1. Main Board 1

Edit   New page

Alan Johnston edited this page on Aug 11, 2023 · 22 revisions

CubeSatSim Main Board Assembly Part 1

In these steps, you will start building your Main board.

To do this, you will need the parts in the BOM https://cubesatsim.org/bom and these tools:

- Safety glasses (to protect eyes while soldering or trimming leads)
- Soldering iron and solder (I use lead-free solder, but leaded solder is easier to work with)
- Needle nose pliers (to bend leads and hold parts)
- Side cutters (to trim leads)

Other tools that are helpful:

- Blue mounting putty (to hold components in place while soldering)

Video

Here is a video showing this assembly step: https://youtu.be/yIlTwG-H96Q

Checklist

The BOM has a sheet "By Steps" which lists the parts needed for each step in order. http://cubesatsim.org/bom If you have a Google account, you can make a copy of this spreadsheet ("File" then "Make a Copy") and check off each part as you install it.

For example, here is the checklist for this step:

|	Ref	Item		Qty	Location	Image
☑		Step 1. Main Board 1				https://github.com/alanbjoh
☐	Ref	Item		Qty	Location	Image
☐		Alt 3a: Main PCB with LPF installed	☑	1		
☐		Alt 3b: Main PCB without LPF	☐	1		
☐	L1	Alt 3b: LPF SMD 13nH 0603		1	Top	
☐	C1	Alt 3b: LPF SMD 18pF 0603		1	Top	
☐	C2	Alt 3b: LPF SMD 16pF 0603		2	Top	
☐	R2	Alt 3b: 1k Ohm resistor		1	Top	
☐	J3	USB0-C breakout		1	Bottom	
☐	S1	Pushbutton switch, DPDT, CFP2-2RC4-AW		1	Bottom	
☐	X1	SC1464-ND 3.50mm Headphone Phone Jack S		1	Bottom	
☐	R11	1k Ohm resistor, 1/8 Watt, 20% tolerance		1	Bottom	
☐	LED1	Green LED 5mm clear lens		1	Bottom	
☐	R13	220 Ohm resistor, 1/8 Watt, 20% tolerance		1	Bottom	
☐	LED3	Red LED 5mm clear lens		1	Bottom	
☐	R12	100 Ohm resistor, 1/8 Watt, 20% tolerance		1	Bottom	
☐	LED2	Blue LED 5mm clear lens		1	Bottom	
☐	J1	GPIO 20x2 female stacking header extra long		1	Bottom	
☐	R3,R4	4.7k Ohm resistors, 1/8 Watt, 20% tolerance		2	Top	

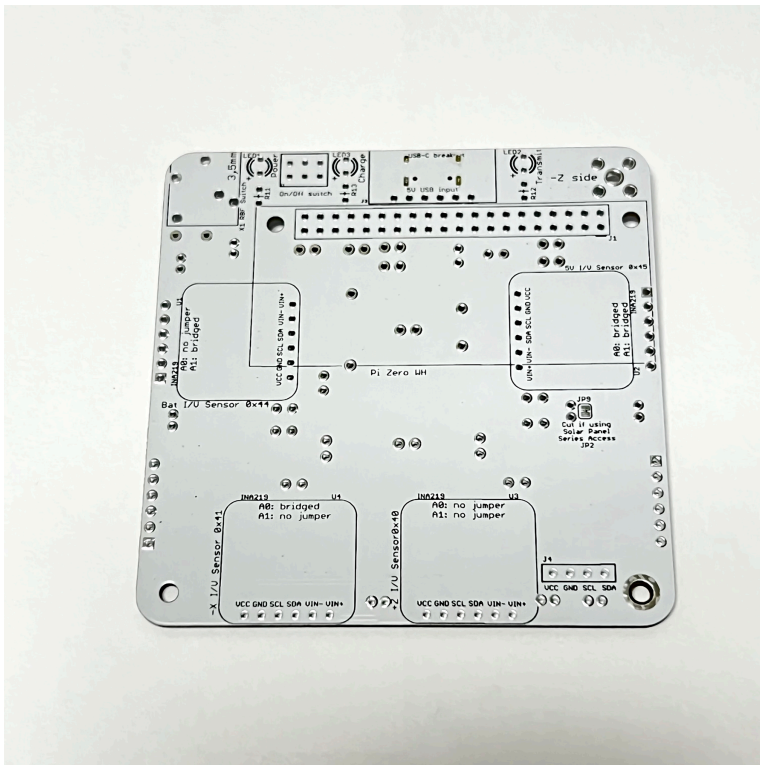# Main Board Instructions

Clone this wiki locally

https://github.com/alanbjohnstc

The PCB has components on both sides. This is the top of the PCB, with the AMSAT logo.
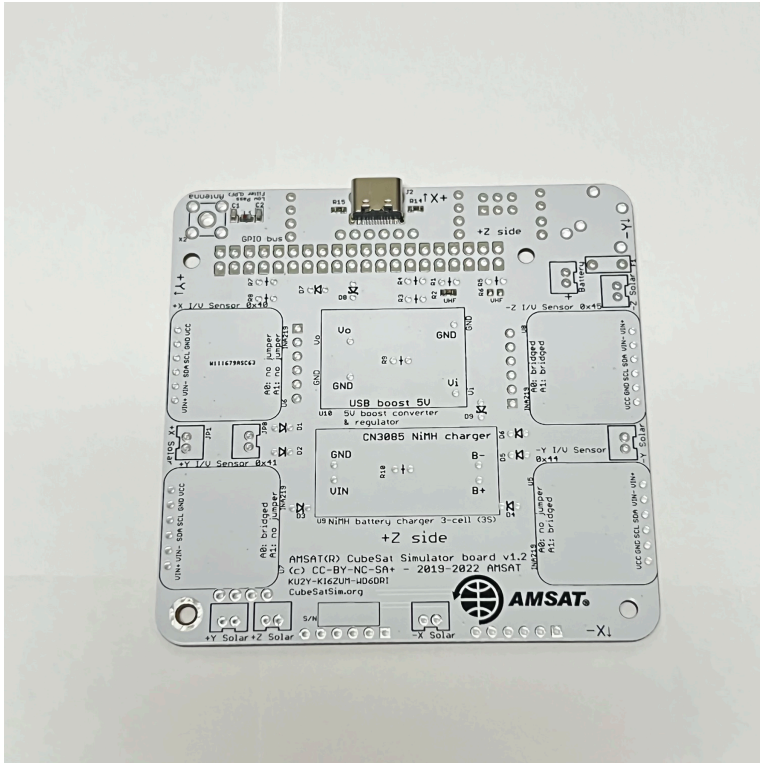
This is the bottom of the PCB.

