

# v1.3.2 6. Solar Board

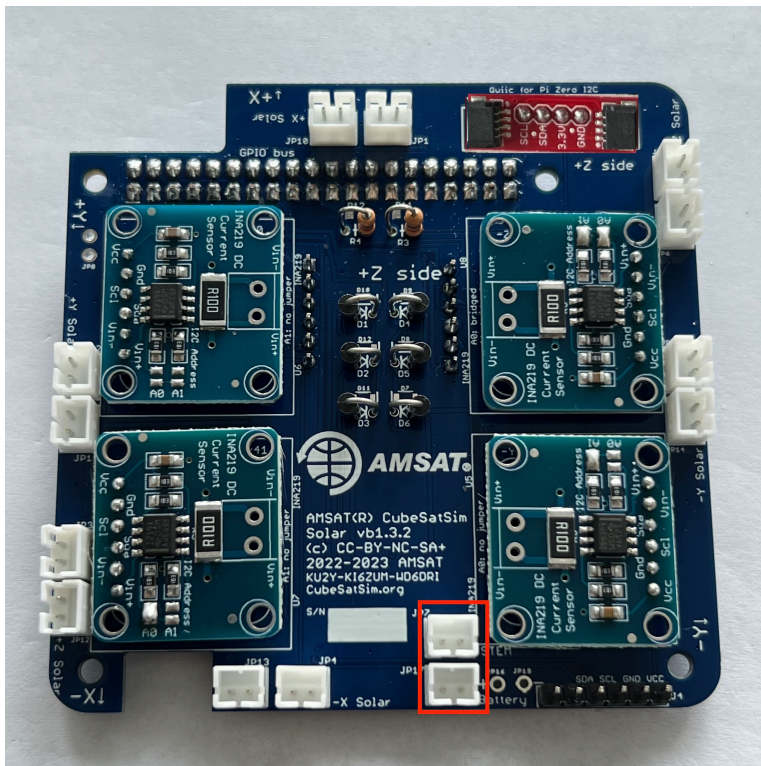
Edit New page

Alan Johnston edited this page 1 minute ago · 23 revisions

If the images on this page fail to load, you can [download a PDF of the page here.](#)

## 6. Solar Board

These steps are to build the CubeSatSim Solar PCB:



Here is the Solar PCB v1.3.2:

▼ Pages 129

Find a page...

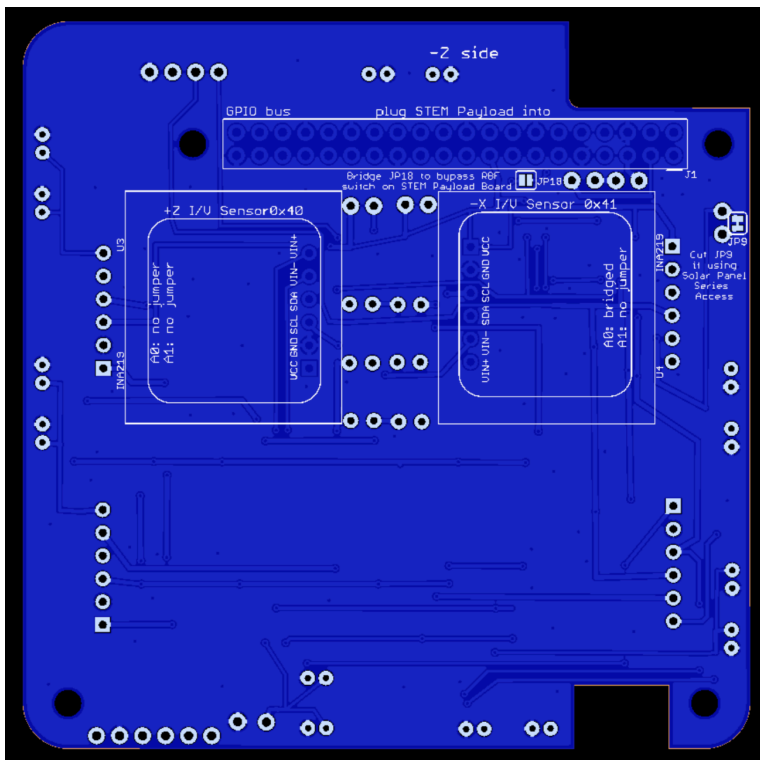
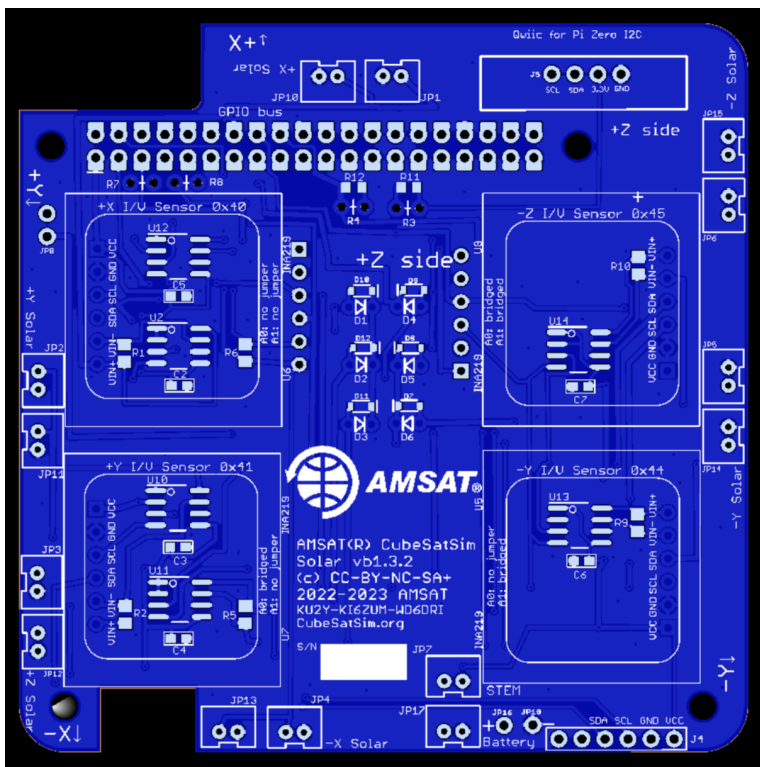
- ▶ Home
- ▶ 1. Main Board 1
- ▶ 2. Software Install
- ▶ 3. Ground Station
- ▶ 4. Main Board 2
- ▶ 5. Battery Board
- ▶ 6. Solar Panel and Frame
- ▶ 7. Main Board 3
- ▶ 8. STEM Payload Board
- ▶ 9. Board Stack
- ▶ 9. Index
- ▶ Creating the CubeSatSim Raspberr...
- ▶ CubeSatSim Lite
- ▶ CubeSatSim Loaner User Guide
- ▶ CubeSatSim Mini

Show 114 more pages...

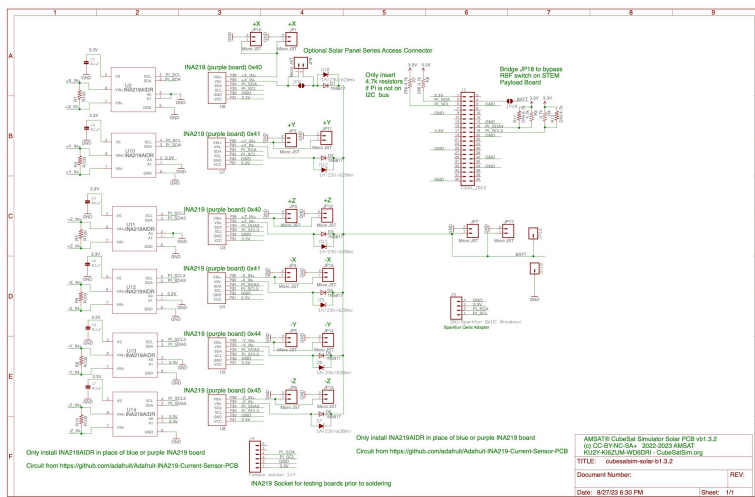
+ Add a custom sidebar

Clone this wiki locally

<https://github.com/alanbjohnst>



Here is the schematic:



[https://github.com/alanjohnston/CubeSatSim/blob/beta-v1.3.1/hardware/beta-v1.3.2/cubesatsim-solar-b1.3.2\\_schematic.pdf](https://github.com/alanjohnston/CubeSatSim/blob/beta-v1.3.1/hardware/beta-v1.3.2/cubesatsim-solar-b1.3.2_schematic.pdf)

You will need 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)
- glue, such as super glue gel to attach the JST connectors to the PCB

Other tools that are helpful:

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











## Checklist

The BOM has a sheet "By Steps" which lists the parts needed for each step in order.

[https://docs.google.com/spreadsheets/d/1Ta5UaJcinGozcheROrkfwXdGSDUZrXvQ1\\_nbIBdIIOY/edit?usp=sharing](https://docs.google.com/spreadsheets/d/1Ta5UaJcinGozcheROrkfwXdGSDUZrXvQ1_nbIBdIIOY/edit?usp=sharing)

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:

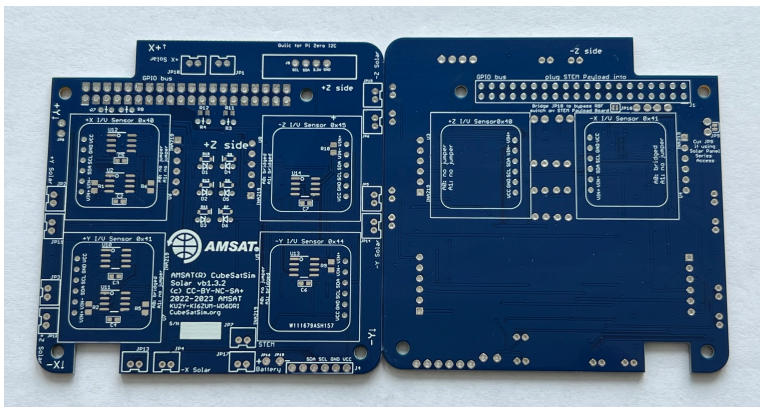
<input type="checkbox"/>	Step 6. Solar Board		<a href="https://github.com/alanbjc">https://github.com/alanbjc</a>		
<input checked="" type="checkbox"/>	Ref	Item	Qty	Location	Image
<input type="checkbox"/>		Solar Board PCB	1		
<input type="checkbox"/>	D1-D7	1N5817 diode	6	Top	
<input type="checkbox"/>	R3, R4	4.7k resistors	2	Top	
<input type="checkbox"/>	J1	20x2 female GPIO header non-stacking	1	Bottom	
<input type="checkbox"/>	U3, U4	Blue INA219 High Side DC Current Sensor Br	2	Bottom	
<input type="checkbox"/>	U5, U6	Blue INA219 High Side DC Current Sensor Br	4	Top	
<input type="checkbox"/>	JP1-JP3	Micro JST 2 pin connectors	14	Top	
<input type="checkbox"/>		QWICC connector	1	Top	
<input type="checkbox"/>		1x4 male breakaway header	1	Top	
<input type="checkbox"/>		JST jumper cable	1	Top	

## Video

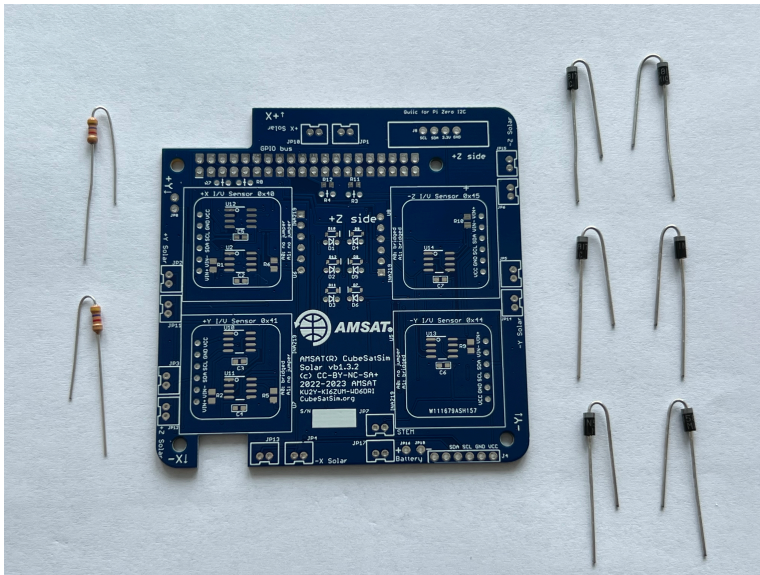
Here is a [video of this step](#).

