ZReach Z-Wave Controller Reference Design

# Introduction

## Features

* Public GitHub repository – Open-Source Repository – MIT License
* KiCAD schematic & PCB layout
  + Easily imported into other tools like Altium
  + Includes gerbers for immediate production
* Roughly hockey puck in size with an antenna poking out of the center 

Not the real thing - but similar with USB cable

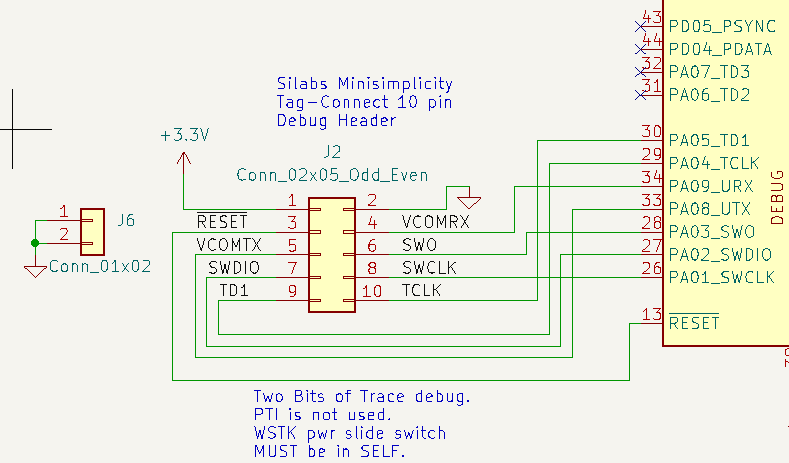
* EFR32ZG23 based +20dBm & CP2102N USB->UART
* Firmware is the binary SerialAPI in the SDK
* Documentation with full details for customization and replication
* Ideally pre-certified by the Alliance and used as a test case for the SerialAPI controllers & CTT
* Development budget: $30k
  + Schematic, PCB layout, 10 protos, functional testing, RF range testing, Documentation
  + Target Assembled/tested kit cost less than $50 (not including enclosure)
    - Target BOM cost under $10@10K
* White Paper on Antenna Best Practices with simulation results from antenna expert
* Development Time Line
  + 10 weeks from agreement to delivered prototypes

# Theory of Operation

## Z-Wave MCU

The UART Rx/Tx pins are connected to the CP2102 thru 100 ohm resistors to allow the Tag-Connect debug cable to drive the pins during manufacturing. RailTest is needed to send/receive UART commands during crystal calibration.

A Tag-Connect 10 pin header with alignment pins is used for programming and test during production and debug. Two Trace pins are wired to the header to enable Trace debugging using the adaptor board in the [etm\_zwave](https://github.com/drzwave/etm_zwave) repo. The PTI pins are not wired as Z-Wave typically uses a separate standalone Zniffer. The 2 ground pins are often needed for Trace debugging to improve signal integrity.



The ZG23 DCDC switching power supply is utilized to convert the 3.3V to 1.8V for the VDD power supply. The USB 5.0VDC is passed thru a linear LDO to drop the supply voltage to 3.3V. There will be significant power spikes of about 130mA when the radio transmits at +20dBm. The LDO and USB must be able to handle that sudden change cleanly to ensure good RF range. The PAVDD is powered with the 3.3V supply to be able to power the RF amplifier to +20dBm. The analog supply and IOVDD are on the 3.3V to provide better analog voltage range and connectivity with other external devices. The internals of the ZG23 are powered with 1.8V. Sensitive power supplies have additional ferrite beads and capacitors to ensure clean supply voltages.

## Antenna Filter and Match

The RF path from the ZG23 to the RF50 signal is straight from the Silabs DevKit reference designs. This path filters the RF and shifts the high impedance pins to be close to 50 ohms. From there to the SMA connector is a standard 3 element pi filter used to match the impedance of the antenna. The values for these 3 elements will be determined once the PCB and antenna have been tuned. A U.FL footprint is in this path to make tuning easier via a U.FL cable to a network analyzer. The U.FL is only mounted to bare PCBs to help with tuning. Thus, it is listed a NM in the schematic and is why there is a red X thru it. The SubGx1 pins are not used.



## USB-C Interface

A Silicon Labs CP2102N USB to UART chip is used to connect the SerialAPI to the host computer. A USB-C connector is used but only USB 2.0 is supported. USB3 is not supported or needed as the data rate of the UART is 115200 baud. The design follows the reference example in the CP2012 datasheet. This [article](https://hackaday.com/2023/08/07/all-about-usb-c-example-circuits/) was used as a guide.

## Power Supplies

USB-C provides +5V power in the typical application of a controller. The +5V is reduced to 3.3V via an LDO to power the on-chip switching regulator on the ZG23. The radio is powered from +3.3V to achieve a transmit power of +20dBm to the PAVDD pin.

When prototyping a low-power end device, the battery holder can be populated and connected to a CR123A or similar 3V battery. No reverse battery protection is provided so the battery holder must ensure the polarization. The LDO and USB devices are not populated when developing a battery powered device.