The silkscreen outline markings indicating which side to mount the parts.

Before any components are soldered onto the board, the Surface Mount Device (SMD) components should be installed, if they aren't already soldered on. C1, C2, and L1 make up the Low Pass Filter (LPF). The purpose of the LPF is to filter out all the undesirable radio frequency (RF) emissions besides the ones at 434.9 MHz UHF (i.e, harmonics, sub-harmonics, and spurs). J2 is the USB-C charging connector, which also has resistors R14 and R15 to set the current limit. The resistor R2 allows the software to detect the presence of the LPF and allow radio transmissions. Here is a closeup of the SMD components after they have been mounted:
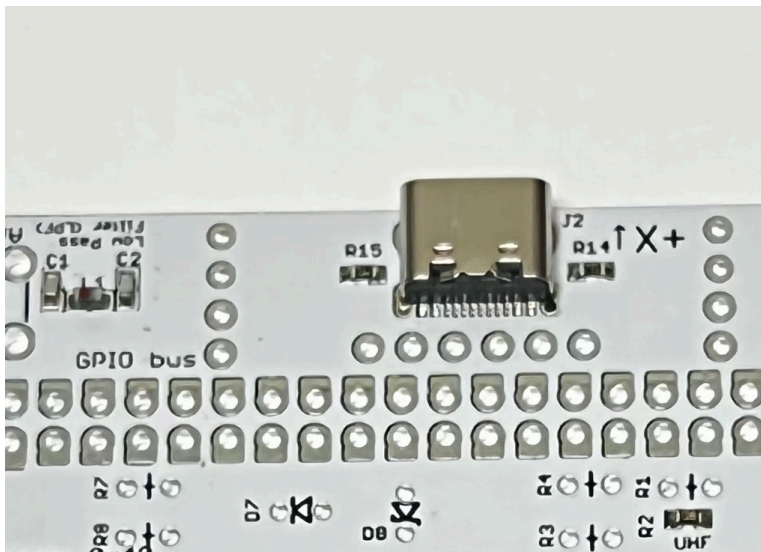
If you don't have the ability to do SMD soldering, you can instead use an external Band-Pass Filter with SMA connectors. For example, this BPF works quite well https://www.amazon.com/gp/product/B07R8Y1PM4/ . For the USB-C connector, you can use J3, the Sparkfun USB-C breakout https://www.sparkfun.com/products/15100. And instead of R2, you can install a 1k resistor in R1.

The instructions for the tape measure monopole or dipole are in the Board Stack instructions.

Here's the schematic for the Main board for reference:
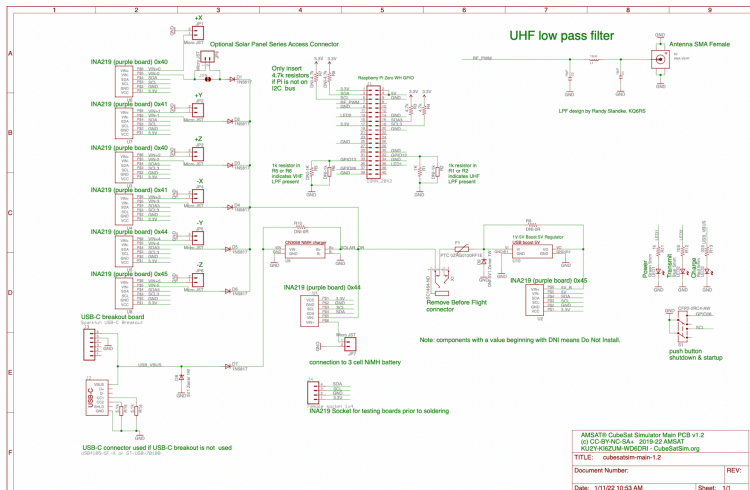
First, we will start soldering the components on the bottom of the PCB. The printed outlines on the board indicate that a part should be installed on that side, with the pins soldered on the other side. For components with multiple pins, solder just one pin first, then flip the board over and verify that it is flush with the board and straight. It is a lot easier to fix this with only one pin soldered than with multiple pins. It is also one last check to make sure you are mounting it on the right side. For some parts, such as resistors or diodes, mounting on the wrong side won't matter. For others, such as LEDs and switches and connectors, mounting on the wrong side will make it more difficult to mount the solar panels. But other parts, such as a the stacking GPIO headers or the INA219 boards, must be mounted on the correct side, so double check carefully.

