Water Leak Detector

PCB Assembly Instructions

For PCB v2

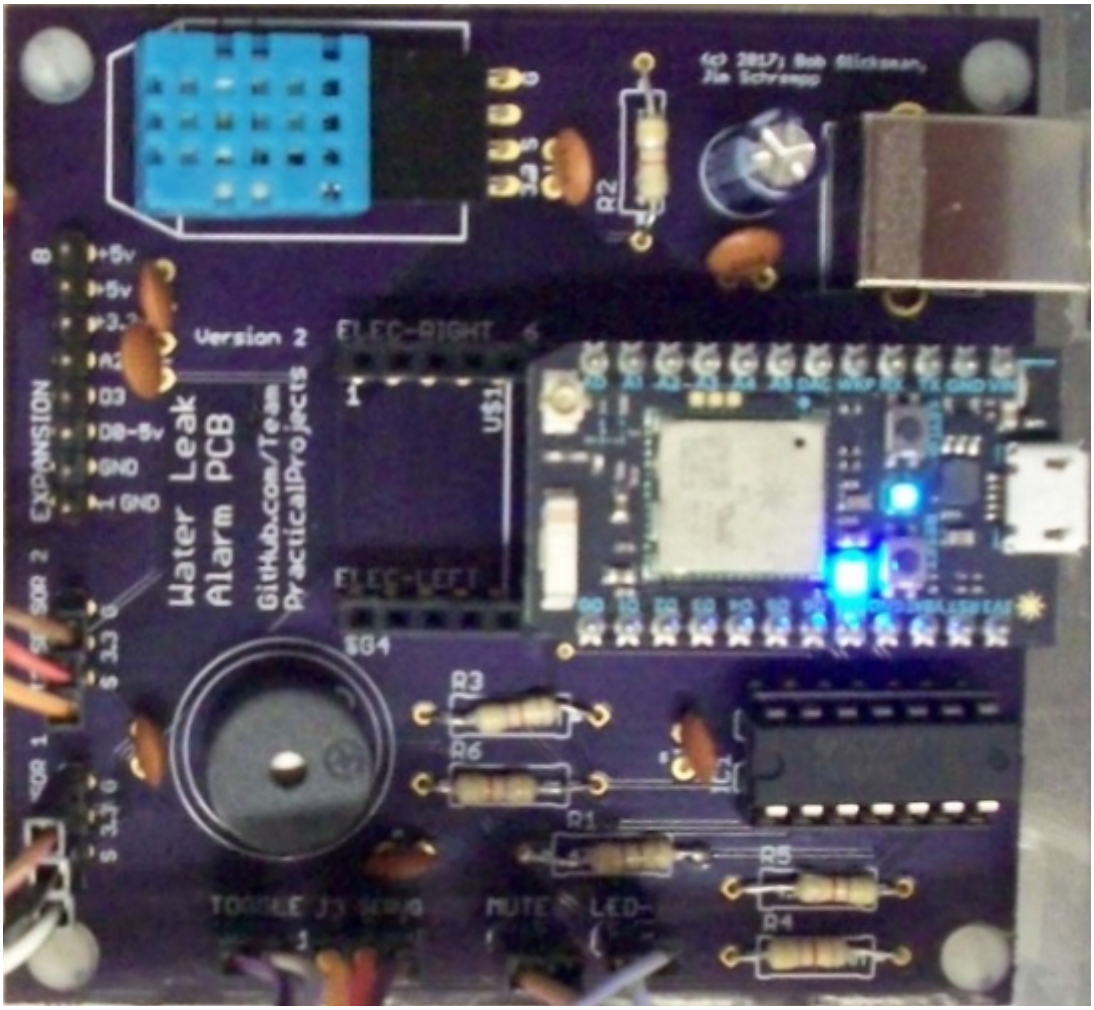
By: Jim Schrempp and Bob Glicksman; updated 9/13/2017

Table of Contents

Purpose

This document will provide step by step instructions for assembly of the Team Practical Projects Water Leak Detector printed circuit board. This process will take about 90 minutes.

You can find complete information about this project at:
https://github.com/TeamPracticalProjects/WaterLeakSensor/tree/master



Completed WLD PCB

What you will need

In addition to the electrical parts listed in the bill of materials spreadsheet, you will also need the following:

1. Soldering Iron
2. Solder
3. Diagonal cutters
4. Small nose pliers
5. Wire strippers
6. Masking tape

Step by Step Instructions

Double check the PCB

If you have a multi-meter it is a good idea to double check the PCB before starting the build.

