Adapter Shield Build Instructions

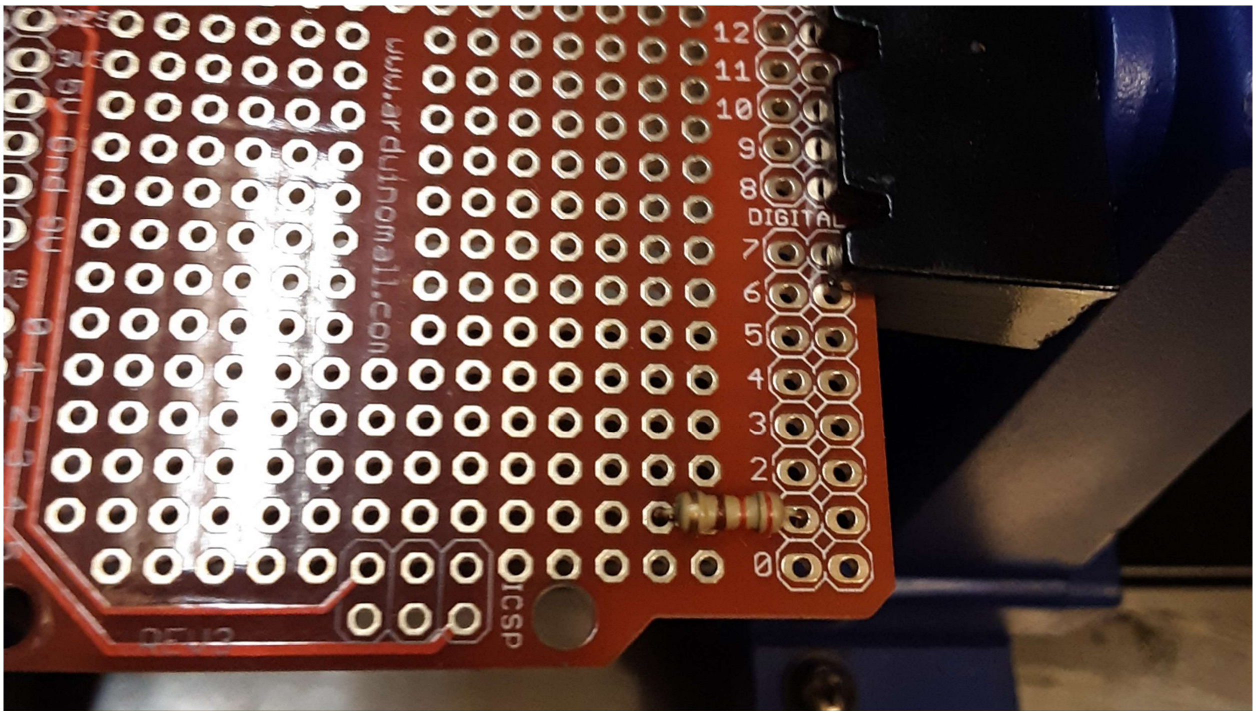
Contents

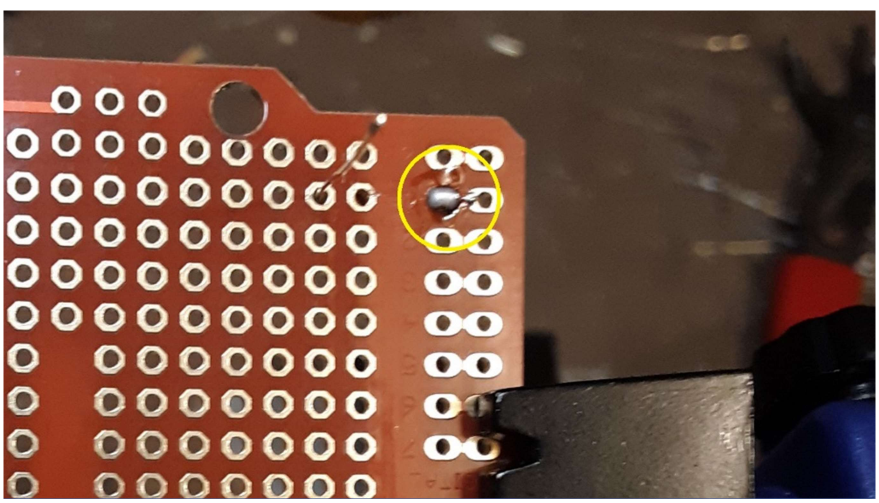
Part A – Bluetooth Connector

Bluetooth modules are generally powered by 5VDC power, but the RXD port can only handle 3.3VDC, so a voltage divider is required to protect the module from the Arduino’s 5VDC signal. This is the hardest part of the build due to the small size of the jumpers needed in order to power the Bluetooth module.

Step 1 – Place 220 Ω resistor

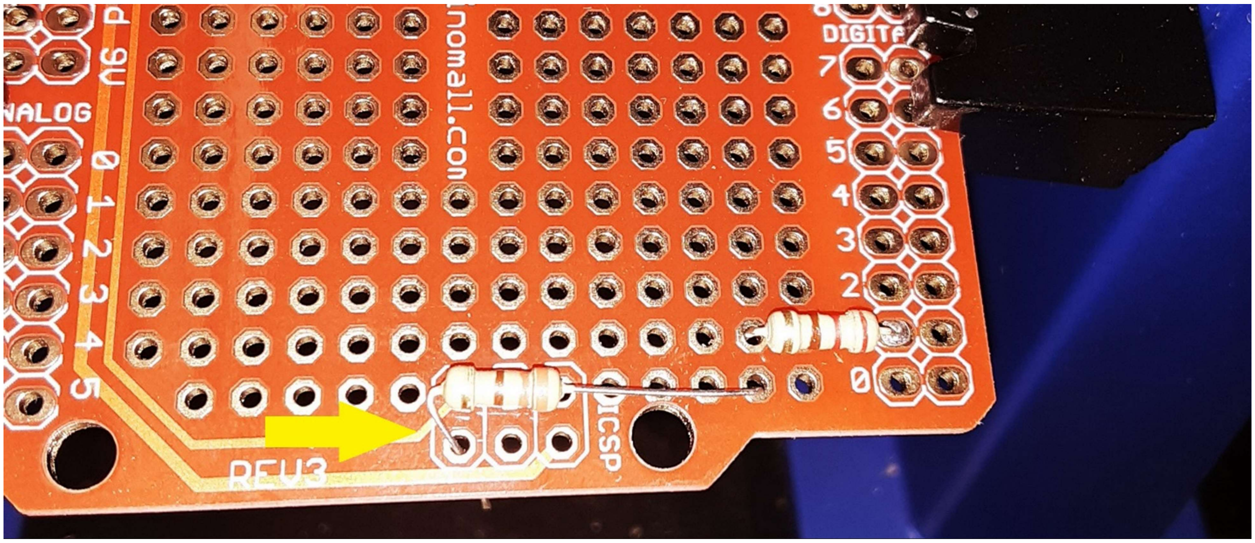
Place 220 Ω resistor at pad 1 and solder in outboard lead <u>ONLY</u>. A piece of tape will hold it in place when the board is flipped over for soldering.

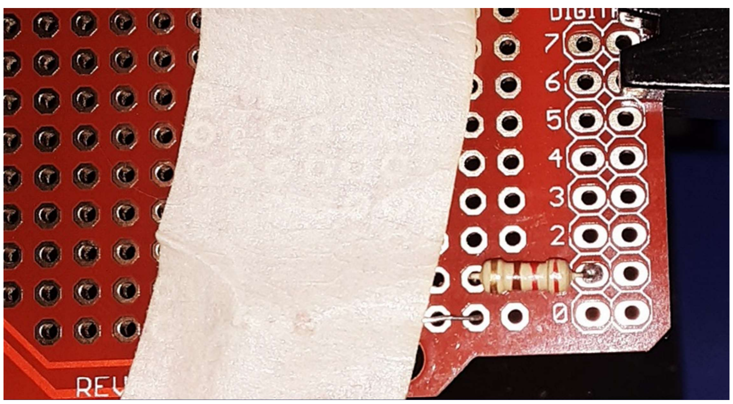


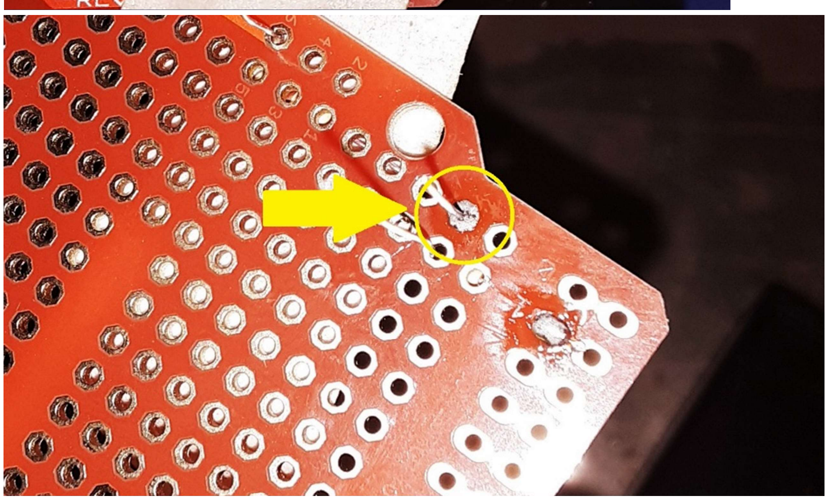


Step 2 – Place 330 Ω resistor

Put 330 Ω resistor in place and solder outboard lead ONLY. Bend the inboard lead carefully to align with the ground connection hole. Use tape to hold resistor in place for soldering. This solder connection is mechanical only in order to hold the resistor in place.

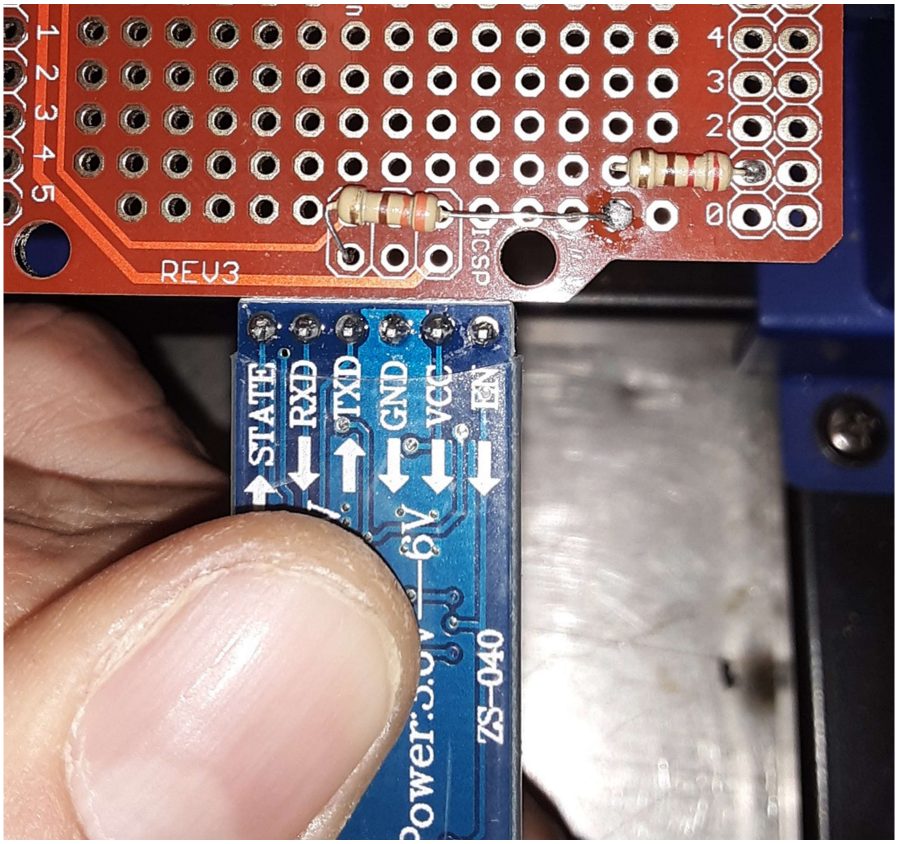






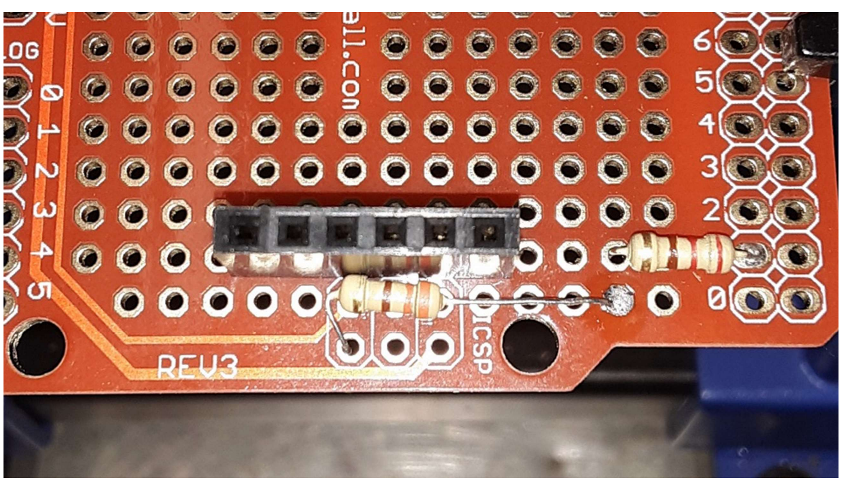
Step 3 – Check pin alignment

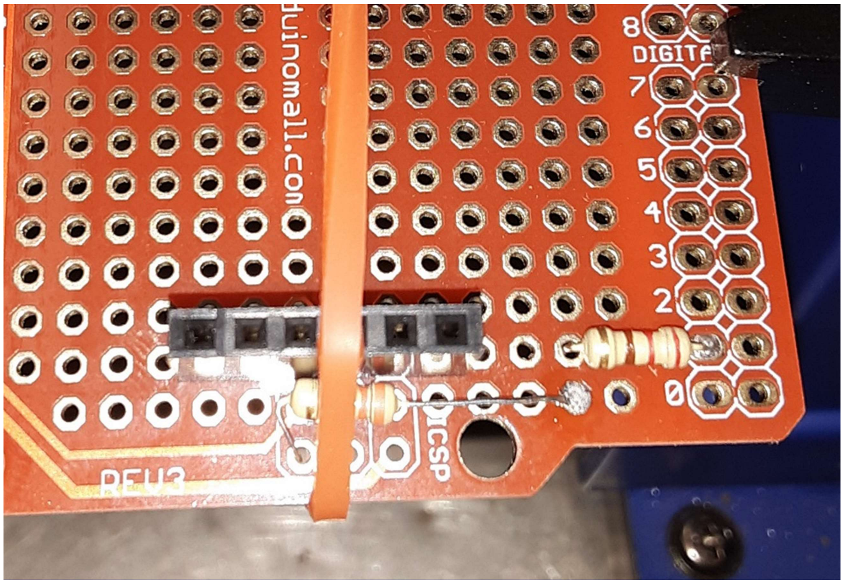
Check pin alignment of the Bluetooth module to verify header placement (next step). An HC-05 module was used here, but pin outs seem to be standard.

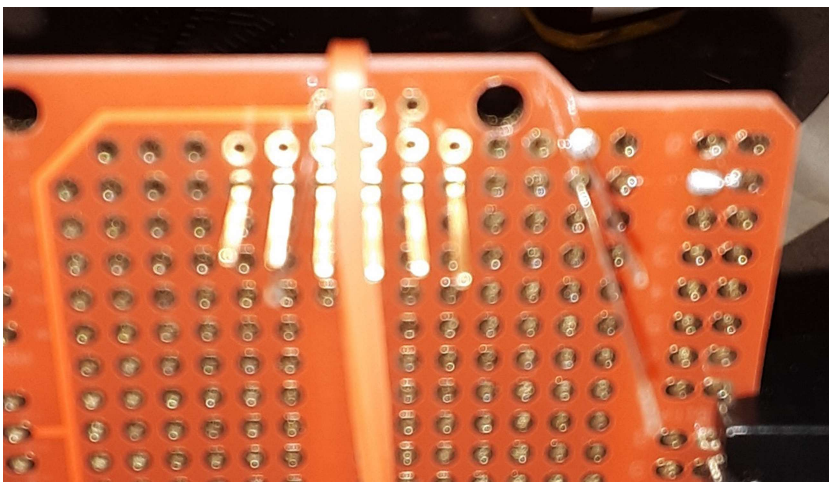


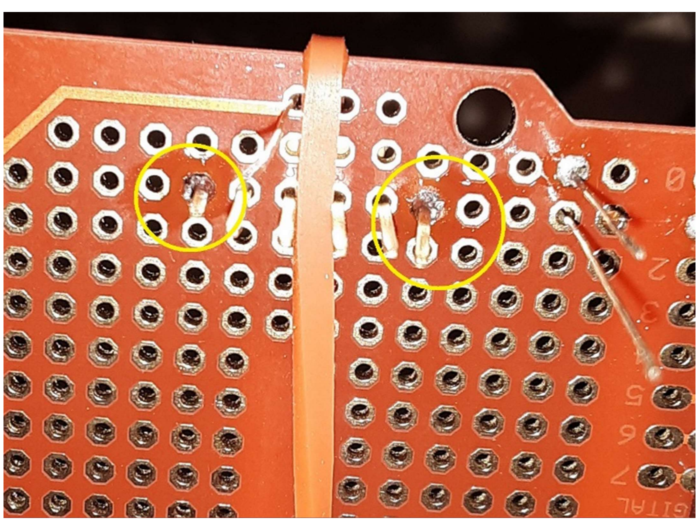
Step 4 – Place header

A rubber band is very handy to hold the header in place for soldering. Solder the outer most pins <u>ONLY</u>. These are only mechanical connections, no wiring will be connected here.



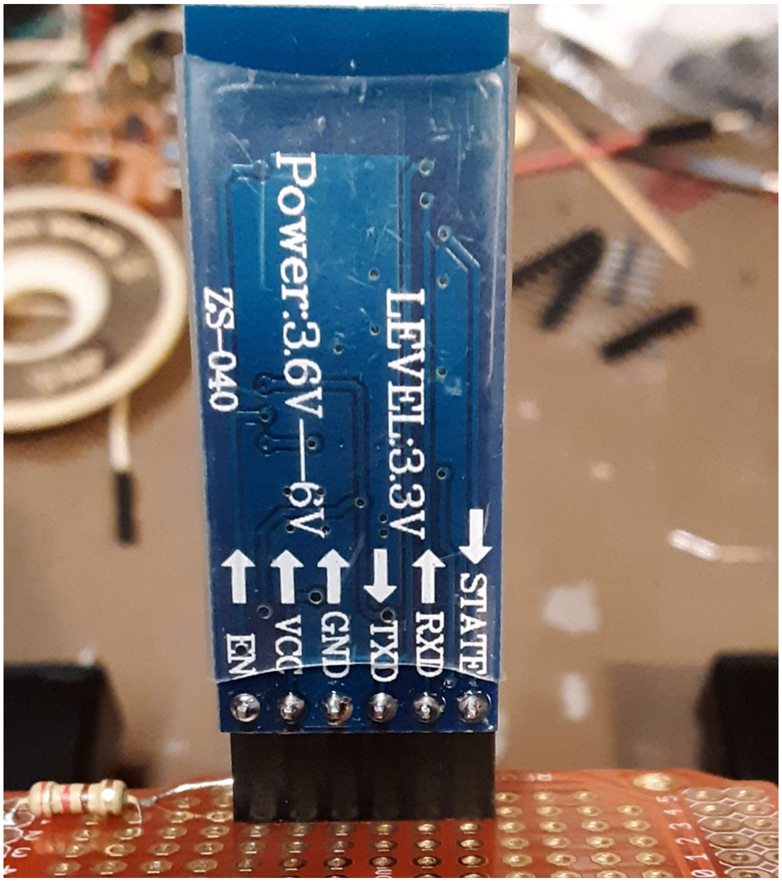

8
7
6
5
4
3
2
0
ICSP





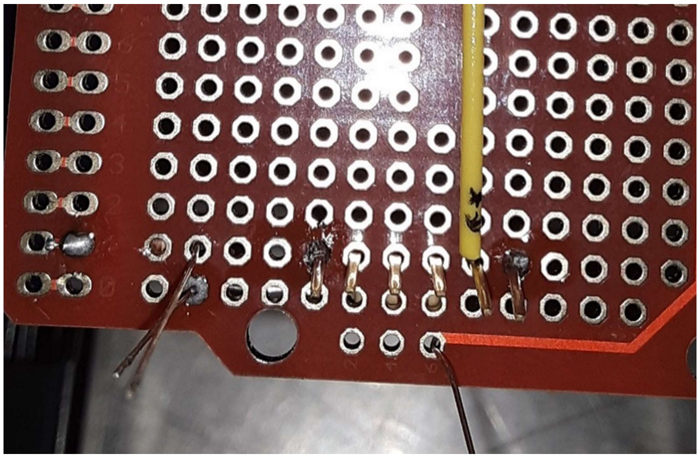
Step 5 (optional) - Alignment check

You can place the BT module in the header now to double check alignment.

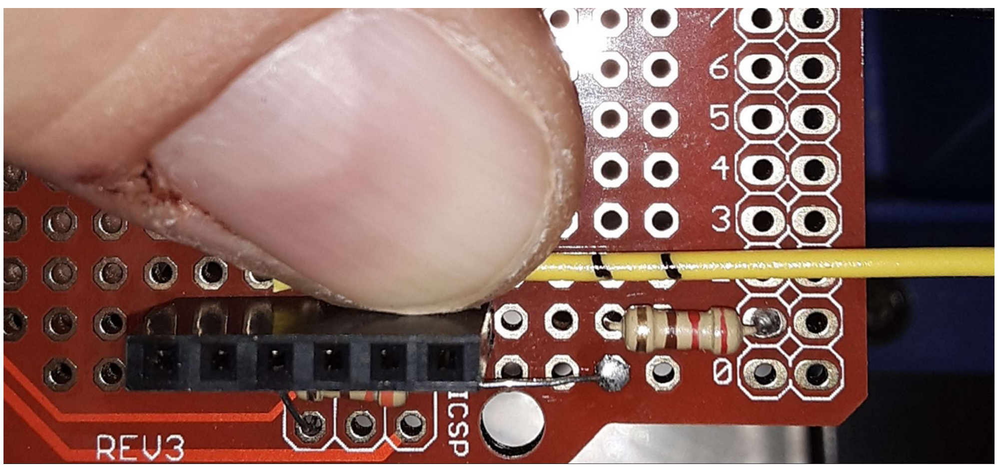


Step 6 – Connect Jumper wire

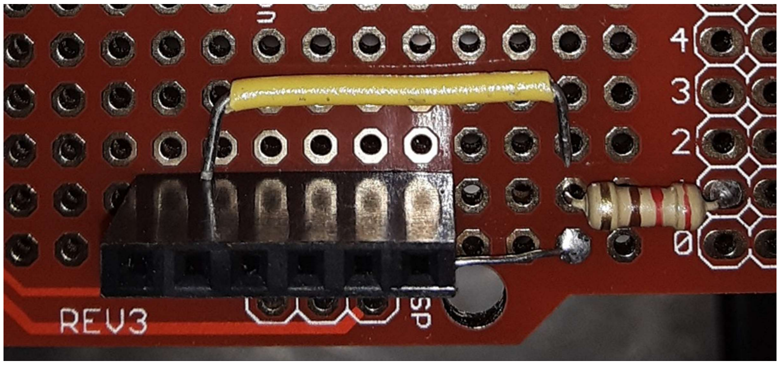
Size, place and solder a wire from the RXD pin over to the voltage divider. This picture is on the bottom side, measuring the amount of wire to strip in order to connect to the header pin. The first mark is at the hole the wire will come through, the second mark is where I stripped back to in order to provide clearance:



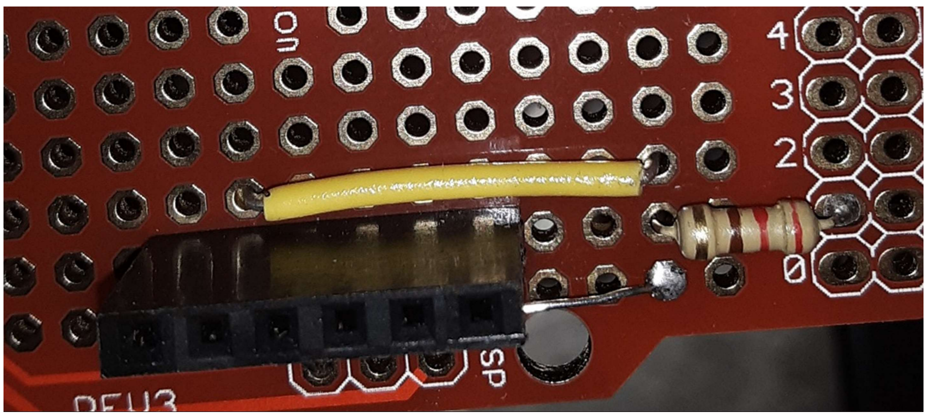
In this picture, the end of the wire was stripped, bent and inserted into the hole in order to allow measuring the size of wire required. The mark closest to my thumb is where I will strip to, the second mark is my cut line.