This photo shows the set of parts to be mounted in this step:
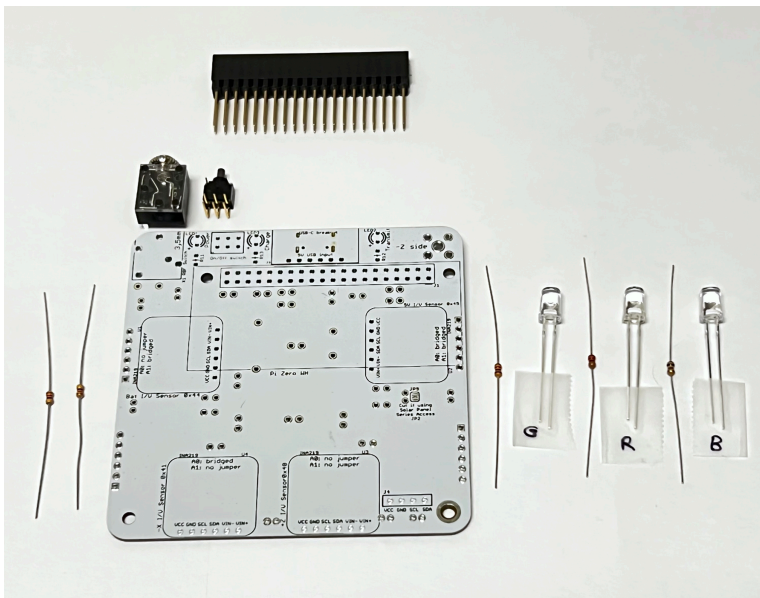
Start with the push button switch S1. This momentary push button switch is used to power up the CubeSatSim when it is off, or to change modes or shutdown when it is running. Note that to allow the button to be pressed when mounted in the frame with solar panels, the switch hangs over the edge. An extra mounting clip doesn't touch the board and can be removed, as shown here:
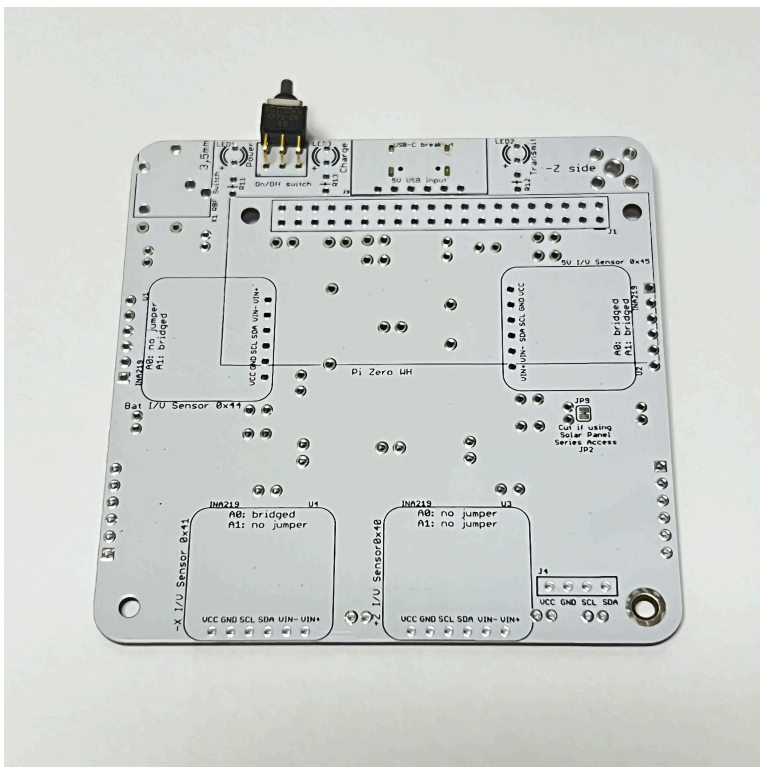
The switch is inserted in the bottom of the PCB (in the rectangle outline) and then soldered on the top of the PCB:
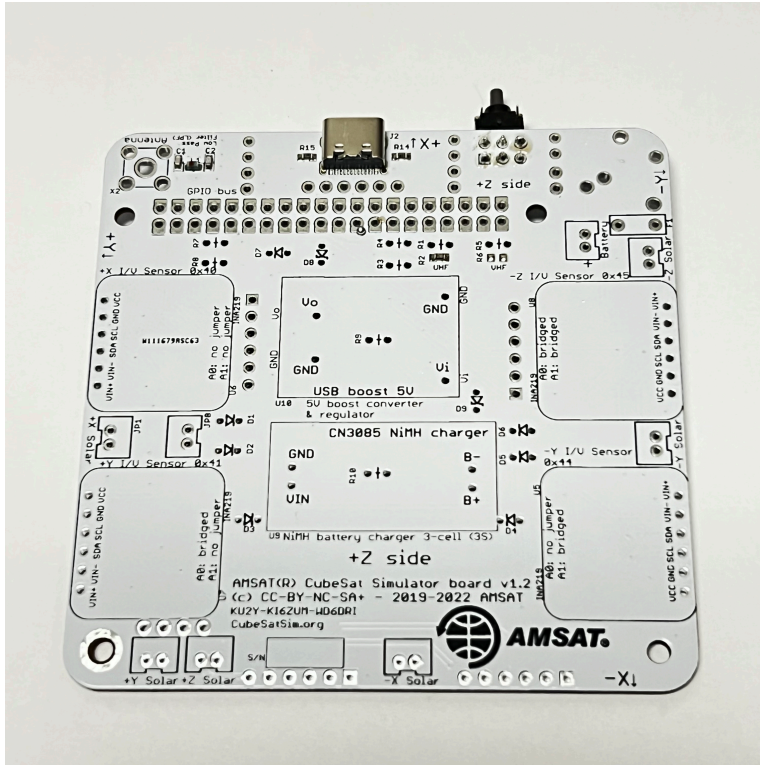
Then, solder X1 the 3.5mm audio jack, which has a contact switch (you can see the contacts with the springs if you look through the clear cover) for our Remove Before Flight (RBF) switch. This switch disconnects 5V power from the Pi forcing the CubeSatSim into a powered down mode for transporting or storage, or, on a real CubeSat, prior to launch.
