GCODe Mini

Construction Manual

Parts list

Part	Quantity	Cost($)
4mm Black Acrylic Sheet	2x2 feet	10
GCODe Mini PCB	1	30
2cm corner bends	8	3.2
M4X15 bolts, nuts	16	0.16
M3X12 bolts, nuts	8	0.04
M3X25 bolts, nuts	6	0.03
4mm washers	10	1.5
Red, Black single-strand wire	40cm each	0.1
HUDZ 5V Buzzer	1	1
47E resistor	1	0.06
1k resistor	2	0.12
3.3k resistor	1	0.06
10k resistor	2	0.12
100k resistor	1	0.06
1 uF electrolytic capacitor	1	0.04
3/8" square staggered lead variable resistor: 10k	1	0.14
0.1” side-entry screw terminal block: 2-pin	4	1.2
5mm red LED	1	0.1
BPW34 photodiode	1	1.00
Male pin headers	32	2.54
Arduino Uno	1	22
1.5m USB B cable	1	3.43
Total cost		76.9

You’ll also need two flat-tip screwdrivers (5mm and 2mm long tips), some pliers, a soldering iron, and some solder wire. We also recommend having a cheap multimeter to check connections. Here’s a quick guide by SparkFun on how to use them: https://learn.sparkfun.com/tutorials/how-to-use-a-multimeter .

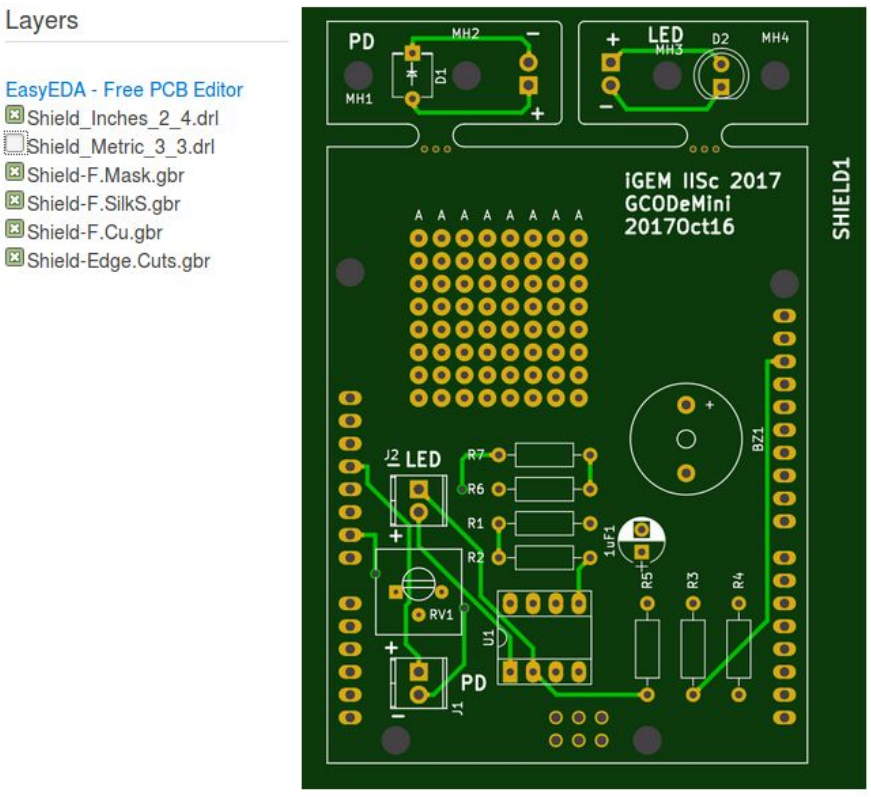
Fabrication Guide

Printed Circuit Board (PCB) Printing

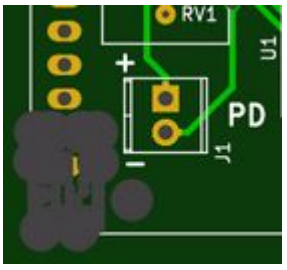
Don’t worry; it’s easy. All you have to do is:

1. Find a PCB fab lab in your city, or online. Here’s an easy place to get you started: https://easyeda.com/order (they're not sponsoring us).
2. Upload the gerber zip (you can find them in the Fabrication Files zip)

3. If online, check that your formats match (sometimes a format mismatch can lead to drill holes appearing in random places, or the PCB getting scaled by a factor of 2.54, etc). Try to get a render of the final PCB according to them. If you’re using EasyEDA, click on the “Gerber Viewer” link that pops up.