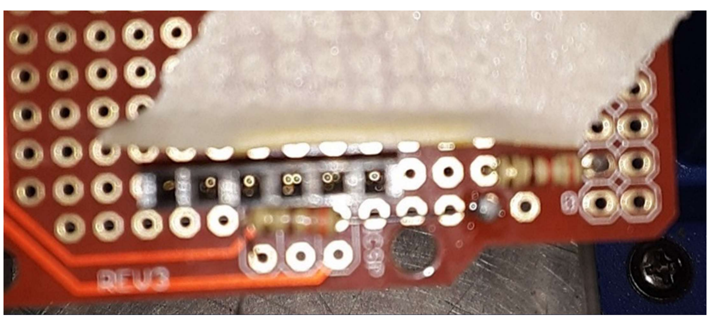


When cut, stripped, and bent the wire should look like this:

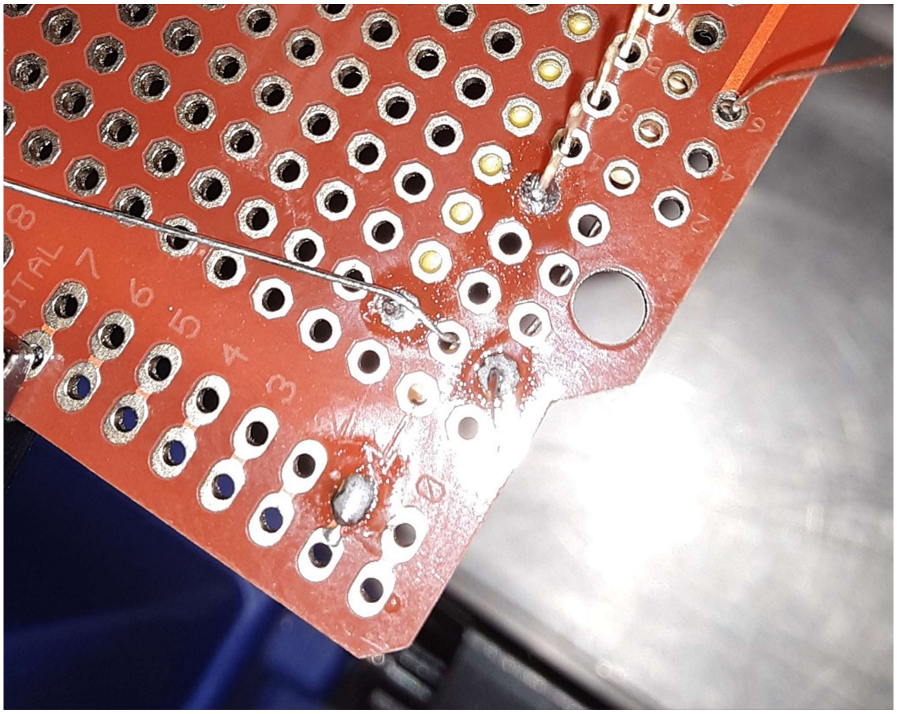


Insert the wire into the board and solder the end next to the voltage divider (the outboard end) in place. Tape again is useful for holding in place.

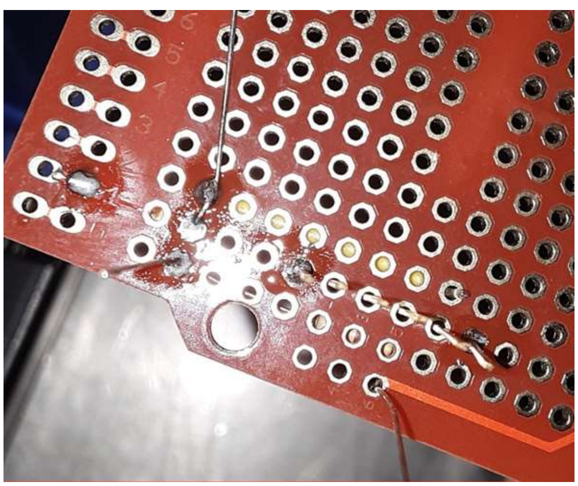




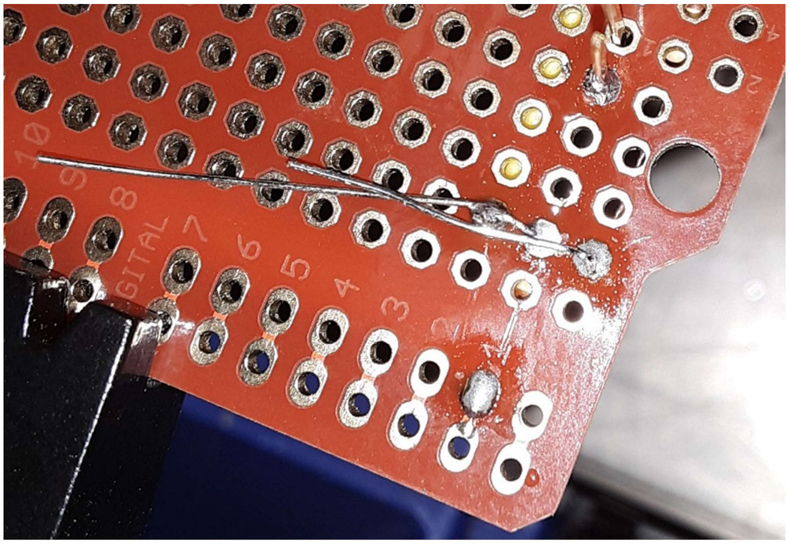
With the initial solder made, bend the leads from both resistors over to the wire and solder all wires together:



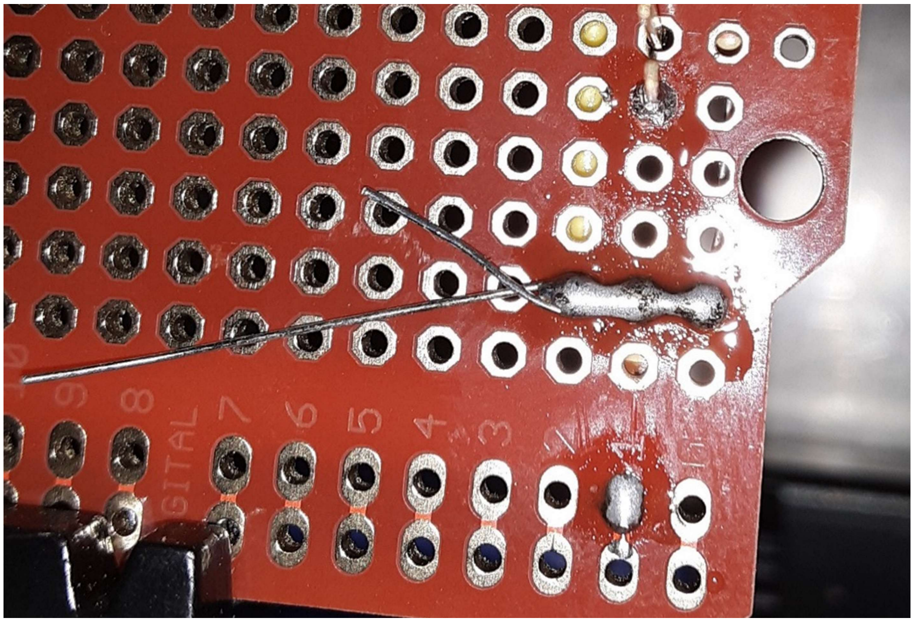
220 Ω ohm resistor lead bent to wire and initial solder made:

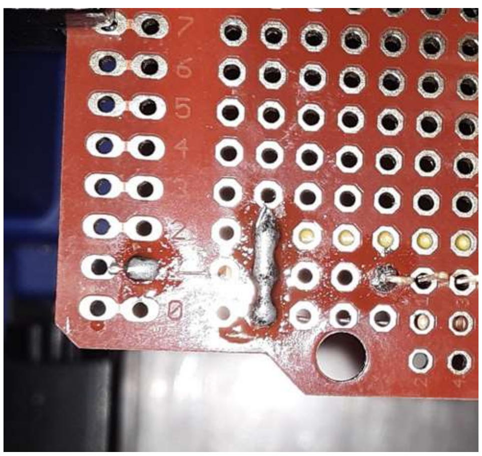


330 Ω resistor lead bent into place:



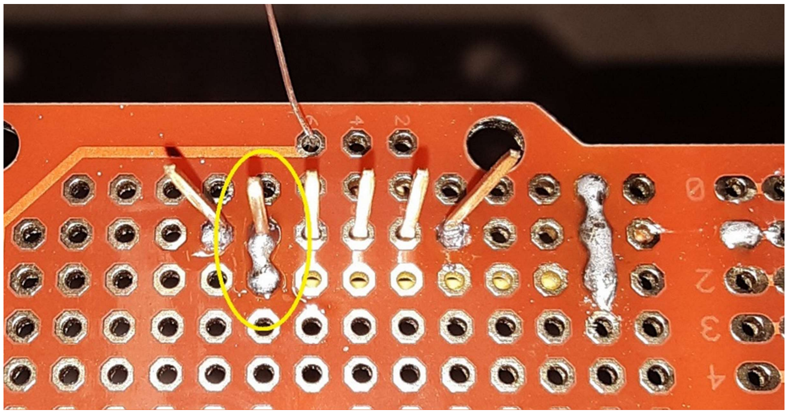
Final soldering at the voltage divider junction. Trim the leads to size after soldering:

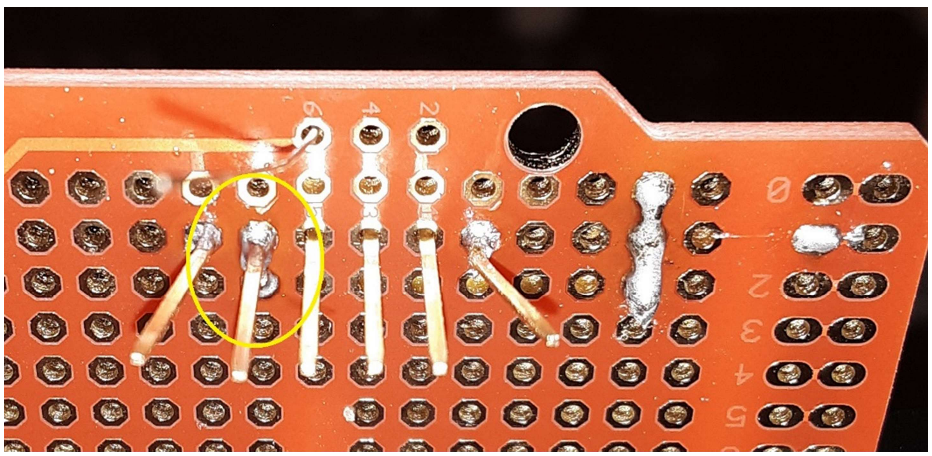




Step 7 – Jumper to Header connection

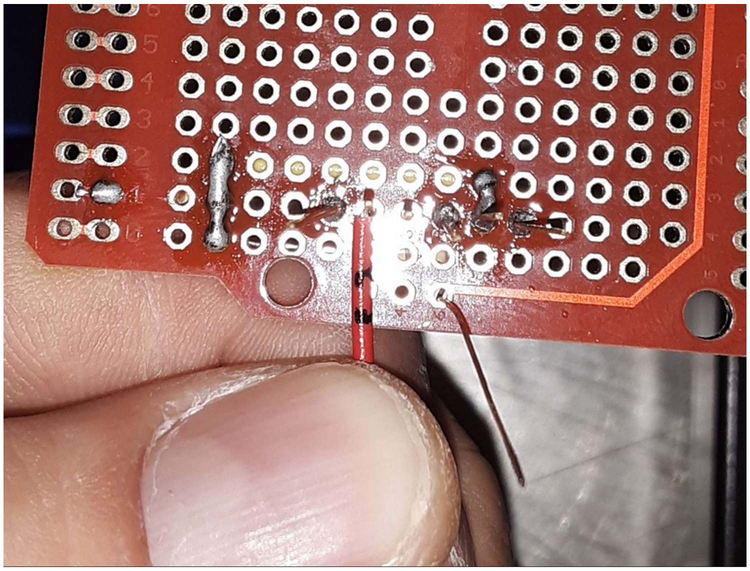
Connect data line to RXD connection on the header. Bend wire over and solder to pin. Solder header pin to the pad on the board at the same time. The end pins on the header can be trimmed to size to make soldering easier:

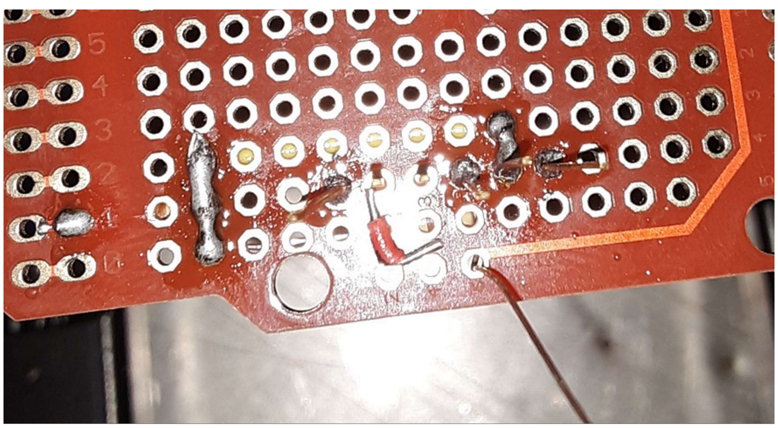




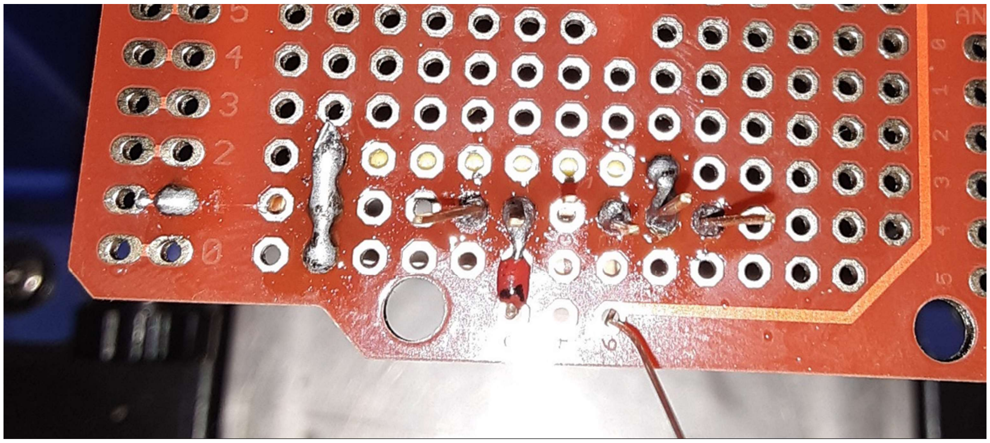
Step 8 – 5VDC Power connection

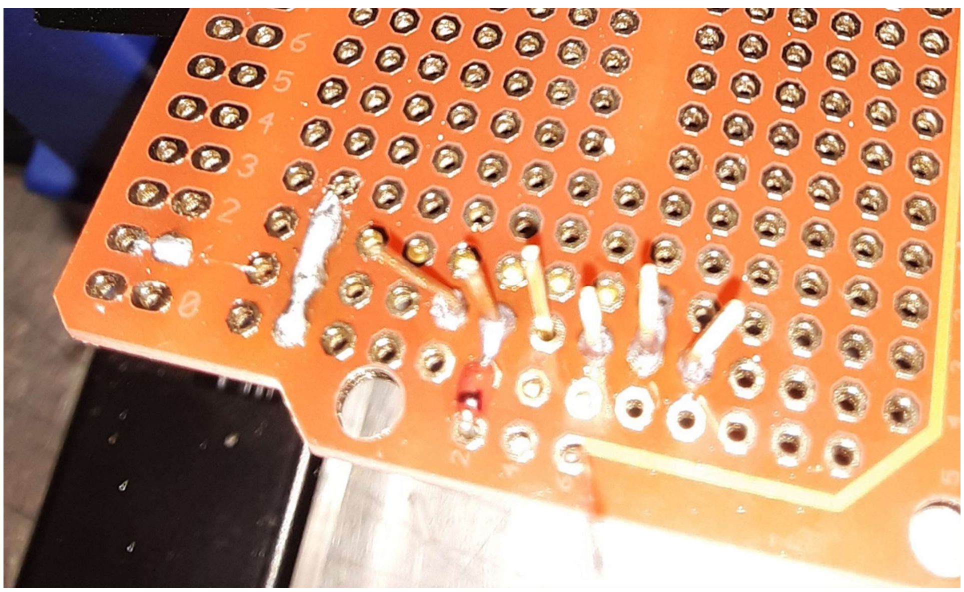
Connect 5 VDC power to the header. This is a hard part because it uses a *very* small jumper to connect the 5 VDC power pad on the SPI connector to the header pin. The mark closest to my thumb is the cut line, the next one in is the strip line:



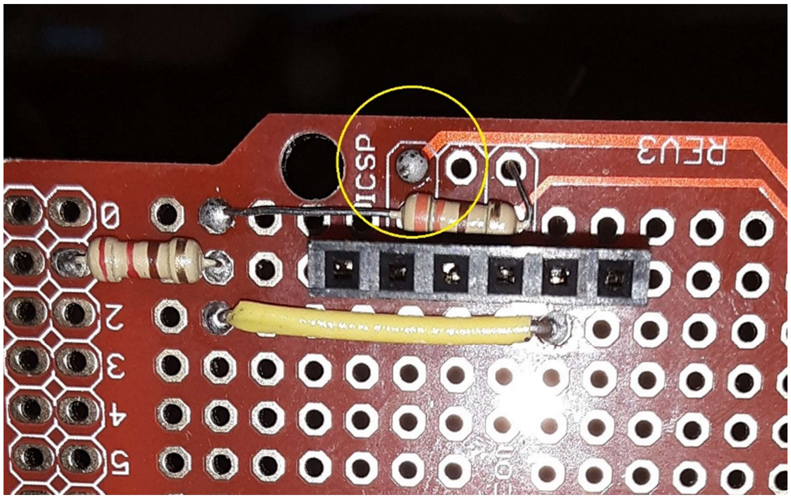


Place the jumper in the 5VDC hole on the board, then solder to the header pin. The easiest way to do this is to place the jumper in the correct hole, then solder the correct header pin to the board. Leave your soldering iron in the solder puddle, and gently push the tip of the wire into the puddle with the end of your solder wire, then remove the soldering iron from the puddle.



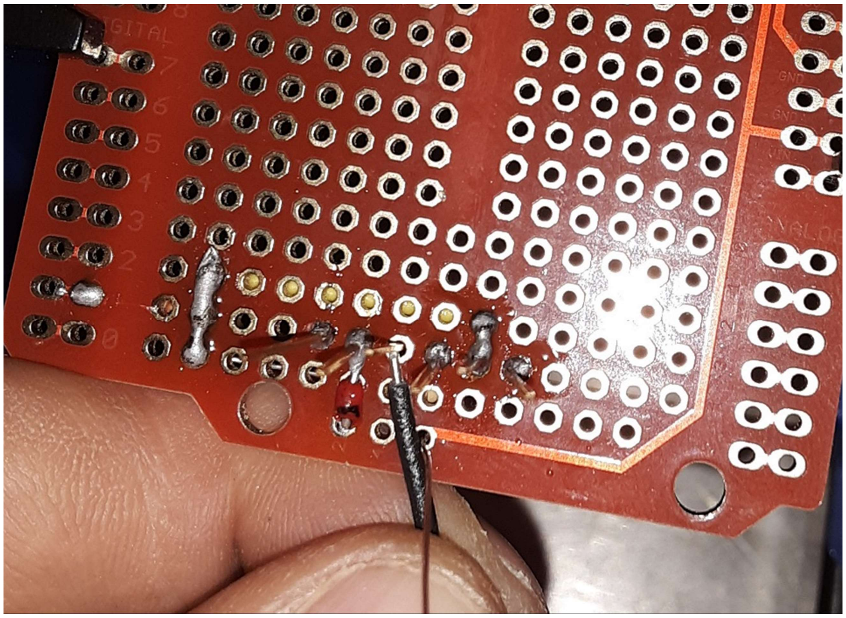


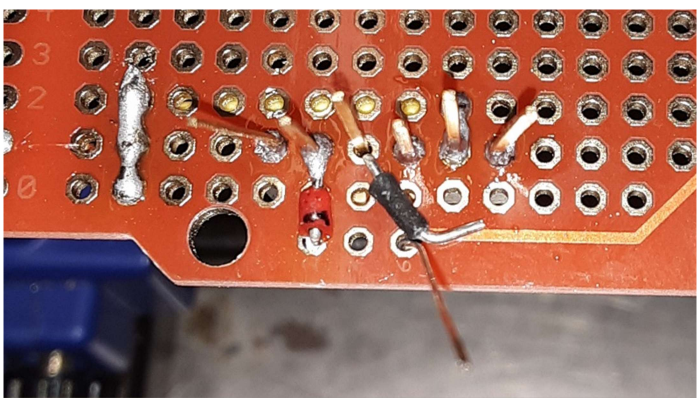
Now flip the board over and solder the wire to the pad on the board from the top:



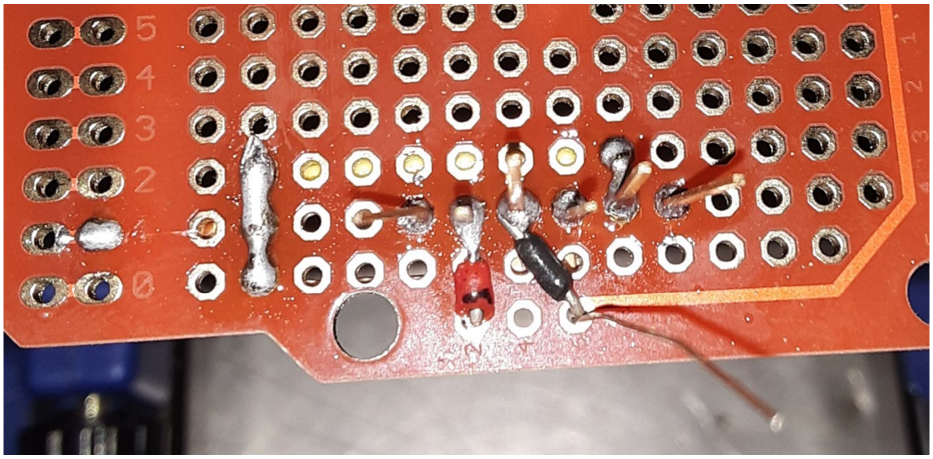
Step 9 – GND connection

Connect the header and voltage divider to ground. This is another hard step due to the small size of the jumper. Also, the jumper wire will go into the same hole as the 330 Ω resistor lead, so your wire can be no bigger than 22 AWG solid core.

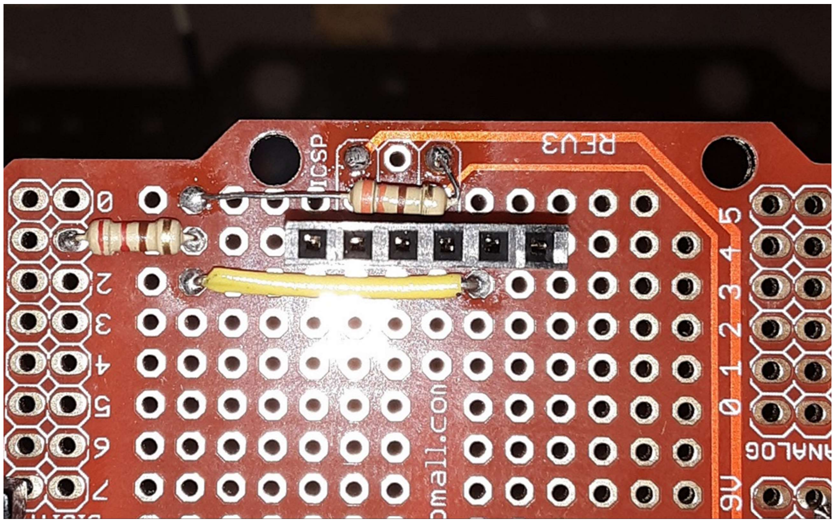




Solder the header pin and end of the jumper to the board. Use the same technique as used on the red wire, i.e. solder the pin to the board, keep the puddle liquid, then push the jumper into the puddle.

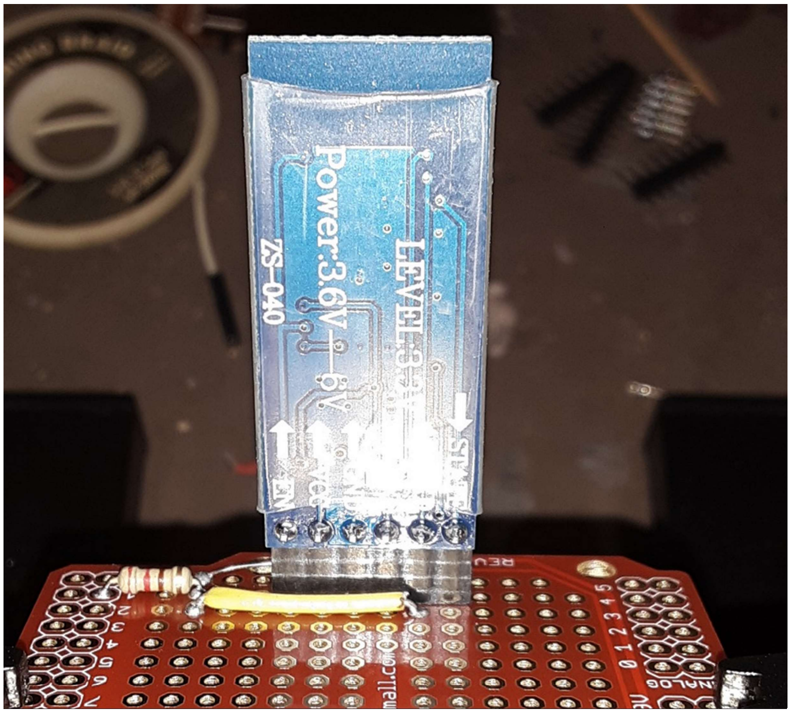


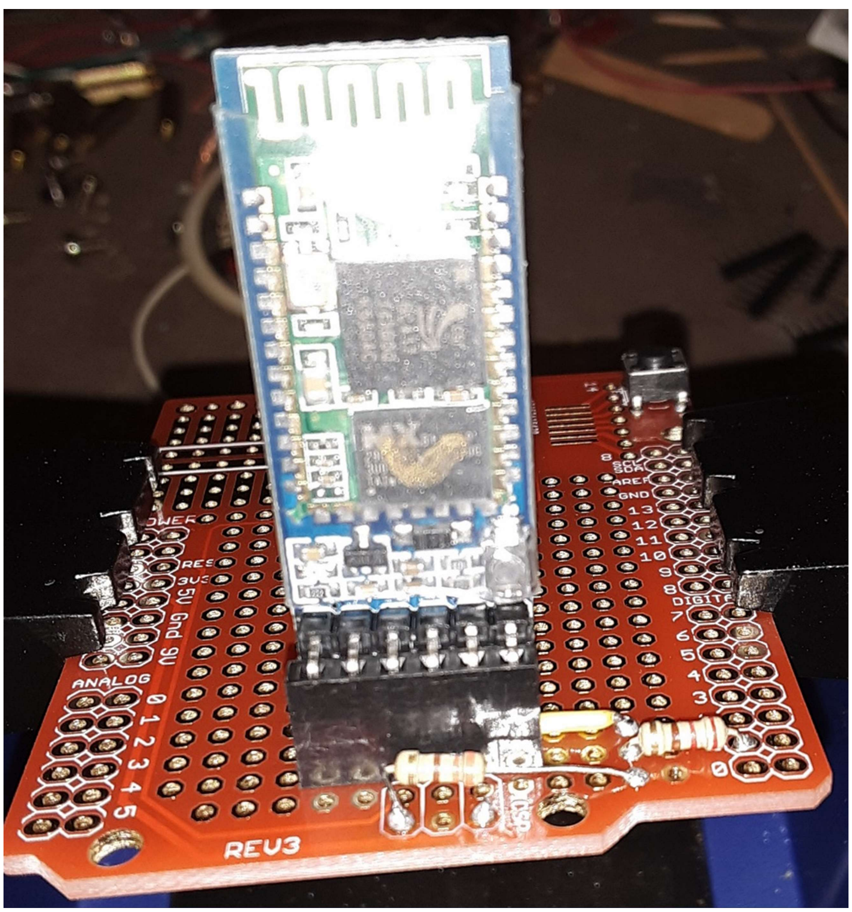
Flip the board over and solder the jumper and 330 Ω resistor to the board from the top. When done, flip the board over and trim the resistor lead on the bottom.