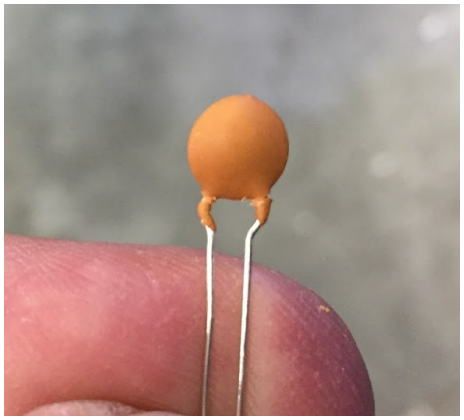
Locate the 8 pad Expansion header on one edge of the PCB. Use your multi-meter in resistance measuring mode to test that there is no continuity between the pads labeled GND and the pads labeled +3.3v and +5v. If you find that there is connectivity between these pads, then you have a bad PCB.

Resistors and Capacitors

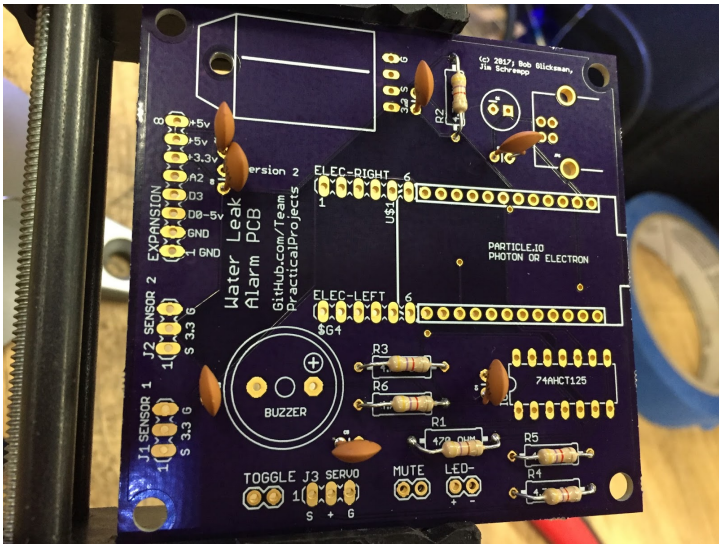
Locate the single 470 ohm resistor. Locate the one position on the PCB for the 470 ohm resistor (“R1”). Bend the resistor leads to fit the holes on the PCB, insert the resistor, and solder it in place. Trim the extra wire with the diagonal cutters.

Now locate the 4.7K ohm resistors (“R2” through “R6”). Bend the leads of each to fit the locations on the printed circuit board. While not required, it is considered good practice to have them all face the same direction on the board. Solder them in place and trim the extra wire.

Find the 0.1 micro farad disc capacitors (there are 7 of them). Again, it is just good practice to have them all face the same direction. Place them on the PCB and solder them in place. Trim the extra wire.

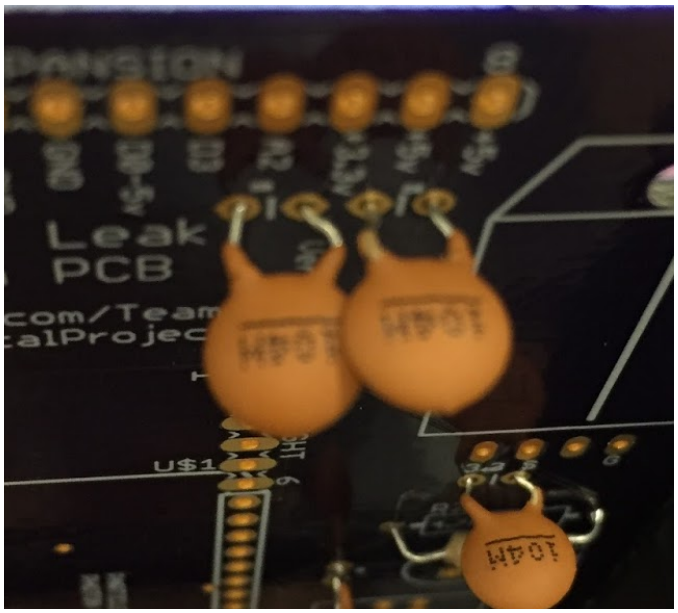


You may need to bend the leads in slightly, as shown here.



PCB with resistors and disk capacitors in place.