4. Here, we had to untick the metric drill file “Shield_Metric_3_3.drl” to get it to look proper. Otherwise there was this little mess at the bottom left:



5. Tell them the specifications of the PCB - this one is 5.5 * 8,5 cm; and in the EasyEDA case they were to use the inches drill file, not the metric one.
6. Send the payment
7. Receive your PCBs

Printed Circuit Board (PCB) Soldering

Just a few things to remember while soldering your parts:

1. Make sure to consult the 3D render above, and more importantly, the PCB PDF. It has all the markings needed to solder the parts. Note, however, that the 3D file for the 3-legged variable resistor and the LM358 IC are missing in the above render. They are to be installed, however.

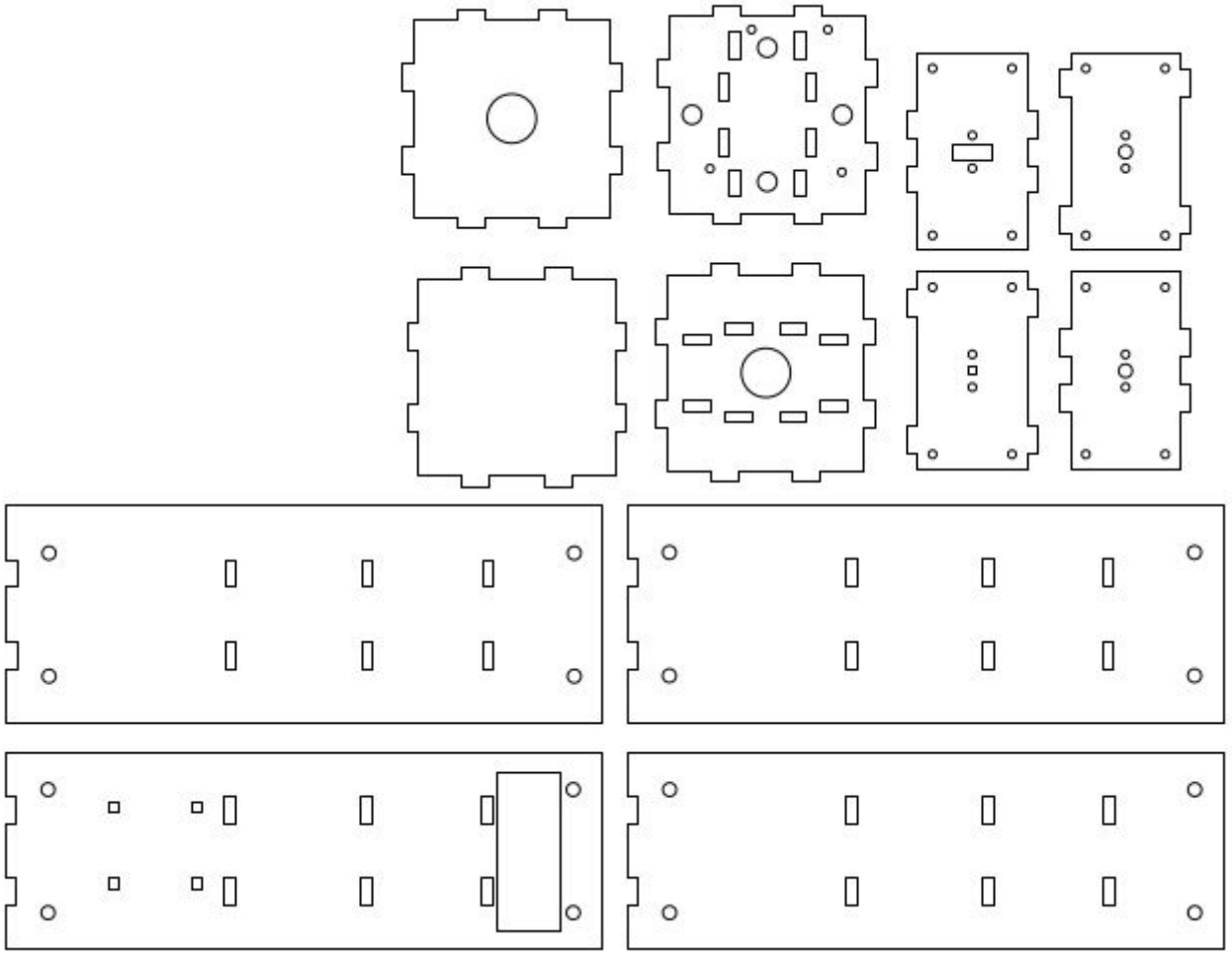
2. The buzzer does have a polarity - the longer terminal should go towards the top of the board (the top of the buzzer has a + sign too).
3. Break the PCB along the following lines (marked in red). It is intended to be snappable, but It's still quite strong, so be careful.