If the header you used was a stackable kind, like this one, trim the header leads on the bottom of the board to size.

Your Bluetooth header and voltage divider are now complete! That was the hard part!

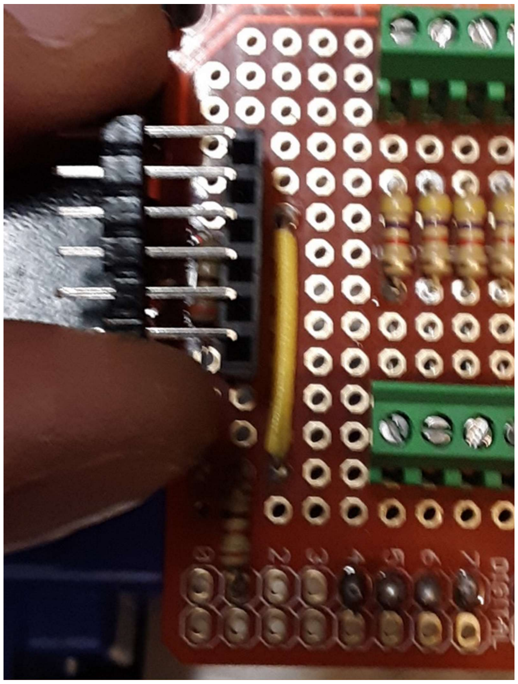


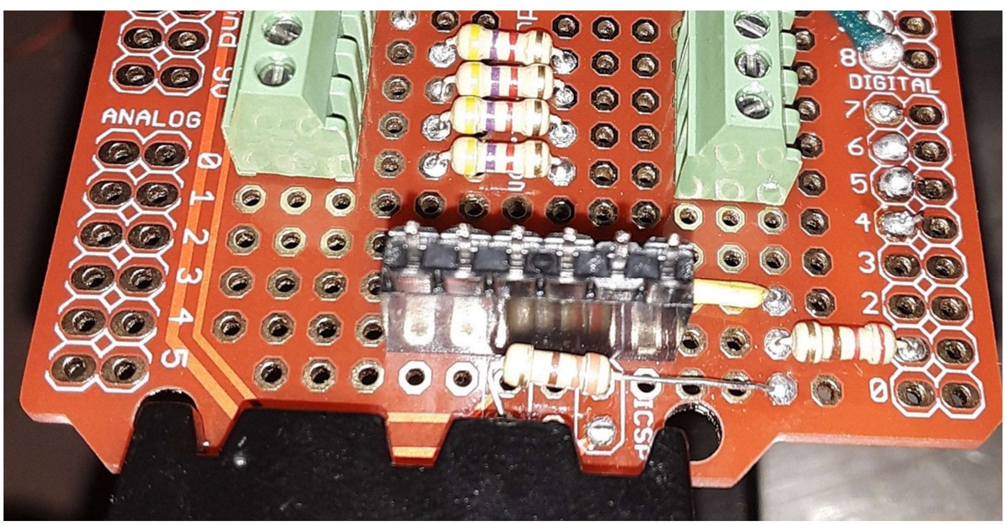


Part A1 – Testing the Bluetooth circuit

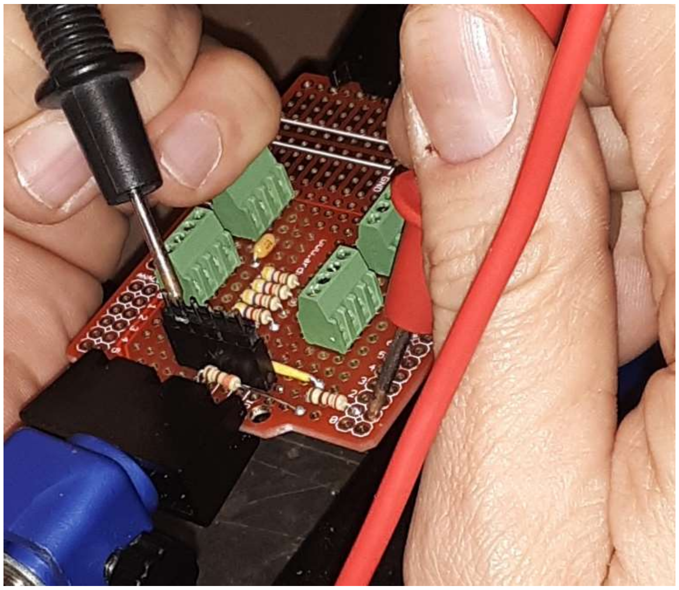
It's a good idea now to perform continuity checks on the circuit you just built. A cheap multimeter and a 6 pin header are all that are required for this.

Testing setup. Insert a 6 pin header into the stackable header as shown:



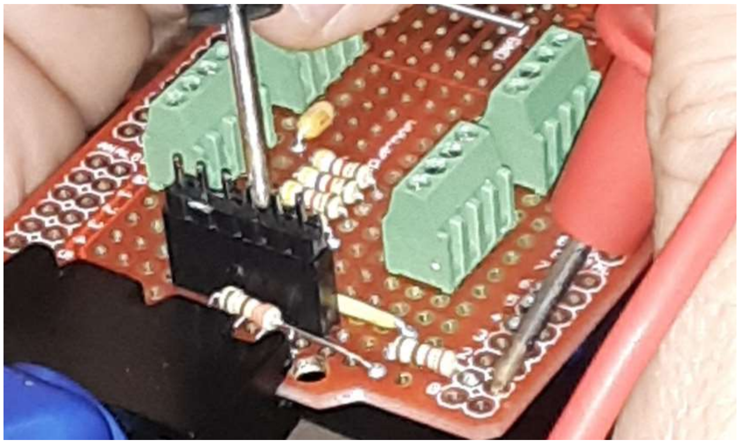


Check the resistance from pad 1 on the board to the RX pin:

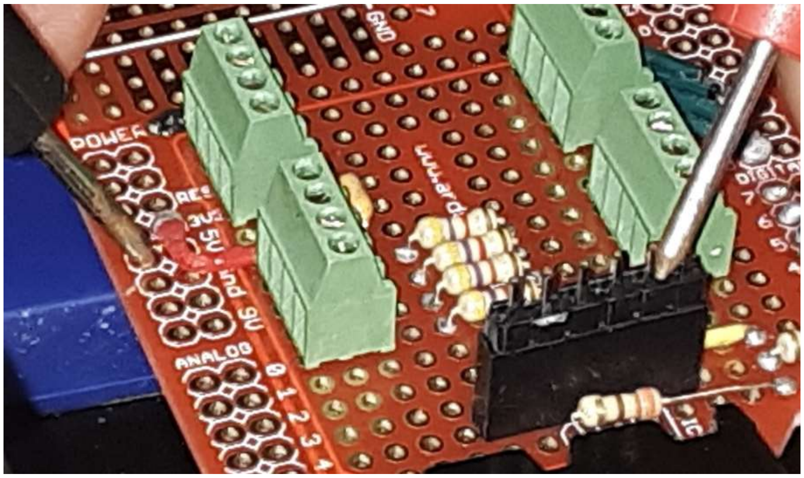


You should see about 220 Ω on the meter:

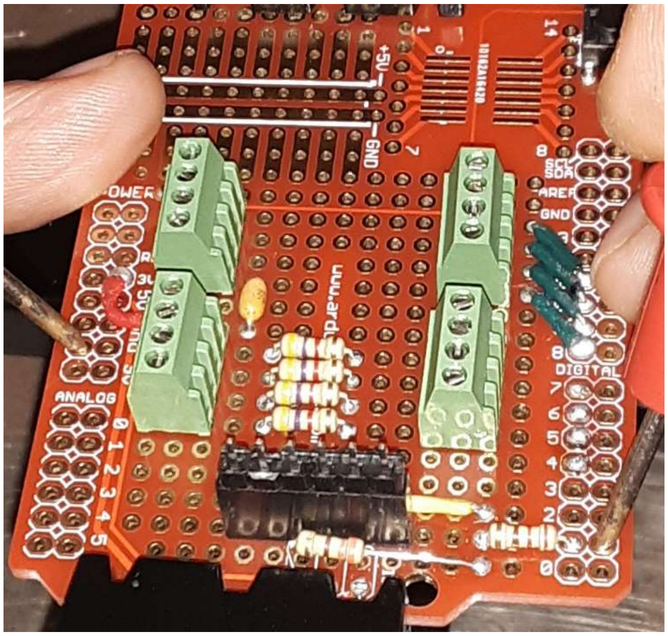
Now check pad 1 to the GND pin on the header. Your meter should read about 550 Ω:



Now check the 5VDC power pin on the header to the 5VDC pad on the board. There should be very little if any resistance:



Now check pad 1 to the GND pad on the board. Note that there are 3 GND connections on the board, but they are not all connected together. You will have to perform a continuity test between the GND pin on the header and all 3 GND pads to determine which one the board connects to.



Your meter should read approximately 550Ω again:

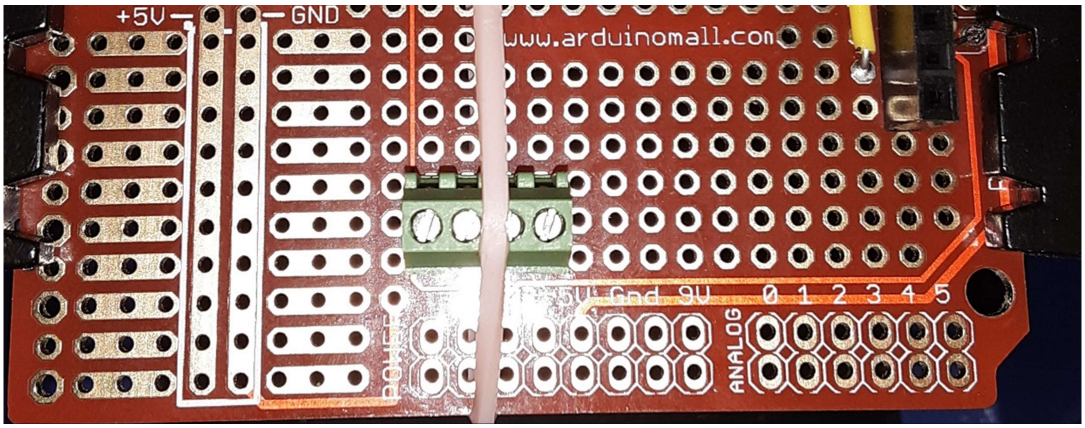
Those are the only connections that should currently be made. Set your meter to its highest resistance setting and check all other pads to make sure that all other readings indicate an open circuit (i.e. infinite resistance).

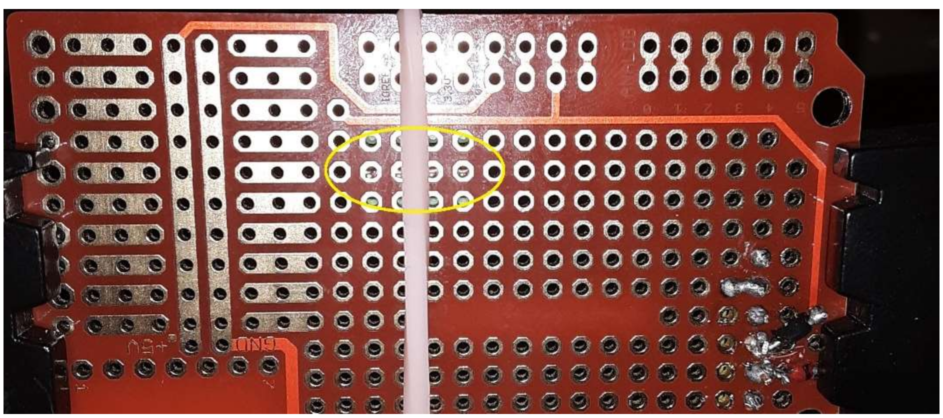
Part B – Scale Power Connectors

Every wire on each USB breakout board connects to a different connector in this design. These 2 connectors will have a Vcc and GND wire from each board connected to them. For DC power connections, I use Red for Vcc and Black for GND.

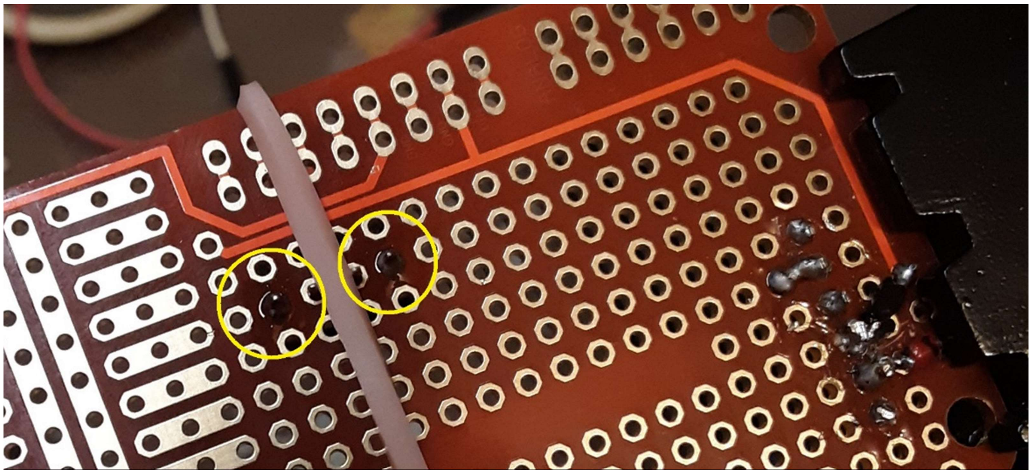
Step 1 – The GND connector

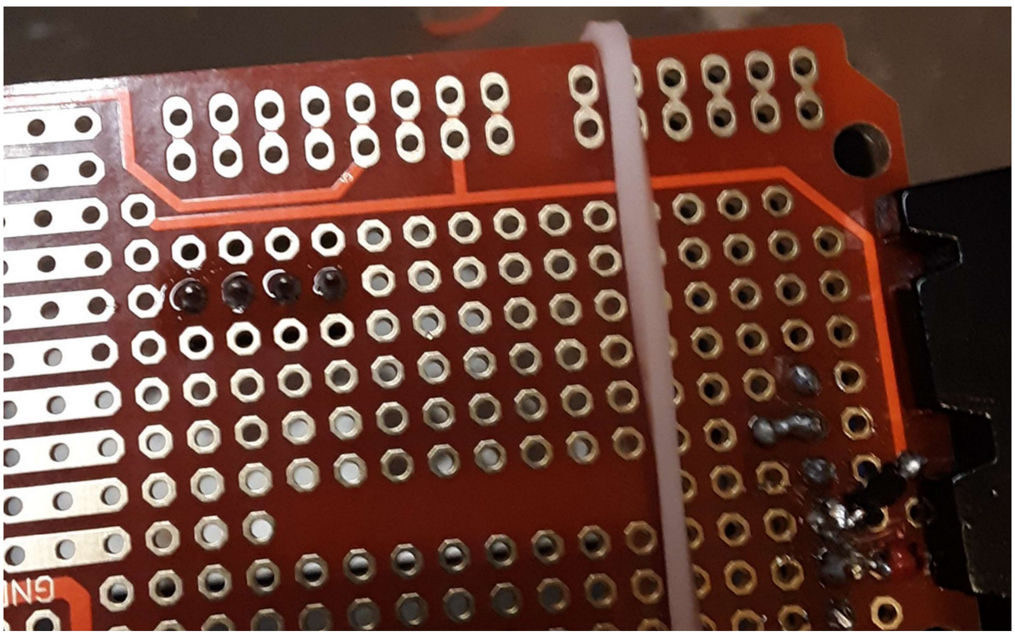
Place a 4-screw terminal block on the board as shown. A rubber band is very handy to hold the block in place.





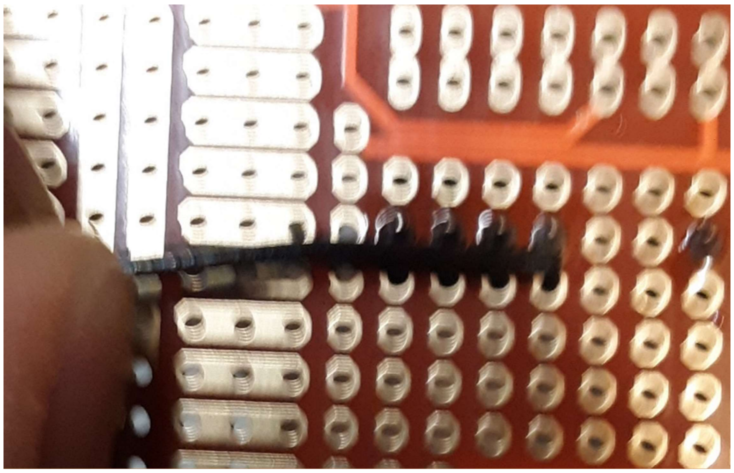
Solder connector pins to the board. It is helpful to solder the outer pins, then remove the rubber band to solder the inner pins.

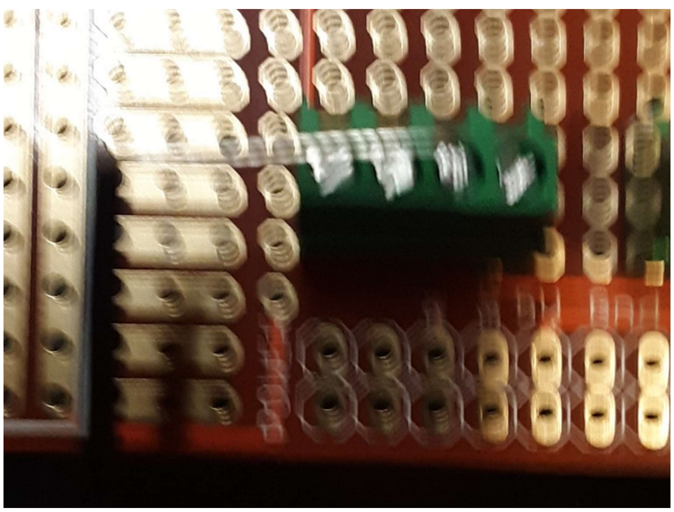




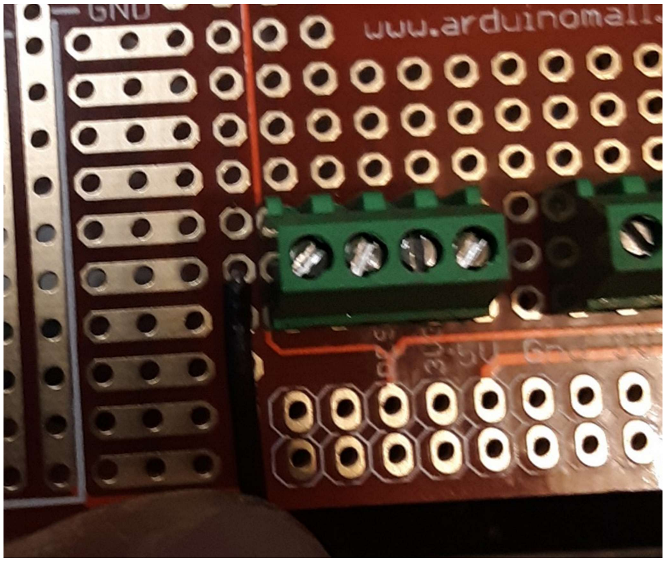
Now measure, cut and strip a jumper wire to connect all 4 pins to a GND point on the board. This particular board has one next to where the connector is located. I apologize for the blurry pictures, I didn't realize they were blurry when taking them.

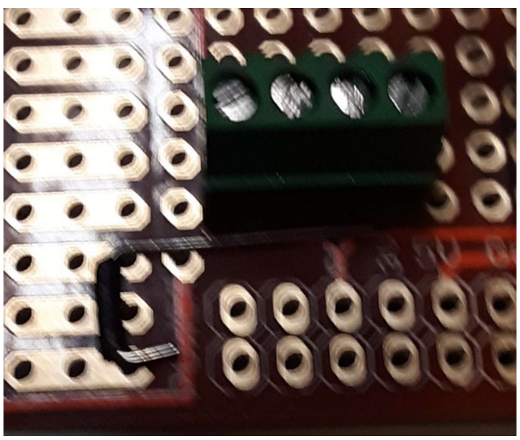
On the bottom of the board, measure and strip wire that will connect all 4 pins and reach the hole next to them, then put a 90° bend on the end. The wire will come down through the hole in the board next to the block on top and will connect all 4 pins together:

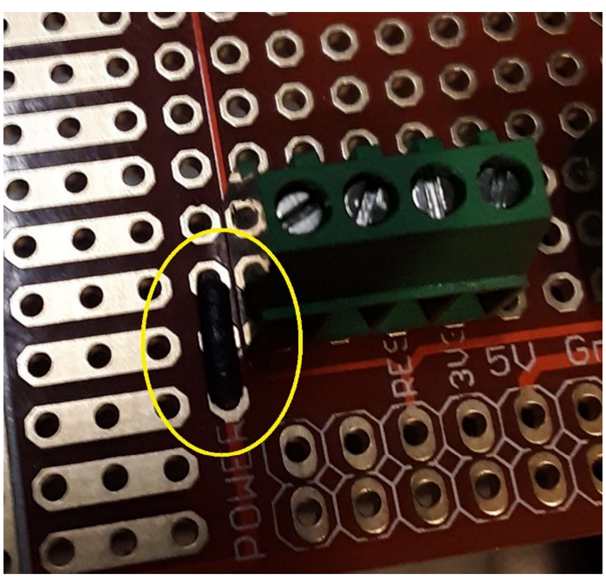




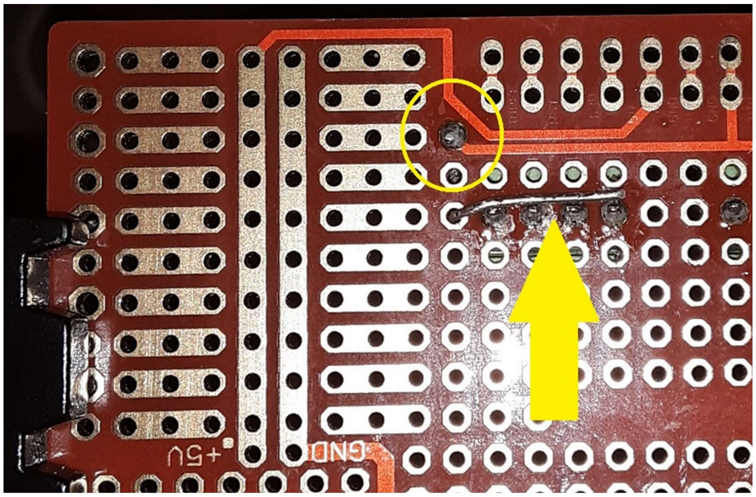
Now put the bent end into the hole next to where the pins are and measure to a convenient GND point. Cut and strip, then place a 90° bend in this end now:



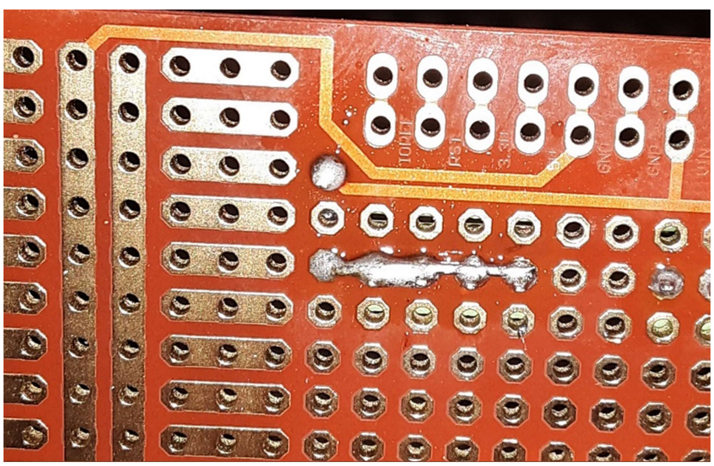




Place tape on the jumper to hold it in place, then turn the board over and solder the end of the jumper that is in the GND hole on the board. After that joint is made, bend the other end of the jumper over so that it is contacting all 4 pins of the screw terminal:



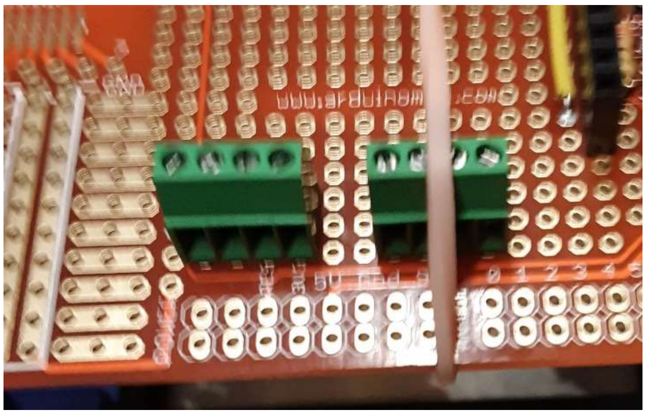
Now solder the end of the jumper to all 4 pins of the screw header:

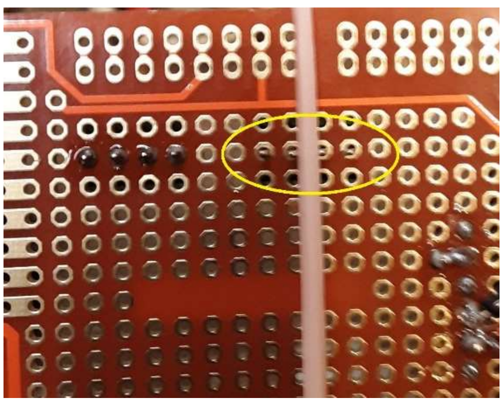


Step 2 – The 3VDC power connector

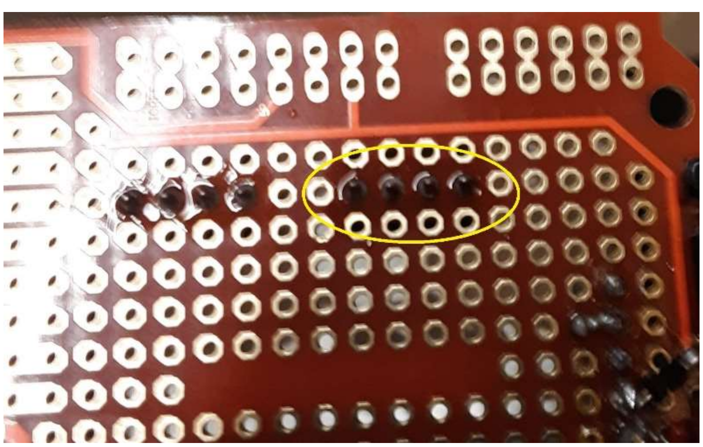
This step is pretty much identical to the GND connector, only the jumper will connect to the inboard 3VDC pad on the board.