Note that there is one place where two of the capacitors are placed very close together. The two leads that are near each other are both grounded so it does not matter if those two leads touch. If we make a Revision 3 of the PCB then we will correct it there.

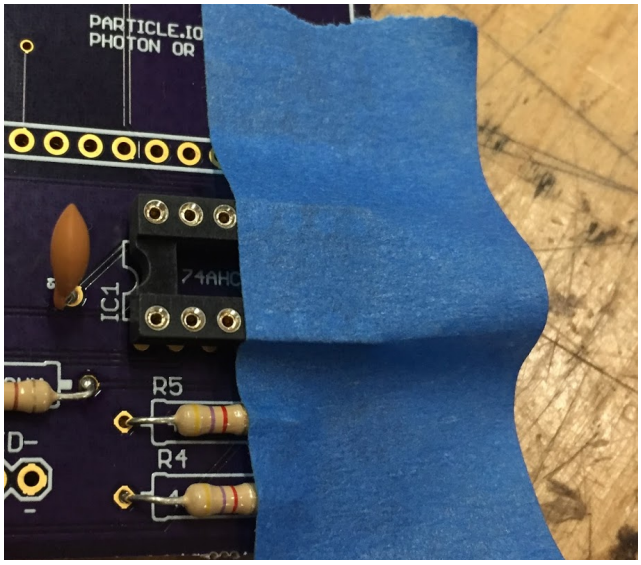


Two capacitor leads are close, but in this case
it does not matter if they touch.

Find the one electrolytic capacitor and its location on the PCB. NOTE the polarity of the capacitor and the holes on the PCB. The negative side is the side with the silver stripe and it must be connected to the “-” round pad on the PCB, with the other side connected to the square pad on the PCB. Solder the capacitor in place and trim the extra wire.

DIP Socket

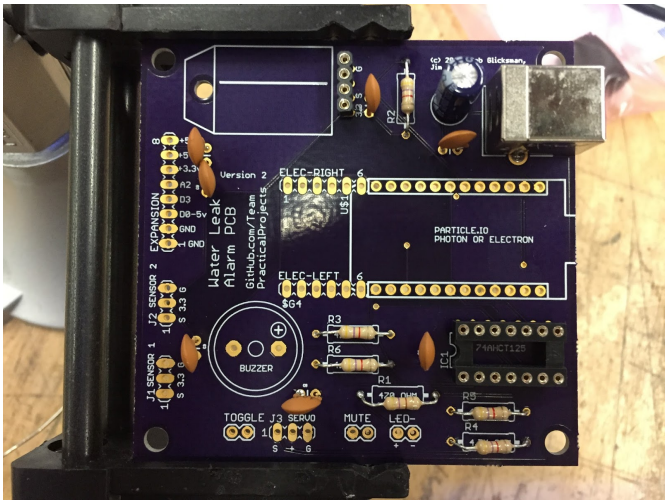
Locate the dual in-line socket. Note that one end of the socket has a small divot in it. Note that the icon on the PCB also has a small divot in it. Orient the DIP socket so that the divot is in the location indicated on the PCB. Use a piece of masking tape to hold the DIP in place and solder it to the board.



DIP held in place for soldering.

USB Connector

Locate the USB connector. Place it on the PCB. Note that you will have to press gently but firmly to seat the big case pins into the PCB. Solder the four pins in place. Then solder the two big case pins to the PCB. Be sure to flow enough solder to fill the big holes.