## PCB Ground Plane

A very important part of an IoT radio is the ground plane. Ideally the ground plane is perpendicular to the antenna and is half-wavelength. At 868MHz the wavelength is 34.5cm and 920Mhz is 32.6cm. The exact dimension isn’t that critical as long as it’s close. The ½ wavelength of 17cm but a square is more cost effective than a disk, thus divide by the diagonal (1.41) to get a 12cm on a side or roughly 5”. The ground plane fills the entire bottom side of the PCB to ensure there are no eddy currents or blockages. The next layer is used to distribute power. The upper two layers are used for interconnection of the ICs. All layers are flood-filled with ground as much of the area is open to provide a large ground plane for ideal RF performance. A smaller PCB size can be easily prototyped by simply making the PCB smaller with the potential of a loss in RF performance.

A related calculation is the width of the infeed trace. That it calculated using one of the many coplanar waveguide calculators such as: <https://chemandy.com/calculators/coplanar-waveguide-with-ground-calculator.htm>.

For [OSHPark](https://docs.oshpark.com/services/four-layer/), the relative dielectric constant is 3.61. The thickness of the prepreg is .2021mm. Usually the gap is set to .3mm then try various Widths until the calculator comes up with close to 50 ohms. In this case, it’s .44mm. I might want to lower the gap to .25 or even .2 but that might make the impedance too low.

## RGB Color LED

An RGB Color LED can be used to provide visual status of any sort. Utilizing this LED requires customizing the SerialAPI. The LED can be used when Z-Reach is used as an end-device for Indicator CC support.

* Red = PC04 – Low=ON, High=off
* Green = PA00
* Blue = PA10

DETAILS TO BE ADDED HERE ON USING PWM to change the color. NOTE THE SCHEMATIC has Blue and Red backwards.

## Learn Mode Switch

PC05 is connected to a SPST momentary pushbutton switch. This pin is an EM4WU pin so it can wake the chip from EM4. End Devices use this pin to enter Classic Inclusion/exclusion mode and press and hold for more than 5 seconds to factory reset.

## Optional Connections

Extra connections are incorporated in the design to allow prototyping of many other types of Z-Wave devices. These connectors and devices are normally NOT mounted for SerialAPI applications.

* I2C pins on a QWIIC connector easily attached many Sparkfun sensor and display boards
  + SCL=PB00, SDA=PB02, select Fast Mode (400K)
  + Use the I2CSPM peripheral to communicate with QWIIC devices
* 32KHz crystal supports Time CC. The on-chip low-speed oscillator drifts by a few minutes per day making it not usable for schedules or other time-of-day uses. The crystal reduces drift to a few seconds per day.
* Two 5 pin GPIO headers provide generic connections to other devices

# Board Status:

Rev A: scrapped – USB chip difficult to solder due to pads under the IC.

Rev B:

* 1: Working – not CTuned
* 2: Working – not CTuned

# Reference Documents

1. EFR32ZG23 Datasheet
2. EFR32xG23 reference manual
3. Z-Reach github repository
   1. See docs directory for this document and the Z-Reach datasheet
4. [CP2102N](https://www.silabs.com/documents/public/data-sheets/cp2102n-datasheet.pdf) Datasheet

# Journal

Details of the development and timeline are described here in reverse chronological order. The SE firmware is 2.1.2 but GDK 4.4.1 has 2.2.4 so I upgraded the SE. Now it gets to main but immediately ends up in HardFault\_handler. I switched to Ozone and it ends up at the WFI like it’s supposed to. WTF. And now so is SSv5. WTF. And it now has a QR code. Now to enable debugprint. Select it, then wait at least 1 minute. Then install it. Wait another minute or more. Select IO Stream: EUSART(0). Wait another minute. Click on Install. Wait several minutes. Doesn’t seem to do anything. Nothing asking me for which GPIOs to use. Ask it to build an has nothing to do so nothing has changed as far as SSv5 is concerned. Uncomment DEBUGPRINT and attempt to build fails because it needs sl\_iostream\_write. Give up on DEBUGPRINT. I don’t really need it but it is often handy.

## 2024-04-04 – Rev B PCB Debug

### Antenna Smith Charts

David Zima mentioned that some antennas may assume a ground plane but other may not. Thus, I used one of the Rev A boards, cut the trace to the SMA, soldered another SMA to the bottom of the board which then can be screwed into my VNA to generate a Smith chart for each specific antenna below. I used my Agilent N9921A FieldFox spectrum analyzer in VNA mode. Calibrated the open connector (or the connector with the GND plane) to zero out the connector, then screwed the antenna into the N9912A and had it plot the smith chart from 850 to 950MHz. Used the Log Mag chart to find the minimum and set the marker there.

SETUP:

Eleven antennas were tested of various sizes and cost:

The setup used an Agilent N9912A Vector Network Analyzer. Two passes were made with each antenna, one with a GND plane and one without. The GND plane used is a Rev A PCB with an SMA connector soldered to the bottom of the board and the RF trace on the PCB was cut.