Place a 4-screw terminal block on the board as shown. A rubber band is again very handy to hold the block in place while soldering:

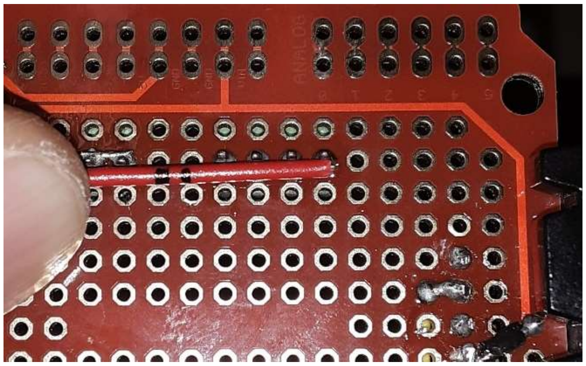




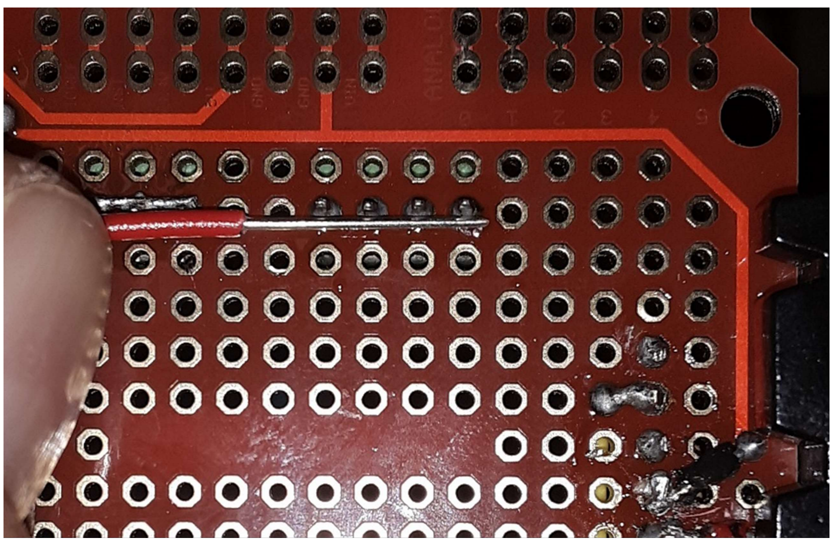
Solder all 4 pins to the pads:



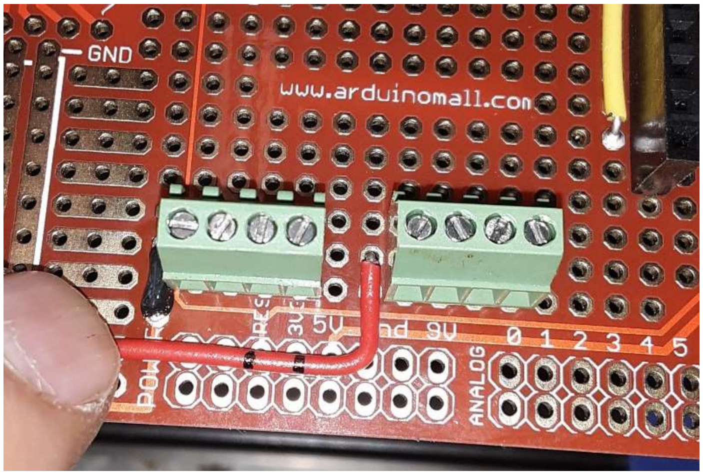
Now we will make another jumper similar to the GND jumper. On the bottom of the board, measure the length of wire needed to go from the hole next to the pins and long enough to connect all 4 pins together. The mark closest to the end of the wire is at the hole it will thread through, the other one is the amount of wire to strip:



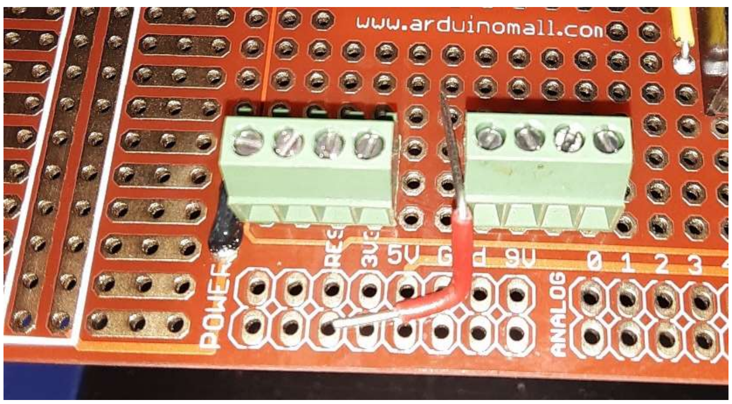
Stripped piece:



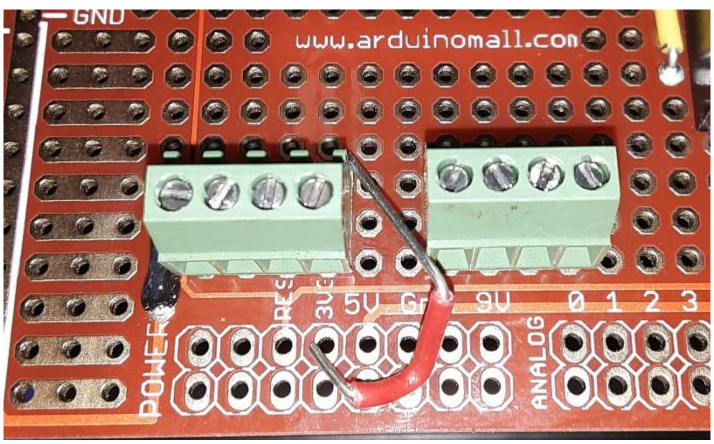
Place a 90° bend in the end of the wire and place it in the hole. Now measure to the inboard 3VDC power pad. The mark closest to my finger is the cut mark, the other one is the amount of wire to strip:



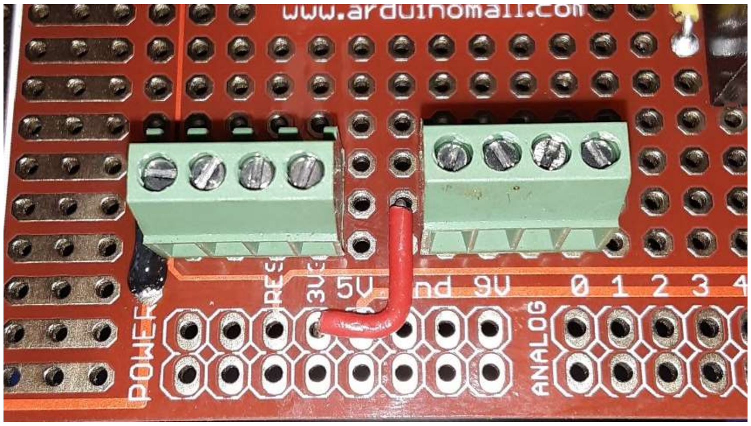
Jumper ready to place. The end just stripped will go in the inboard 3V hole, NOT the outboard hole:



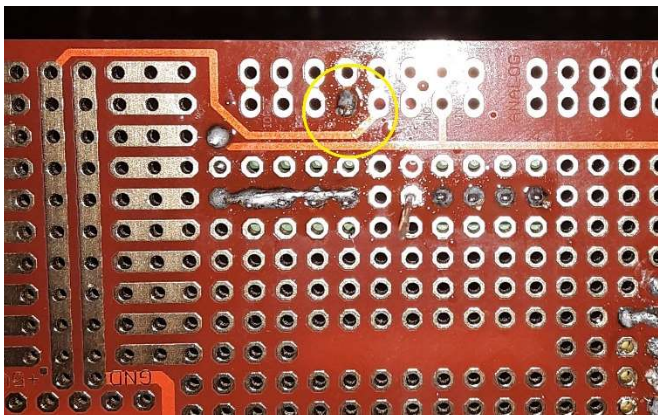
Now place a 90° bend in the end just stripped, pointing down in the same direction as the other end:



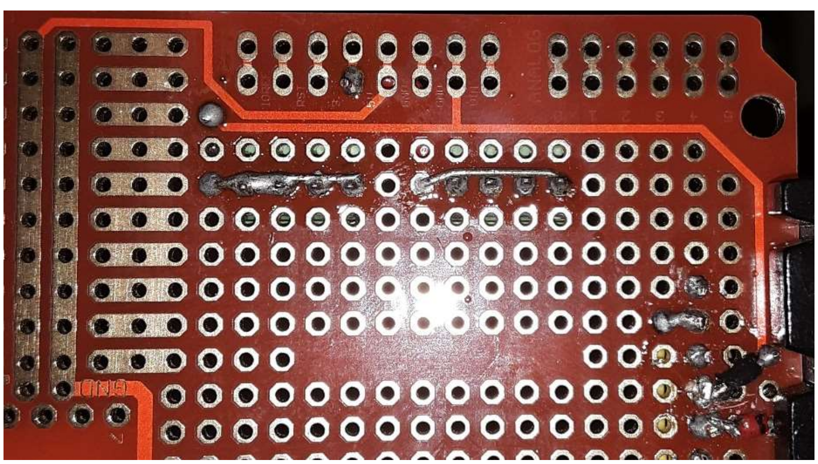
Then put it in place and hold down with tape:



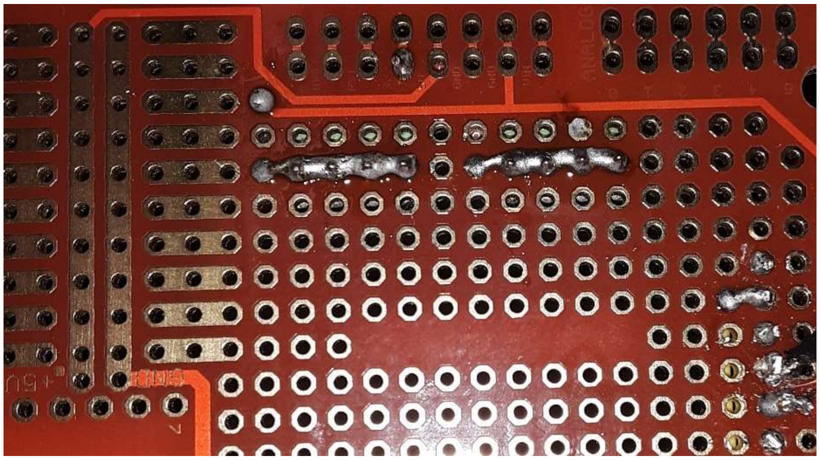
Now solder to the 3VDC pad:



Bend the long end of the jumper over so that it touches all 4 pins on the connector:



And solder it to all 4 pins:



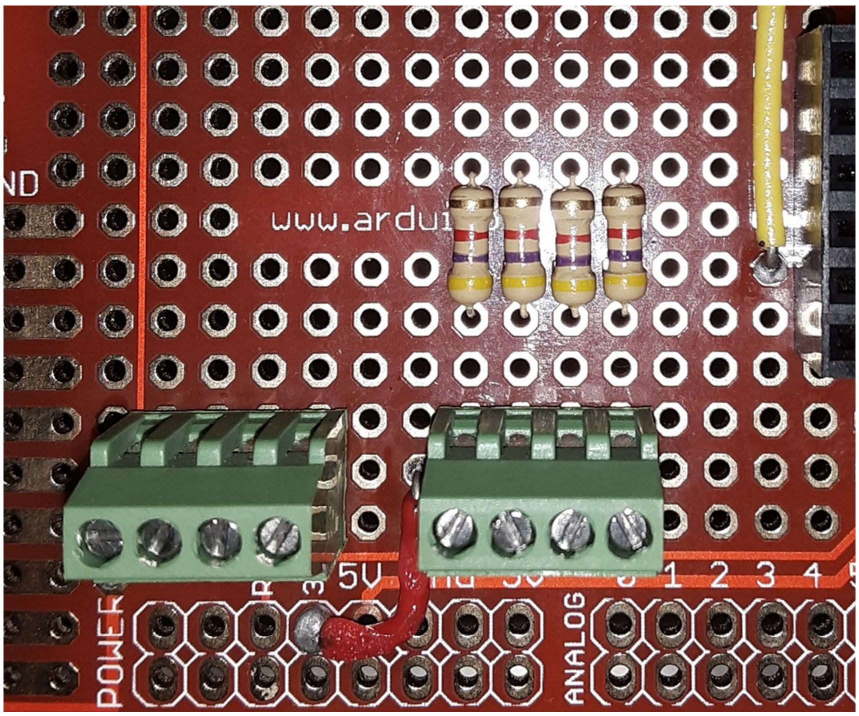
The power rails are now complete.

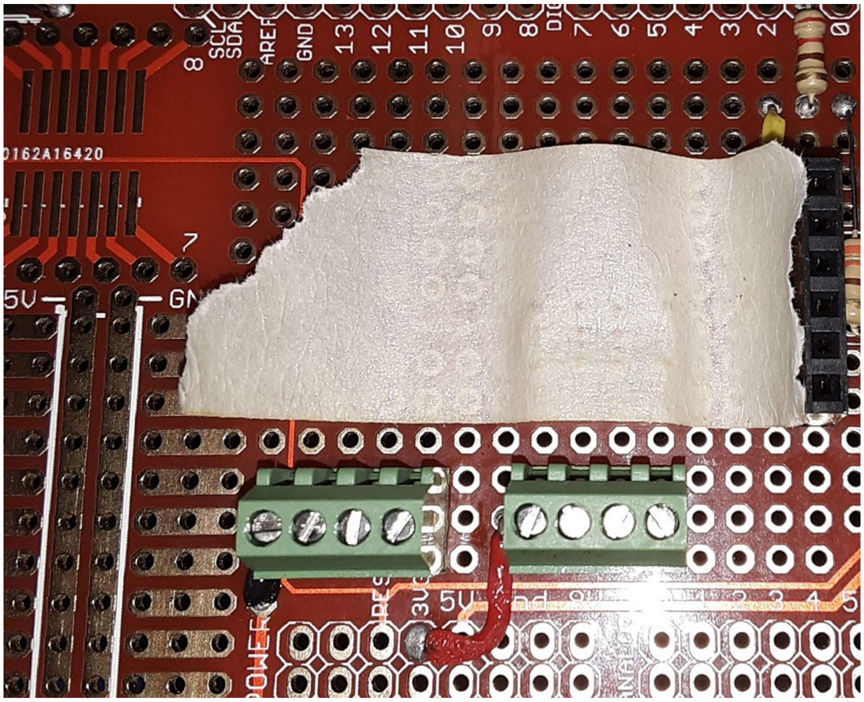
Part C – Pull-up Resistors

The scales require a voltage to be applied to the clock line. This voltage gets pulled low when the read head is sending a bit for the receiver to read. The scales operate at 3VDC, but an Arduino operates at 5VDC. To protect the read head, we will supply that voltage to the clock lines from the 3V power rail we just constructed, using 4.7kΩ resistors to limit the current draw.

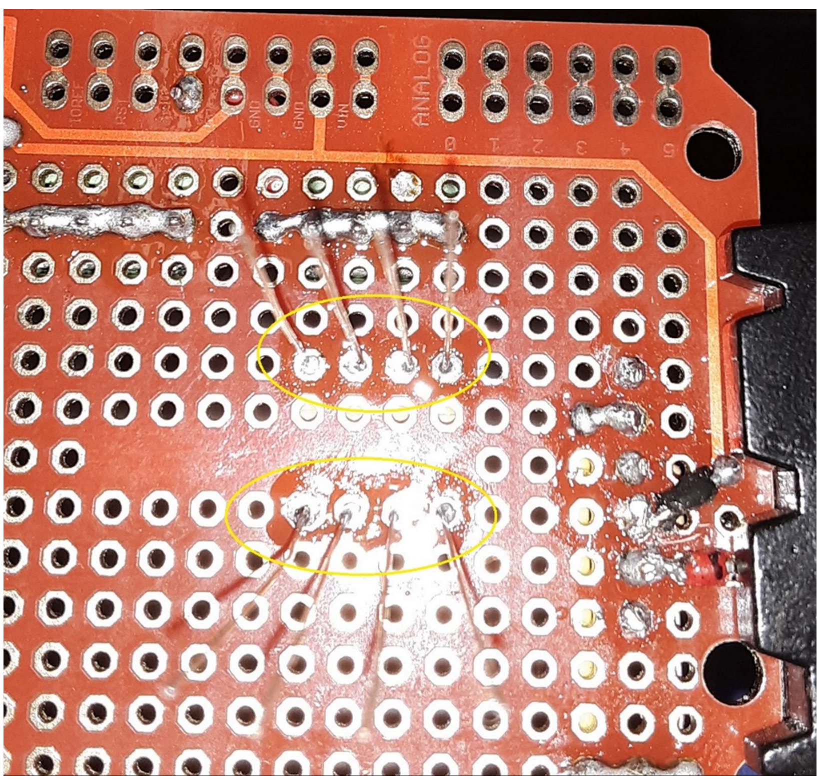
Step 1 – Place Resistors

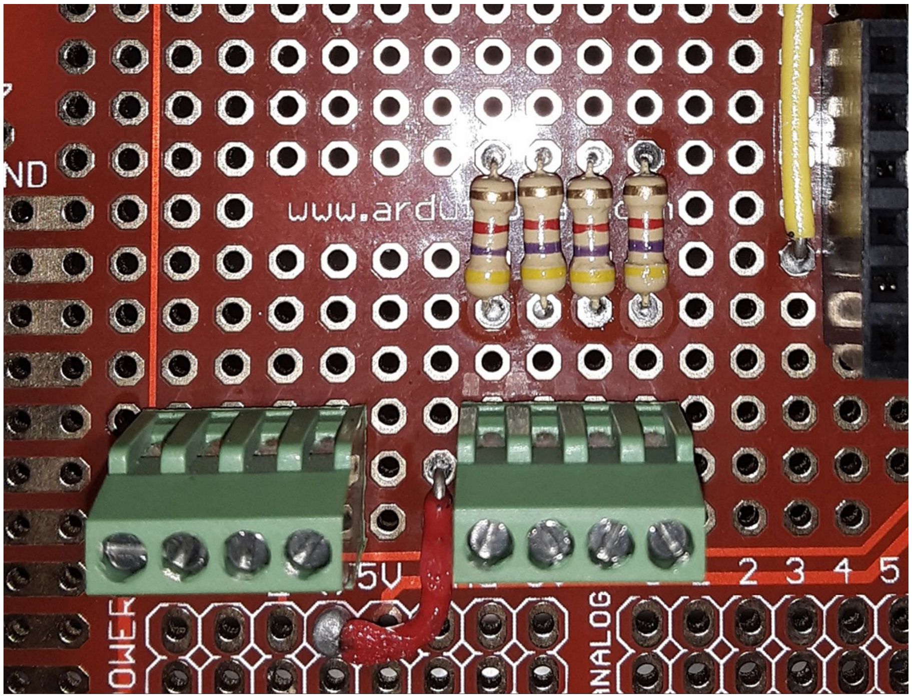
Place (4) 4.7kΩ resistors in the board as shown, then hold in place with tape:





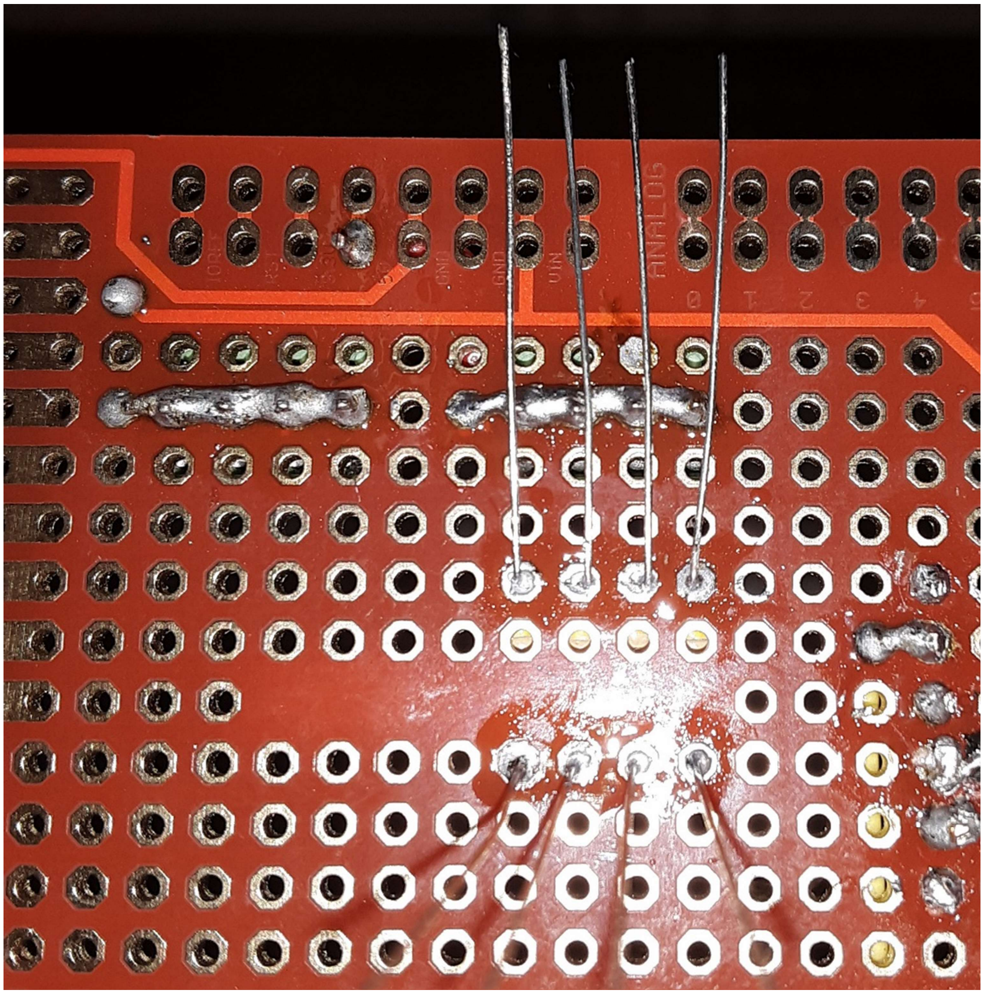
On the bottom side of the board, solder the leads in, but do NOT trim them yet. These joints only provide a mechanical joint.



