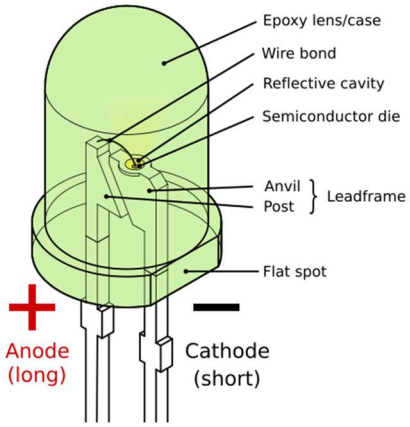
Resonator

- ☐ Insert the blue ceramic resonator **X1** into place next to the IC socket for **U1**.

LEDs

LEDs must be installed in the correct orientation to work. On most LEDs, there are two easy ways to identify the cathode (negative) side of the component. The cathode is marked with a flat at the base of the LED, and if the leads are of unequal length, the shorter lead is the cathode. You'll see that the silkscreen outlines on the board for LED1 through LED8 have a flat side, so align the shorter lead and flat spot on the actual LEDs with that.

- ☐ Insert each 3mm LED into one of the **LED2**…**LED5** locations on the board. Install them in any order you like, but make sure the short lead and flat spot at the base align with the flat spot on the component outline on the circuit board.
- ☐ Solder one lead of each LED first, then verify everything is seated and lined up to your satisfaction. If you need to make adjustments, melt the solder and quickly reposition the LED as needed. Don't get the LED too hot or it won't survive. Solder the remaining lead on each LED and then trim the leads.
- ☐ Insert the large 10mm RGB LED, **LED1**, into its position. **The longest lead goes into the hole marked +**. The leads are spaced very closely, so use care not to create a short circuit between them. A little extra flux can be very helpful to prevent inadvertent bridges.

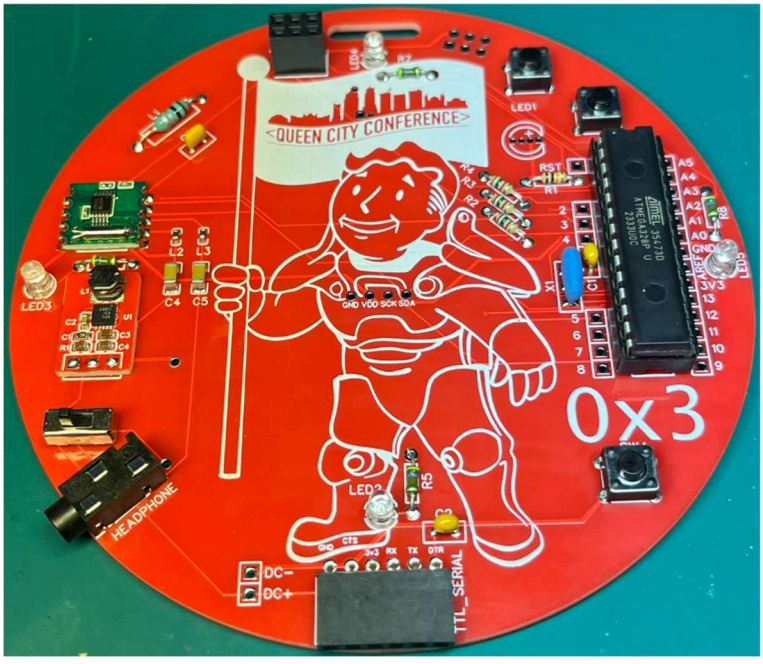


SAOV2 Header

- ☐ Insert the small 2x3 female header into the **SAOV2** position (labeled on the rear side of the board), near the top of the silkscreened flagpole. **Do not confuse this with the angled ICSP pins next to SW2.**
- ☐ Solder one pin first, verify the header is fully seated, and then solder the remaining 5 pins.

Insert the Microcontroller

- ☐ Straighten the pins on the ATmega328PU IC, preferably using a pin straightener. It can also be done very carefully on the surface of a table with a ruler or other flat object to bend the pins so they are parallel and ready to go into the IC socket.
- ☐ Carefully line up the pins and place the ATmega328PU on the socket, with the notched end matching the notched end of the socket, next to **SW3**.
- ☐ Slowly insert the IC by applying light pressure with your fingers, moving back and forth along the length of the IC so that both ends are seated at the same time.



OLED Display (Optional)

This board supports a 0.96 inch OLED display, directly mounted to the 4 holes at the center of the board. Compatible OLED displays are readily available on Amazon, or you can buy one in the village at a modest markup, which helps to pay for all of the equipment, tools, and supplies available there. If you want to install an OLED, it's easier to do before mounting the battery holder, but can be done afterward with a little caution.

- ☐ Apply a small piece of double-sided tape to the bottom edge of the OLED display (opposite side from the pins). This will help support the display.
- ☐ To install the OLED display, position it so the 4 pins are at the top of the display and drop it into the holes on the badge.
- ☐ Turn the badge over and solder the 4 connections and then trim the leads.

Check Everything Over

Before you install the battery holder, re-check your solder joints, especially the ones for **R5**, **LED2**, and **SW4**, since those will be under the battery holder after you mount it. If you're building your kit in the village, now is a great time to call one of our volunteers over to help you check your work.

Battery Holder

- ☐ Peel the backing from the double-sided tape on the battery holder.
- ☐ Carefully insert the battery holder leads through the back side of the board, taking care that the end with the leads contacts the board first.
- ☐ Press firmly on the end with the tape to attach the battery holder.
- ☐ Solder the battery holder leads on the front side of the board, and then carefully trim them. The headphone jack is very close, so use care not to damage it during this process.

Final Test

- ☐ Install three AA batteries and slide the switch to the "ON" position. If the RGB LED lights up, you have succeeded. If not, immediately remove the batteries and start troubleshooting (if you're building your badge in the village, ask a volunteer for help).

External Power Supply (Optional)

Solder pads located near the serial programming port have been provided if you decide you want to turn your badge into a permanent desk memento. Any DC power supply (use a regulated power supply unless you install a voltage regulator) with an output of between 2 and 5V would work here.

With an external power supply attached, the on/off switch selects between external and battery power, so you'll want to remove the batteries.

Circuit Schematic