The setup shown here has the GND PCB attached. The QuickCal calibration of the N9912A was used to zero out the effect of the connectors and ground plane prior to testing.

|  |  |  |
| --- | --- | --- |
| **Part Number** | **With GND Plane** | **Without GND Plane** |
| **[TI.92.2113](https://www.digikey.com/en/products/detail/taoglas-limited/TI-92-2113/11197416)**  $6.20 – gain 1.21dBi – 198mm  Comments:  Looks like the one shipped with Silabs DevKits  Relatively insensitive to hand nearby unless touching.  VSWR under 1.5 |  |  |
| [**TS.89.4113**](https://www.digikey.com/en/products/detail/taoglas-limited/TS-89-4113/15284364)  $40.56 – gain 3.5dBi  Comments:  Directional antenna with 3.5/2.7dBi gain in 1 direction – Tomahawk shaped.  Not a candidate for Z-Reach but interesting for comparison purposes.  Not sensitive to nearby hand until touching | Basically, the same since the antenna is on a short cable so the GND plane is not close enough to impact the response. | Adding the ground plane made little difference |
| [**ANT-8/9-MMG1-SMA-1**](https://www.digikey.com/en/products/detail/te-connectivity-linx/ANT-8-9-MMG1-SMA-1/16592298)  $5.66 – gain 2dBi – 83mm  Comments:  Cable to a small magnetic mount that will stick to iron.  Insensitive to hand unless almost touching the antenna.  Advantage here is the PCB could be quite small but include a metal disk to stick it to.  VSWR is under 2 (min of 1.5) for much of the range  Wide frequency range so the same antenna is OK for EU and US |  |  |
| [**ANT-916-CW-RH-SMA**](https://www.digikey.com/en/products/detail/te-connectivity-linx/ANT-916-CW-RH-SMA/1962849)  $8.89 – gain -1.3dBi – 51mm  Comments:  Very Small.  Hand closer than 1” to make a significant difference in smith chart  Price is the same for 1 or 10,000. |  |  |
| [**FW.95.B.SMA.M**](https://www.digikey.com/en/products/detail/taoglas-limited/FW-95-B-SMA-M/6362786)  $9.13 – gain 2.7 dBi – 226mm  Comments:  With GND VSWR is 1.1  Sensitive to hand 1” away  Slim and stiff but quite long |  |  |
| [**DELTA12C/X/SMAM/S/S/17**](https://www.digikey.com/en/products/detail/siretta-ltd/DELTA12C-X-SMAM-S-S-17/6096292)  $9.65 – gain 3dBi – 130mm  Comments:  Classic Rubber Ducky  Terrible VSWR barely below 3  Very little sensitivity to a hand but it’s not good in the first place |  |  |
| [**JCG402LR-2**](https://www.digikey.com/en/products/detail/jc-antenna/JCG402LR-2/15814458)  $2.42 – gain 2dBi – 110mm tiltable  Comments:  Relatively insensitive to hand nearby until touching the base. The tip was not sensitive implying there’s no wire up there. Touching close to the base brings the impedance down right to 50 ohms. Thus it should be possible to tune the antenna to the desired band.  Lowest cost antenna.  Better without the ground plane! |  |  |
| [**2600130083**](https://www.digikey.com/en/products/detail/w%C3%BCrth-elektronik/2600130083/11680415)  $14.88 – gain 0.9dBi – 179mm  Comments:  Covers 868 thru 928.  Highly impacted by a hand nearby without a GND, less so with GND.  VSWR above 2. |  |  |
| 868 antennas: |  |  |
| [**ANT-868-CW-HW-SMA**](https://www.digikey.com/en/products/detail/te-connectivity-linx/ANT-868-CW-HW-SMA/5592340)  $8.74 – gain 0dBi – 135mm  Comments:  One of the better antennas  VSWR well below 2 for a wide frequency range  Hand has little impact more than 1” away  There is an identical antenna at 916MHz - ANT-916-CW-HW-SMA |  |  |
| [**FW.86.B.SMA.M**](https://www.digikey.com/en/products/detail/taoglas-limited/FW-86-B-SMA-M/6362785)  $9.13 – gain 2.8dBi  240mm  Comments:  Thin fixed (similar to the 915 one above)  Hand has little impact until less than 1” away  Needs the ground plane to have VSWR below 2.  One of the larger antennas |  |  |
| [**FLEXI-SMA-868**](https://www.digikey.com/en/products/detail/rf-solutions/FLEXI-SMA-868/5845738)  $5.12  Comments:  Short Rubber Ducky  One of the cheaper options, it’s also not very good based on the smith charts. Especially with a large GND plane. |  |  |

### Conclusion:

* Recommended antenna is the same one shipped with the Silabs DevKit
  + [TI.92.2113](https://www.taoglas.com/datasheets/TI.92.2113.pdf) - [$6.29](https://www.digikey.com/en/products/detail/taoglas-limited/TI-92-2113/11197416?s=N4IgTCBcDaICoEkB0BOMSwEZMGYAEIAugL5A) @1k – +1.21dBi – 198mm – 915MHz
  + [**TI.85.2113**](https://www.digikey.com/en/products/detail/taoglas-limited/TI-85-2113/11197203) is the 868 version
  + Reasonable cost, good response,
  + Needs more testing
  + More to come here
* Second choice is the antenna with a cable and magnetic mount
  + [ANT-8/9-MMG1-SMA-1](https://mm.digikey.com/Volume0/opasdata/d220001/medias/docus/6146/ANT-89-MMG1-SMA-1.pdf) – [5.66@1K](mailto:5.66@1K) - +2dBi – 82mm – both 915&868

Where to test the RF Range? Testing at the beach is pretty terrible due to sand and sun. Much better if the range can be down a river with a paved sidewalk alongside. The River Rest Park in Haverhill MA has a 1+ mi view down the Merrimack river though there are trees in the way at various points. Along the Buttonwoods Riverside Park.

## 2024-04-03 – Rev B PCB debug

Still trying to get the SwOnOff project to work. Tried again but this time I just waited for several minutes between each step to see if it’s just SSv5 being so incredibly slow. To start off, I built just the SwOnOff and only changed the region to US\_LR and it sits in all sorts of odd loops in RAILINT and BURTC. Board #1 does not have a QR code so the code hasn’t finished initialization. It does get to Main().