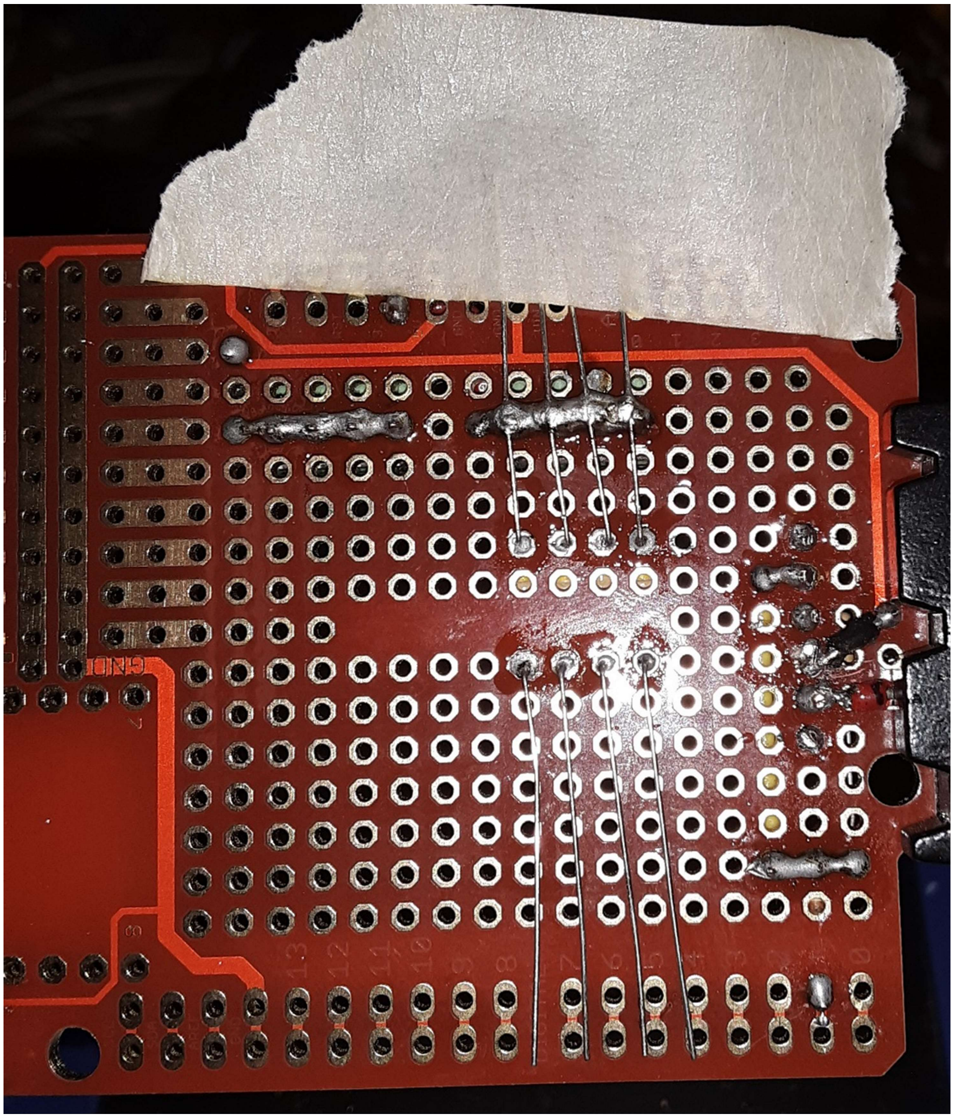

5V
ANALOG
2 3 4 5

Step 2 – Connect leads to power

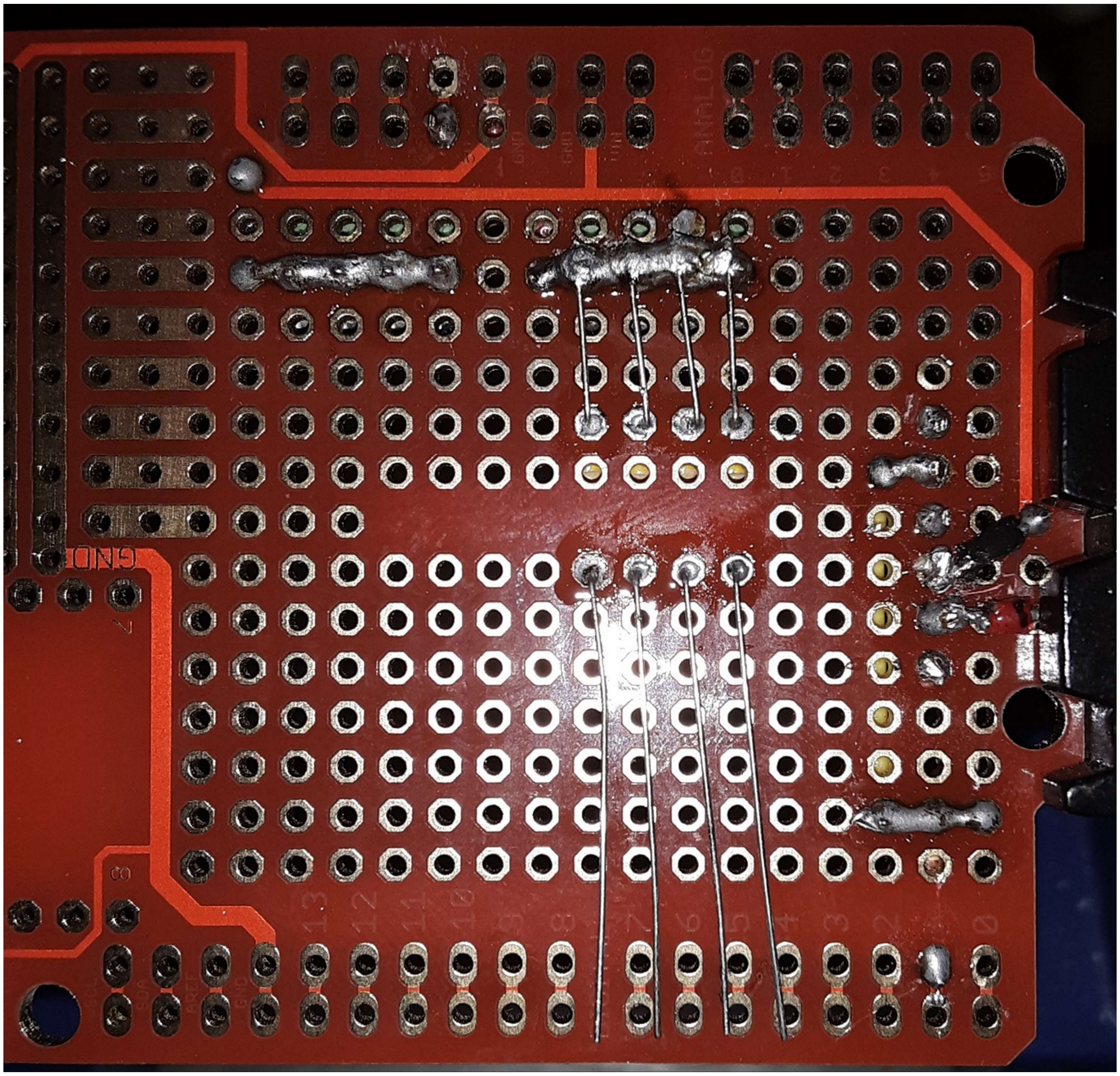
Take the leads that are closest to the 3V rail and bend them over to it, then solder to the 3V rail. Tape is handy to hold the ends of the leads down while soldering:



I bent the leads on the other end down, too, in order to make soldering easier:



Now trim the ends just soldered down to size. Do NOT trim the other ends yet!

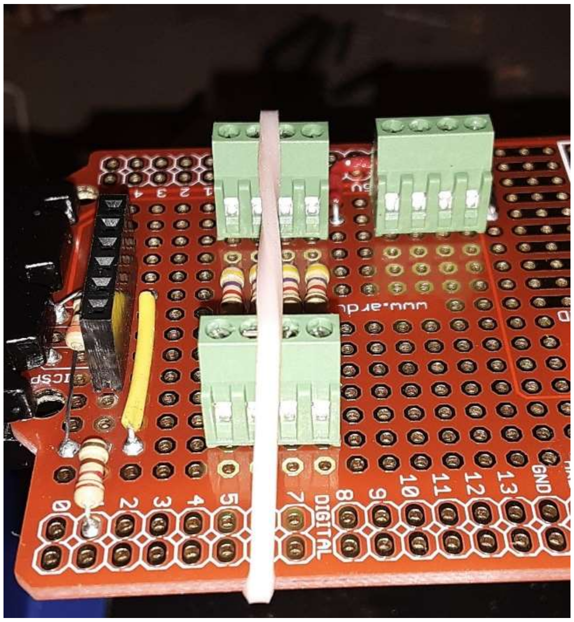


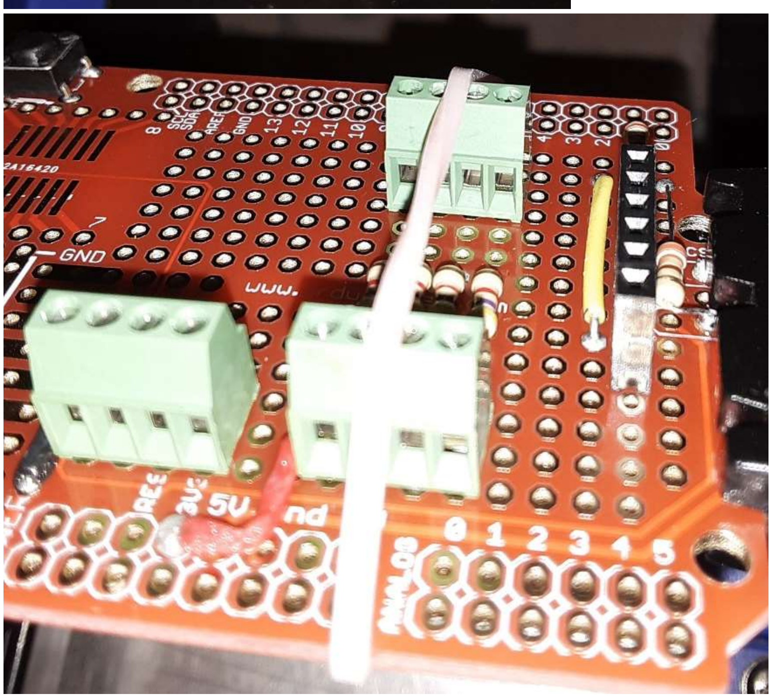
Part D – Clock Line Connections

The last part installed the pull-up resistors, now we will install the connector for the clock lines, attach the pull-up resistors, and make the final connection for each line to its appropriate pad. My color code for clock lines is White. The D- line on the USB breakout boards will connect here.

Step 1 – Place the terminal block

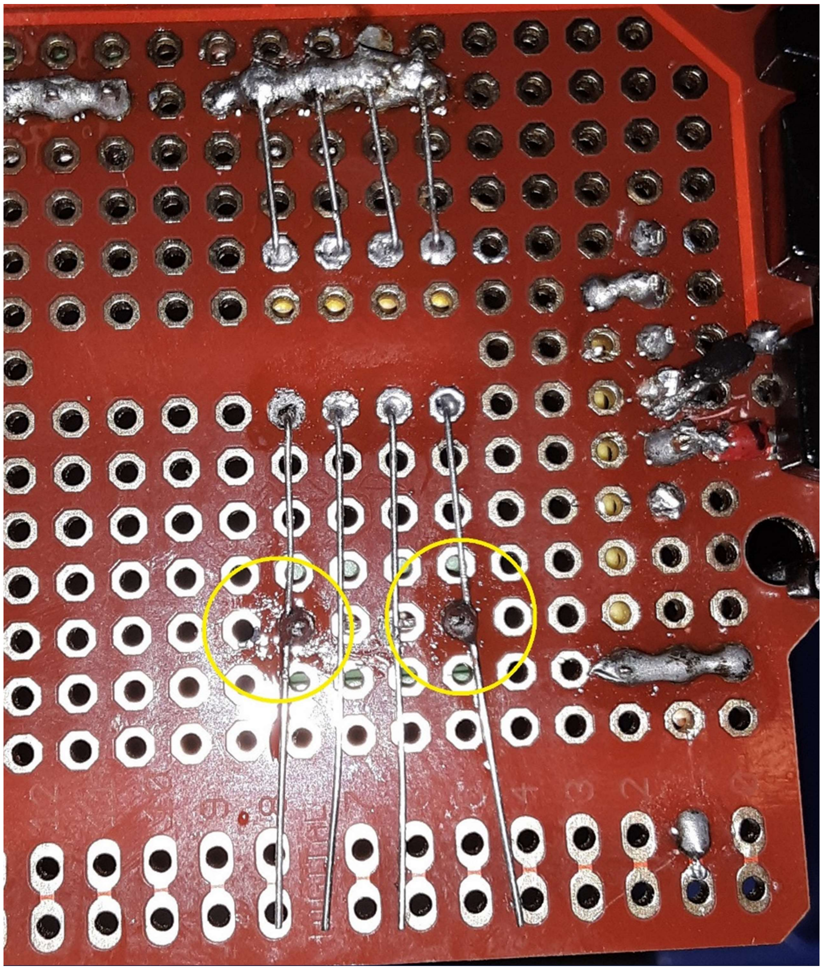
Place a 4-screw terminal block as shown, holding in place with a rubber band as before:



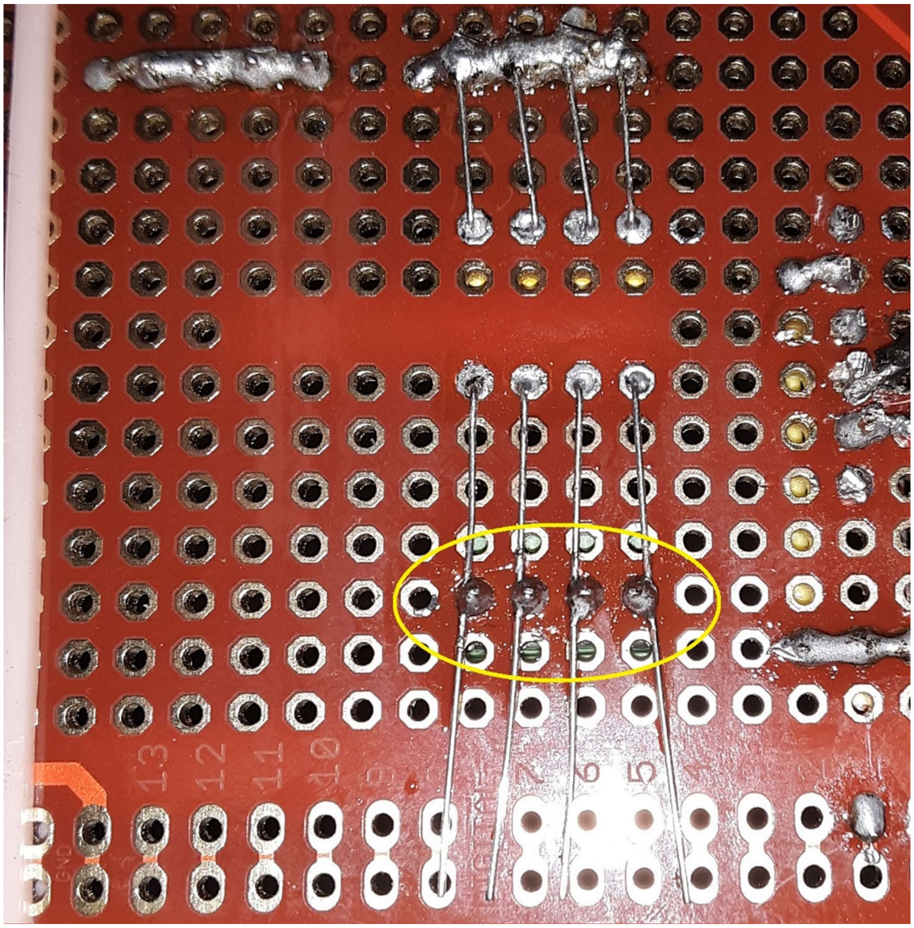


Step 2 – Connect to Pull-up resistors and solder

Solder the outer 2 pins to the board, and at the same time solder the long leads on the outermost resistors to those pins. This allows rubber band removal:

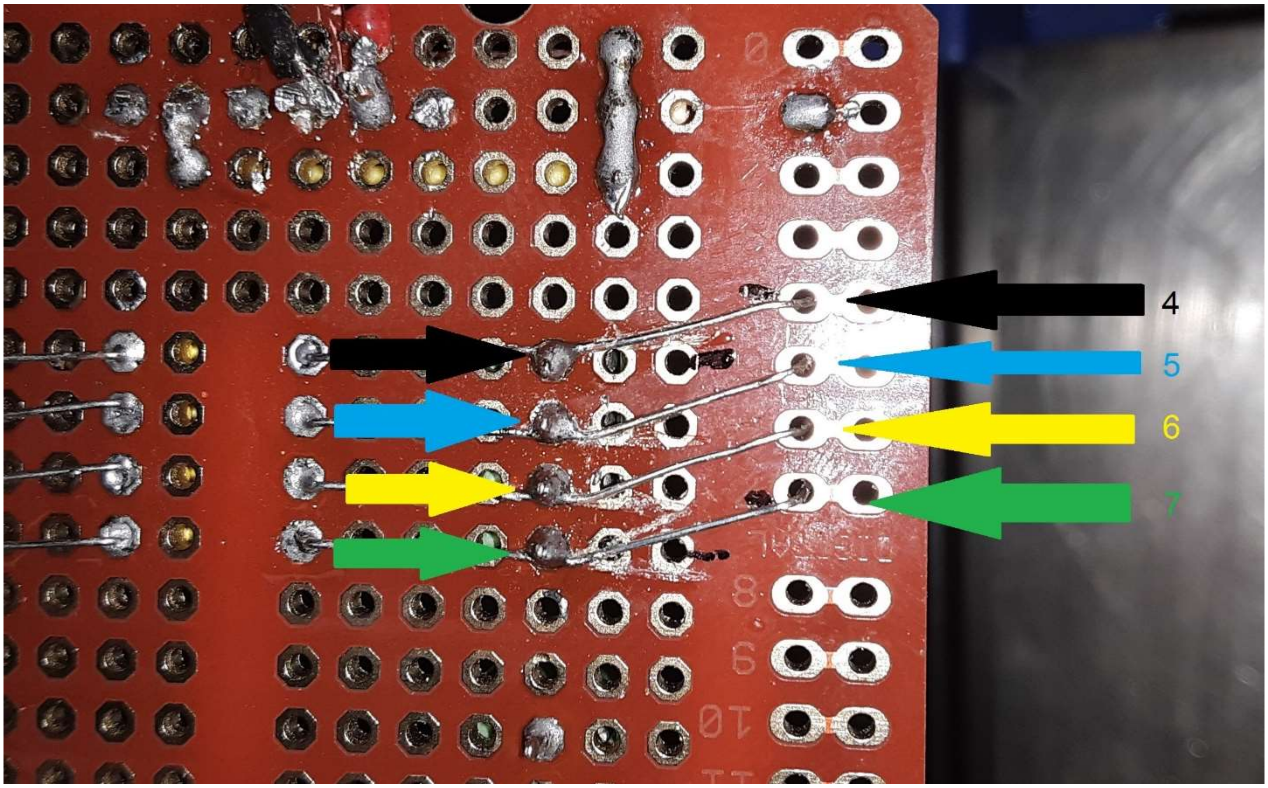


Solder the remaining 2 resistors to their respective pins:

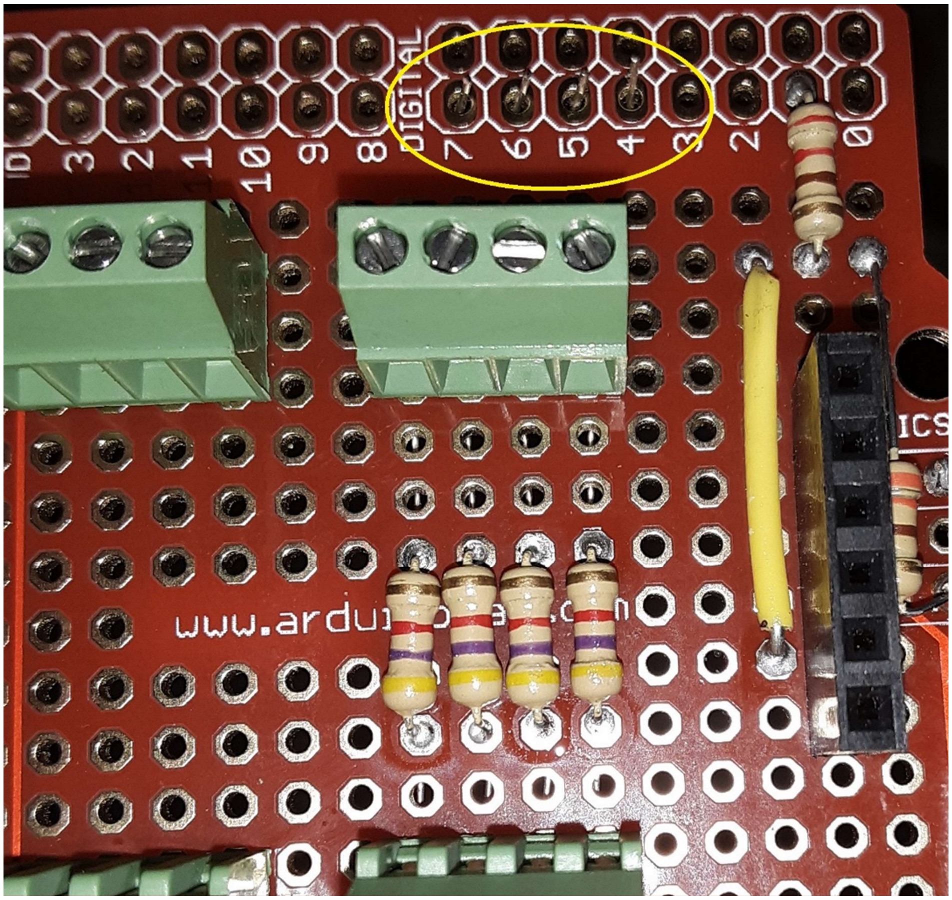


Step 3 – Connect to pin pads

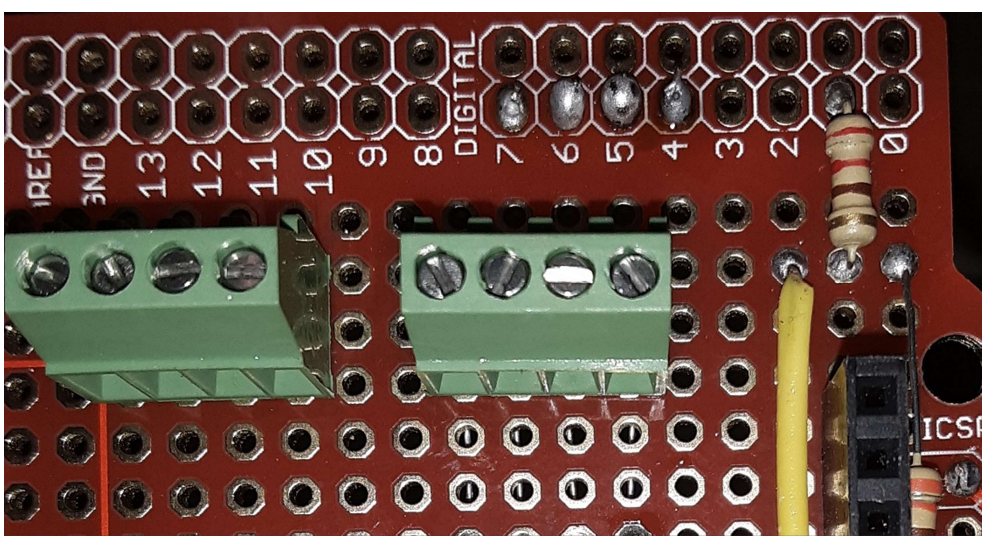
Bend the remaining ends of the resistors into their respective holes in the board. The leads go to the inner pads on the board, connecting to pins 4, 5, 6, and 7. This picture is color-coded to show which lead goes in which hole:



Top side view with leads in place:



Solder those ends from the top, then trim any excess:

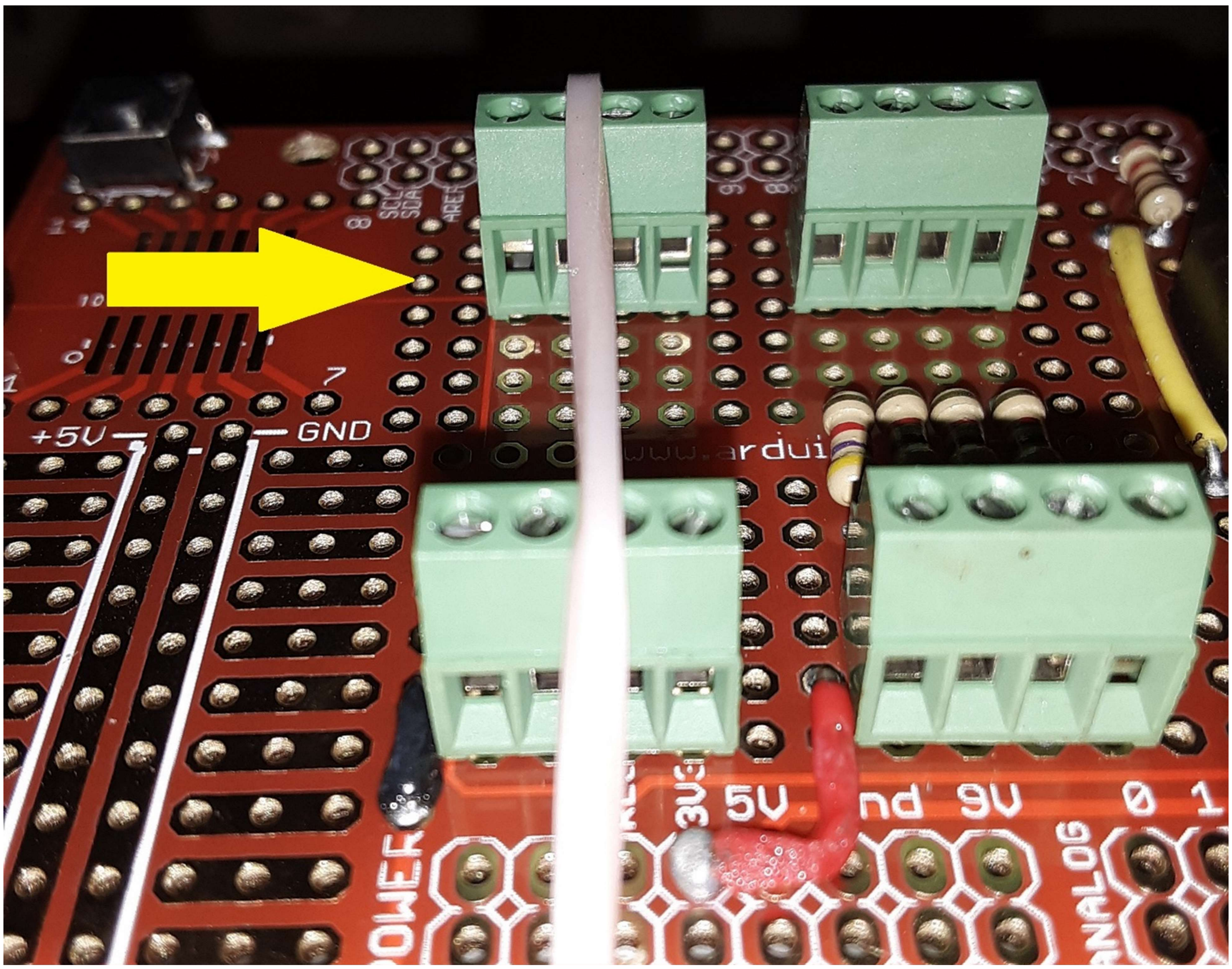


Part E – Data Line Connections

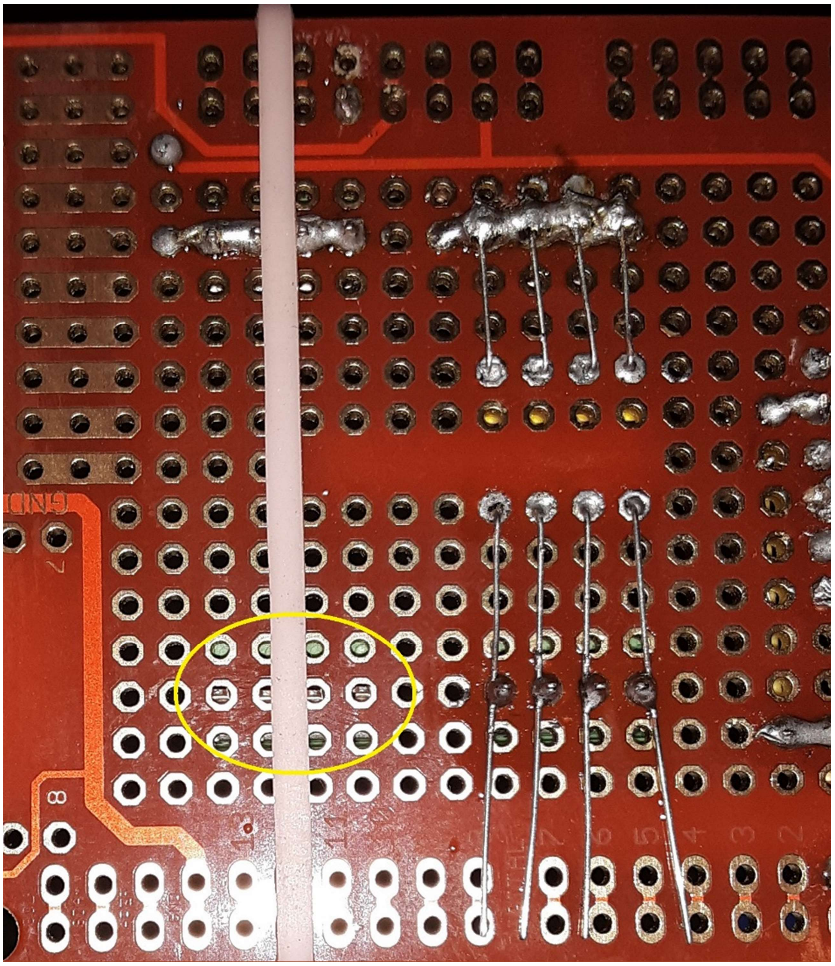
The last connection we need to make to the read head is for the actual data line for each scale. My color convention for data lines is Green. This connector is where the D+ line on each USB breakout board will connect. The connections made here are straight-thru, since the 3V signal coming from the read-head is well over the 2V detection threshold of the Arduino.