## Steps

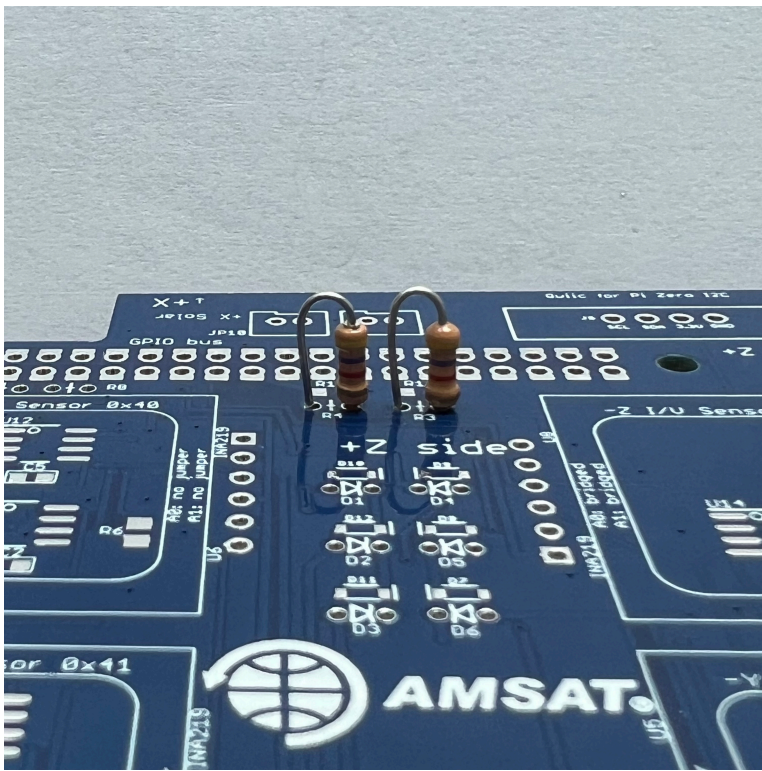
Start with the blank PCB, shown here top on left with AMSAT logo, bottom on the left:



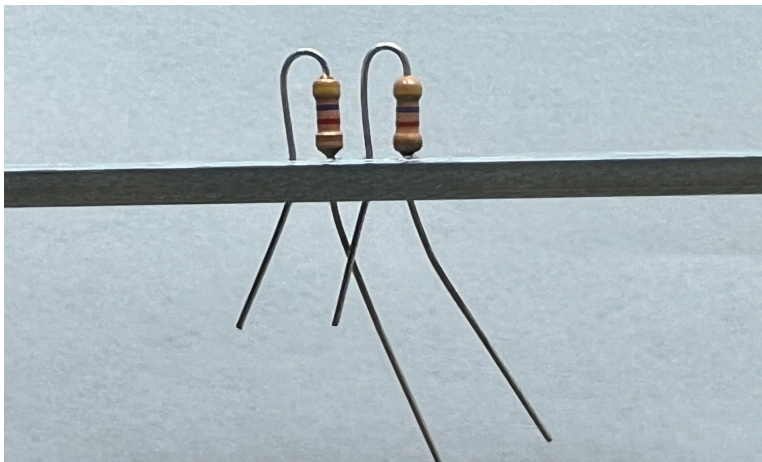
The 1N5817 diodes D1 - D6 and 4.7k Ohm resistors R3 and R4 are installed vertically:



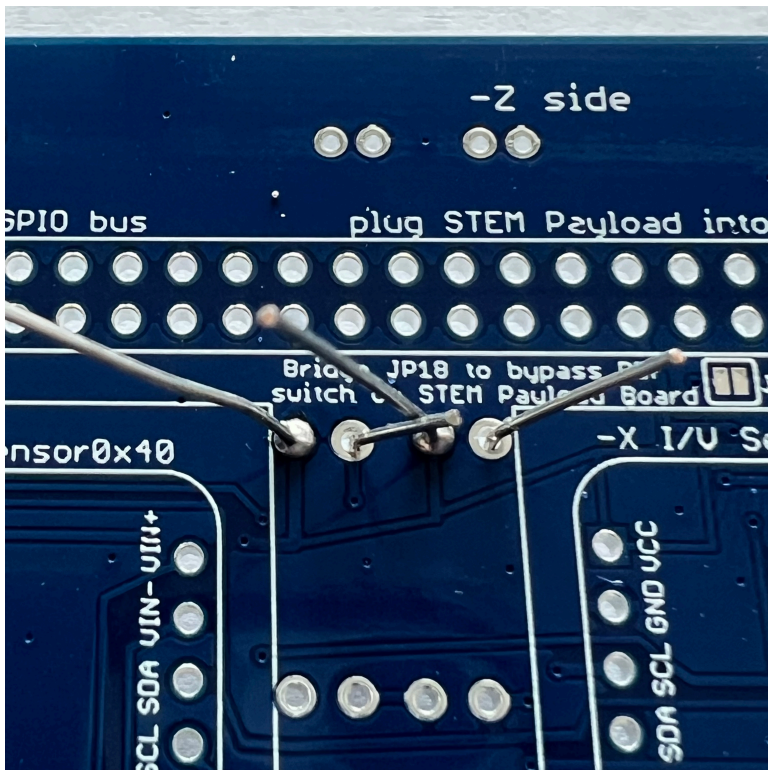
First insert the 4.7k Ohm resistors R3 and R4 (color bands yellow blue red):



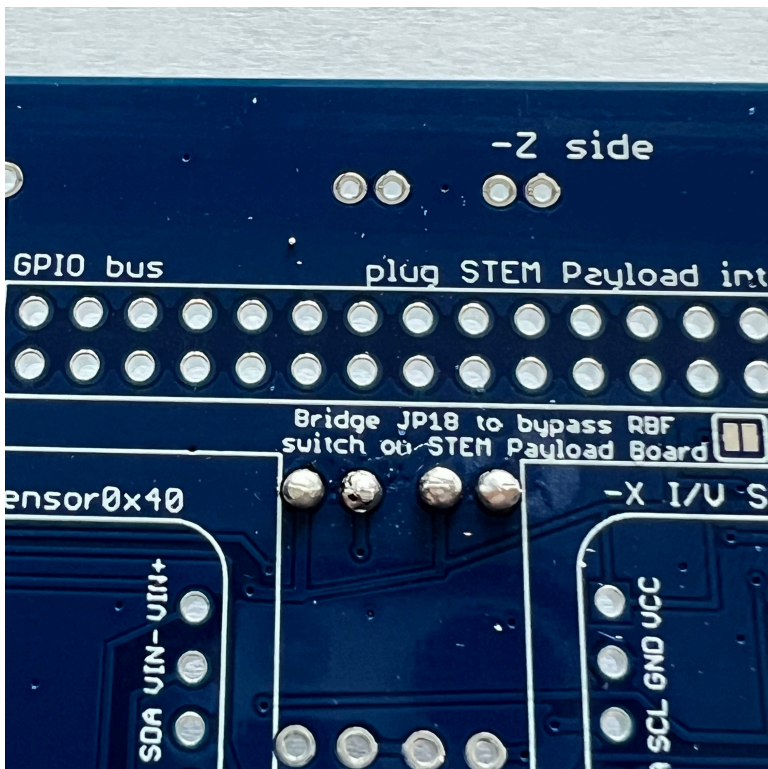
Bend the leads on the bottom of the PCB so they are held in place:



Solder one pin on each resistor then check the top of the PCB that the resistors are straight and inserted all the way:

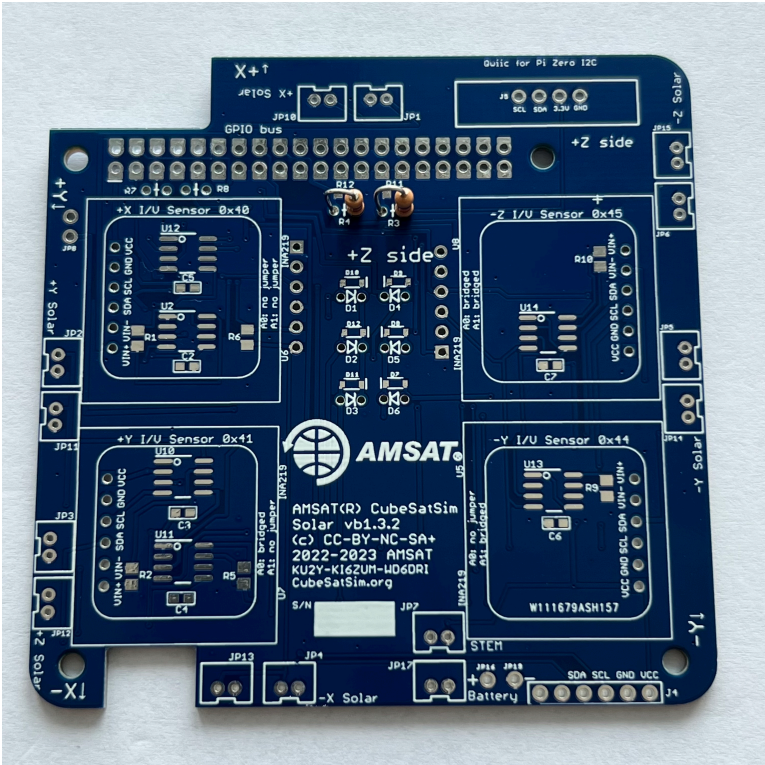


Then solder the other pin, the trim the leads:



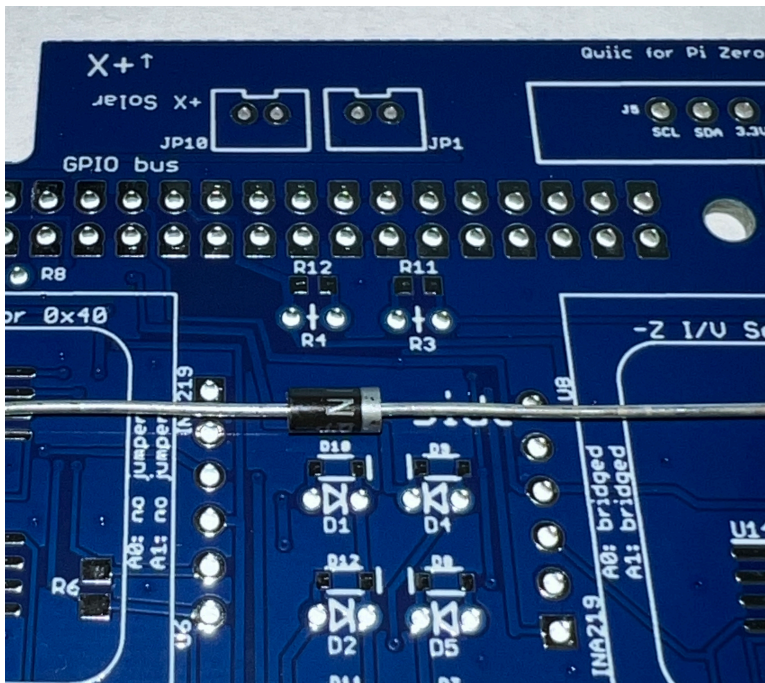


Here's how the board looks:

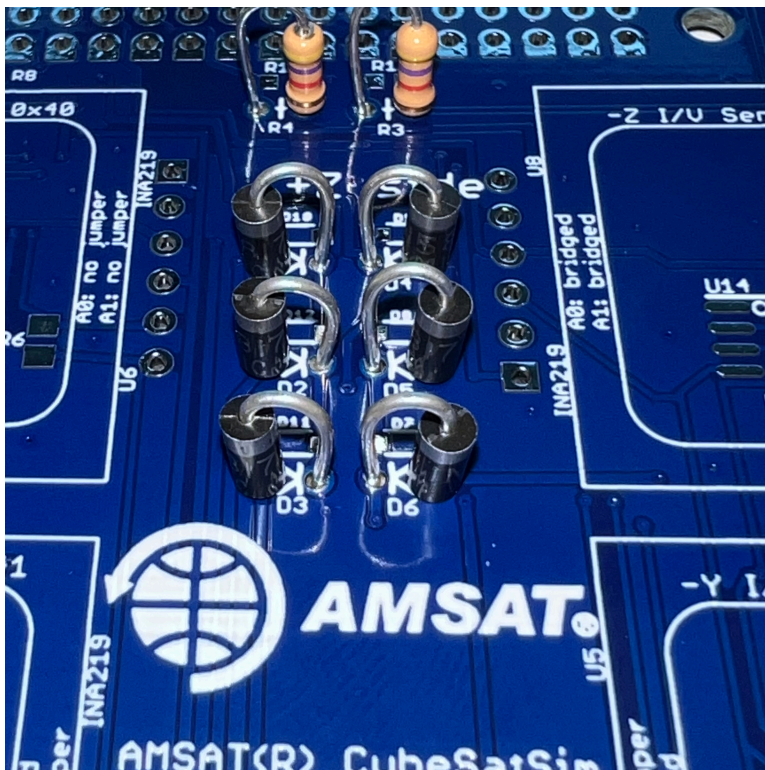


Next, solder the six diodes. The polarity of this diode is marked by a white band, which is on the right side in this photo, which matches the schematic symbol on the PCB for D1:

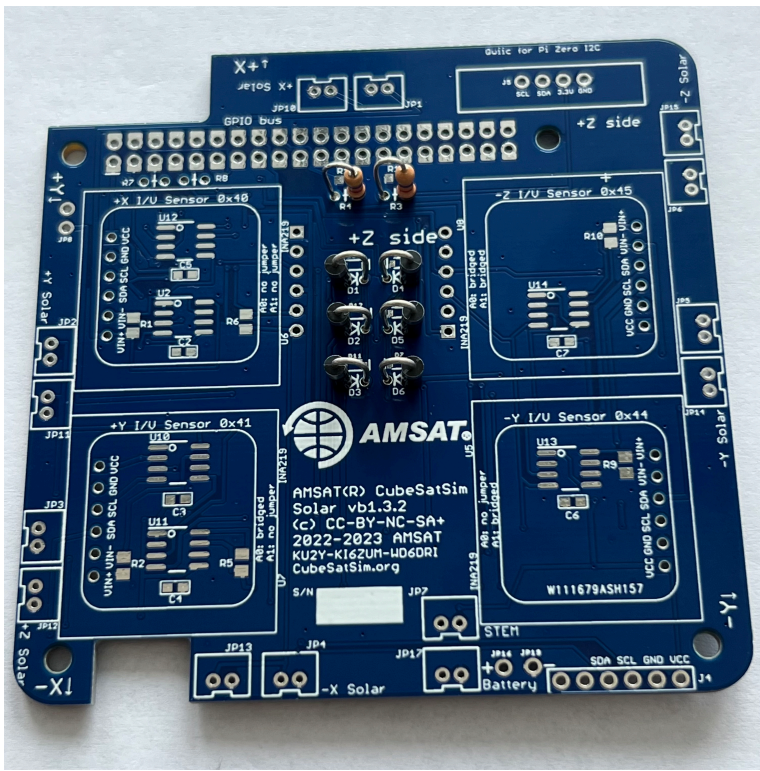




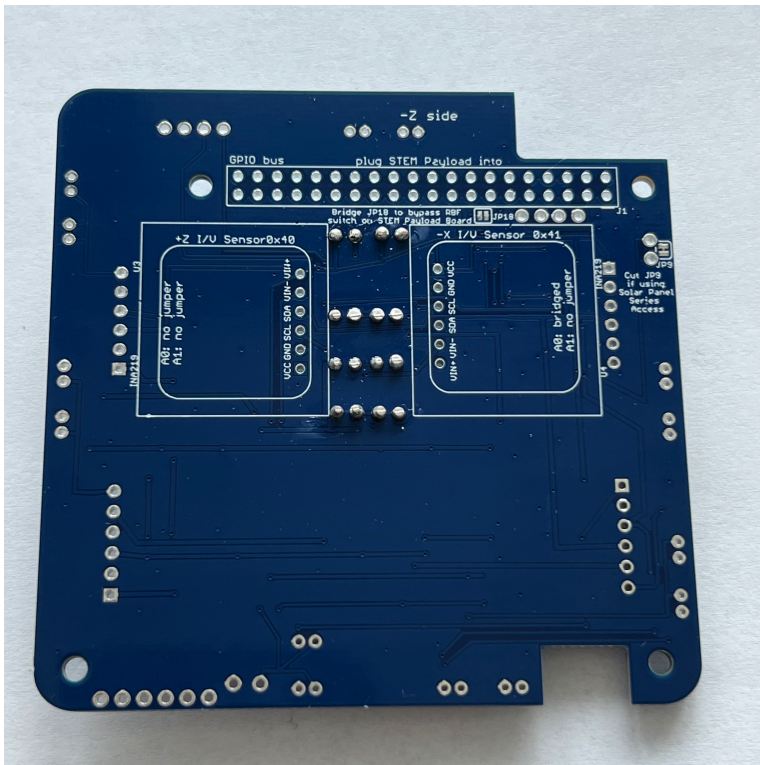
Here is the polarity of the diodes:



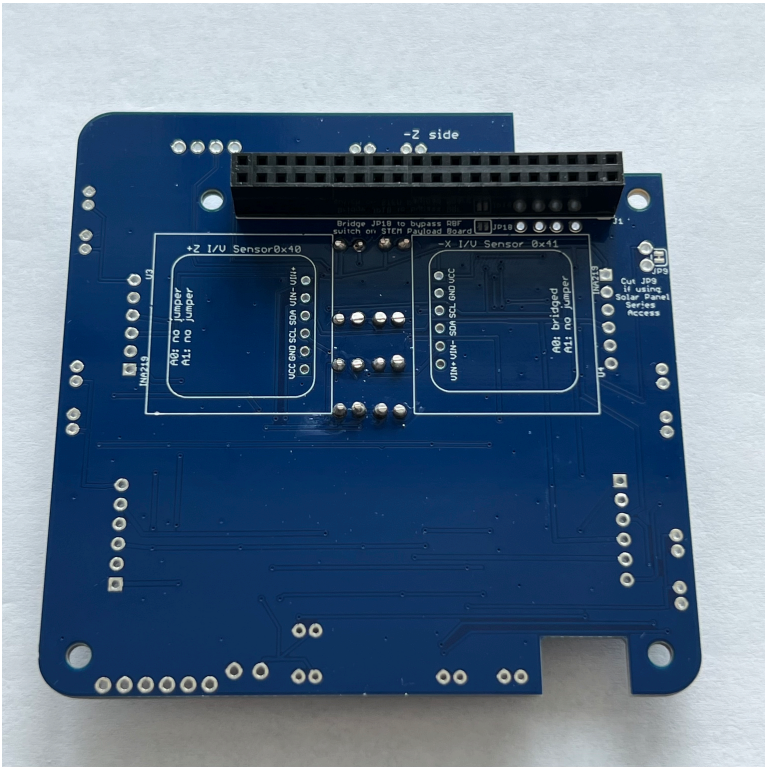
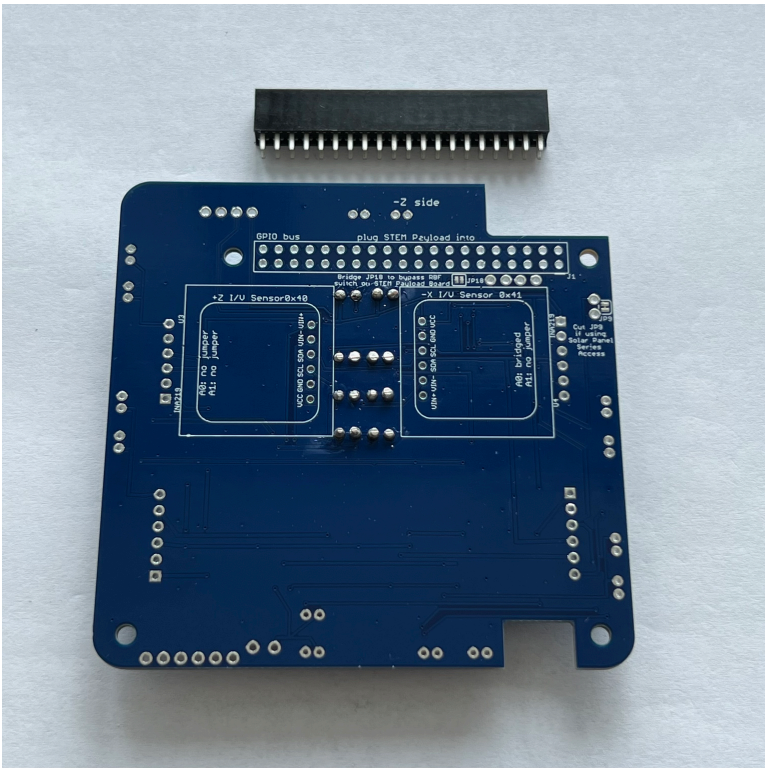
They are installed on the PCB top then soldered on the PCB bottom. Solder one lead on each diode then flip the board over and double check polarity, insertion, and straightness before flipping board back over and soldering the other lead:



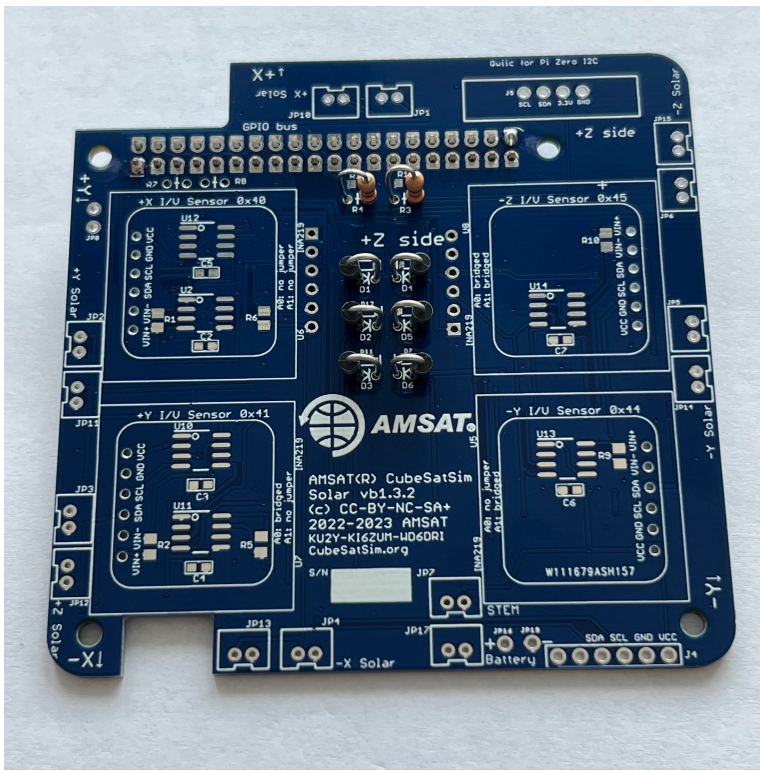
Trim the leads after soldering:



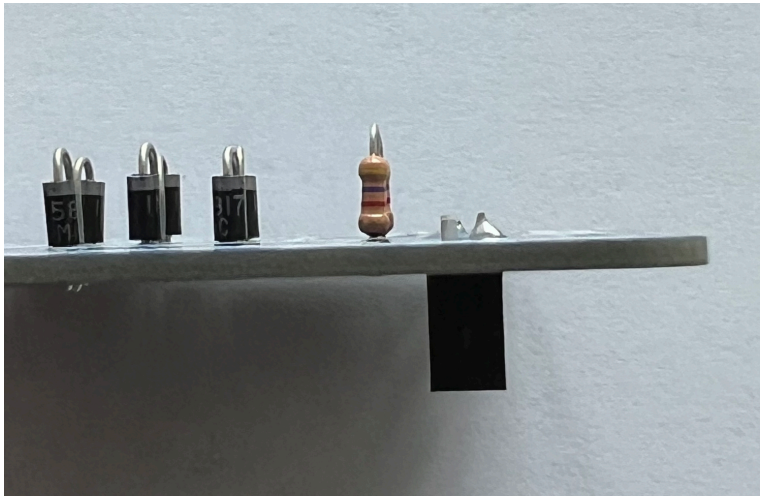
On the bottom, install the 2x20 GPIO female header J1.

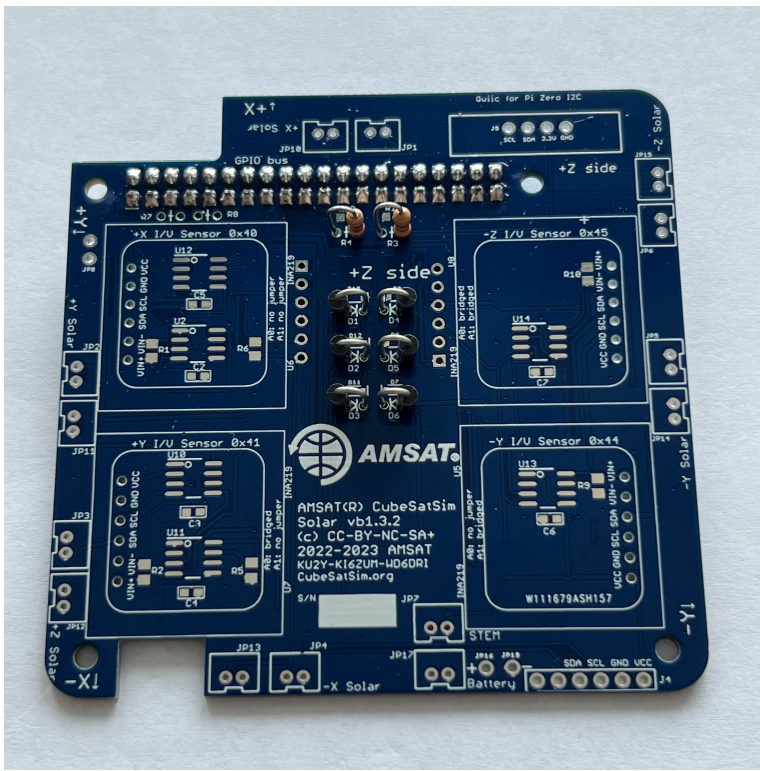


First solder only one pin on each side:

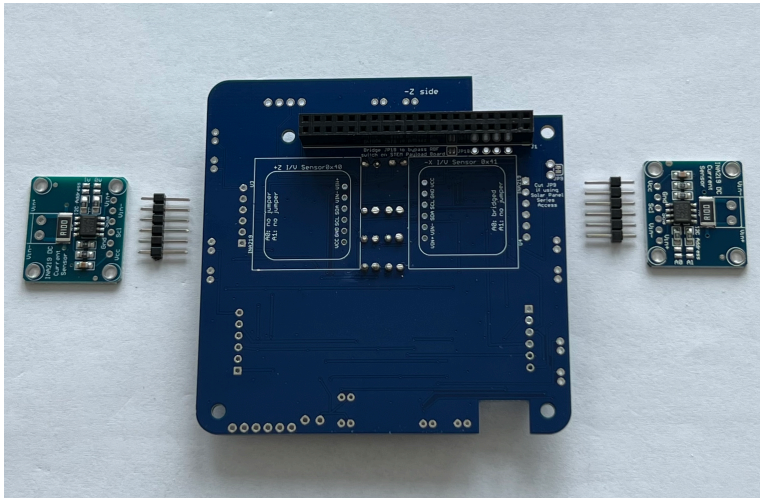


Verify it is fully inserted and straight before proceeding to solder the other pins:

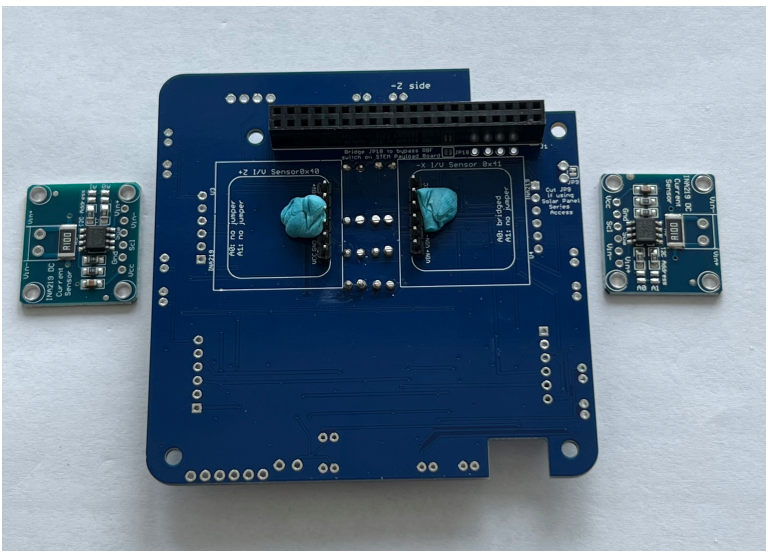




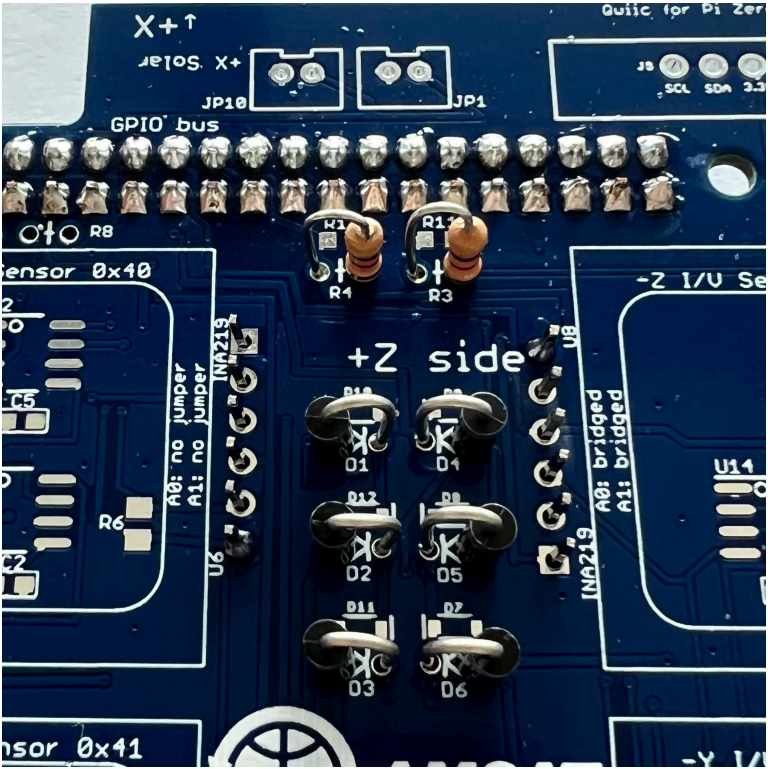
Also on the bottom, install two INA219 boards U3 and U4 and their 1x6 male pin headers:

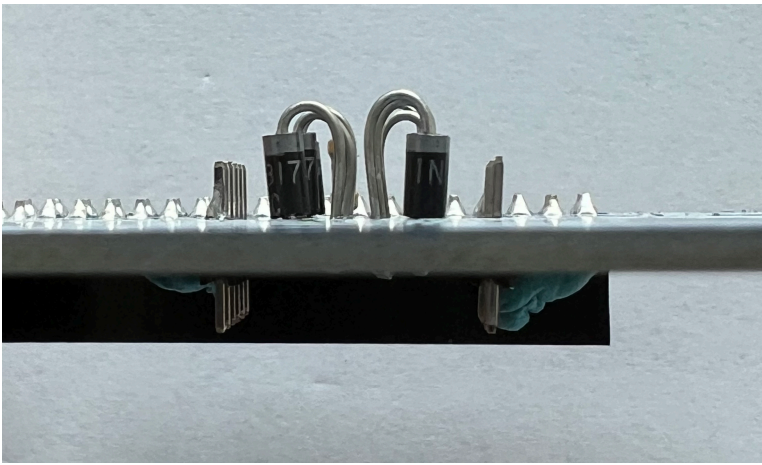


Insert the pin headers, long pins down and hold in place with blue putty while you solder them on the other side:

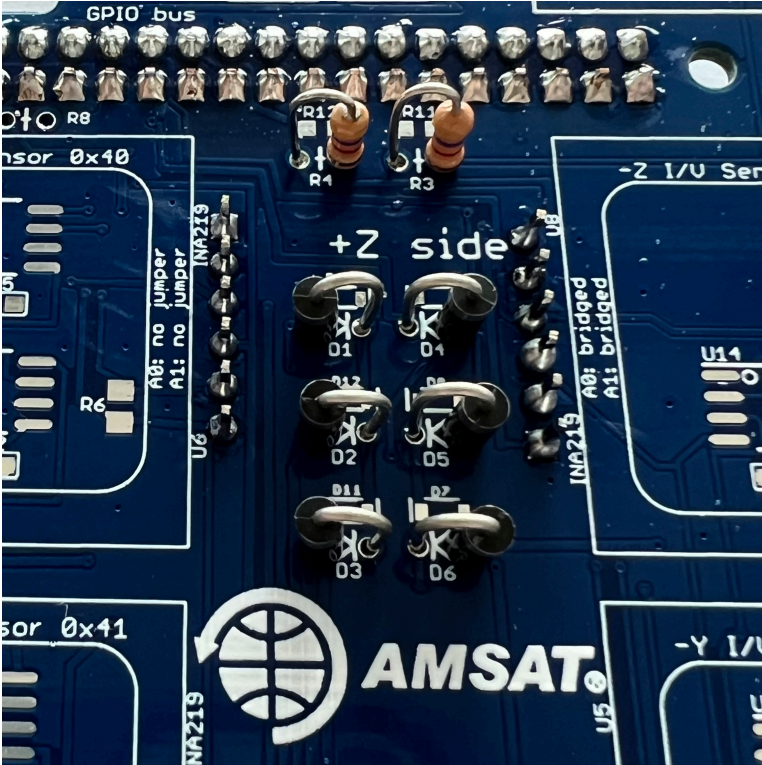


Solder one pin on each header, then check to make sure they are fully inserted and straight:

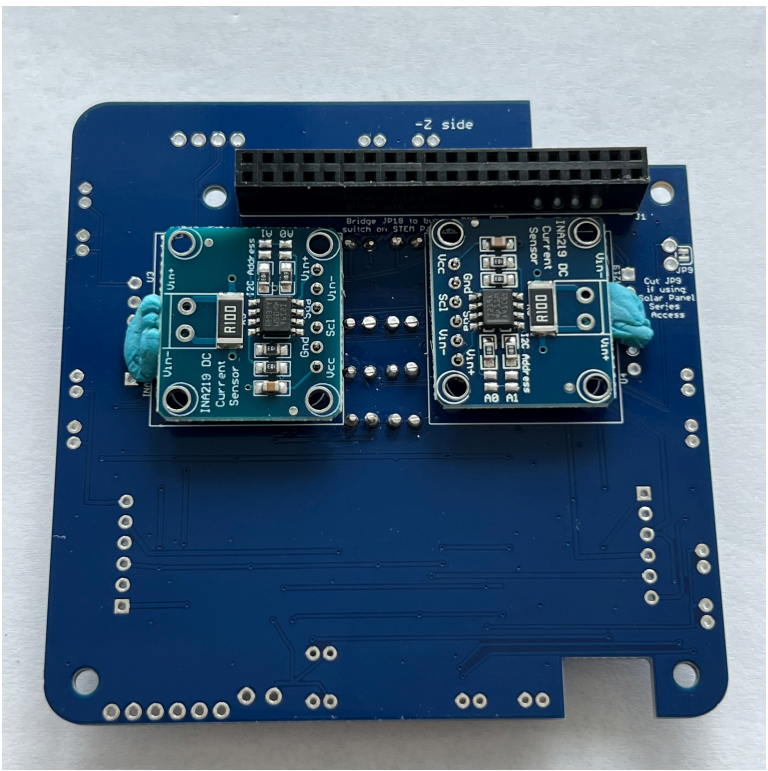




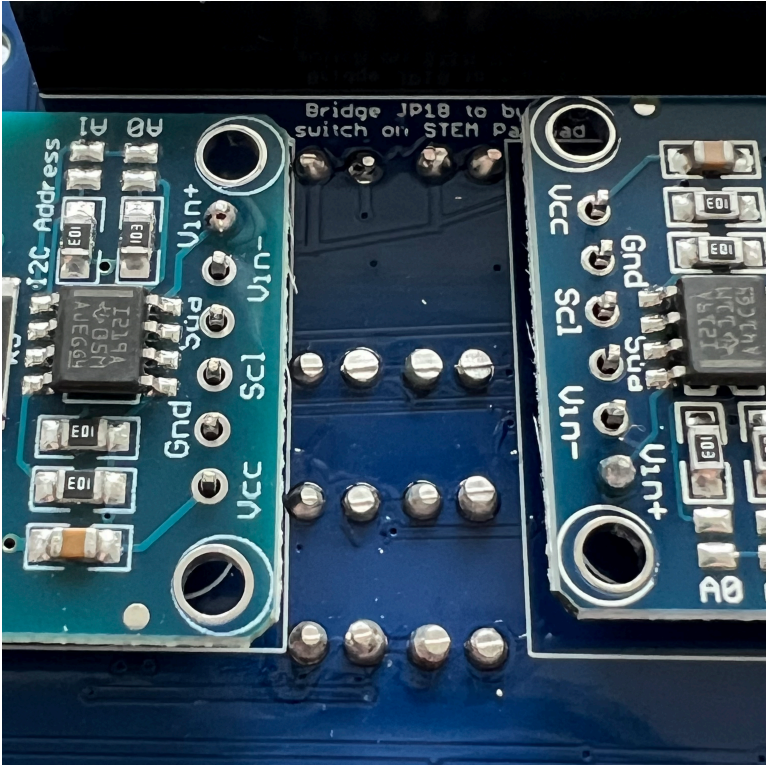
Solder the remaining pins:



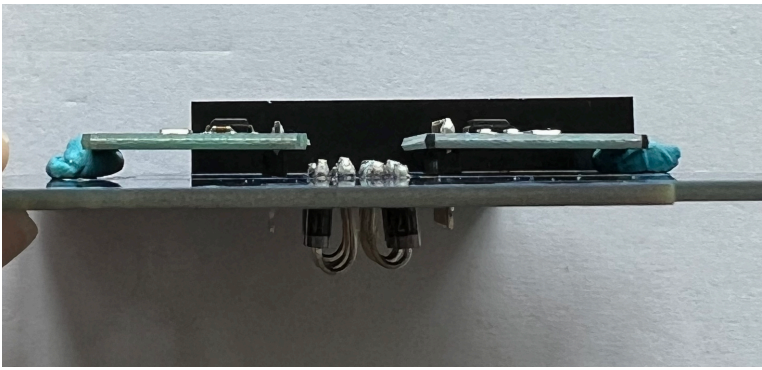
Then, on the bottom, install the INA219 boards and hold in place and level with blue putty:



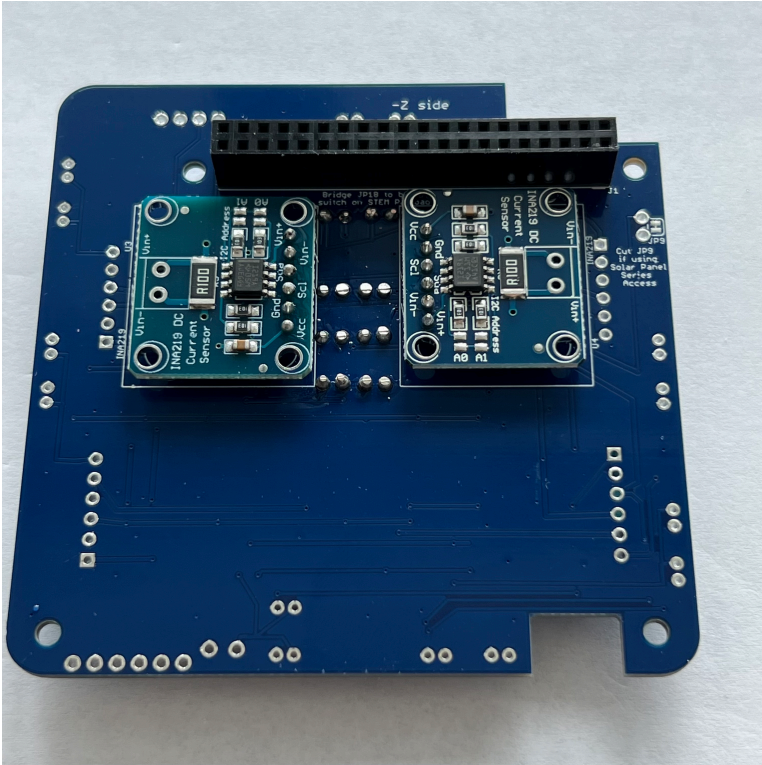
Solder one pin then check for straightness:



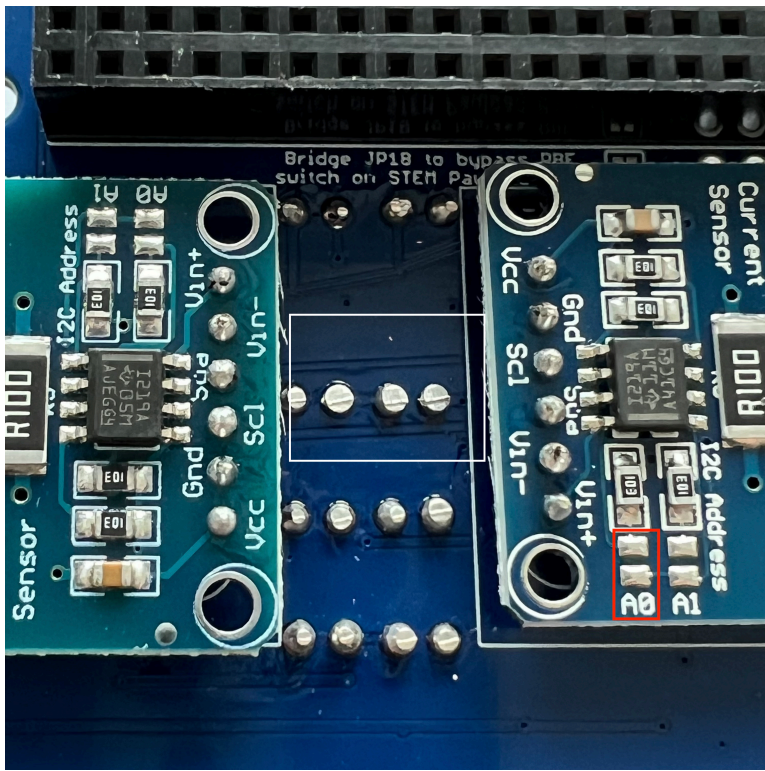




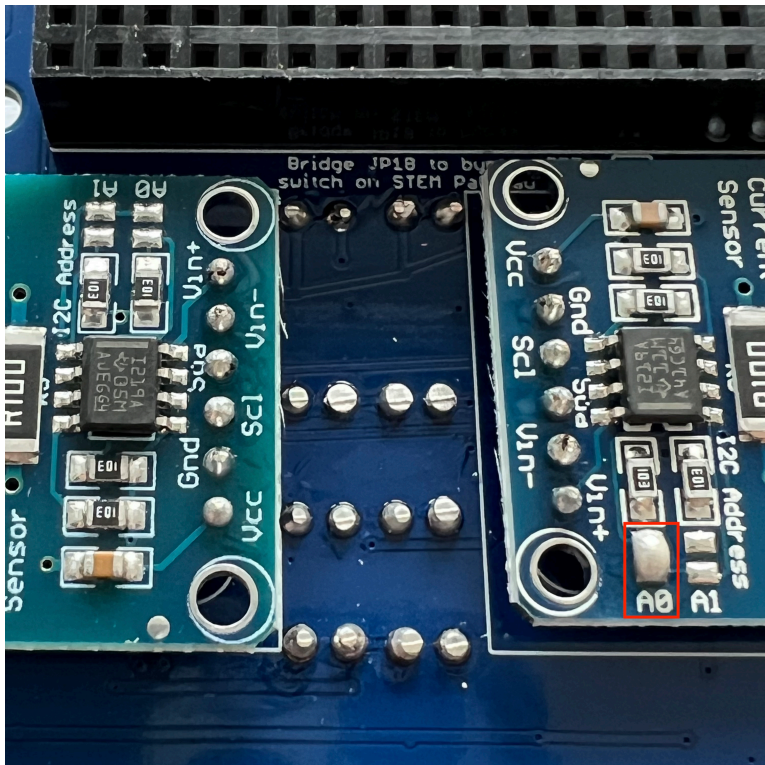
Solder the remaining pins:



One jumper needs to be soldered on right INA219 (marked here in red):

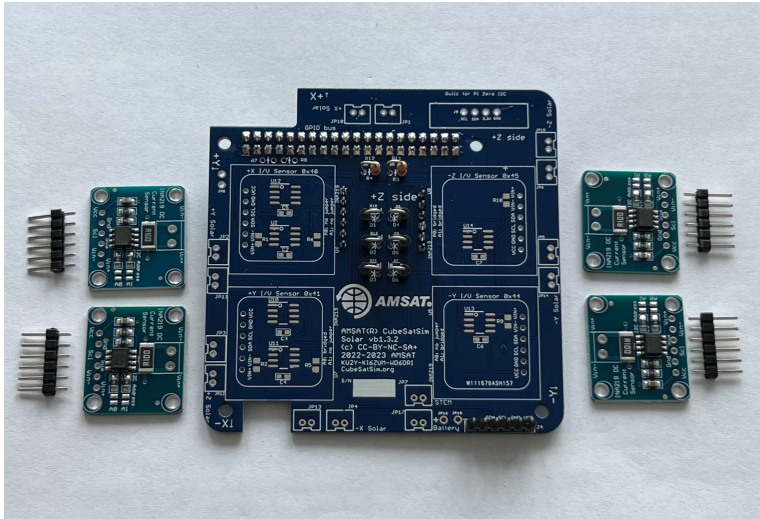


Add a small blob of solder on the pad so that it is bridged (shorted):

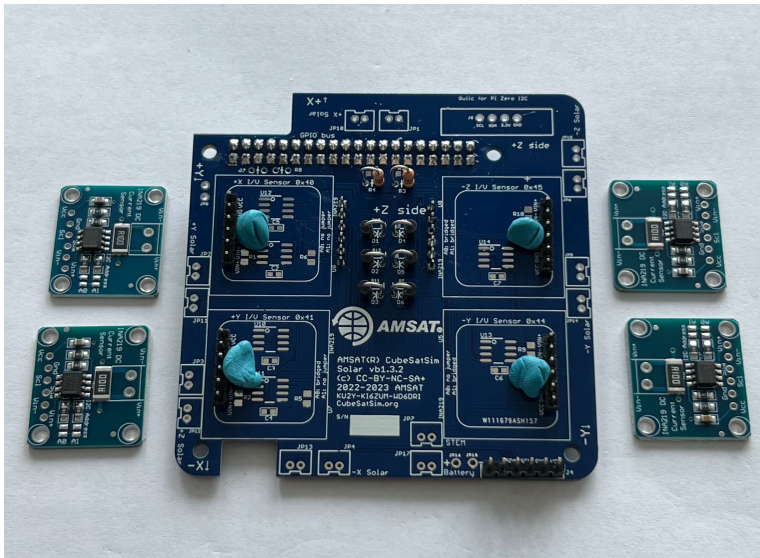


This completes the bottom of the PCB.

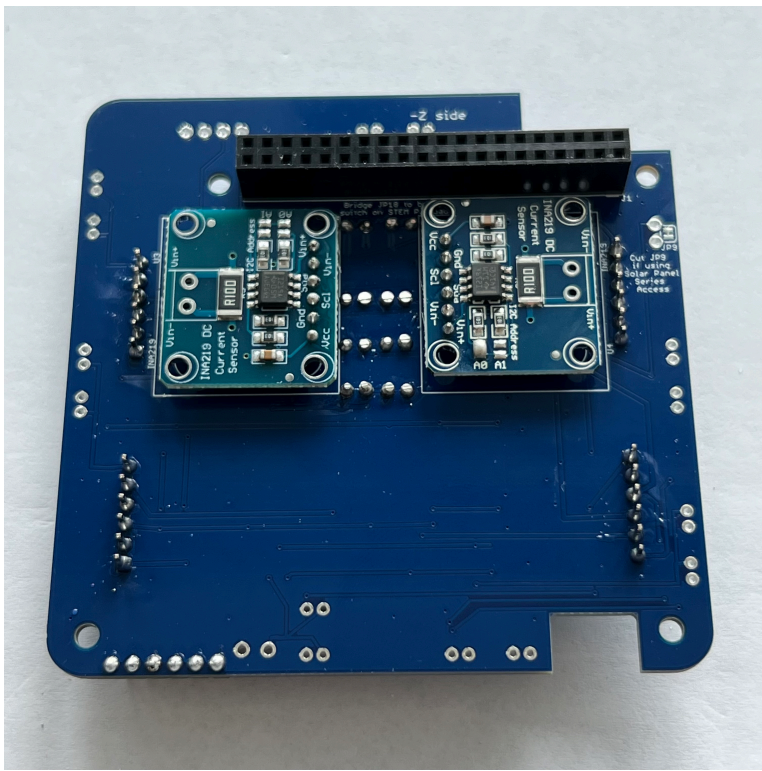
Flip the PCB to the top and install the four INA219 boards U5, U6, U7, and U8 and their 1x6 male pin headers.



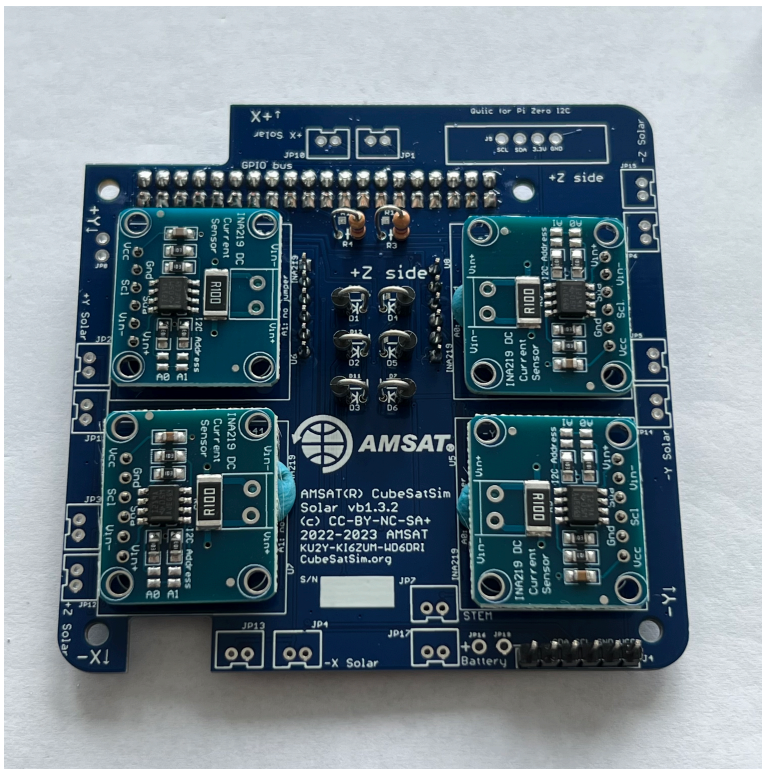
Insert the pin headers, the long side of the pins into the board. Hold in place with blue putty:



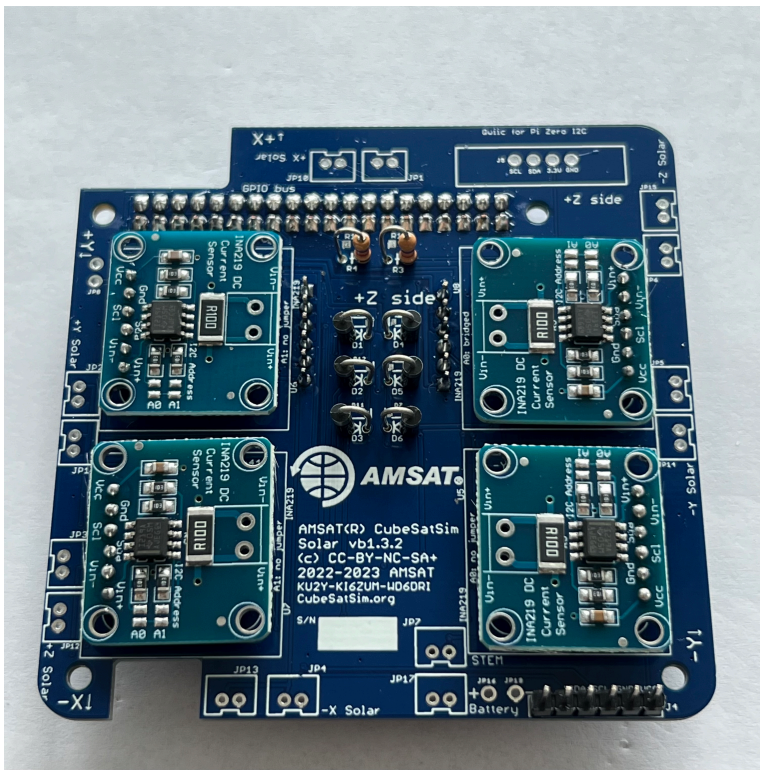
On the bottom of the PCB, solder one pin on each, then check for straightness. Then solder the rest of the pins:



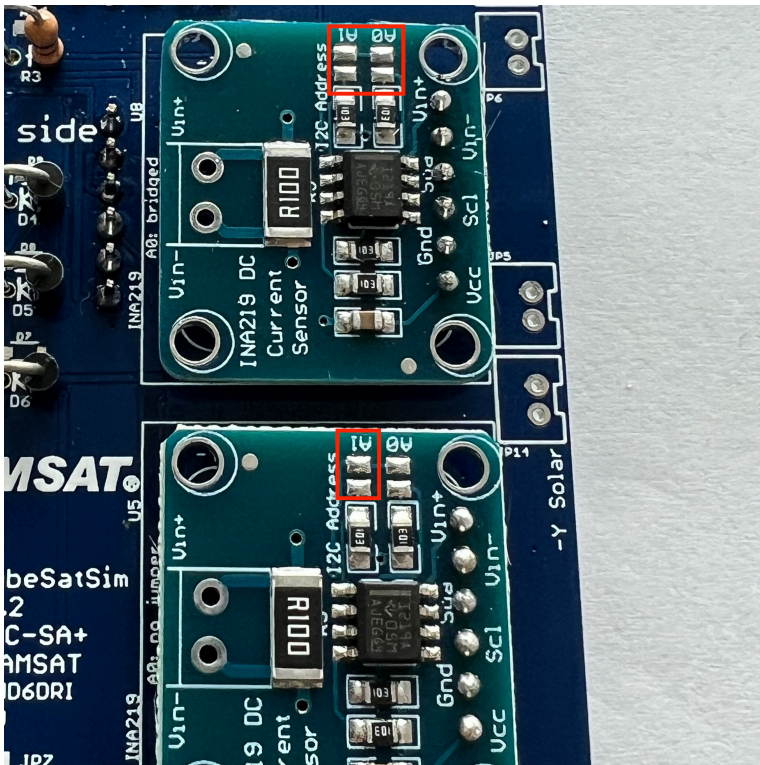
On the PCB top, place the INA219 boards on the pin header and hold straight and in place with blue putty:

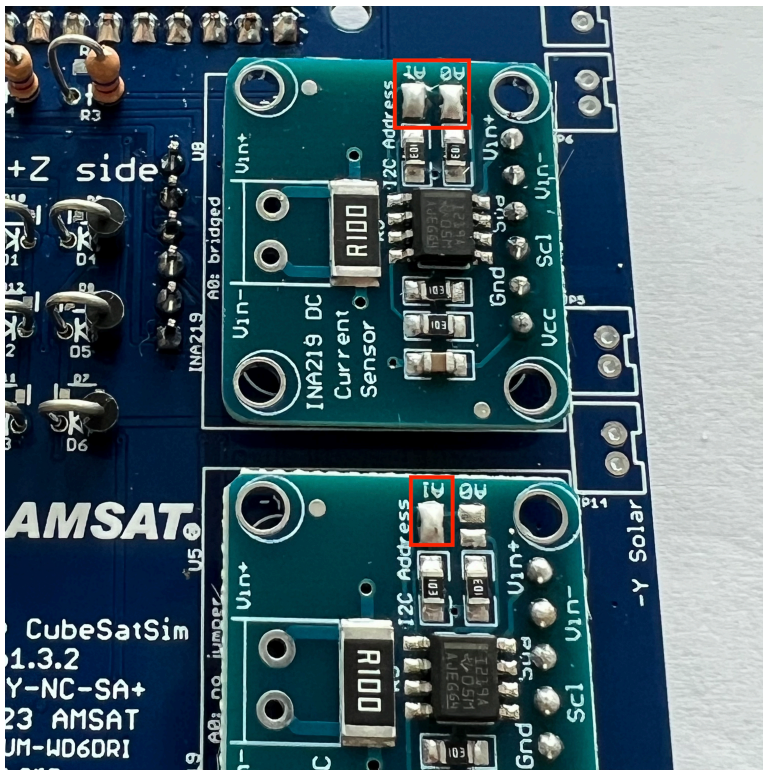


Solder one pin on each, check for straightness, then solder the rest:

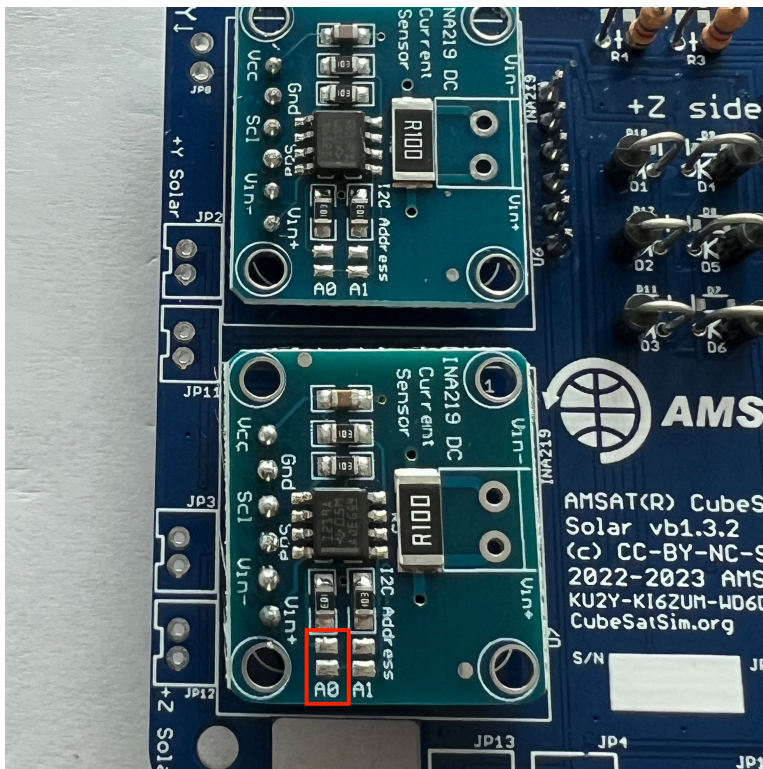


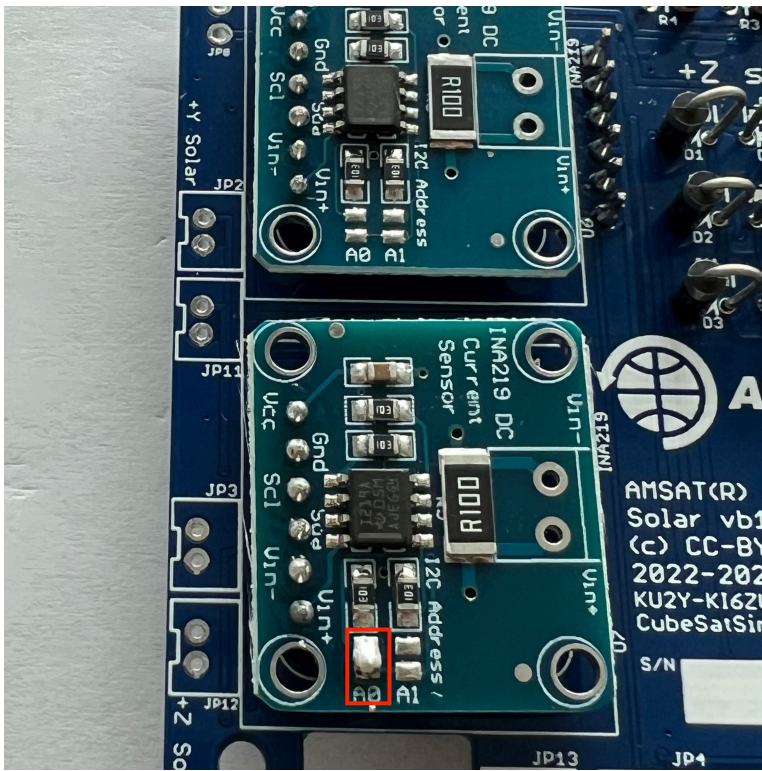
Next, the jumpers need to be set. On the right side, 3 jumpers need to be bridged:



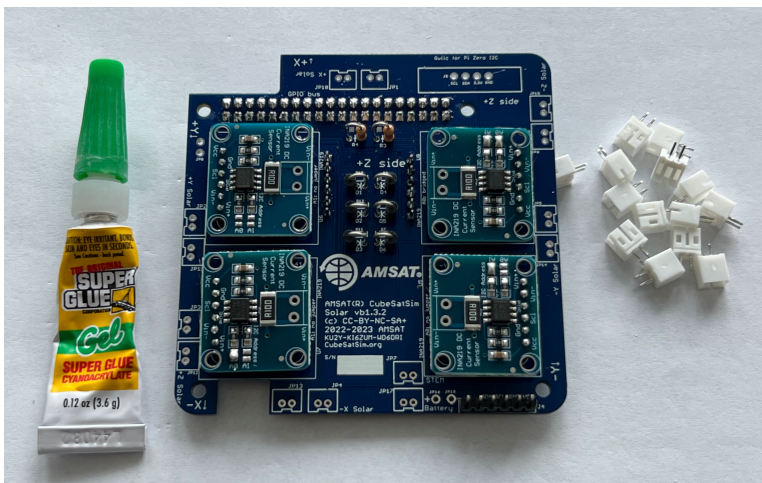


On the left side, one jumper needs to be bridged:

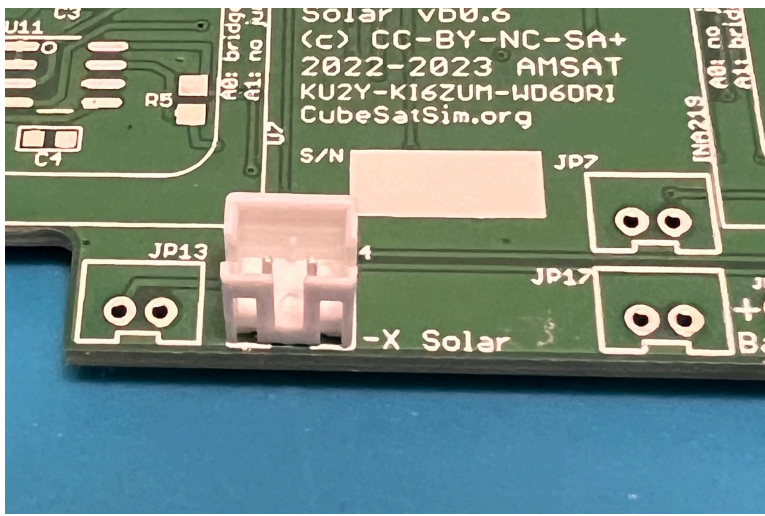




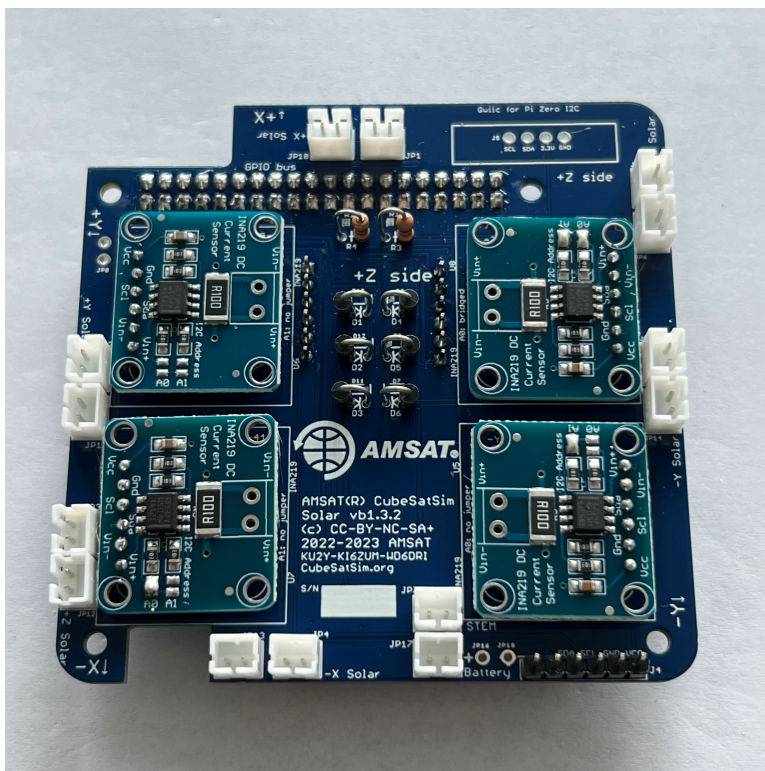
Next, install the JST connectors JP1, JP2, JP3, JP4, JP5, JP6, JP7, JP10, JP11, JP12, JP13, JP14, JP15, and JP17 on the top of the PCB. Some connectors click in place. If yours don't, you should add a drop of superglue gel or other adhesive to secure it to the PCB so it won't pop out when you are unplugging solar panels:



Note that these connectors have a polarity. Make sure the slot in the connector faces to the outside of the PCB:

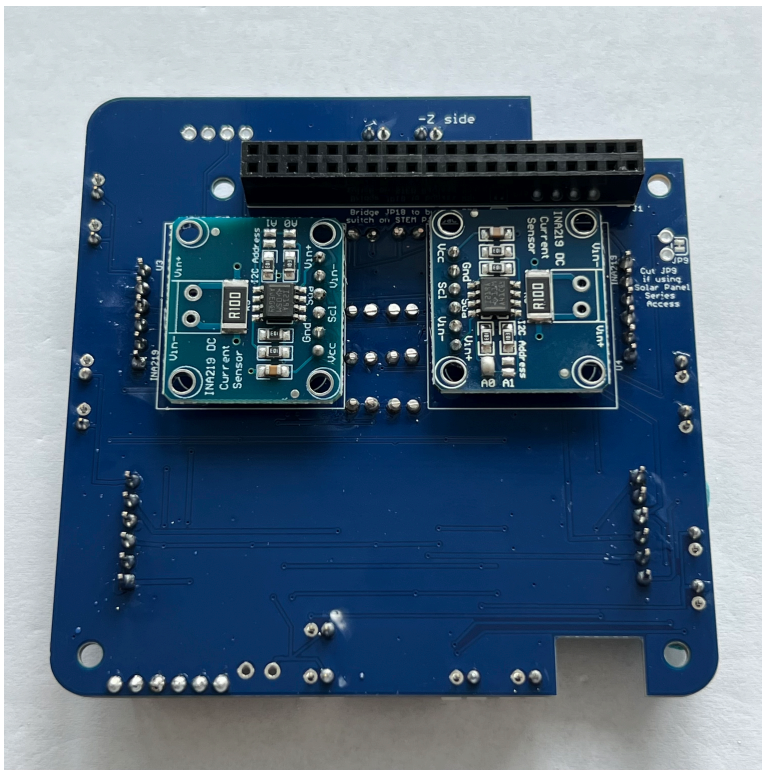


Here's how the board looks with them all installed:

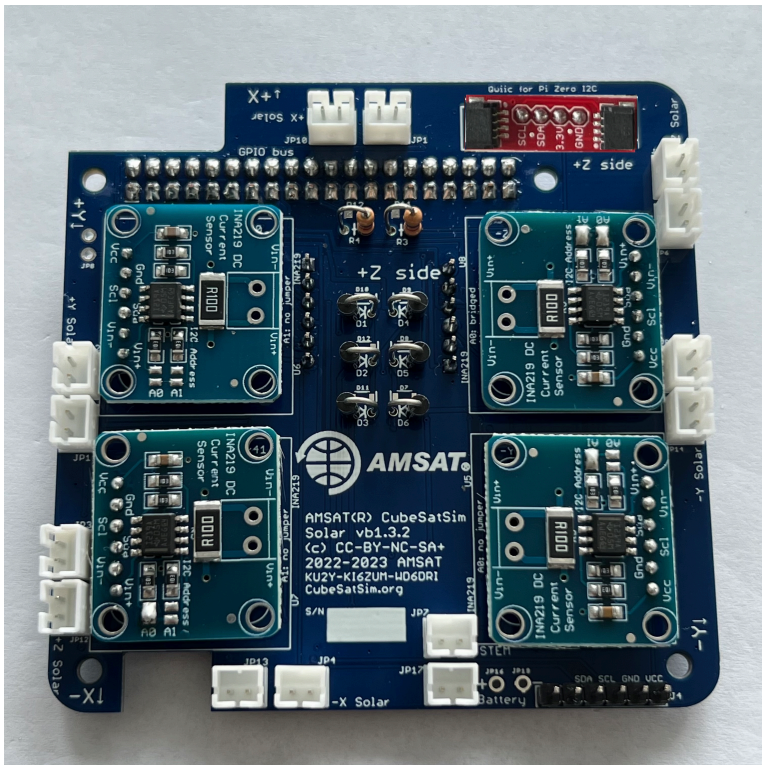


Flip the board over and solder one pin on each one then flip back and check insertion, orientation, and straightness. Then solder the other pin:

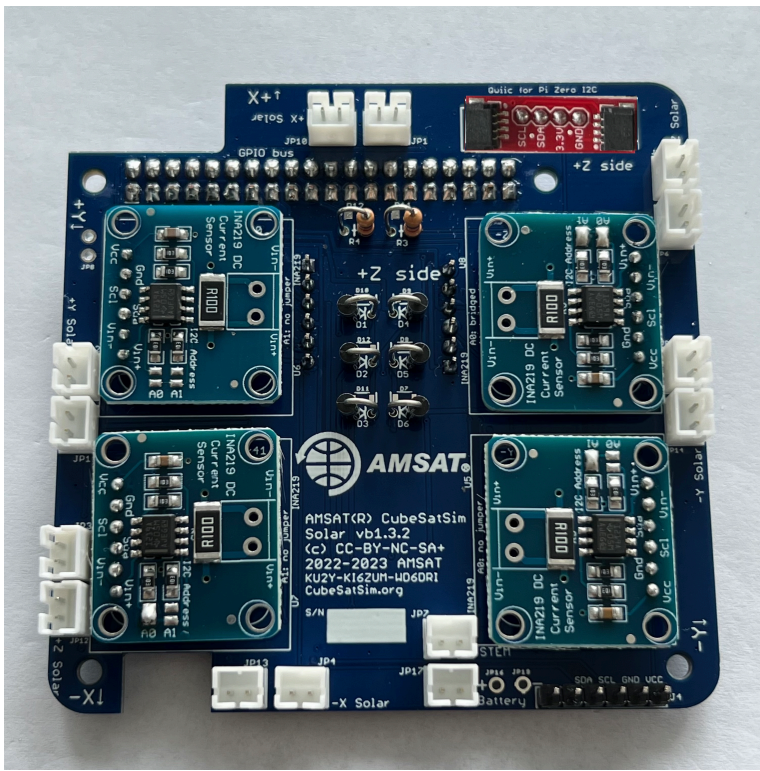




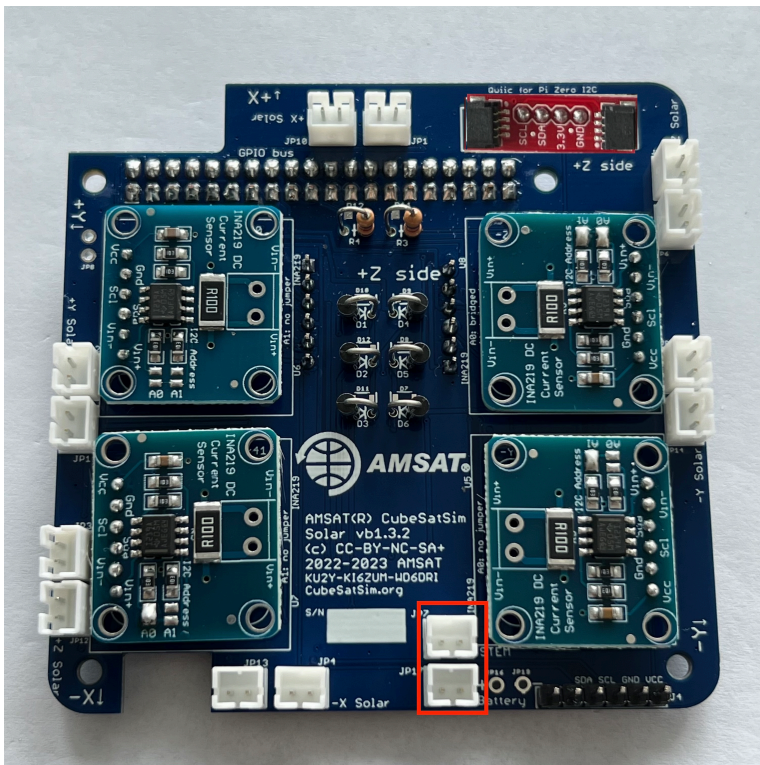
Next, install the Qwiic connector (red board) using the 1x4 pin male header J5.



Here's the complete board:



The two JST in the red box are for the JST jumpers to the Battery and STEM Payload boards that you will connect in the Board Stack step. Both connectors are the same - it doesn't matter which is used.



Plug the JST cable into one of the connectors. You will plug in the other end into the STEM Payload board when you build the board stack.

The next step is [Step 7. Solar Panels and Frame](#)

+ Add a custom footer