Using Ozone to manually set GPIOs to check out the LED. Green works, Red and blue are backwards but work. I’ll just change the documentation. PC05 the momentary switch for Learn Mode works. I’ll check out I2C later. Right now I need to connect Basic On/Off with turning on the LEDs so we can run the ERTT.

Nothing in the .SLCP to configure binary switch CC. Looking thru the code in app.c is a p\_switches structure which apparently needs something to connect to. P\_switches comes from cc\_binary\_switch\_get\_config(). This comes from SwitchOnOff\_hw.c which again comes from p\_switch. How is this then threaded to a GPIO? SWITCH\_STATUS\_LED is from APP\_LED\_A which is BOARD\_LED1 which is simply an enum of 0 (in board.h). The board\_getLedPort uses m\_led\_info[led] to get the gpioport. This in turn uses LED\_INFO(LED1). But in this project m\_led\_info is never assigned a value. Create a project using a TBZ and see if it’s assigned somewhere. AHA! The key is the LED\_INFO is a macro that expands LED1 into several other #defines with LED1 as part of the name. Talk about obfuscation! The file radio\_no\_board.h says to Use Configuration Wizard in Context Menu to set the various LED values. But WHERE in the Wizard? Z-Wave Target Board has the configuration of the LEDs and buttons. According to the ReadMe.md, BTN0=toggle state of the switch, BTN1=learn mode, LED0=state of switch LED1=indicator CC. Thus, set BTN1=PC05 and LED1=PA00 (green) and LED2=PA10 (blue) (active low). The LEARN button and the blue LED comes on when pressed! And sending an On/Off turns on/off the green led! So the code is much more tightly integrated into SSv5 now. However, following the code is nearly impossible! This is enough to begin some range testing using the ERTT.

Next step is to get the I2C QWIIC connector checked out and connected to a GPS to enable tracking an end device to get quantitative range measurements. There is a Peripheral Driver I2CSPM in the wizard but it is polled and blocking which is not great. Should be interrupt driven. What does the TBZ multi-level sensor do? RHT sensor is enabled in Platform/Board/Board Control. The Si70xx driver is installed. The I2CSPRM is enabled and configured. The Reference Clock frequency is set to 0 (does that matter or does 0 select the HFXO?) – no documentation of course so who knows. There is a short doc on docs.silabs.com but doesn’t explain the configuration in the SLC. The GPIOs are configured. Back to the SwOnOff project – install the I2CSPM. Selected Fast Mode (400K) and assigned the SCL and SDA pins. The project compiles and the I2C pins show up in the PinTool. The I2C peripheral is initialized in sl\_driver\_init so it should be working. The only interface is via the I2CSPM\_Transfer function in sl\_i2cspm. I did verify the power and gnd connections on the connector so the PCB is a go.

## 2024-04-02 – Rev B PCB debug

Built the OTA bootloader which fortunately just builds using the defaults. Created a SwOnOff project to quickly check out the LED and QWIIC connectors. Once those are checked out, I can order more boards. First step is to get DEBUGPRINT working via a custom board. With the DevKit, the USART is used for debugprint but with a custom board the project fails to build because of “undefined reference to sl\_iostream\_write”. Click on the .SLCP file, search for IO STREAM, select Services/IO Stream which then opens a popup with a variety of choices – use IO Stream:USART. That didn’t work and I cannot get any other USART to configure as SSv5 says Selected Modules are “none” with no other options. So, I deleted the project and started again but this time I enabled Iostream before trying to compile.

This time, when I Installed Z-Wave Debug Print a popup asked for which IO Stream I wanted to install so I selected EUSART. But seems like nothing happened. Closed SSv5 and reopened and debug print is not enabled so something did not save. Closed SSv5 again and opened it but this time I installed IO Stream USART first then enabled debugprint. That seems to have worked. The project compiles. Not exactly sure how that worked and it’s terrible that I have to basically randomly try various things to find one that works. All because the Silabs environment is way too heavily weighted towards their devkits and NOT customer boards!

The SwOnOff project does not boot and gets stuck in BOOTLOADER\_TEST\_FAULT\_HANDLING for some reason? I get different results each time I run. It’s not in a WFI which is where it usually is. Using Ozone I still have no idea what’s going on. The BURTC is running but seems like it’s in some sort of loop.

## 2024-04-01 – Rev B PCB debug

How to build the bootloader for Z-Wave for a custom PCB instead of a devkit? The problem with the devkits is they all use hi-security “B” parts which most users don’t need or want. The mid-security “A” parts are cheaper and firmware written for an A part will run a B part but not the reverse. Thus, it is important to compile the firmware for the A part. Downloading the devkit pre-compiled sample apps gets stuck in a loop presumably waiting for a high security feature. Using the Silabs search button didn’t find any usable articles. However, using google to search for “z-Wave 800 series build Z-Wave bootloader” bring you to: <https://community.silabs.com/s/article/Creating-OTA-bootloader-for-Z-Wave-800-series>. This KB is a bit light on the topic but at least gives some guidance. But we need an OTW version for the serial API. Add “OTW” to the google search and it pops up: <https://community.silabs.com/s/article/z-wave-700-otw-of-controller> .