Step 1 – Place terminal block

Place a 4-screw terminal block on the board, holding down with a rubber band again:



View of the bottom side:

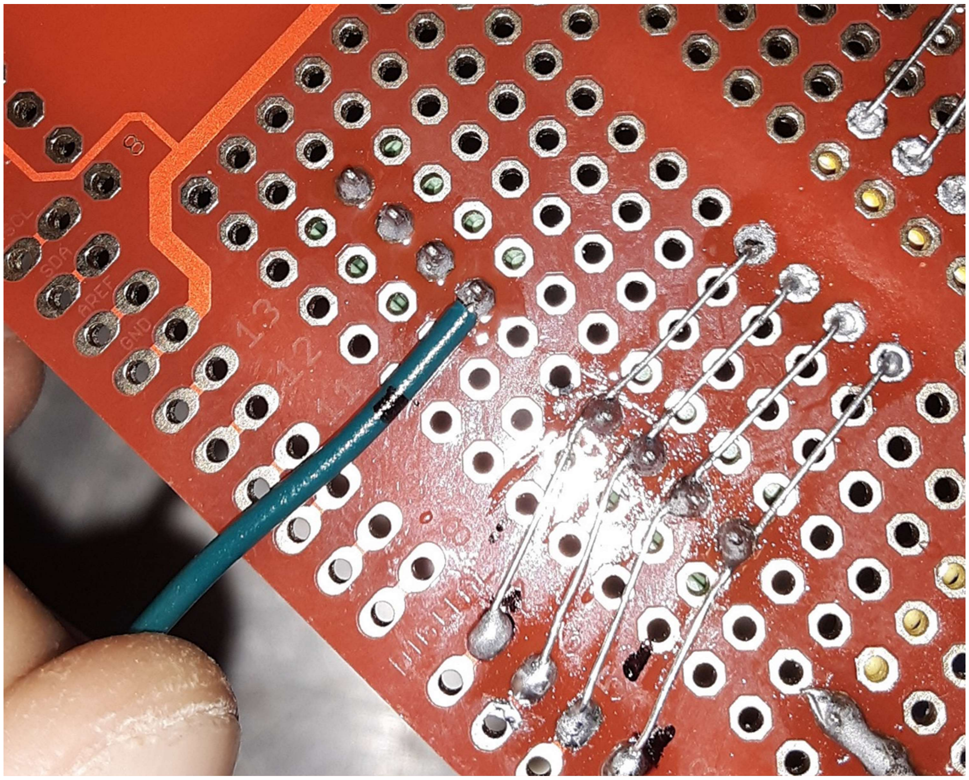


Solder all 4 pins to the board, soldering the outer 2 first to allow rubber band removal.

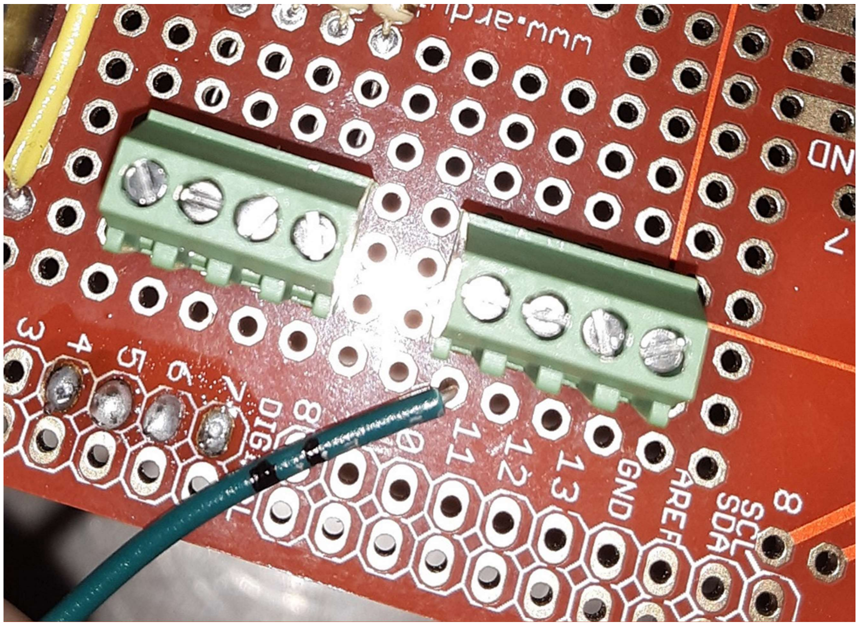
Step 2 – Make and install jumpers

Next we need to cut 4 jumpers to connect the pins to their respective pin pads.

First, measure and strip a wire end long enough to go from the board via hole over to the pin:

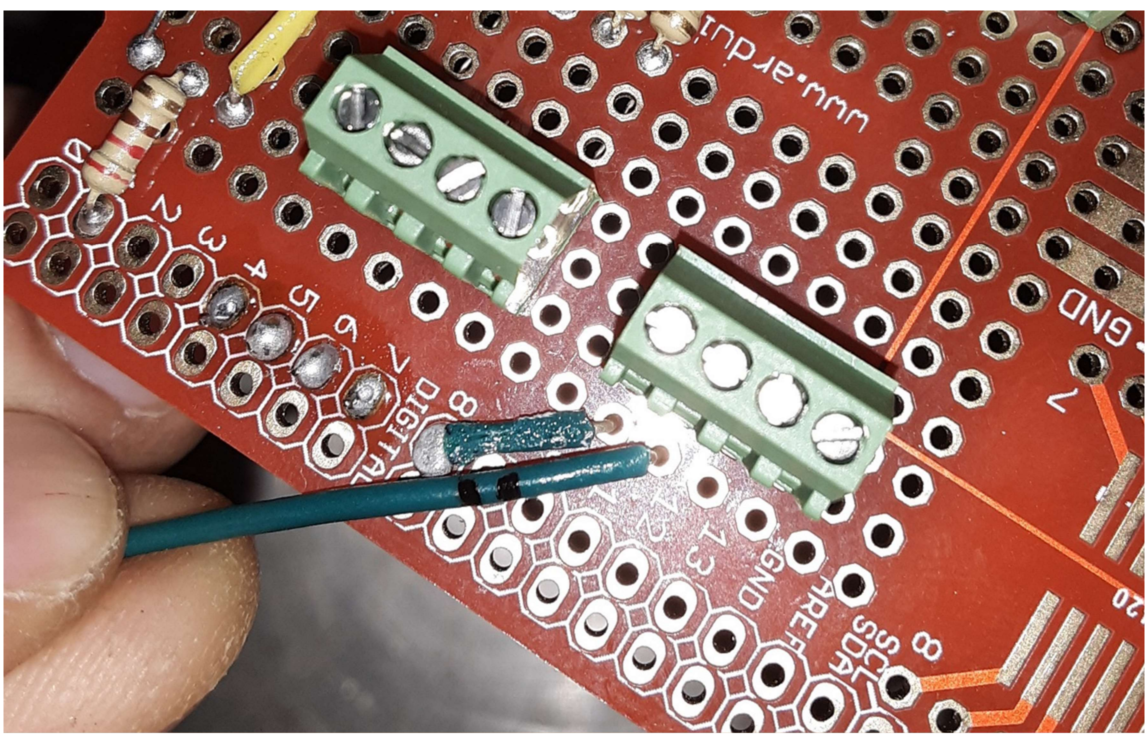


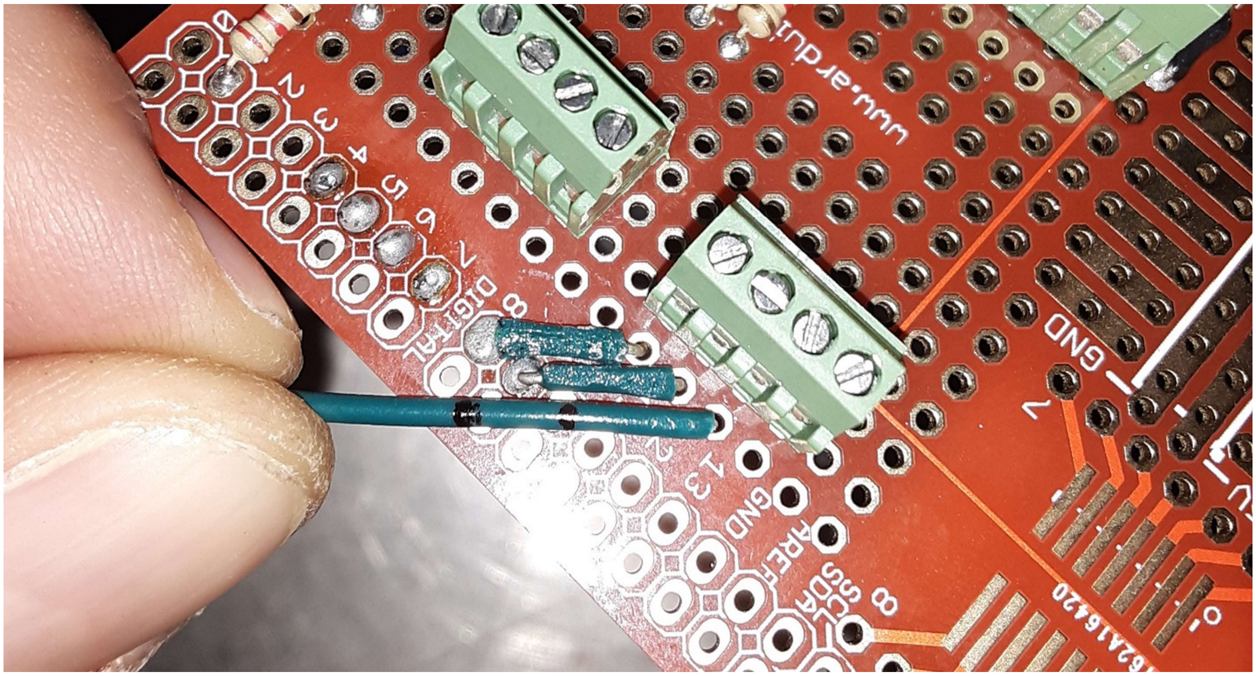
Strip that end and put a 90° bend in it. Insert that bend in the top side of the board and measure for overall jumper length. These jumpers will connect to pads 8, 9, 10, and 11, respectively. Leave plenty of wire to go through the hole, it will be soldered in from the board bottom. This one almost wasn't long enough:



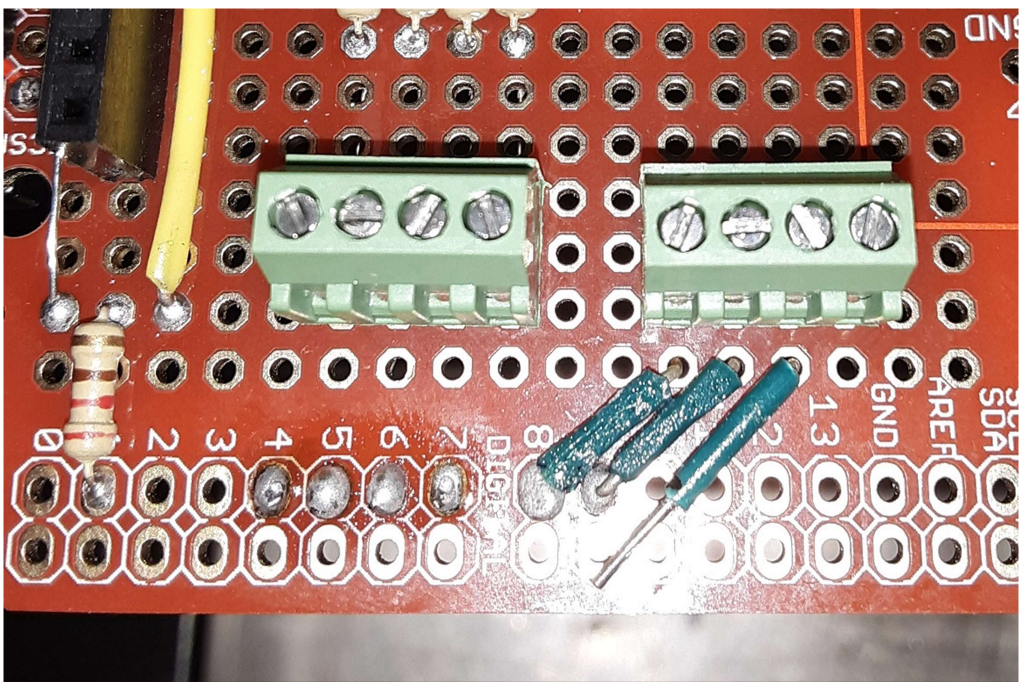
Solder from the bottom at the pin 8 pad connection. Tape is useful to hold the jumper in place.

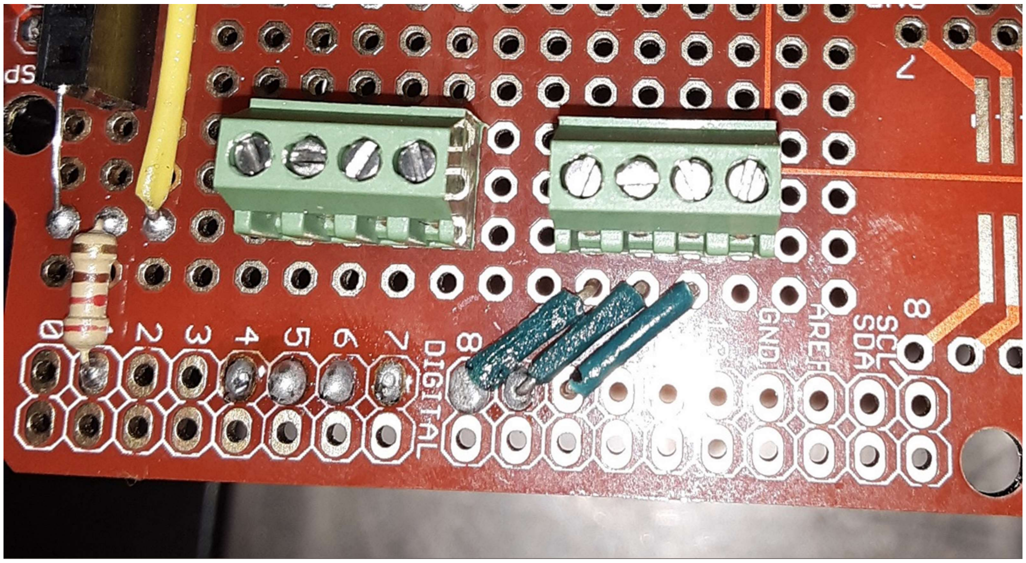
Repeat this 3 more times, once for each pin:



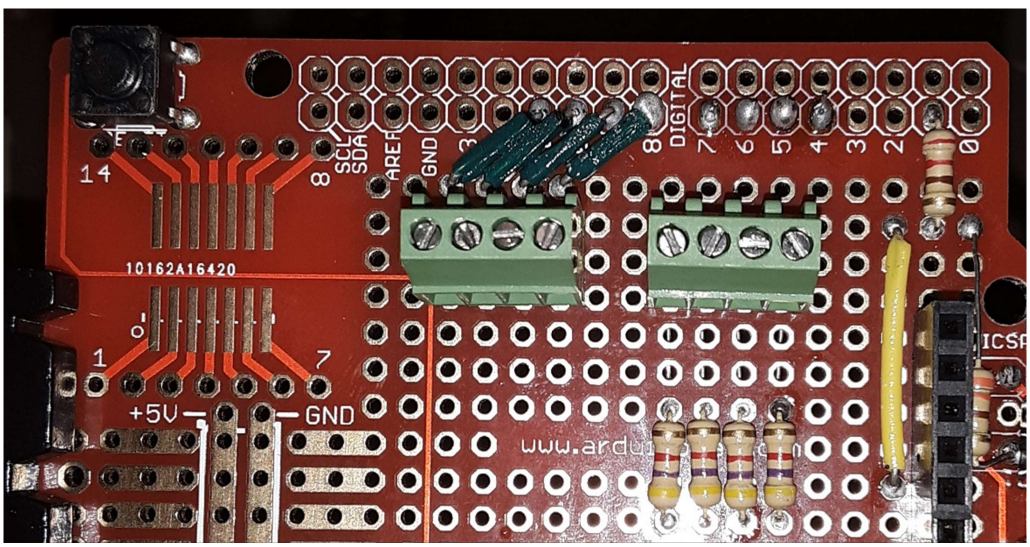


This is a MUCH better length. It can be trimmed later.

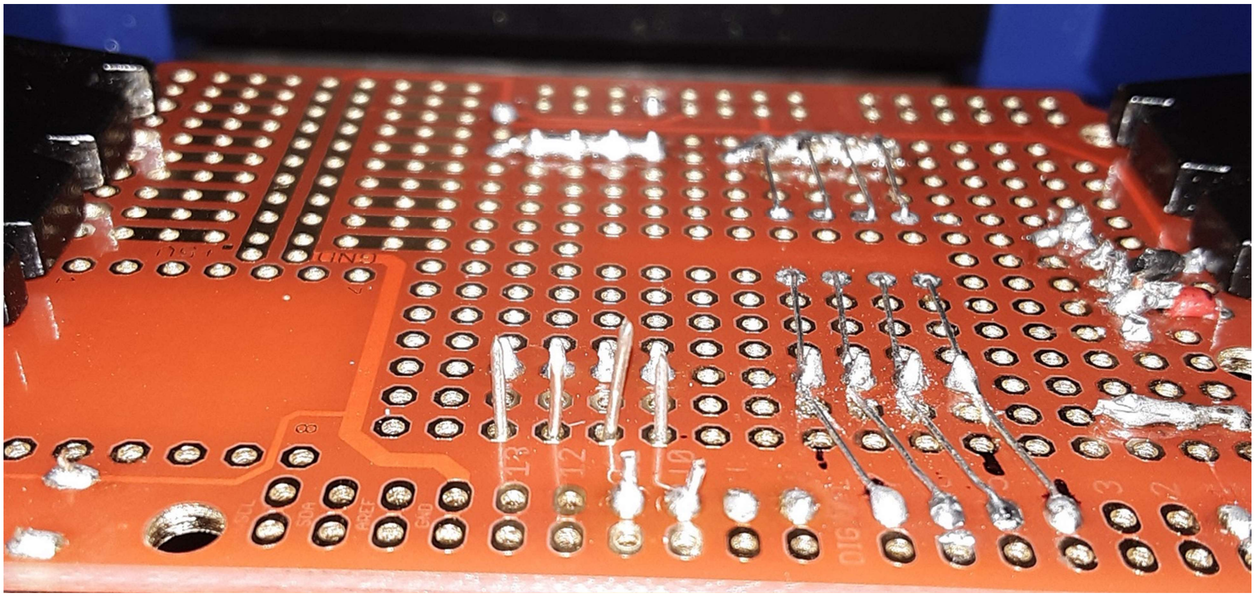




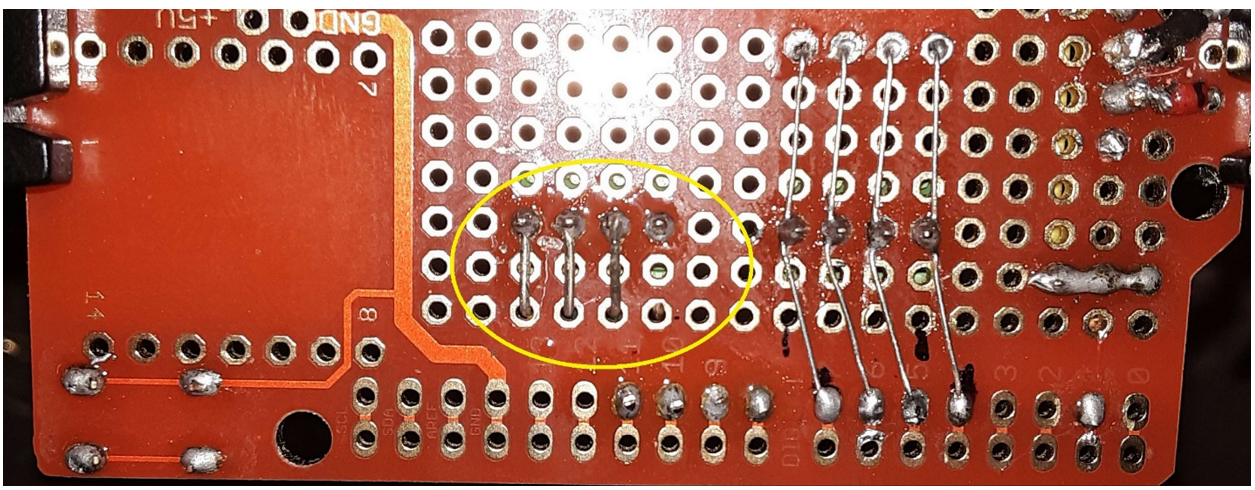
The last jumper in place:

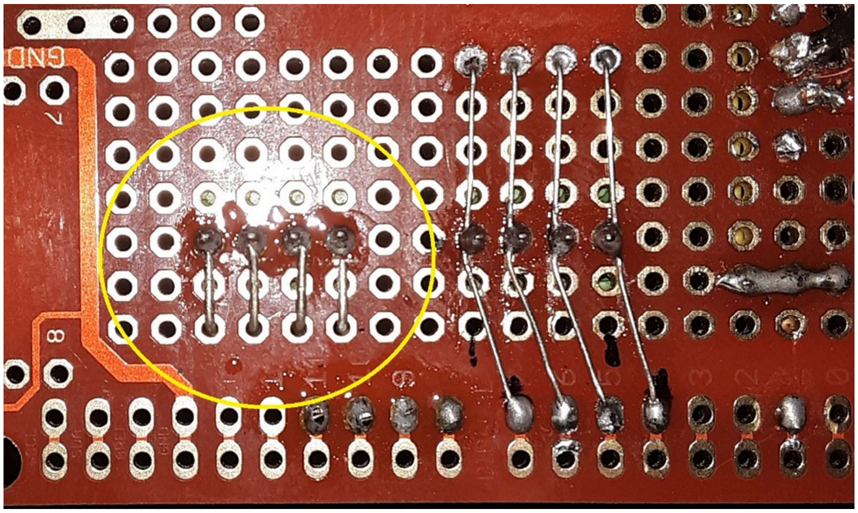


With all 4 jumpers in place, you will be left with unsoldered long lead ends on the bottom of the board:

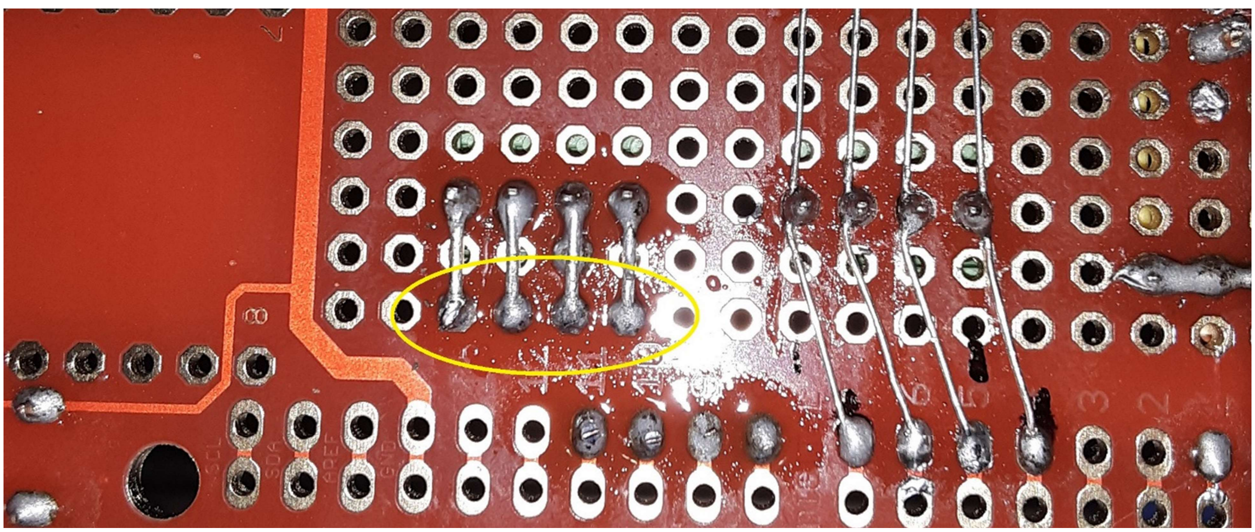


Bend those over to their respective pins and solder the connection:





Now solder the thru-board via for mechanical strength. Add solder to any joints that may not have a good electrical connection:



Trim any excess wire from the bottom side of the board.