4. While soldering, make sure the screw terminals on the main board face outside.

Laser Cutting

Laser cutting is among the simplest machining tools for beginners, or for rapid prototyping.
It's as easy as 1-2-3!

1. Find a laser cutter in your city (online sources might exist, but you're on your own there)

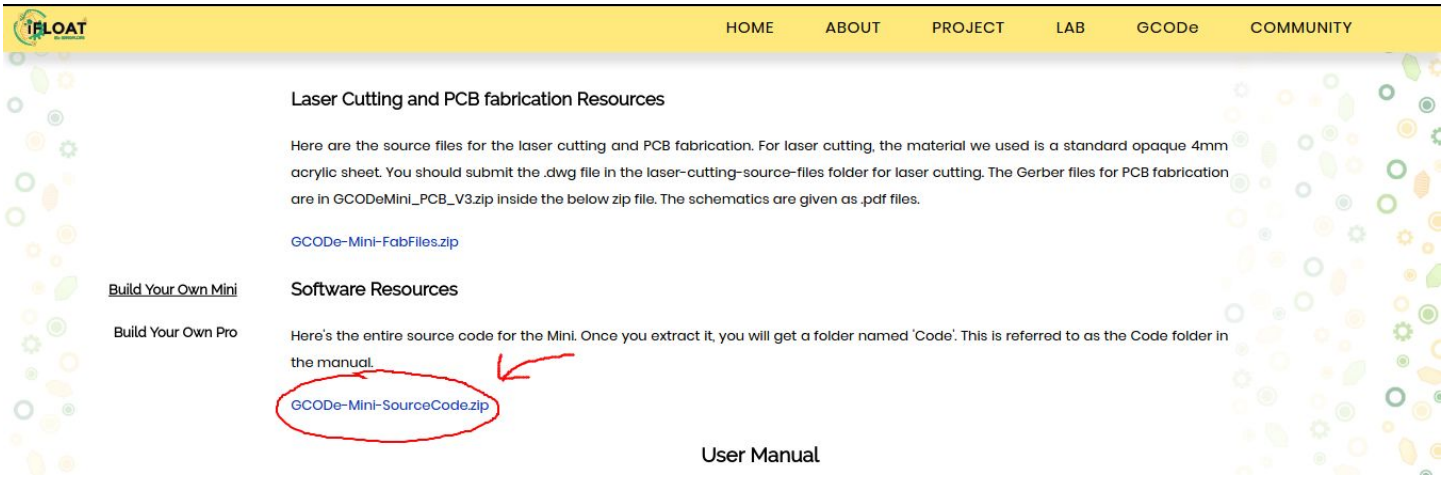


2. Submit the .dwg file in your Fabrication Files folder, and the 4mm acrylic sheet
3. Let it be cut, and take the parts home!

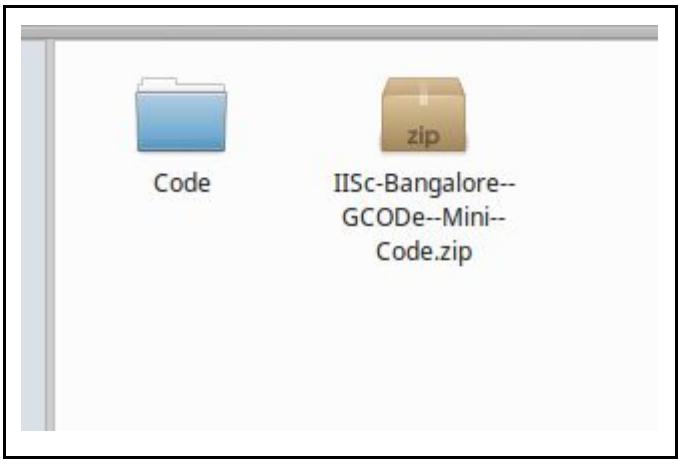
Machine Assembly and Calibration

Get the code

Download and unzip this Folder for the source code.