This last article says to look here for the pre-compiled bootloaders: \demo-applications\protocol\z-wave\Apps\bin\

I filed a “feedback” on the website that the binary files are not in this directory.

I have a blog article in the works on how to build the bootloader. I was able to build the bootloader once I configured the Bootloader UART Driver component. The Pintool shows the Rx/Tx pins correctly.

[UG489](https://cn.silabs.com/documents/public/user-guides/ug489-gecko-bootloader-user-guide-gsdk-4.pdf) Bootloader User Guide is the main reference document but it does not explain the key requirements for Z-Wave. The document does not mention Z-Wave at all.

Next step is to build the SerialAPI in the same method – build it using a devkit and copy the necessary options into the custom board project. Built it and it chooses the standard UART pins by default so the project just builds without needing any configuration. Connect the PCC to it and it works and can join a node!

Board 1 debug: Ohm meter shows zero ohms between D2 pins 2 and 3 which are the 2 USB data lines. Short was under the USB-C connector. Touch soldered that to open the short and now the board shows up as a CP210x driver COM port.

Articles on how to implement [I2C](https://community.silabs.com/s/article/z-wave-700-how-to-implement-i2c-into-the-z-wave-embedded-framework?language=en_US) and [SPI](https://community.silabs.com/s/article/z-wave-700-how-to-implement-spi-into-the-z-wave-embedded-framework?language=en_US).

Next step is to get a SwitchOnOff into board 1 and then see how far can they reach. Need to quickly test out the LED and QWIIC connectors then order 9 more PCBs if they are all OK. Maybe quickly check the antenna tune and have to ctune both boards.

## 2024-03-29 – Rev B PCB assemble and debug

Three boards arrived today and I built 2. Still forgot to order 7.2pF caps but I had 7.5 which are close enough for now. Everything fits and the new refdes line up with the BOM.

The ZG23 soldered well but had a few shorts and both boards had 1 side slightly raised which in turn had several opens. For prototype builds I think removing some of the solder paste of the EP would make the chip not stand up. Maybe also push it down harder to make sure all 4 sides will quickly apply surface tension to hold it down. The footprint is a standard footprint so I assume for volume production the amount of paste would be more consistent and the IC would be applied with more pressure.

Board 2 connects via USB! The CP210x driver loads. Board #1 comes up as an unknown device. Touch soldered the USB connector which looks fine now – had some shorts originally and looked OK before I touched them up again. Device Manager reports the Device Descriptor Request Failed.

For now, I’ll focus on board #2 and come back to board #1 later.

The demo files are stored at: C:\SiliconLabs\SimplicityStudio\v5\offline\com.silabs.sdk.stack.super\_4.4.1\protocol\z-wave\demos. I downloaded the BRD4204 version of the SerialAPI solution but PCC does not recognize it. I can’t debug it either as all I have is the binary which is in some sort of loop and not at the EM1 WFI so the firmware is stuck somewhere.

But this has an A part and there may be other special things on the PCB that don’t match my board (I did install a 32KHz crystal). I need to build a bootloader and a SAPI using the actual part which will take some time to figure out how to configure both the bootloader and SAPI.

## 2024-03-18 – Rev A PCB debug – Rev B update

What are the possible reasons the CP2102 doesn’t connect to a PC? The most likely problem is that D- isn’t connected or is shorted to something as pin 5 is one of the recessed corner pins. First thing to view is to see if reset is being held low during power-on. Then look at the D+/D- pins during power on and see if they toggle. Not much else I can debug as the CP2102 is fully encapsulated – no oscillator or anything else coming out. Configuration is done via USB. Check that pin 13 (WAKEUP\_N) is high which it should be. Should see the TX/RX uart pins toggle as well by the ZG23 sending power on commands. On power up I see the USB pins toggling probably sending a wakeup sequence. I also see the ZG23 sending the wakeup commands on the TX. Reset quickly follows powerup so I think that is fine. I posted this [question](https://community.silabs.com/s/question/0D5Vm000005tCdsKAE/cp2102-unresponsive) to the community page. I searched the community but didn’t find anything similar.

I might as well fix all the issues and release Rev B as I can’t see what else might be wrong. Either the CP2102 connects, or it doesn’t – no way to debug it if it fails.

Rev B issues are in github:

1. The refdes are not ordered relative to their position on the PCB. I forgot to run the reorder after placement so the refdeses are the random order they were placed on the schematic.
2. C11 and C12 are 220pF caps but have a 0603 footprint instead of 0402. The 0402 can be installed in the 0603.  
   Fixed in Schematic, not PCB.
3. The color LED is the wrong pinout in the schematic - the LED needs to be a common anode, not cathode. The footprint has the wrong pinout for the selected LED - QLSP14RGB\_B (7 cents) which is a good LED. Might have to make a custom footprint and symbol to fix it.

* A LED I've used before is: CLV1A-FKB-CJ1M1F1BB7R4S3CT ($0.38) Though it does have 3.2V forward drop which won't work if on battery power where the voltage might be only 2.4V.

1. The infeed track width to be 50 ohms should be .44mm instead of .3. Move L1 to the right just enough to get the flood fill to fill in between L1 and L2. Ideally squeeze a via in there. The traces out of the ZG23 have to start at .2mm (same size as the pad) but should widen immediately to .44mm.
2. Switch from the CP2102N QFN20 to the QFN24 package. The QFN20 has 4 pins that are below and recessed slightly within the package. The QFN20 is a standard pin arrangement so the solder joints can be visually inspected and reworked as needed.
3. The QWIIC connector footprint has thru-holes but the part is SMT.
4. Label the signals on the Tag-Connect
5. Add an optional reset switch (normally not mounted). It's just super handy during debug to be able to reset the chip by pressing a button.