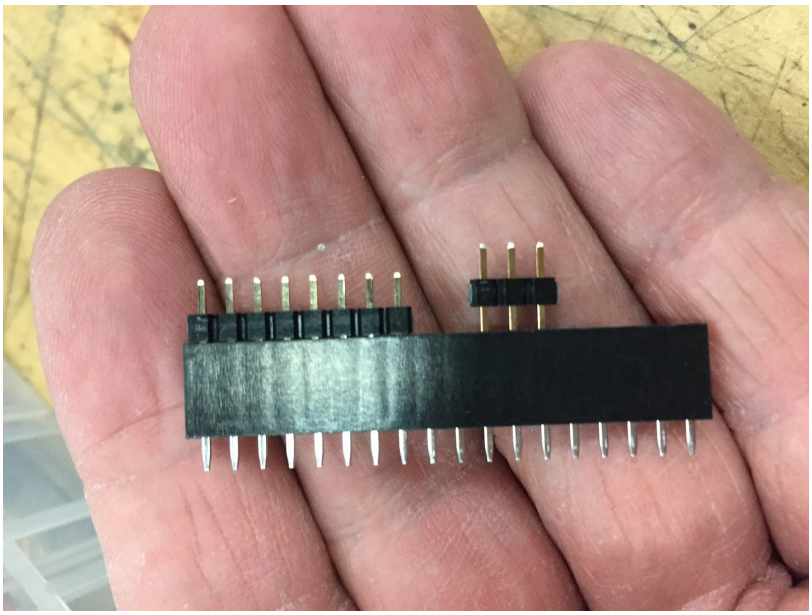


USB connector is soldered in place

Side Headers

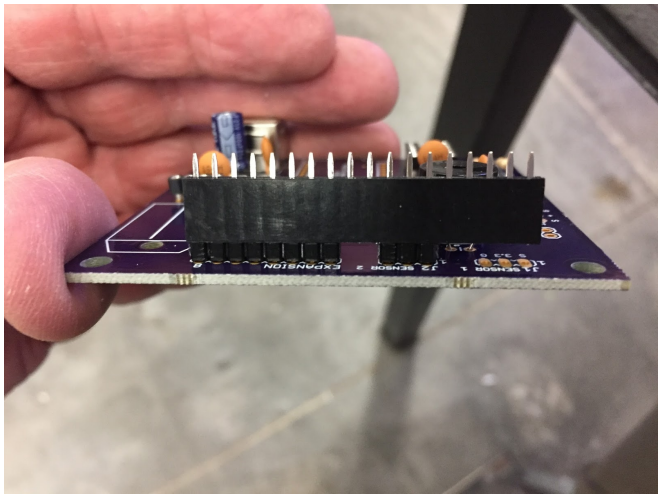
Side A

Locate the snap apart male pin headers. Make three pieces: one 8-pin and two 3-pin headers. Locate one of the Photon headers. We will use the Photon header to hold the other headers in place while we solder them. Place the long pins of the 8-pin and 3-pin headers into the Photon header; do this so that the small headers are the correct distance apart to fit into their locations on the PCB.

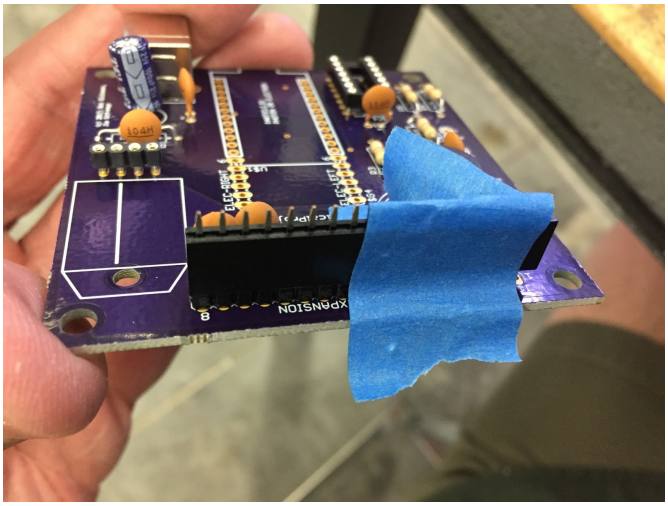


Two snap apart headers in the Photon header

Now place the snap apart header pins into the PCB and use a bit of masking tape to hold them in place.



Headers in the PCB using the Photon header to hold them in place.

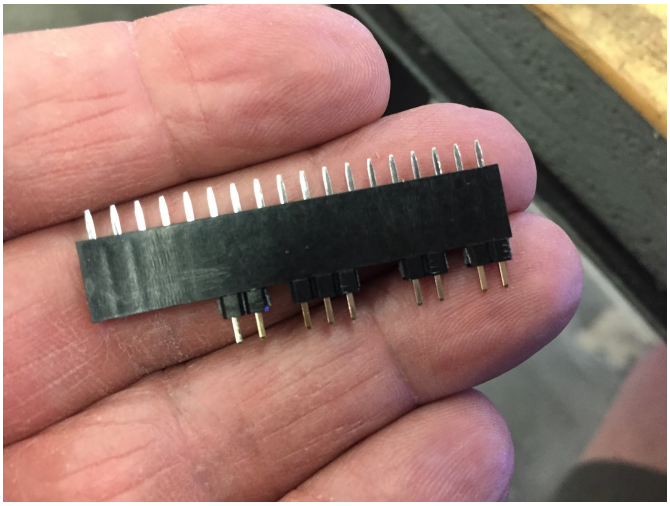


Ready to solder.