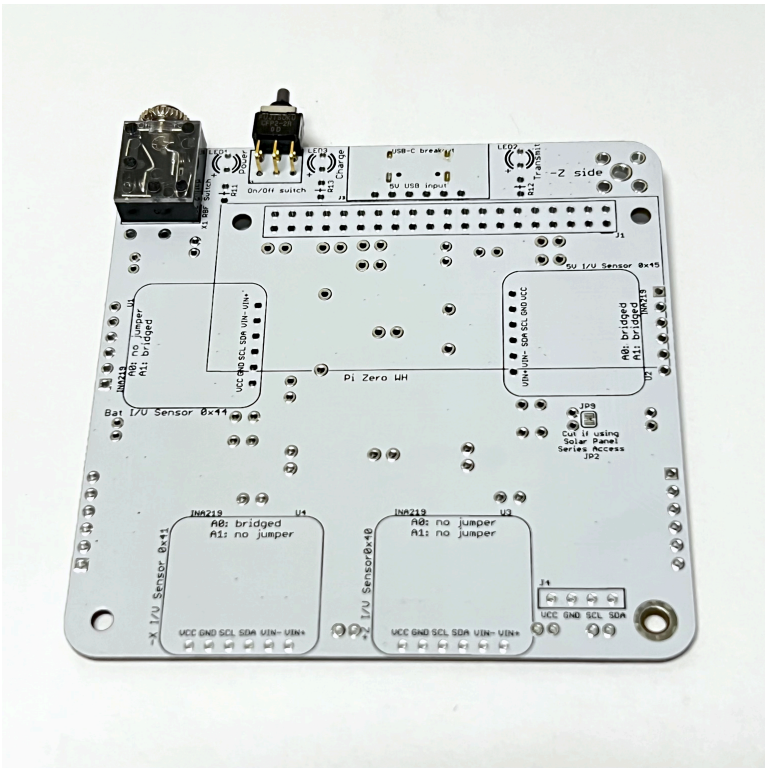
Here's how it looks on the top of the PCB with the two components mounted before cleaning the flux:
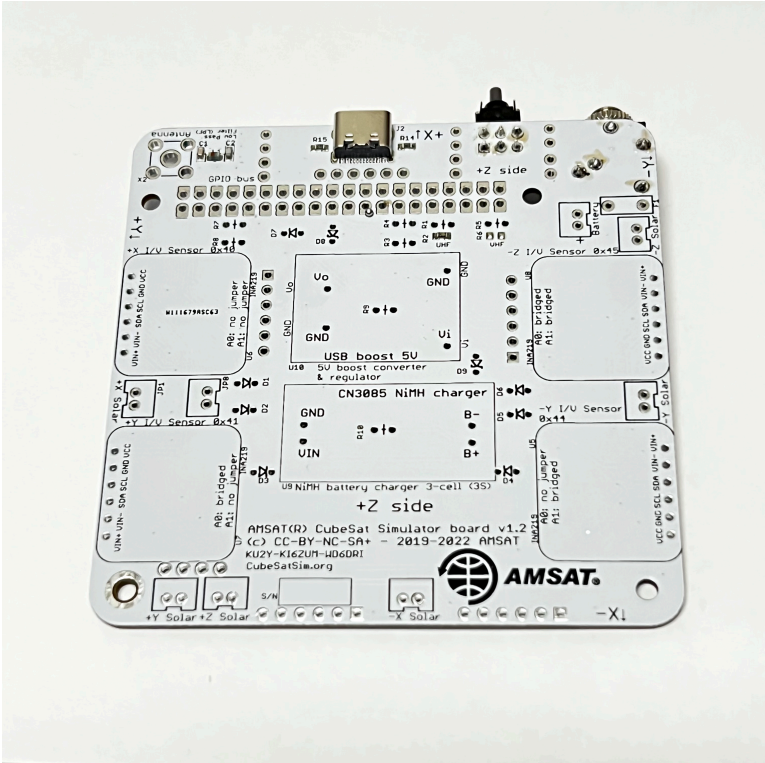
Next we will solder the resistors R11, R12, and R13, and the three LEDs LED1, LED2, and LED3. If you are using clear lens LEDs, be careful not to mix up the colors! If you are not sure, test them with 3.3V and resistor R11 to see what color it is. LEDs need to b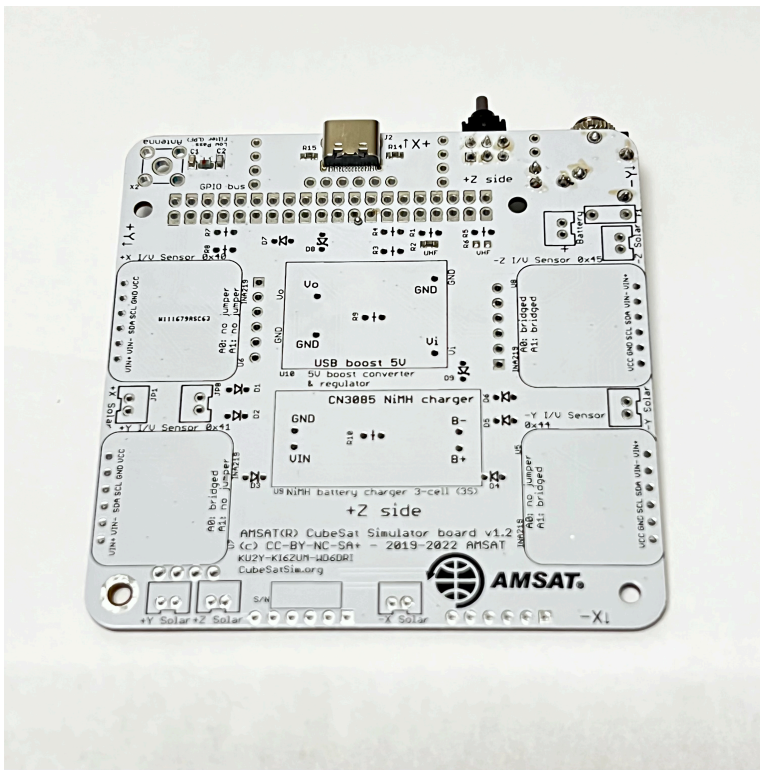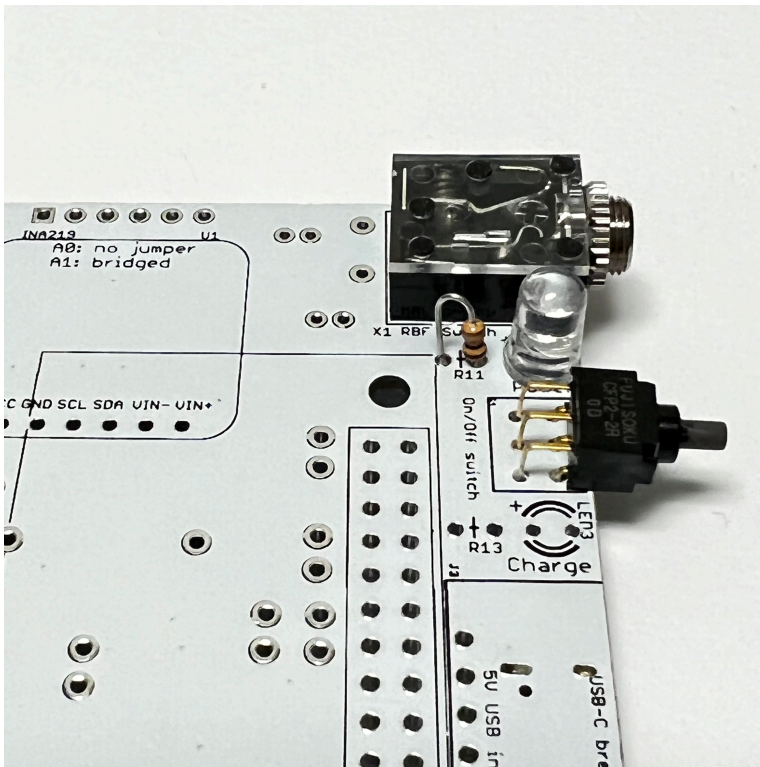e installed with the correct polarity (one lead is + polarity and the other lead is - polarity) or it will not illuminate. The longer leg on the LED is the '+' lead and should be away from the edge of the PCB. Also, if you look at the LED lens from the top, it is circular but there is a flat side that marks the '-' side. So the flat side of the LED should be towards the edge of the PCB.