Rev B upgrades the KiCAD database to V8.0.0. This required updating a number of symbols and footprints which are much more standardized in V7 – specifically pin 1 markers are in most footprints.

Somehow pins 29 and 30 are backwards in Rev A? I corrected this in Rev B. TD1 and TCLK.

The U.FL connector for whatever reason has a DRC error in it that the solder mask bridges 2 nets. I made it a little smaller. As a result, there is an exclusion that the footprint doesn’t match the one the library.

The 32KHz crystal also has an exclusion because I edited it again as the silk screen lines are too close to the pad.

QWIIC connector – easier to just modify the one I already have to be SMT than try to import one. Well, turns out in KiCAD V8 there already is a JST connector library and it has the correct footprint!

DRC Report: The 2 symbols were slightly fixed to prevent errors. The extra footprints are the fudicials.

\*\* Drc report for ZReach.kicad\_pcb \*\*

\*\* Created on 2024-03-18T15:06:45-0400 \*\*

\*\* Found 2 DRC violations \*\*

[lib\_footprint\_mismatch]: Footprint 'Crystal\_SMD\_3215-2Pin\_3.2x1.5mm' does not match copy in library 'Crystal'.

Local override; warning (excluded)

@(207.0000 mm, 114.0000 mm): Footprint Y1

[lib\_footprint\_mismatch]: Footprint 'U.FL\_Hirose\_U.FL-R-SMT-1\_Vertical' does not match copy in library 'Connector\_Coaxial'.

Local override; warning (excluded)

@(203.0000 mm, 97.2000 mm): Footprint J4

\*\* Found 0 unconnected pads \*\*

\*\* Found 2 Footprint errors \*\*

[extra\_footprint]: Extra footprint

Local override; warning (excluded)

@(174.4000 mm, 74.6000 mm): Footprint Fid1

[extra\_footprint]: Extra footprint

Local override; warning (excluded)

@(223.0000 mm, 145.2000 mm): Footprint Fid2

\*\* End of Report \*\*

Release board for fab via OSHPark and ordered a new stencil. Updated BOM with new RefDes.

## 2024-03-15 – Rev A PCB debug

Solder blobs on boards 1 & 2: Cleared them without too much trouble using soldering iron and a solder sucker. The USB connector was the most problematic but seems OK.

Plugging in the Minisimplicity header and powering the board via the WSTK SSv5 is able to detect the part and pull the info from it. Since it’s an A part, it seems none of the precompiled Demos are available so I have to create a SerialAPI and bootloader. Or is simply that this is a custom board so SSv5 doesn’t assume the GPIOs are where they are on a devkit? Naturally the bootloader project won’t compile because there’s no EUSART assigned. This is because all of Silabs projects are very centered on their devkits and if you don’t use one of those then you are 100% on your own with no help at all.

I built a BL and SAPI using the ZGM230 DK2603 board as the target. Flashed the binaries into the board and it powers up and has configured a DSK so it must be doing something. But it sits waiting for the BURTC to sync which it never will because I don’t have the 32KHz oscillator installed. But since I built the project using a devkit board it assumes it is setup. Ugh. Pintool doesn’t show anything in the SAPI project (or the BL). I can’t easily copy that to an empty SAPI project. I then create a blank SAPI project then try to figure out what else needs to be configured. The version via the devkit installs Platform->board->starter kit->BRD2603A which in turn enables a ton of stuff but not in a way I can then copy parts I need in a blank project. I created another SAPI project and this time it compiled! WTF? There are no VCOM or EUSARTs in the .slcp file? How is the UART being wired thru? How would we change it? It is in the pintool! Not sure how. It does appear to be running. I want to see if it’s coming out the UART. PC Controller does not find it thru the WSTK. Config/serial\_api\_config.h uses USART0 PA08 Tx and PA09 Rx.

USB is not working – the PC doesn’t detect a plugin in either orientation. Power is working. +5V VBUS comes in and the LDO makes 3.3V just fine. But nothing shows up in Device Manager on my PC. Using SSv5 Console it doesn’t connect. Not sure what that means. Is Tx/Rx backwards? The CP2102 is a QFN20 but the 4 corner pins are BELOW the package and not visible at the edge. Would be much better to use the QFN24 instead which is slightly larger but all the pins come to the edge of the package and none are recessed. They are the same price on Digikey and we have to room on the board. USB D+ pin is on the corner of the package. I would have thought the CP2102 will show up as a USB device when plugged into a computer without any interaction via the UART. I could wire an FTDI to the resistors to get something to work and check more things out. But that isn’t a good test of the RF range since so many things are hacked.

Board #1 & #2 are recognized via SWD and Commander: EFR32ZG23A020F512GM48

Ordered the missing components from the BOM. Not needed immediately. I got a bunch of antennas in the mail today.

## 2024-03-14 – Rev A PCB build

I built 3 boards. Board #3 has been cut down to roughly the minimum size to compare with the full size boards to measure the RF range difference with a smaller ground plane. Basically answering the question – does the additional cost for PCB real estate significantly improve the RF range? Overall size of the reference design isn’t that much of a concern but cost is an important factor though range is the priority.