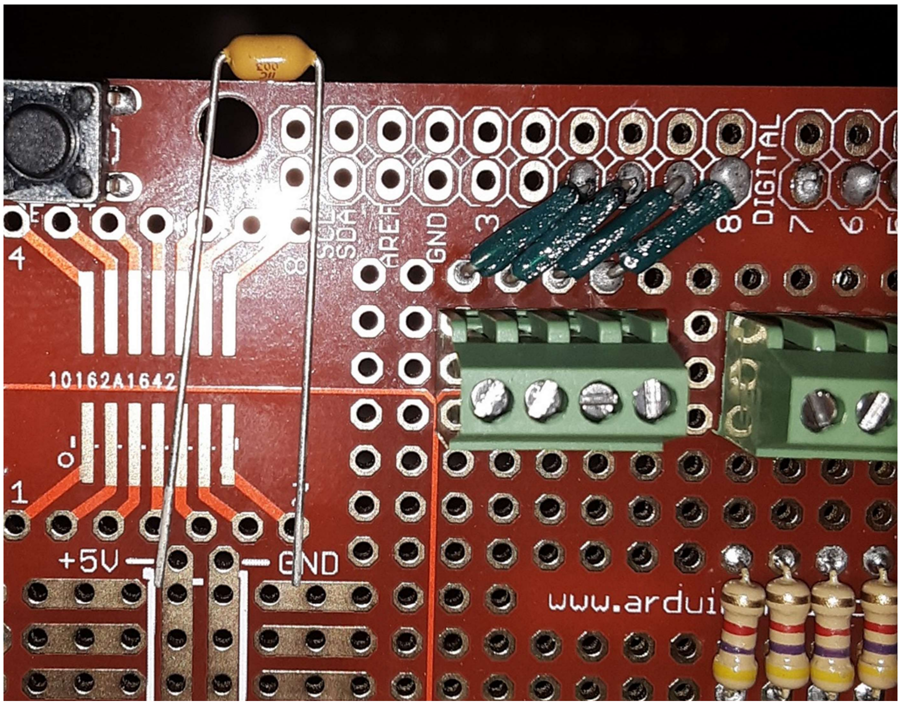
Part F – Adding a filtering capacitor to the power rails

This step isn't strictly required, but it is HIGHLY recommended. It helps filter out any noise in the power rail, which leads to a cleaner signal on the clock lines as well as the actual power rail, itself.

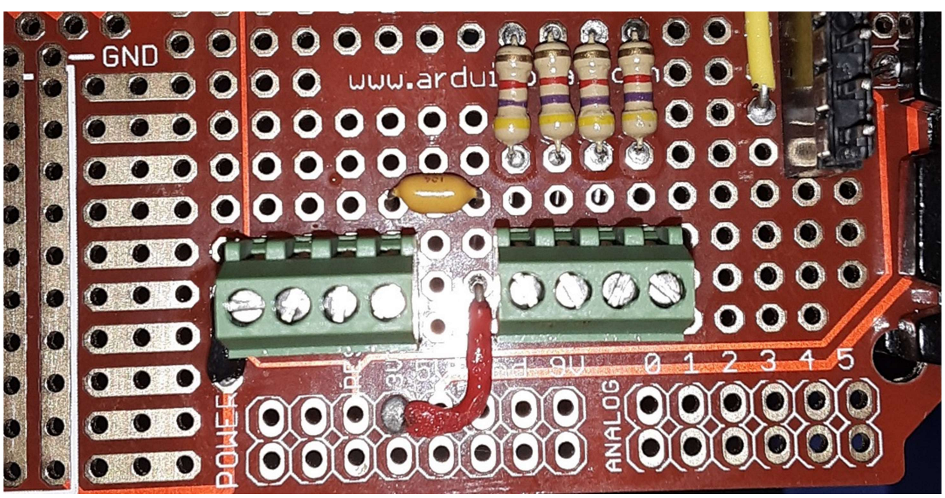
Step 1 – Capacitor installation

I used a capacitor with axial leads. Capacitors with radial leads will not need the first step.

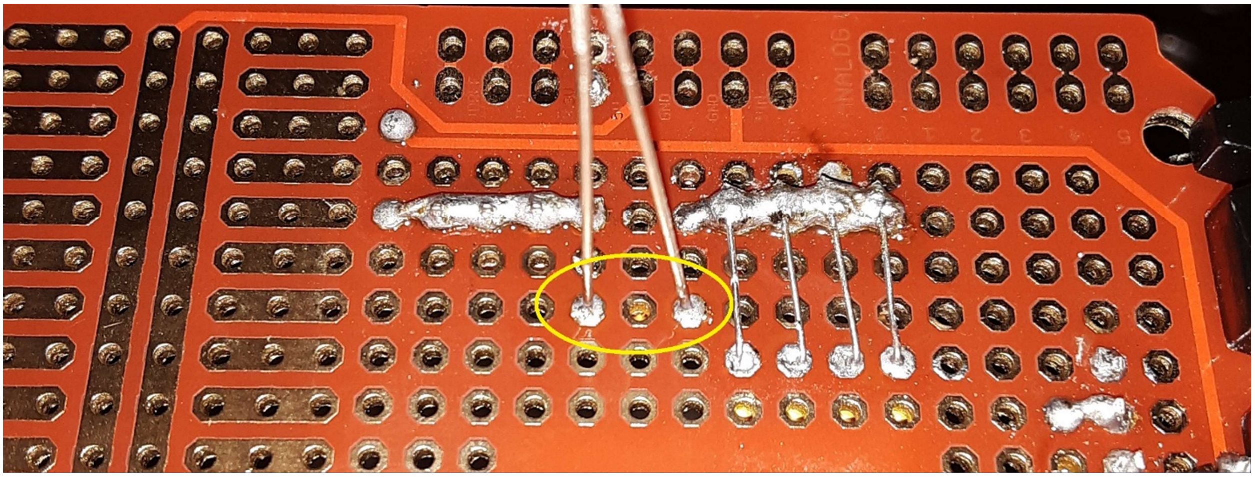
Put 90° bends on the leads of a 0.1 µF capacitor:



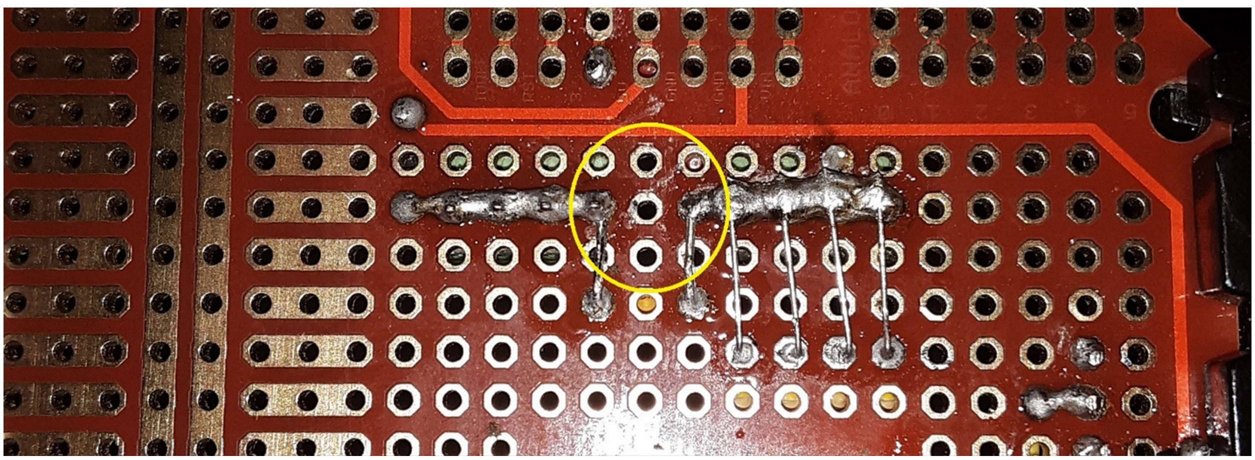
Insert into the board next to the power rails, using tape to hold in place:

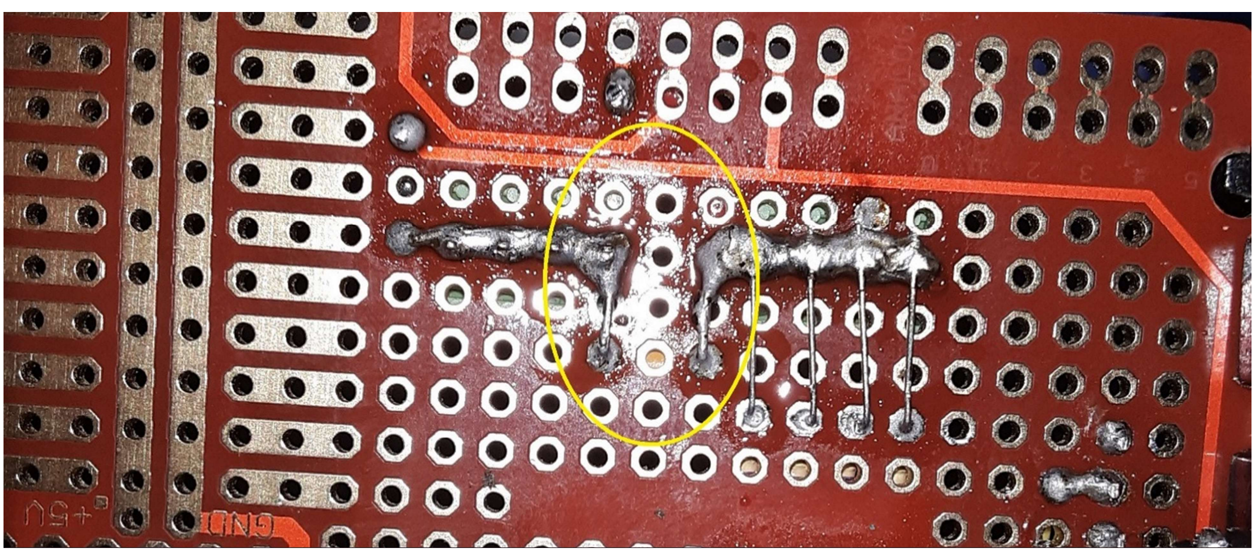


Solder the leads into the vias from the bottom to provide a mechanical connection:



Bend the leads over so that one touches the positive rail and the other touches the negative rail, solder, then trim to length:





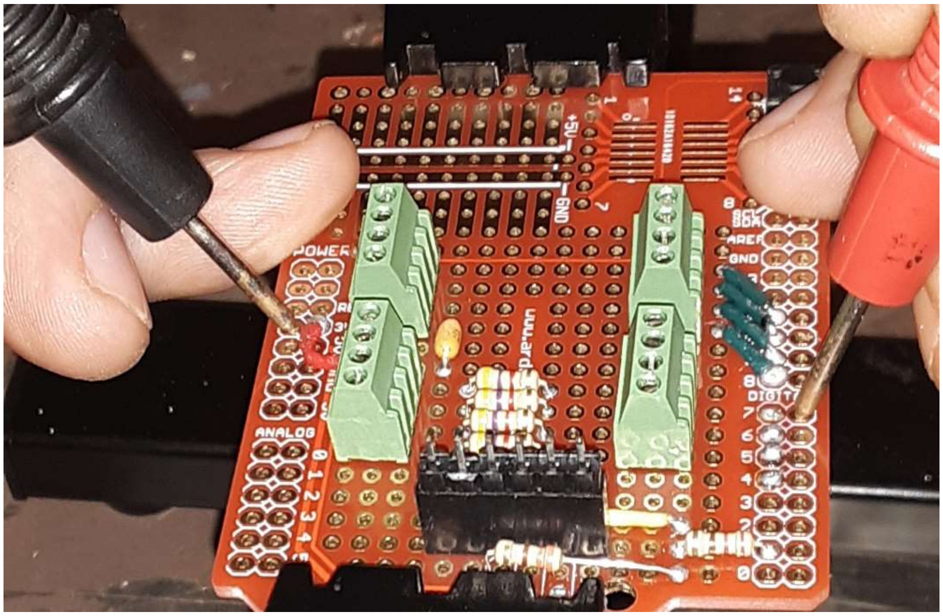
Step 2 – Cleanup

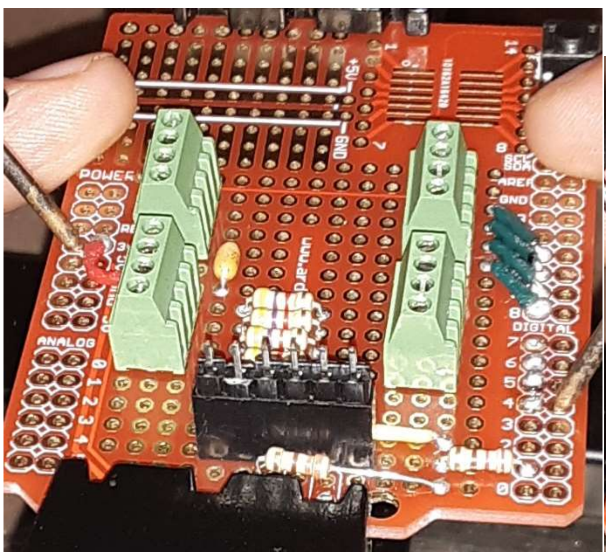
Now is the time to use Q-tips and isopropyl alcohol to clean up all of the flux left on the board. This will help ensure long term longevity by minimizing any corrosion.

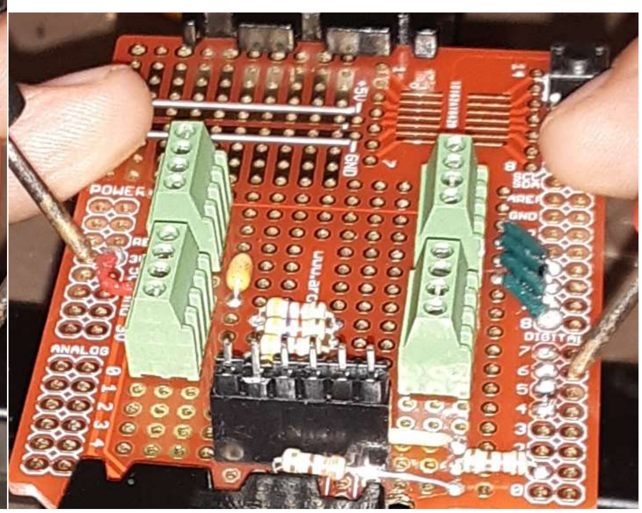
Part G – Testing the connections

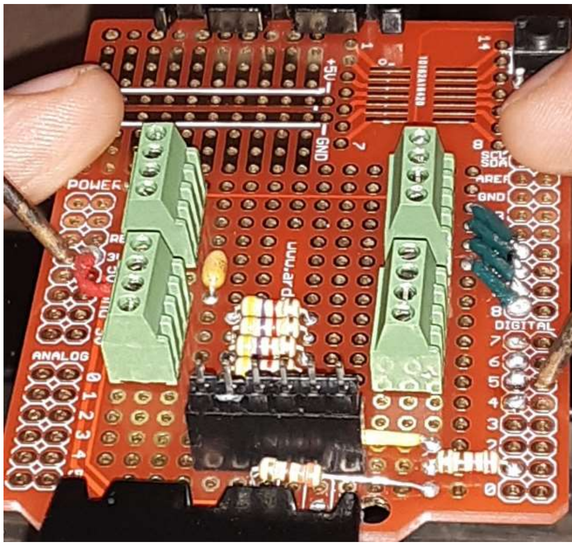
Now is a good time to test all of the connections.

First, check the 3V pad on the board to each of the clock pads on the opposite side. Each reading should be about 4.7 kΩ:

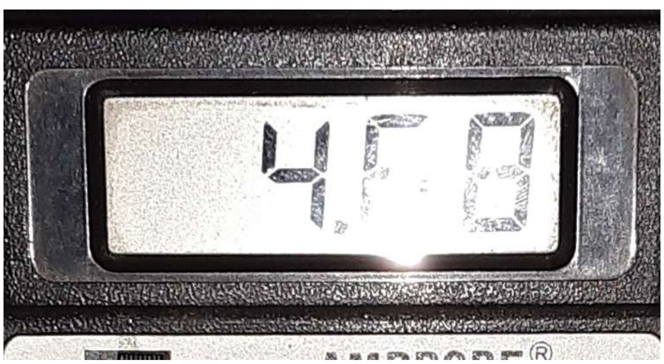




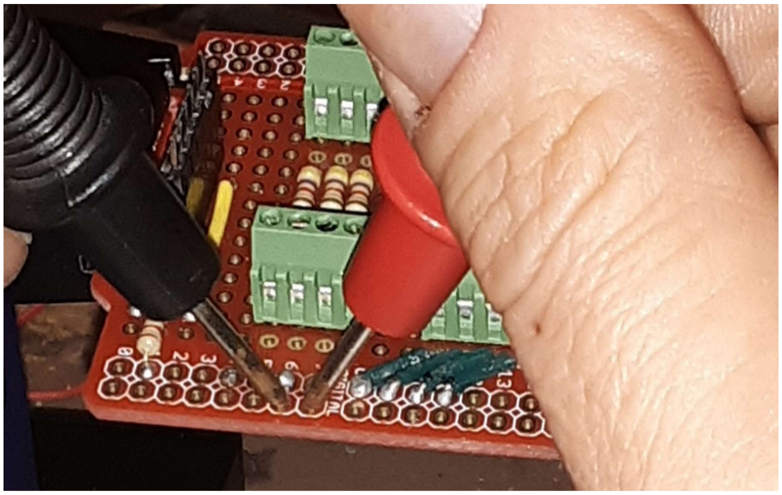



POWER
ANALOG
DIGITAL
GND
AREF

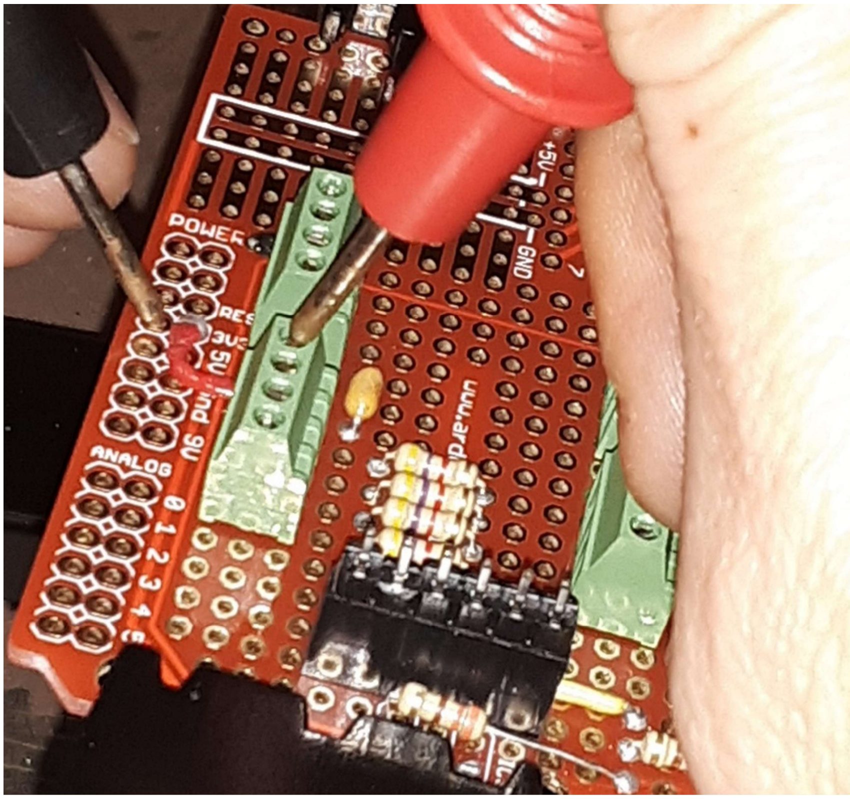

4.63
OFF
ON
AMPROBE
AM-12



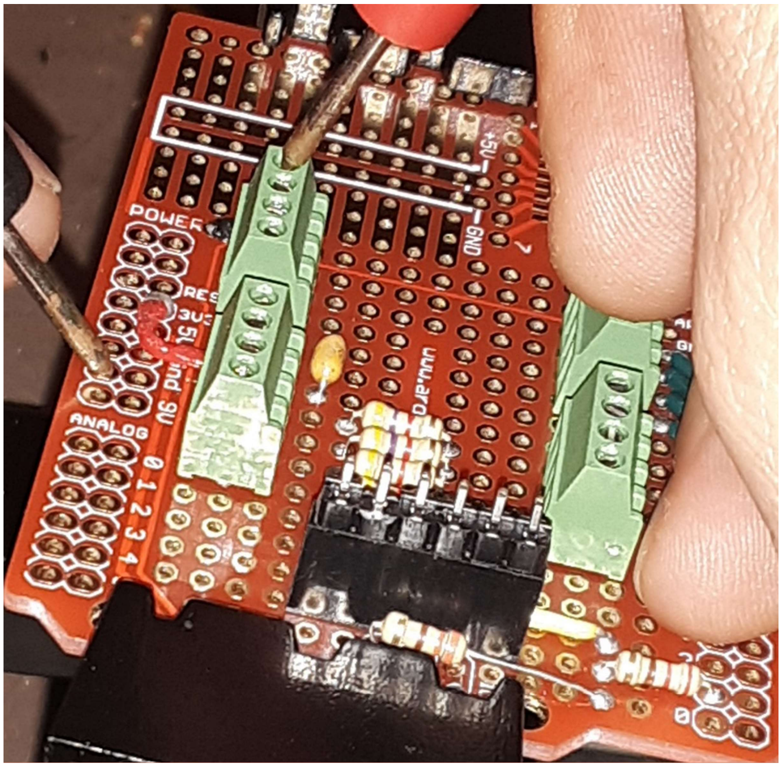
Check the resistance between all of the clock pads. The reading between any given pad and the other 3 should be about 9.4 kΩ:

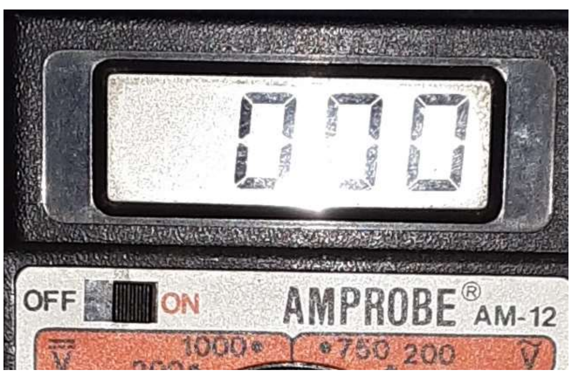


Check the resistance of each screw on the +3V rail connector to the +3V pad. It should read about 0:

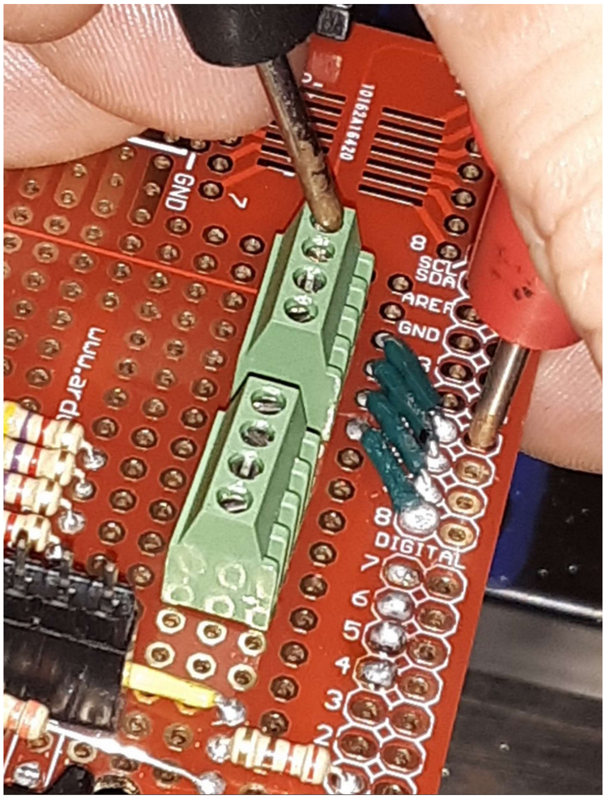


Check each screw on the GND connector to the appropriate GND pad. It should read about 0:





Check each screw on the data line connector to its appropriate pad. The reading should be about 0:



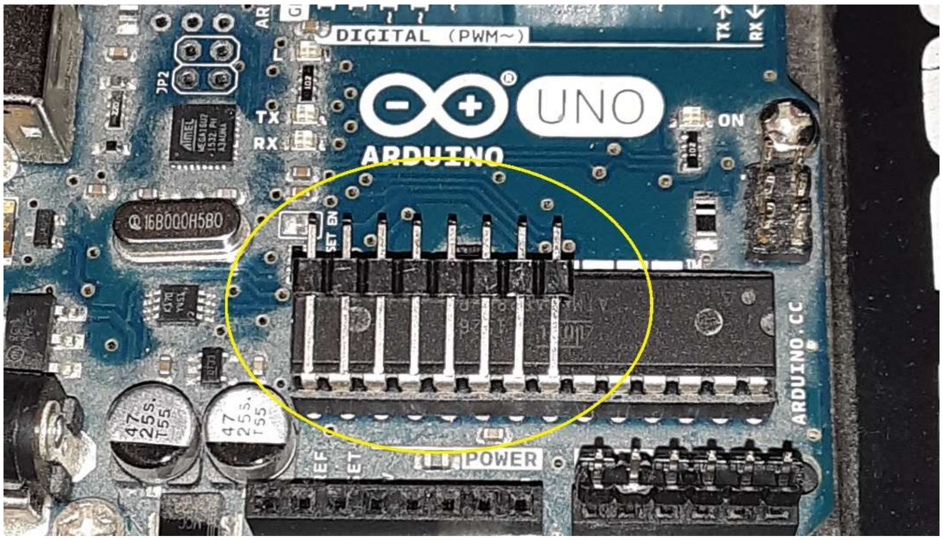
Those are the only connections that should be made up, other than the Bluetooth module checked earlier. Check all other pads to verify there is no connection by setting the multimeter to its highest Ω setting and checking all pads against each other.