To save space on the PCB, all resistors and diodes are mounted vertically, so you have to bend over one of the leads to get a U-shape, allowing it to be inserted into the two closely-spaced holes. This photo shows the green Power LED1 and 1k Ohm resistor R11 (the color bands on the resistor are brown black red - see https://en.wikipedia.org/wiki/Electronic_color_code for details) which are about to be inserted next to the RBF switch. Note the longer plus lead of the LED is on the left here:

Here's how the resistor looks when it is installed on the bottom of the PCB:

After you have soldered the resistor is then soldered on the top of the PCB and, the excess leads trimmed with your side cutters.

The green LED is soldered next to it:

Next, install the red Charge LED3 and 220 Ohm resistor R13 (color bands red red brown),

Then install the blue Transmit LED2 and 100 Ohm resistor R12 (color bands brown black brown):


Here's how the PCB looks on top:
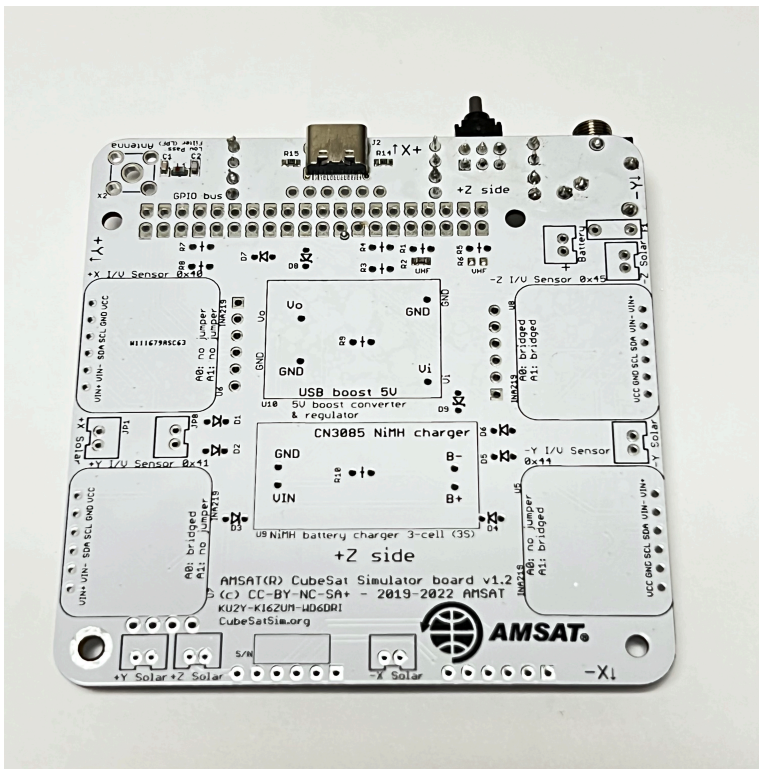
The final step is to install the stacking GPIO header J1. We use the Raspberry Pi GPIO as our spacecraft bus, and it connects the Pi and the three boards together. We will insert the stacking GPIO header on the bottom of the PCB and solder the pins on the top of the PCB.

IMPORTANT: Do not trim the leads on this stacking header, since the long leads are needed to plug the next board on top! It is also important that you don't get excess solder on the top of the GPIO pins or it might be hard to plug then next board in.