I forgot to order 1.3nH inductors and 7.2pF caps. I had 1.2nH and 6.8pF which are close enough. I’ll order the correct ones for the next build. The QWIIC connector I ordered is SMT but the PCB is TH so need to pick a different connector. I didn’t order the 32khz crystals but those are not needed at this point. The QWIIC connectors appear to all be SMT so I’ll change the PCB and keep these components.

The solder paste seemed a bit thick and resulted in several shorts. I ordered the .003” thick mylar stencil. The only other option is a .005” so I am using the thinner one. My solder paste is a little thick as the basement is 55F so maybe warming the solder paste up will help is spread more evenly.

## 2024-03-13 – Antennas

I have a box of various antennas. However, they are not labeled and I don’t have the part numbers nor the specs. Some of them might be 2.4GHz, 868, 915. I really can’t tell. So, I ordered some and will keep the part number with the antenna so we know for sure which one is which. Digikey has 15,000 antennas! Filtering that down to SMA in the 900/868MHz bands with an SMA connector and in stock gets us down to 43. I ordered the following:

|  |  |  |  |
| --- | --- | --- | --- |
| Part Number | Price  @1k | Len mm | Comments |
| [**TI.92.2113**](https://www.digikey.com/en/products/detail/taoglas-limited/TI-92-2113/11197416) | $6.20 | 198 | Looks like the one shipped with Silabs DevKits |
| [**TS.89.4113**](https://www.digikey.com/en/products/detail/taoglas-limited/TS-89-4113/15284364) | $40.56 | 118 | Directional antenna with 3.5/2.7dBi gain in 1 direction |
| [**ANT-8/9-MMG1-SMA-1**](https://www.digikey.com/en/products/detail/te-connectivity-linx/ANT-8-9-MMG1-SMA-1/16592298) | $5.66 | 83 | Cable to a mount |
| [**FW.95.B.SMA.M**](https://www.digikey.com/en/products/detail/taoglas-limited/FW-95-B-SMA-M/6362786) | $9.13 | 226 | Thin fixed |
| [**DELTA12C/X/SMAM/S/S/17**](https://www.digikey.com/en/products/detail/siretta-ltd/DELTA12C-X-SMAM-S-S-17/6096292) | $9.65 | 130 | Classic Rubber Ducky |
| [**JCG402LR-2**](https://www.digikey.com/en/products/detail/jc-antenna/JCG402LR-2/15814458) | $2.42 | 110 | Tiltable – Low cost! |
| [**2600130083**](https://www.digikey.com/en/products/detail/w%C3%BCrth-elektronik/2600130083/11680415) | $14.88 | 179 | Covers 868 thru 928 |
| 868 antennas: |  |  |  |
| [**ANT-868-CW-HW-SMA**](https://www.digikey.com/en/products/detail/te-connectivity-linx/ANT-868-CW-HW-SMA/5592340) | $8.74 | 135 |  |
| [**FW.86.B.SMA.M**](https://www.digikey.com/en/products/detail/taoglas-limited/FW-86-B-SMA-M/6362785) | $9.13 | 240 | Thin fixed (similar to the 915 one above) |
| [**FLEXI-SMA-868**](https://www.digikey.com/en/products/detail/rf-solutions/FLEXI-SMA-868/5845738) | $5.12 | 73 | Short Rubber Ducky |

## 2024-03-03 – Component selection & ordering

Exported the BOM out of KiCAD and used it to manually create the BOM. Ordered enough components for 10+ boards. I already have 15 ZG23s from previous projects and samples. Entered an Issue on github that a couple of the caps are 0603 but should be 0402. The 0402 will still fit but better to have the proper footprint. I expect to spin the PCB at least once more.

The plan is to take one of the 3 boards coming in now and cut it to a much smaller size and compare the range. No sense in making the board bigger than it has to be but we are looking for maximum range. If the full size board is at least 3 dBm better than definitely keep the full size board.

KiCAD has been upgraded to version 8.0.0 which came out a week ago. I had to upgrade to checkin the ZG23 component as all libraries were upgraded to V8. The ZG23 will now be included in the KiCAD default libraries. My PR [4289](https://gitlab.com/kicad/libraries/kicad-symbols/-/merge_requests/4289) has already been merged in. I assume it’ll be included in the next tagged release.



## 2024-02-23 – PCB release to fab

There are 3 excluded DRC violations:

1. The Y2 Crystal footprint was slightly modified to clear the silkscreen error in the footprint
2. The 2 Fiducial marks are not in the schematic

These “warnings” can be ignored.

The PCB is 120mm square which I am certain is overkill. But we can cut a board to be smaller and see the net impact. Might also make a different proto board with a smaller PCB. The corners have 10mm radiuses to make a clean point for the enclosure to latch onto the PCB and make it easier to separate the boards when built as arrays.

Three PCBs and a stencil ordered. Expected in about a week. These 3 boards are just for initial testing to make sure the design doesn’t have some major problem like an incorrect pinout. One board will be for tuning the antenna(s), one is a proto and I expect to cut one to be much smaller to see if the large ground plane makes a difference. Might also want to make a 2 layer PCB version which is cheaper. I’ll order components on Monday.

## 2024-02-22 – PCB layout

DRC clean but in need of cleanup and more documentation.

## 2024-02-21 – Schematic capture

Schematic capture completed and ERC is clean.