Now turn the PCB over and solder the headers to the PCB. Remove the tape and pull off the Photon header.

Side B

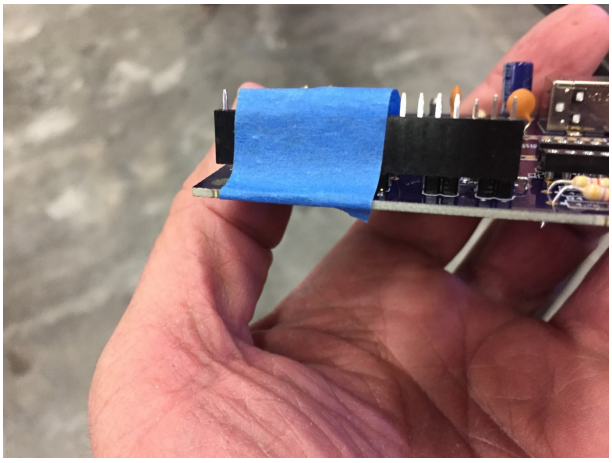
Locate the snap apart male pin header and create one 3-pin header and three 2-pin headers. Insert these into the Photon header at the correct locations to fit the PCB.



Snap apart headers in the Photon header

Note how the snap apart headers are in the correct positions to be inserted into the PCB.

Use masking tape to hold the headers in place and solder them to the PCB.



Second set of headers ready to solder.

Pull the photon header from the snap apart headers.

DHT11 Header

Locate the 4-pin header for the DHT11. This header has round pin holders. Use masking tape to hold it in place and solder it to the PCB.

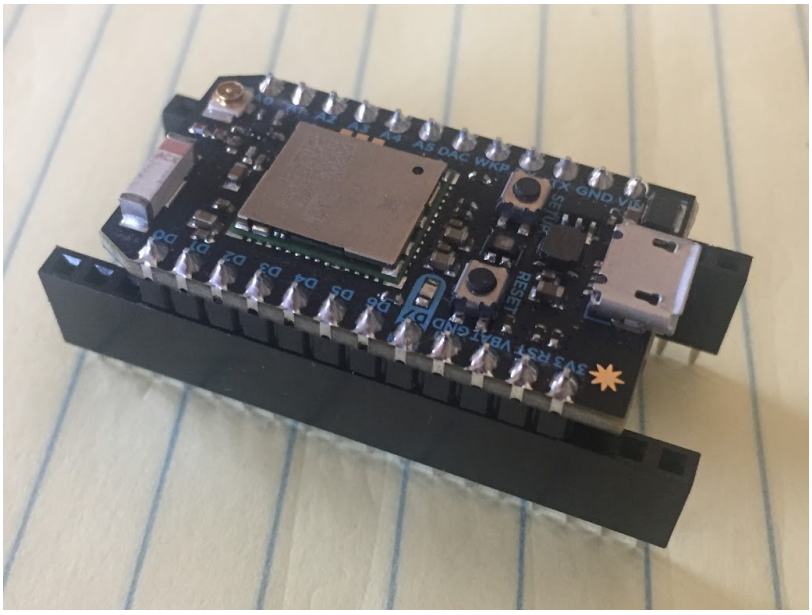
Buzzer

Locate the buzzer. Note the polarity indication on the buzzer and on the PCB board. Insert the buzzer and solder it in place. If there is a protective label over the buzzer, remove and discard it.