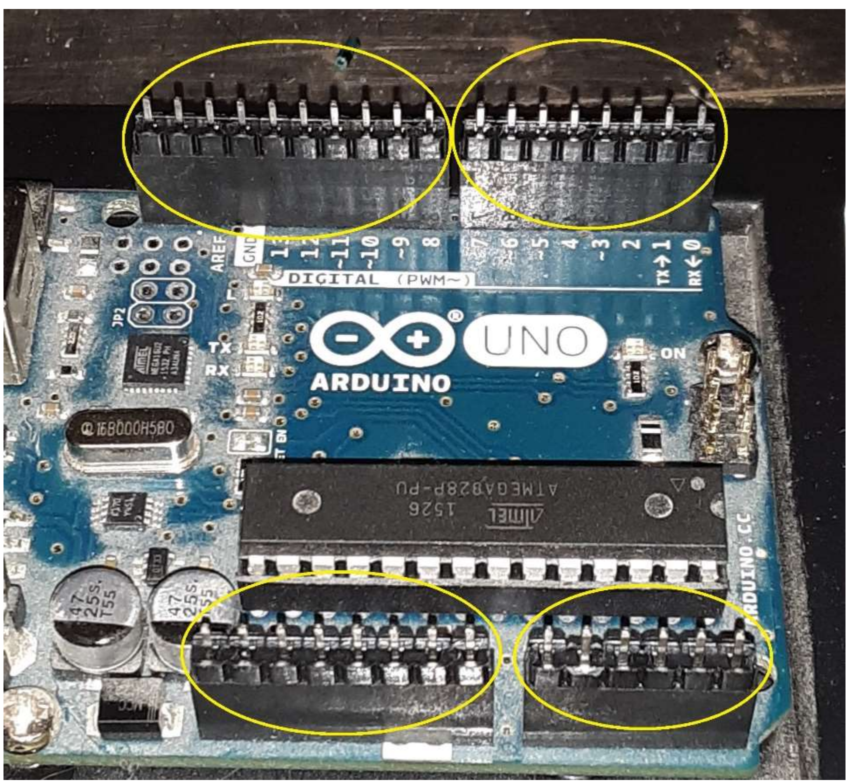
Part H – Soldering in the Pin Headers

The final part of the adapter shield build is connecting the pin headers. Save this step for the absolute last in order to give yourself room to work on the rest.

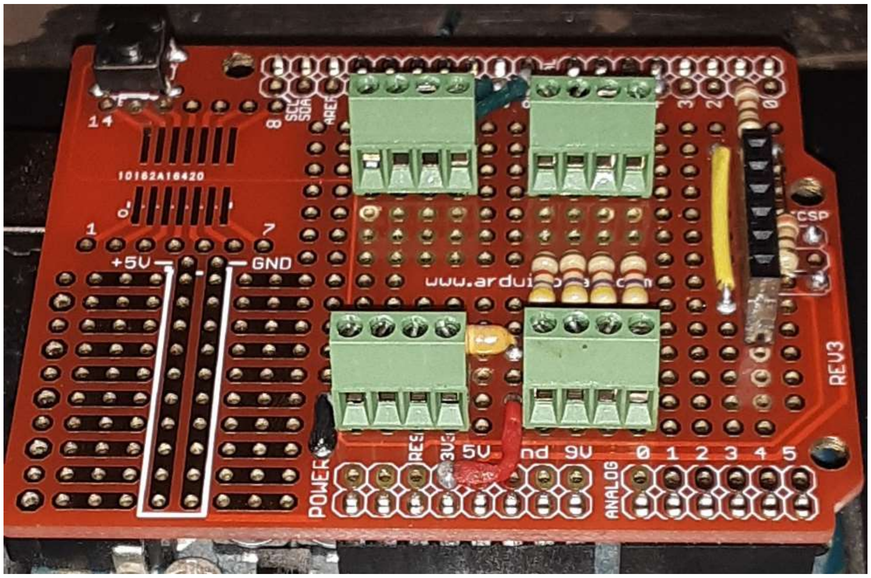
Step 1 – Place pin headers in Arduino

Place pin headers into the Arduino stack headers:

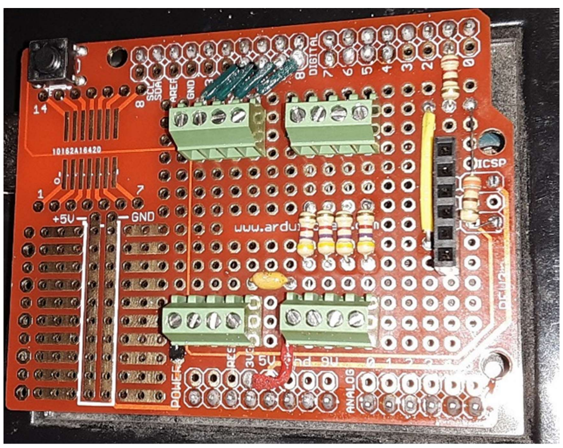


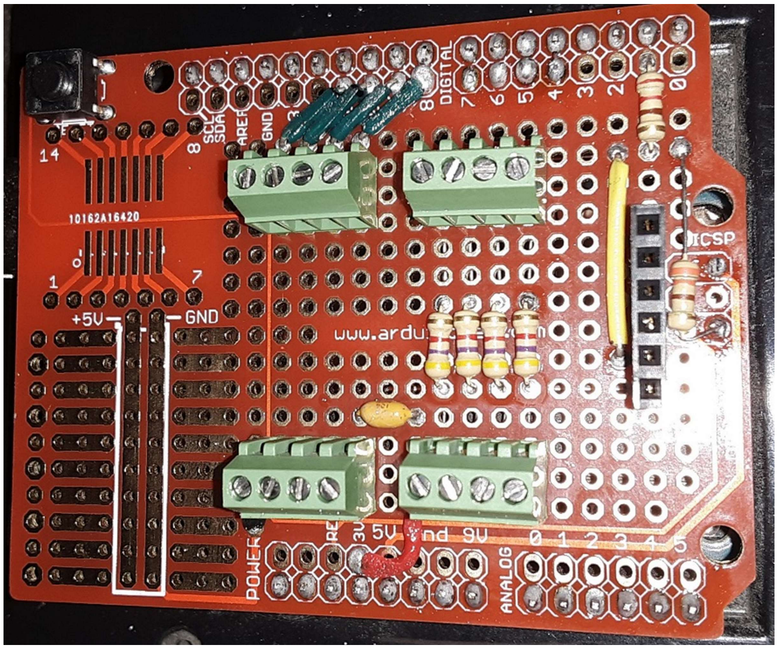


Place the adapter shield on top of the pins:



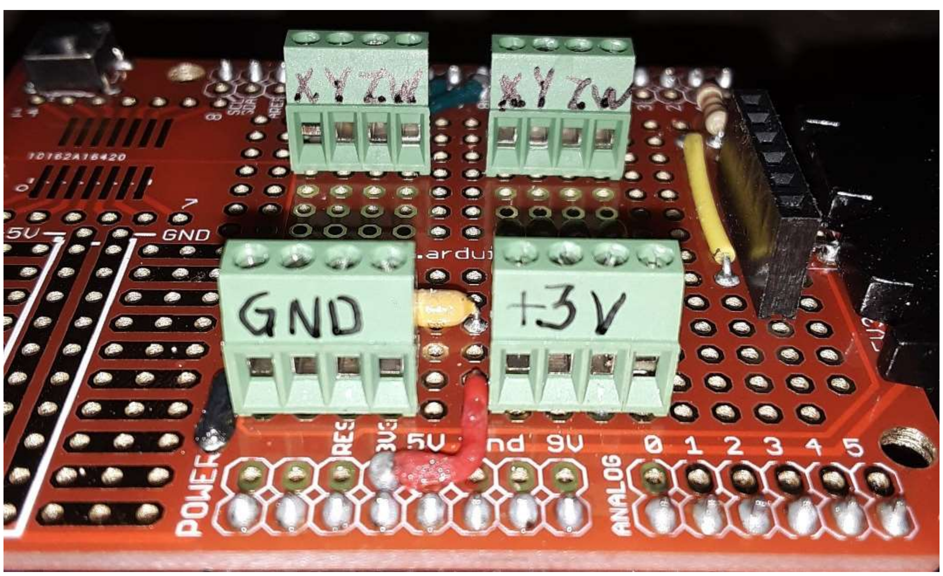
And solder in place:



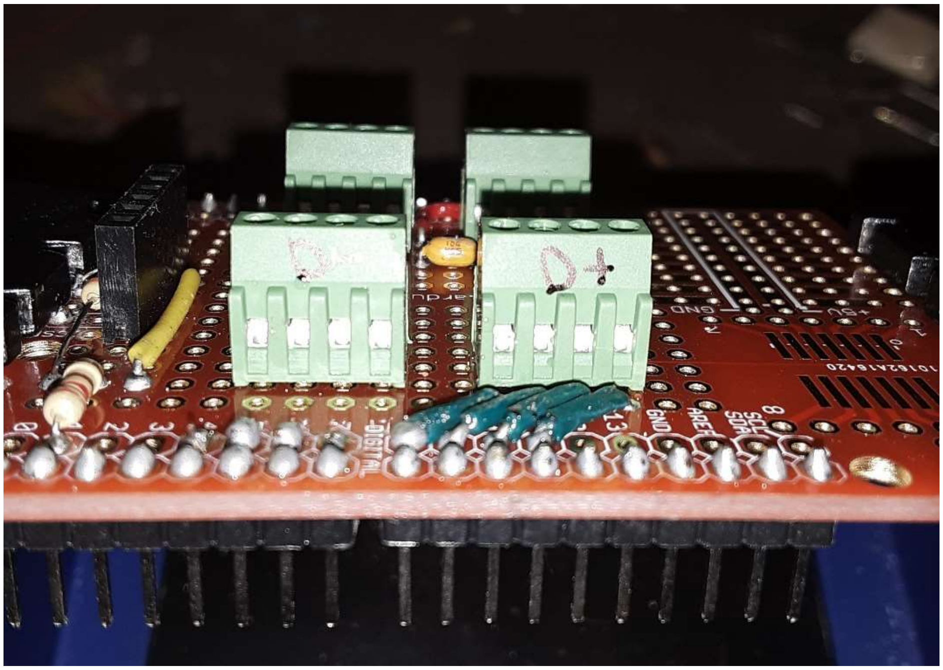


Step 2 – Labeling

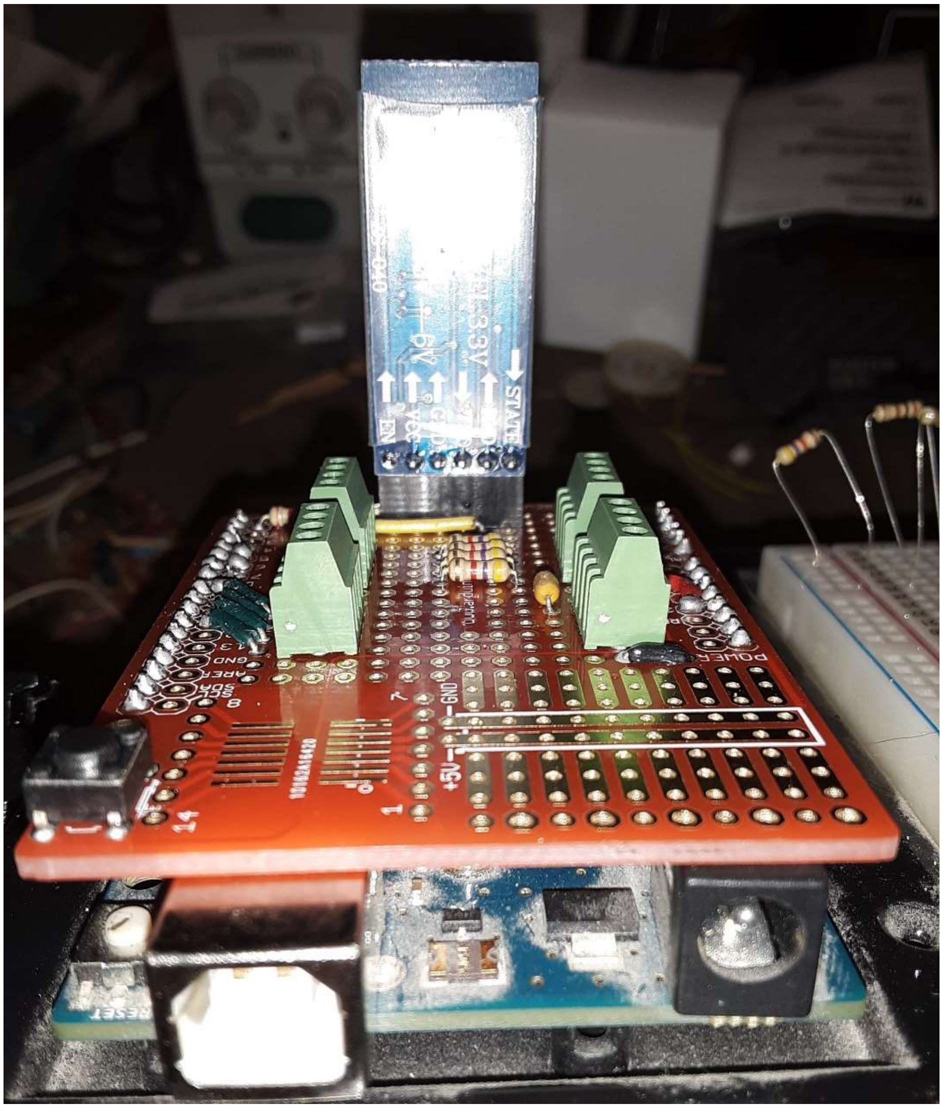
Now is a good time to label the connectors, also. For the Clock and Data connectors, the axes are, from left to right, X, Y, Z, and W. An Ultra-Fine Sharpie works well enough for this:



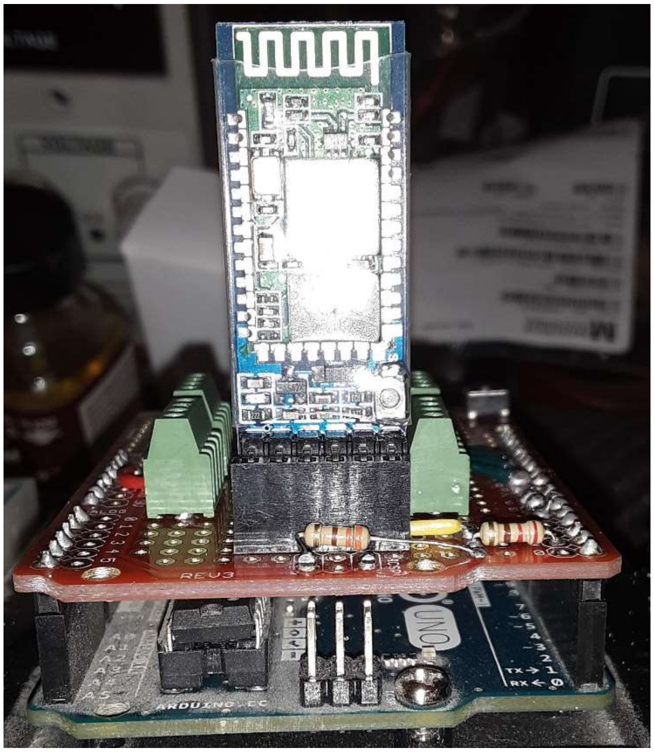
The Clock lines will be coming from the D- connection on the USB breakout board, and the Data lines from the D+ connection. I label the respective connector with those conventions (the terminal block on the left has D- written on it):



The Bluetooth module can be placed in the header now, too:



The other end of the board:



Connection checks can be made again, with the same resistance readings as in the last section.

Part I – USB Breakout Board connections

This part solders in wires to the USB breakout boards. The wires I used here are way longer than recommended. They should be trimmed to size for your enclosure in order to minimize noise induced in the system. The ID pad is unused in this application. Make one for each scale you plan to connect.

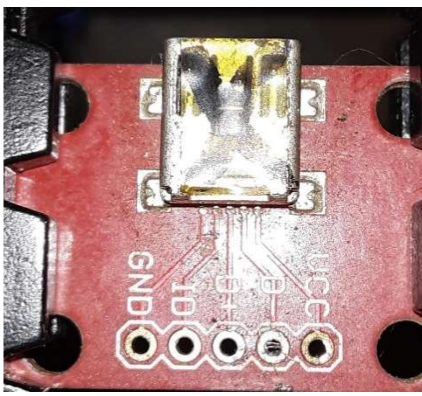
The color coding I use:

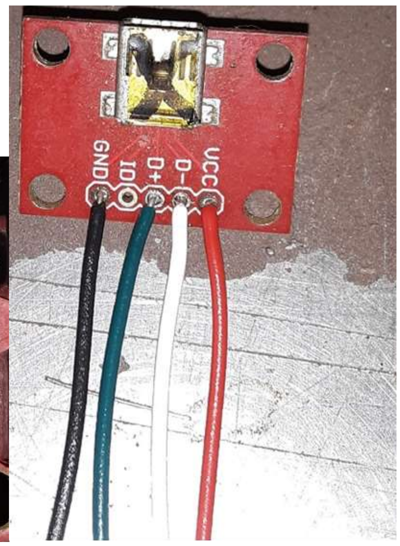
Red: Vcc (+VDC power)

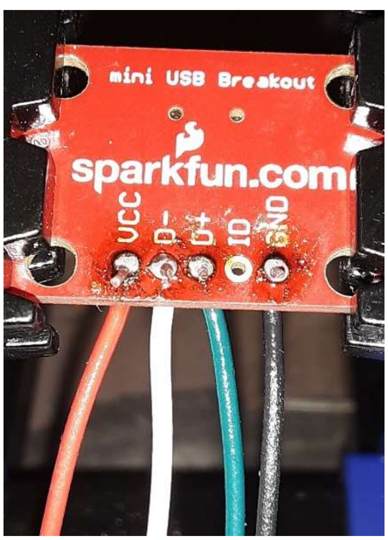
Black: GND (DC ground)

White: Clock signal (D- on board)

Green: Data signal (D+ on board)

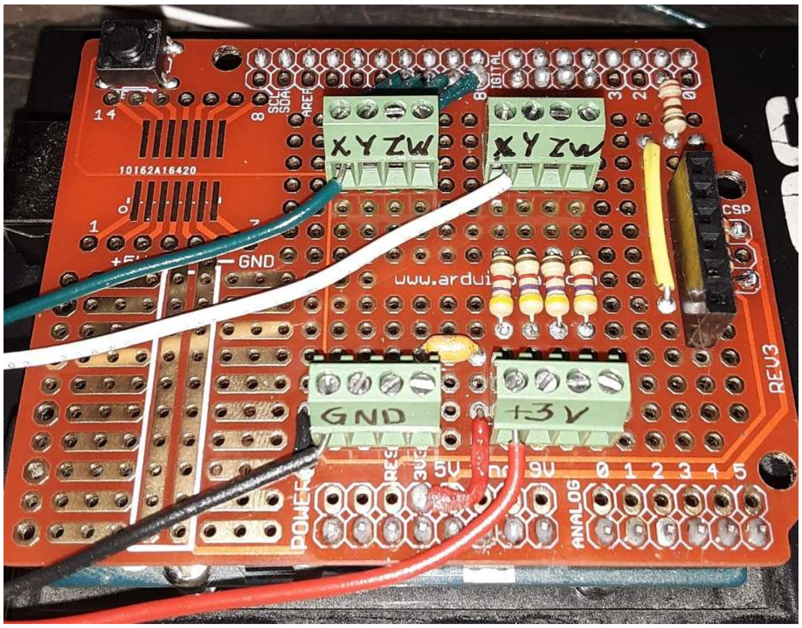


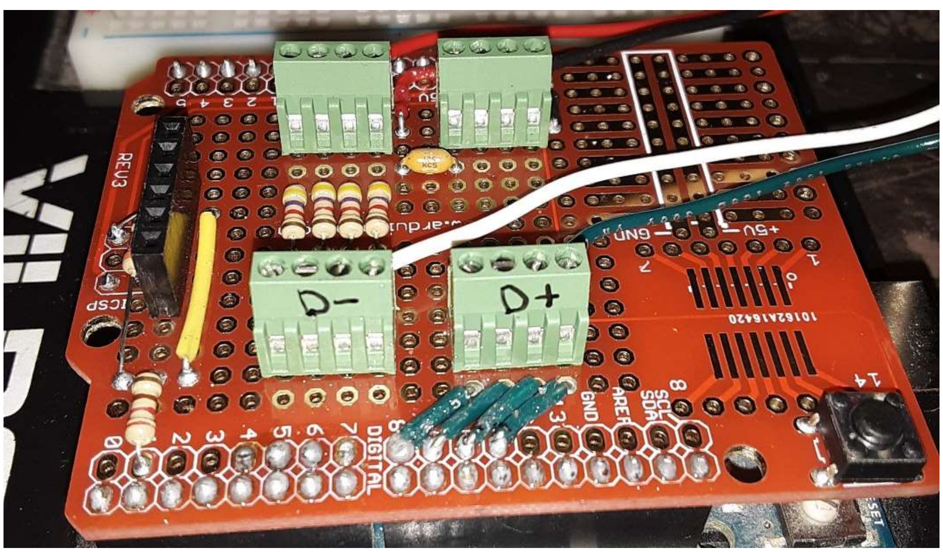




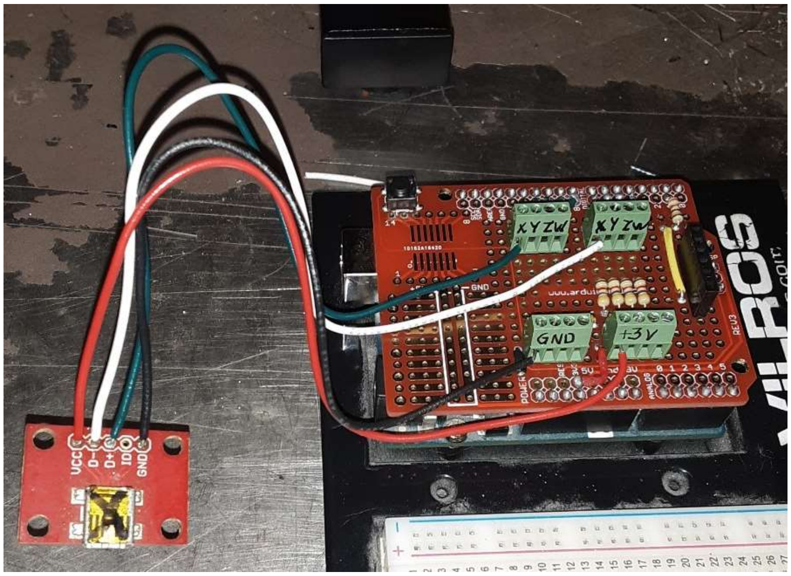
Part J – Final Connections

This section connects each wire on a USB breakout board to its appropriate connector. Each wire on a breakout board connects to a different 4-screw terminal block. The Green and White leads for each board MUST be attached to the corresponding hole in the appropriate connector. These pictures show the connections needed for an X axis connection:





This is a big picture view with an X axis connected:



The wires connecting the USB breakout board to the terminal blocks in this picture are way too long. They should be trimmed to size for your build.

The adapter shield is now complete and the Arduino is ready to be programmed with the accompanying sketch. After that, pair your tablet with the Bluetooth module.

The scales can now be plugged into the USB sockets and TouchDRO can be set up per Yuriy's instructions:

https://www.touchdro.com/resources/getting-started/initial-configuration.html

https://www.touchdro.com/resources/getting-started/scale-calibration.html