Footprints to be picked out:

* USB-C receptacle – GCT [USB4110](https://www.digikey.com/en/products/detail/gct/USB4110-GF-A/10384547)-GF-A is SMT with 2 small mounting pins less than a buck
* [39MHz](https://abracon.com/Resonators/abm8.pdf) crystal – Crystal:Crystal\_SMD\_3225-4Pin\_3.2x2.5mm
* 32KHz crystal – Crystal:Crystal\_SMD\_3215-2Pin\_3.2x1.5mm - SC32S-7PF20PPM
* RGB LED – LED\_SMD:LED\_WS2812B-2020\_PLCC4\_2.0x2.0mm - [QLSP14RGB\_B](https://www.digikey.com/en/products/detail/quelighting-corp/QLSP14RGB-B/15848703)
  + Might need to make a custom footprint
* The RF RCs are 0402 which are cheap and still OK to hand solder. 0201s are tough to handle.
* Other RCs are 0603 and large caps are 0805 which makes them easy to handle and cheap.

PCB board created and a few components placed. Need to adjust the DRC rules to match [OSH Park](https://docs.oshpark.com/design-tools/kicad/kicad-design-rules/).

## 2024-02-20 – Schematic capture

Connected the USB UART to the same pins as the debug header but thru 100 ohm resistors. I could have put the USB UART to a different set of pins but then the firmware could NOT use the standard SerialAPI and would have to be customized with each release. Didn’t think that was a good idea. The series resistors are to provide some isolation when BOTH USB and the debugger are connected. The main issue is that during production, RailTest has to be downloaded and then commanded via the debug connector to the UART to Tx ON for crystal calibration. But since USB is also connected, it is also driving the UART pins. Simply adding some series resistors will provide sufficient isolation for the short time during production. Even during debug it’s not a big deal. During normal operation the resistors will help with EMI. UART speed is only 115.2K baud so the resistors will have no impact on that. The other option is to use a switch but that would be overkill and just 1 more place for things to fail and increase cost.

USB pins require ESD protection as described here: <https://www.digikey.com/en/articles/why-usb-type-c-circuit-protection-is-vital> but this is overkill IMHO. There are many triple TVS diodes but they do not seem to have a common pinout. Prices for these triple diodes are around 30 cents. Only need USB2 level ESD as we do not need to support USB3 which has tighter restrictions on the capacitance of the TVS diode. This [article](https://www.electronicproducts.com/how-to-protect-usb-type-c-connectors-from-esd-and-overtemperature/) has a good description of requirements. Want to have something with lots of 2nd sources and not be locked into a single part. KiCAD doesn’t have any diodes in 0402, mostly SOD-xxx. The CP2102 recommends SP0503BAHTG which is in the KiCAD library and digikey has 114K of them so it is easier to use this part.

The Tag-Connect debug header VAEM is tied directly to the 3.3V from the LDO. This is normally not a problem but could result in both the WSTK and the LDO driving the 3.3V pins. Since they are both at the same voltage it usually doesn’t matter. The WSTK should normally NOT drive the VAEM pin (slide the switch to USB or battery). The only time you would drive the VAEM pin is if you are debugging a battery powered device and want to measure the power which would be done using the AEM in SSv5. But in that case, the LDO would not be installed. The only question is during programming will the LDO short 3V3 to GND? That would require the USB to also be plugged in or powered via a test point which is acceptable but makes things more difficult. However, a functional test of the USB would require the cable being plugged in so maybe it is OK.

There is no QWIIC connector symbol or footprint in KiCad. I copied the ones from the sparkfun repo but I didn’t want to import the entire thing so I created project specific libraries which will also hold some other project specific stuff like the Z-Wave logo and things.

## 2024-02-16 – Project Start

Wrote the datasheet. Started Schematic capture and Theory of Operation (ToO) development.

Lots of LDOs to choose from. Diodes Inc [**AP2125K-3.3TRG1**](https://www.digikey.com/en/products/detail/diodes-incorporated/AP2125K-3-3TRG1/4470777) is 10 cents and will be the initial LDO to use (Digikey has 45K in stock). The TI [**TPS7A0333PDBVR**](https://www.digikey.com/en/products/detail/texas-instruments/TPS7A0333PDBVR/12165108) LDO I’ve used before is 33 cents but does have better Iq, temperature range and Vdropmax but the Diodes Inc is fine.

For the USB-C interface, I used this example: <https://hackaday.com/2023/08/07/all-about-usb-c-example-circuits/>. The classic question is what to connect the shield of the connector to? The Shield is the outside layer of the cable thus it can act as an unintentional antenna and then fail FCC. The example wires it to GND but that is likely to radiate more than leaving it unconnected. Nice discussion [here](https://electrical.codidact.com/posts/279876#:~:text=%E2%88%920-,Connect%20the%20shield%20directly%20to%20ground%20plane,to%20the%20PCB%20ground%20plane.) which mostly recommends connecting SHIELD to GND. There are 2 purposes for this connection – EMI and ESD. For ESD it makes sense to have a solid connection to have the entire board remain at the same potential which might shoot up thousands of volts in an ESD event. But I am more concerned about EMI which in an abundance of caution I put a small ferrite bead leading to the shield to reduce any EMI. Can replace this with a zero ohm resistor later, or leave it open or insert a capacitor or even just short the pads out. I’m not too worried about ESD as ZReach is pretty much plugged in once and left there for the duration.

## 2024-02-07 – Create Repository

Created the Repo, this file and started the hardware design in KiCAD.