Photon Headers

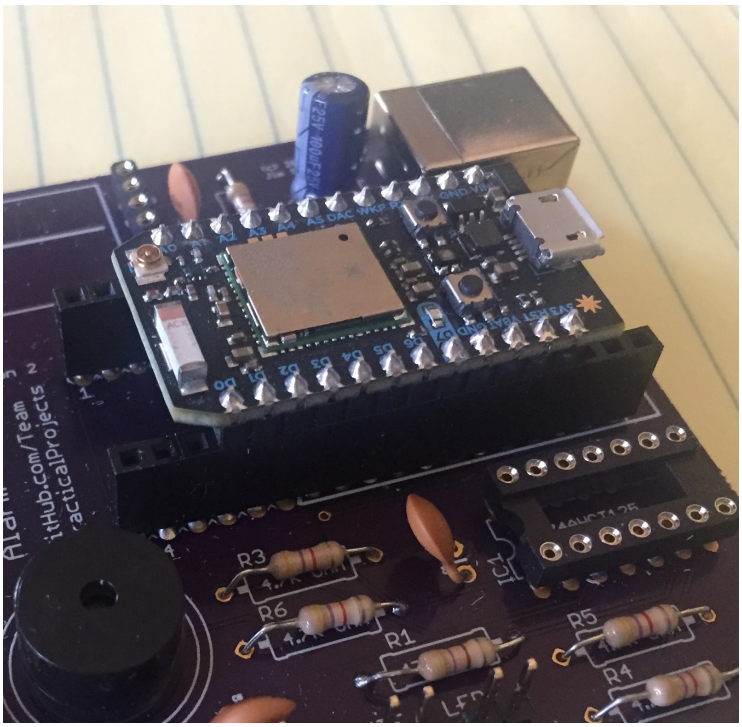
It is critically important that the Photon headers are installed correctly. The procedure here is foolproof (we hope).

First insert the Photon into the middle of the two Photon headers. *Note that this is NOT the correct position when the PCB is finally assembled. This middle position is just used to assist you in soldering the headers to the PCB*.



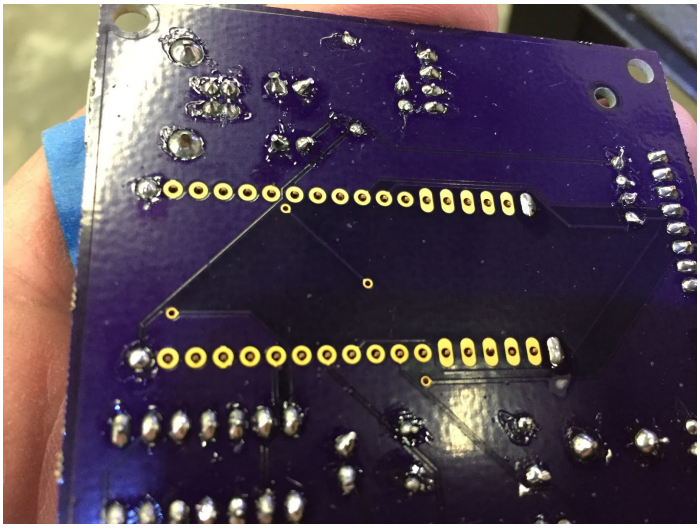
Photon on headers to position them for soldering.

Now place the Photon and headers into position on the PCB.



Photon in headers and on PCB for soldering. NOTE:
This is NOT the correct Photon position for operation!.

Hold the Photon and headers in place with a bit of masking tape. Invert the board and solder the two ends of each header to the PCB.



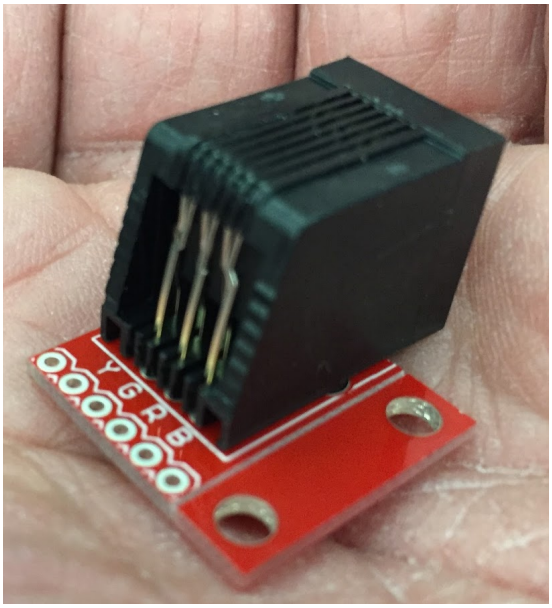
The ends of each Photon header are soldered.

Now **remove** the Photon from the headers and set it aside. We remove the Photon to make sure it is not exposed to high heat while soldering the pins. Solder all the remaining Photon header pins to the PCB.

RJ11 Jacks

Locate four RJ11 jacks and two sets of RJ11 breakout boards.

Break the breakout boards apart so that you have 4 pieces. Insert the RJ11 jacks into each board - you may have to press them gently but firmly into the break out boards. NOTE that the RJ11 jacks are inserted to the side of the breakout boards that say YGRB.