Here's how the stacking GPIO header looks on the top of the PCB ready for soldering. To make it easier solder, the PCB has enlarged pads making it easier to apply heat.
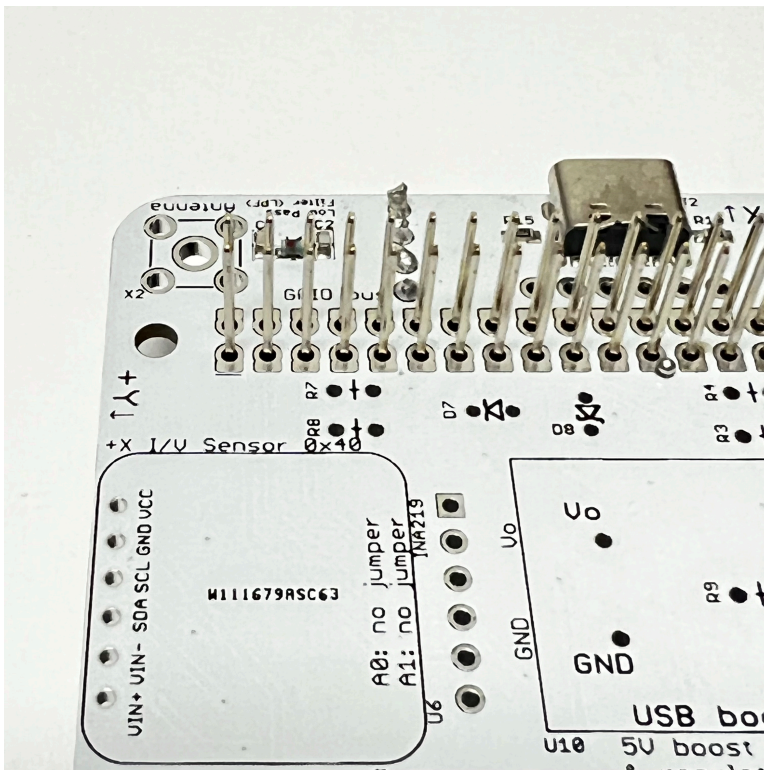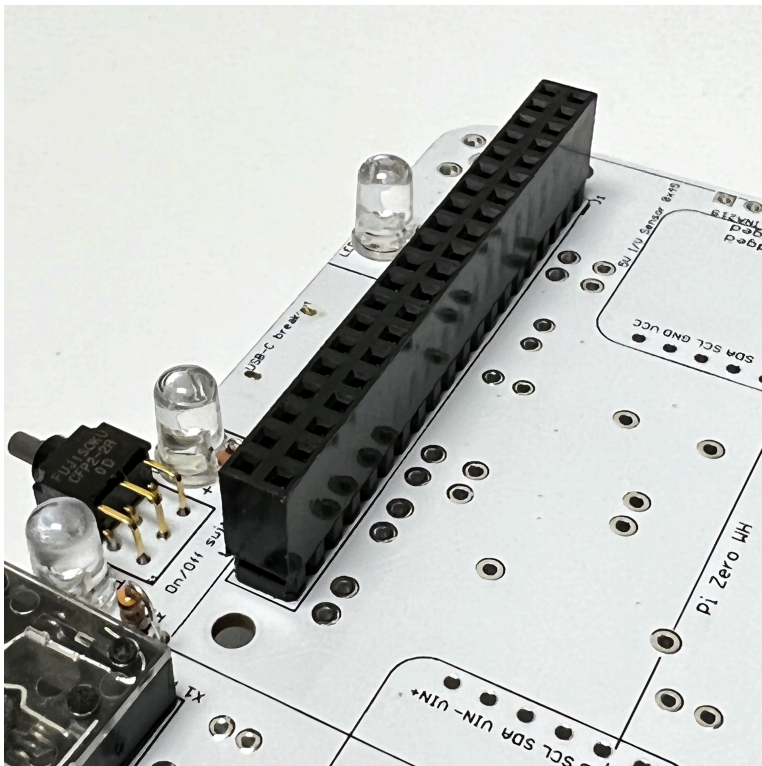
If you place your soldering iron on this pad touching the pin, you should be able to get heat to both the pin and pad to solder it.
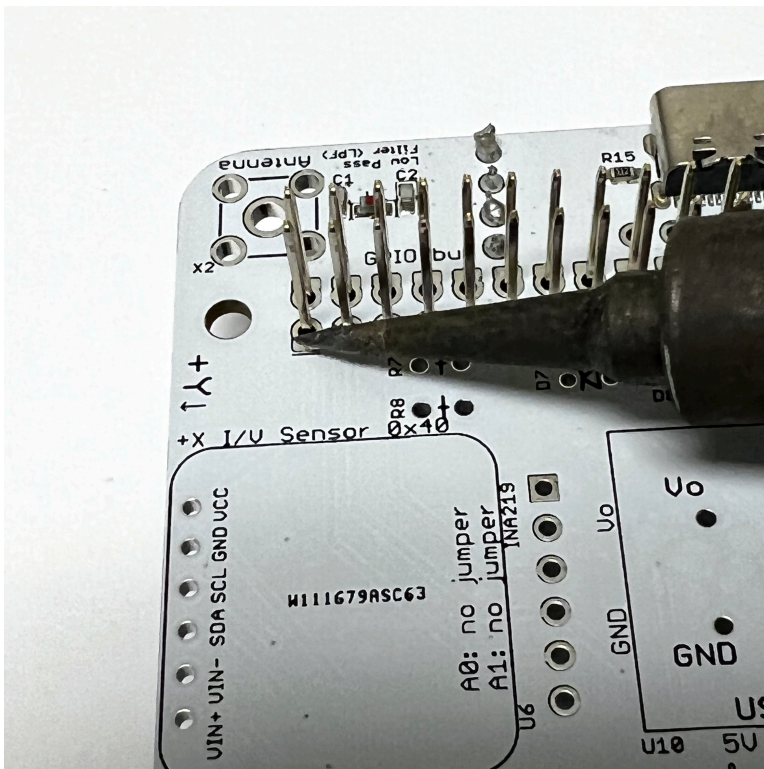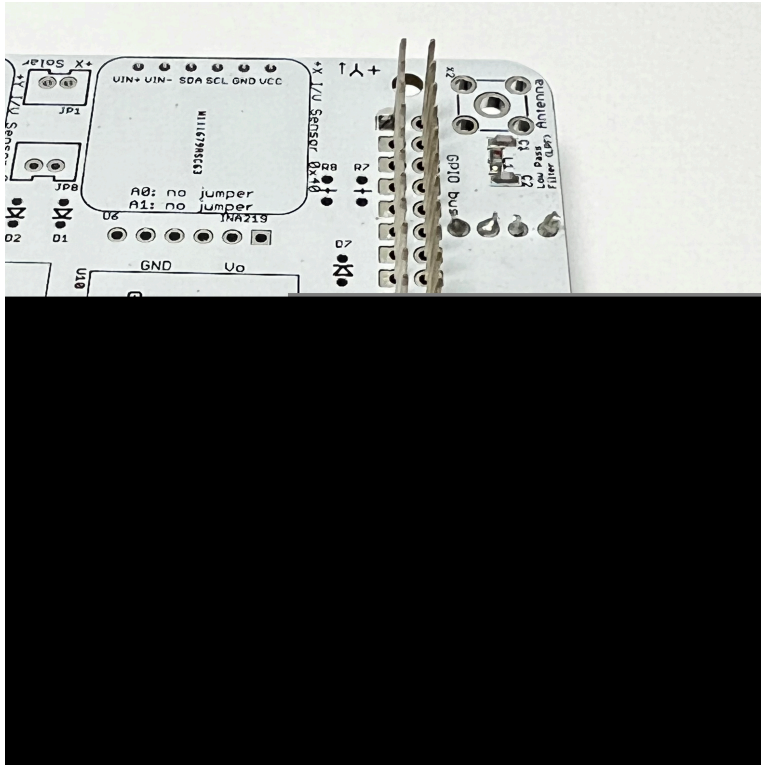
Solder only one pin on the corner of the connector, then double check that it is pressed down all the way and straight before going back and soldering any other pins:
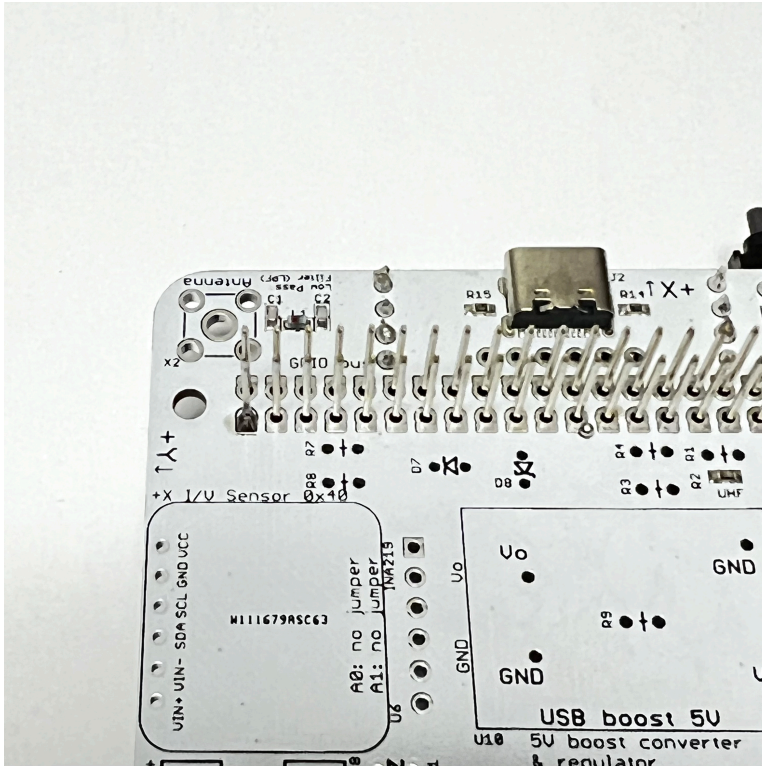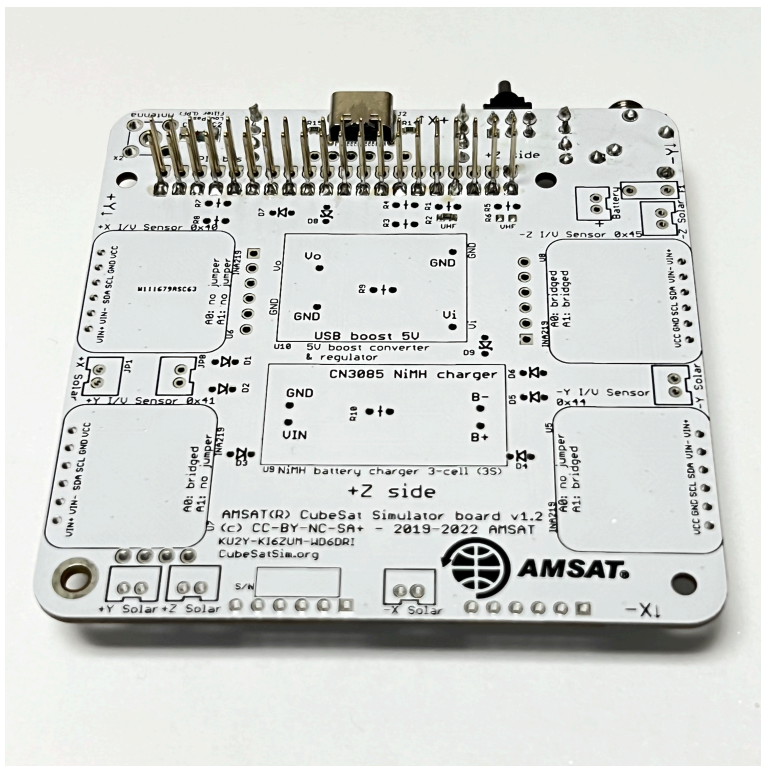