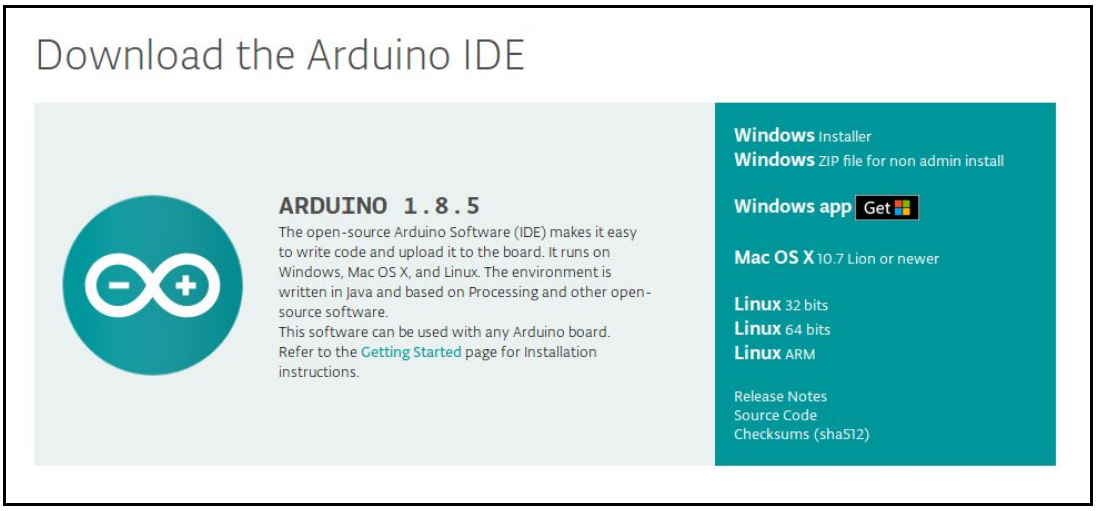
We will call the folder to which you extract this as the “Code” folder.