The RJ11 connector on a breakout board.
You will make 4 of these

Invert the boards and solder the six pins of each RJ11 connector to their breakout board. Now make four 3-pin snap apart male headers and solder them to the RJ11 breakout boards. NOTE: on two of the RJ11 boards you will solder the headers to pins 3, 4, and 5 and on the other two RJ11 boards you will solder the pin headers to pins 2, 3 and 4. Insert the snap apart headers on the same side of the board as the RJ11 connector.

NOTE: The top side of the breakout board has holes labeled YGRB. This labeling corresponds to the numbered hole labeling on the bottom of the breakout board as follows:

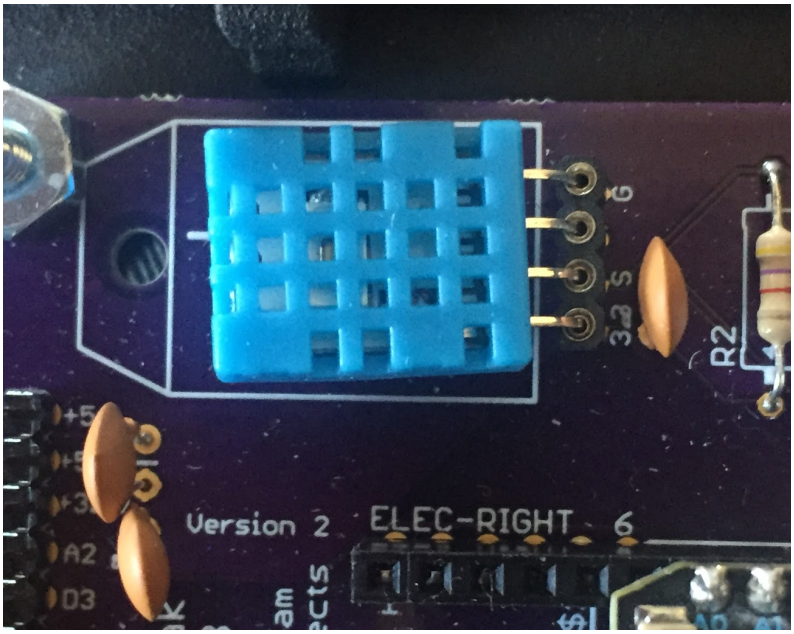
Pin 1: no top label
Pin 2: top label B
Pin 3: top label R
Pin 4: top label G
Pin 5: top label Y
Pin 6: no top label

Components

Locate the 74AHCT125 DIP chip. Use straight pliers to gently bend the pins on one side and then the other so that they will fit into the DIP socket on the board. Note that the DIP chip has a small divot or white dot on one end. Insert the DIP chip into the socket so that the divot or dot aligns with the mark on the socket.

Once inserted, look closely at the pins to make sure they are all in their socket. It is easy to have a pin bend and not be inserted correctly.

Locate the DHT11, a blue rectangular component. Note that one side is flat and one side is full of square holes. Position the DHT11 so that the side of holes is towards the USB power connector. Insert its four pins into the socket. Then gently bend the DHT11 to the side so that it lies close to horizontal as shown in the photo below.



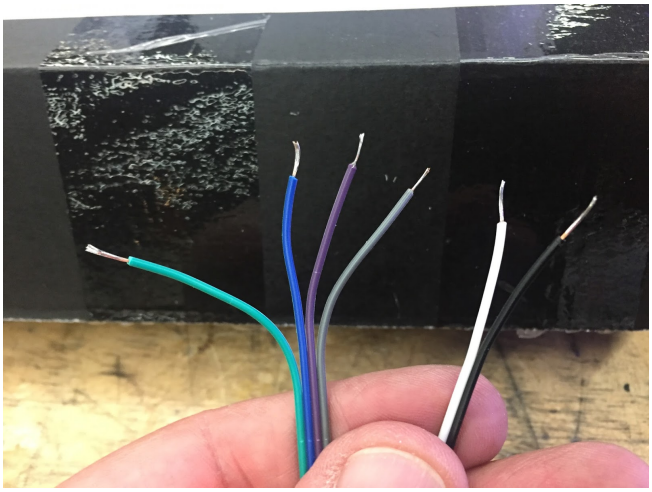
DHT11 inserted into socket and bent over.

Prepare Switches

Since you have your soldering iron hot, now is a good time to attach wires to the panel switches.

Locate the two zip wire pieces, one with 2 conductors and one with 4 conductors. Cut the female connectors from one end of all the wires. Using wire strippers, carefully strip off 0.25 inch of insulation from the ends. Use a wire stripper for 22 gauge stranded wire. It is easy to break the strands, so be gentle. We recommend stripping off 0.125 inch at a time.