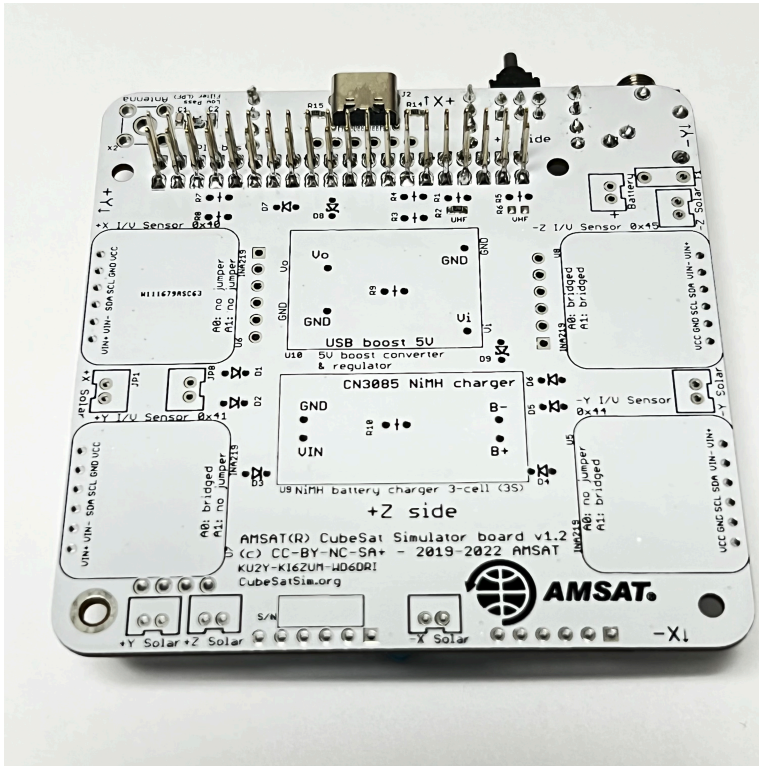Here's how it should look on the bottom of the PCB:

If everything is good, solder one pin on the other corner and check to make sure it is straight and flush with the PCB:
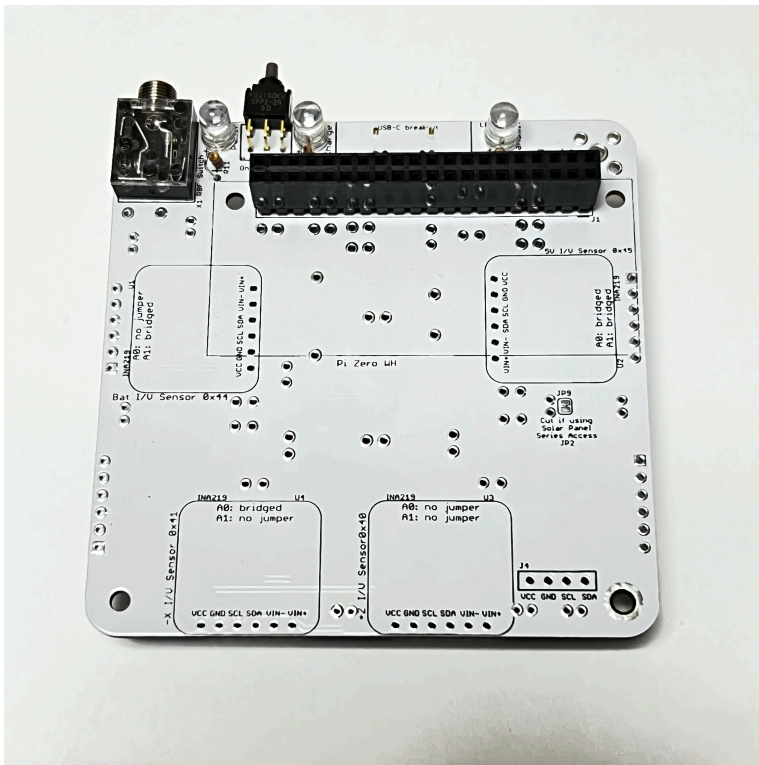

If it looks good, go ahead and solder one row of pins. Here's how it looks on the top of the PCB with one row soldered in:

Then solder the 2nd row:

Here's how it looks on the bottom of the PCB, ready to plug the Pi Zero WH into the female connector:

Next we install the 4.7k resistors R3 and R4 (color bands yellow violet red). They are pull-up resistors on the second I2C bus (pronounced "I squared C bus" which stands for Inter-Integrated Circuit bus) on the Raspberry Pi. Two of these buses are used by the Pi to read the INA219 current and voltage sensor boards we will install in the next section.