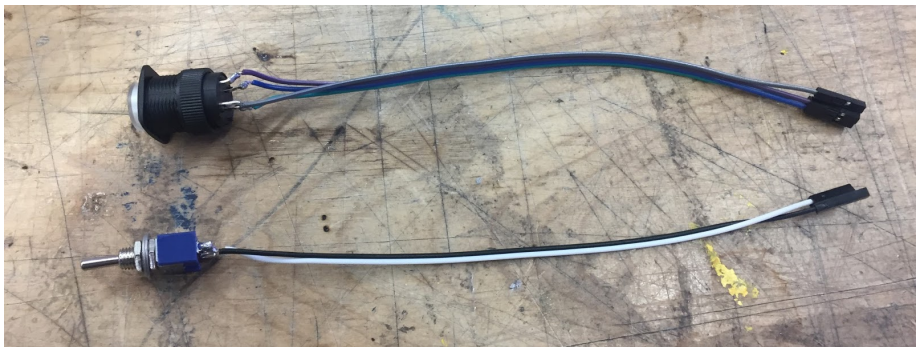
Comb the bare strands to a neat look, twist them slightly if needed. Use the soldering iron to tin the bare ends of each wire.



Two zip conductors stripped and tinned.

Locate the toggle switch. Solder the two conductor zip wire to the center and one end lead of the switch. The end lead should be the lead that is on the same side of the toggle switch as the mounting tab (the slot running down the screw threads of the switch).

Locate the LED push button switch. Solder the four conductor zip wire to the four leads.



Zip wires soldered to switches.

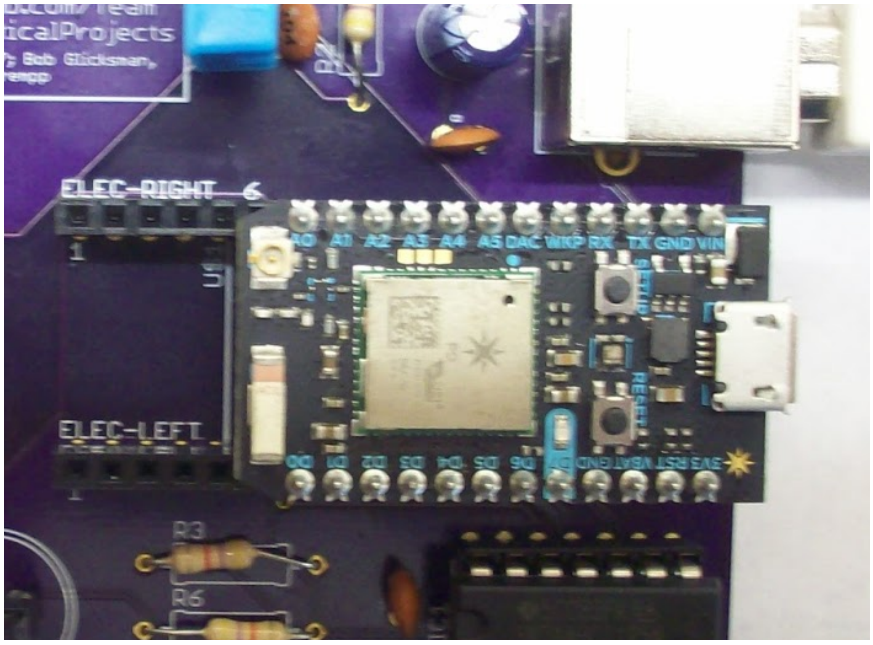
Testing

Carefully inspect all solder joints; a magnifier will help. Make sure each solder joint has enough solder on it. Make sure that none of the solder joints has bridged over to another solder joint.

At this point you should have already set up your Photon and tested it with the BlinkLED app from Particle.io.

Mount the Photon into its socket with the micro USB connector facing forward (the side of the board where the large USB connector is located). Make sure that pin 1 of the photon is as far

forward on the connector as it can go, When the Photon is installed in its socket, there will be 6 empty pins on each side at the back of the Photon. These pins are not used (they are for use with the Particle Electron 3G module).



Photon mounted on PCB for operation.

Connect USB power to the large USB connector on the PCB - not to the connector on the Photon. Your Photon should power up, connect to your Wi-Fi, and run the BlinkLED app. This test shows that the WLD PCB is able to power the Photon.

Your PCB build is now complete. You can move on to the next module build process.