R3 and R4 are the first components we mount on the top of the PCB, so you will solder the leads on the bottom of the PCB. Like all the resistors on the board, they are installed vertically:
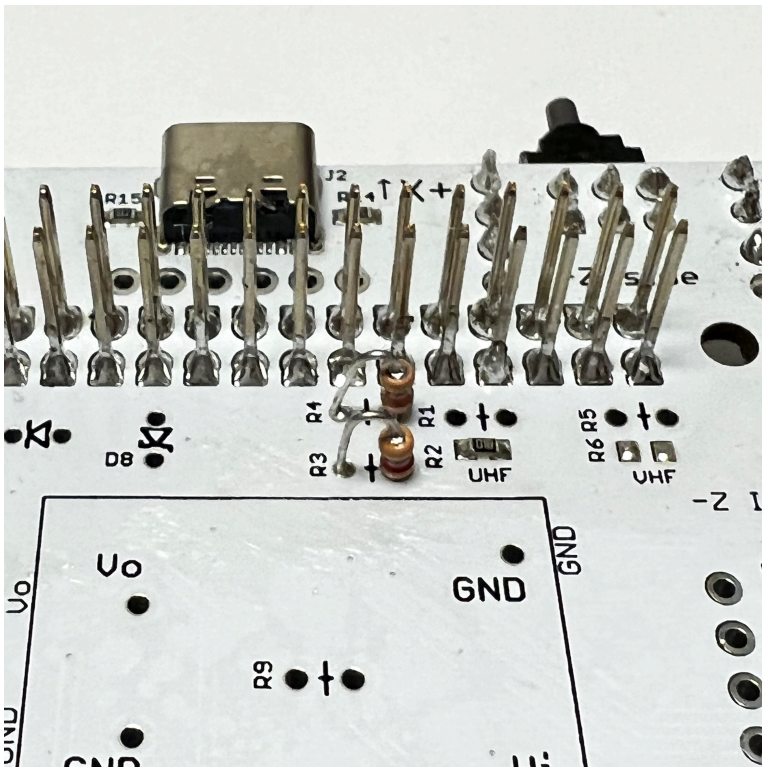
Here is the completed Main board part 1 assembly:
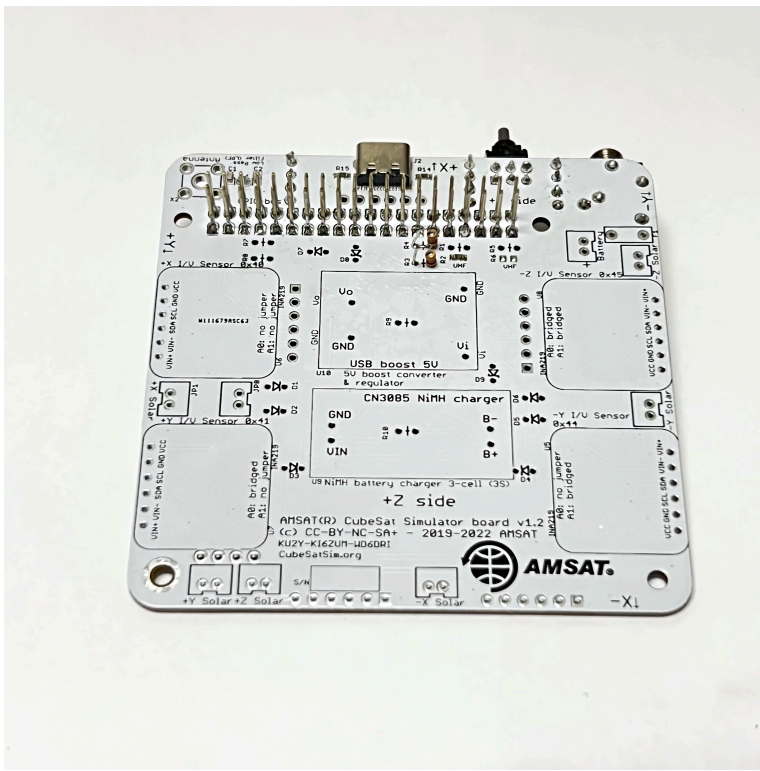
Note that if your PCB had the SMD Band–Pass Filter components installed, there will be a 1k SMD resistor in R2 (labeled "102" for 10 * 10^2 or 1000 Ohms) as shown here:
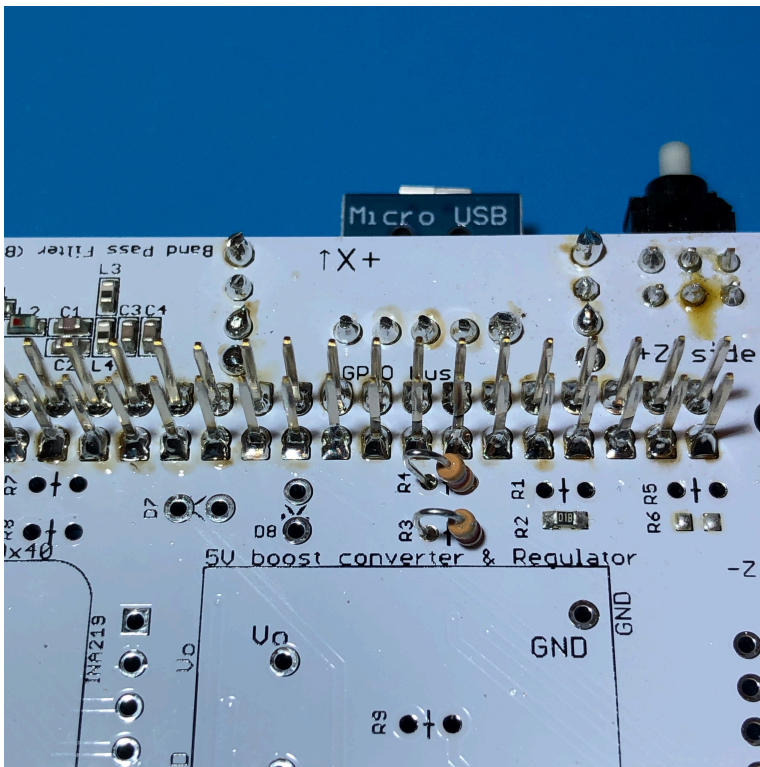
If there is no resistor in R2, you should install a 1k resistor (color band brown black red) in R1 next to it. This resistor is connected to a GPIO input on the Raspberry Pi and is used to sense if there is a Band-Pass Filter installed. If the filter isn't installed, the software will only transmit the CW callsign and will not send telemetry, as the signal might interfere with other stations. R1 or R2 are used to indicate the usual UHF Band-Pass Filter. If you installed a VHF Band-Pass Filter, you would install a resistor in R5 or R6.

You have completed Part 1 of building the Main board.

Next step is to [Install the Software](#).

+ Add a custom footer