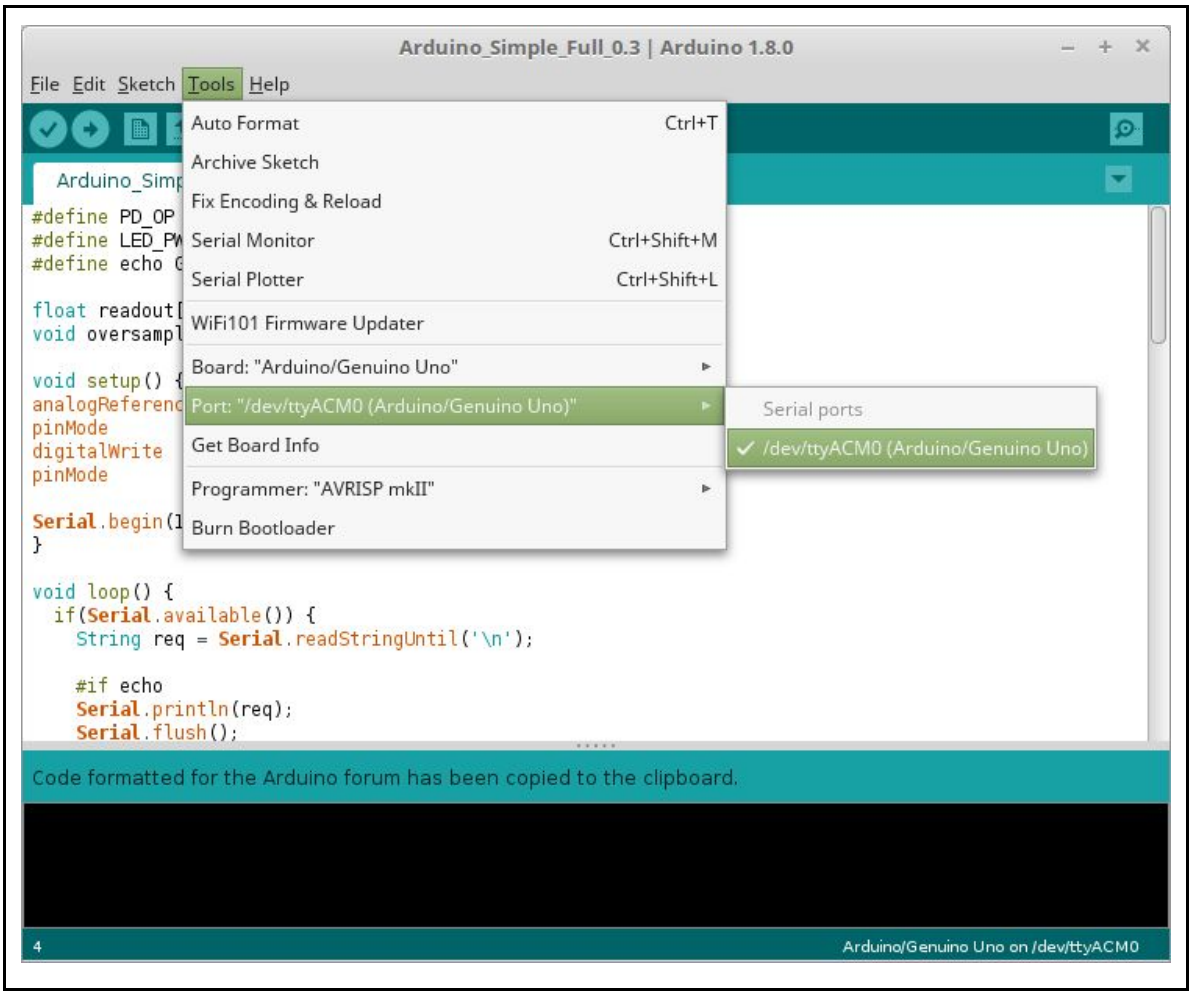
Arduino code upload

First, we’ll have to upload the code to the Arduino, because the PCB will damage the Arduino board[1] if this is not done first.

1. Obtain the Arduino IDE for your OS (Windows/Linux[2]) from https://www.arduino.cc/en/Main/Software. Don’t worry, it’s not scary like some other IDEs.



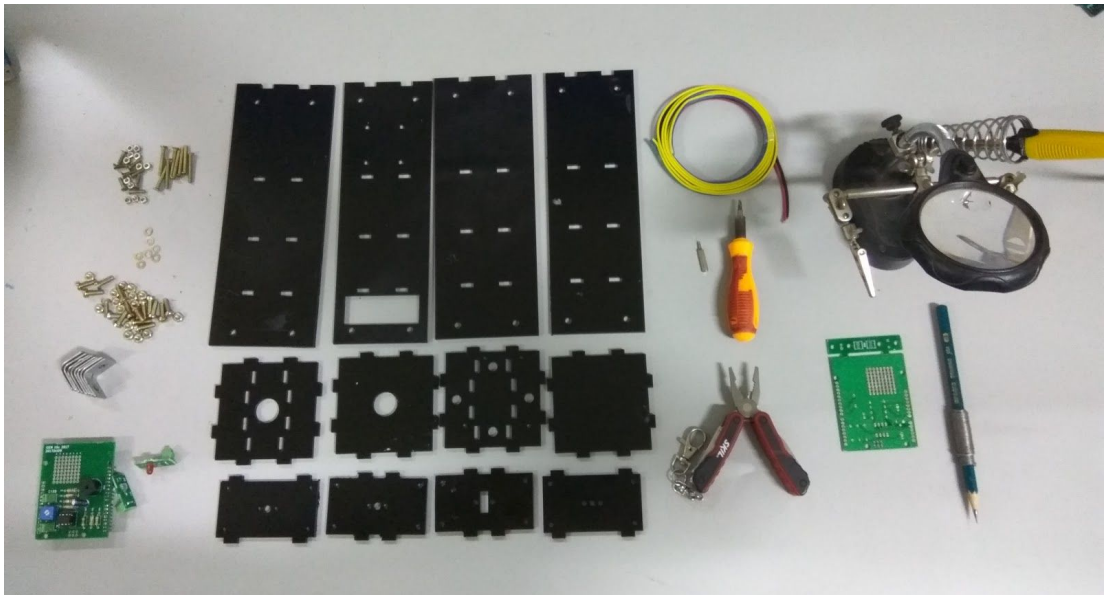
2. Install it. (Mostly involves hitting next)
3. In your Code folder, open the GCODe_Mini_Setup.ino file in the folder of the same name.
4. Double-click it to open it in the Arduino IDE.
5. Connect the Arduino Uno to your PC using the USB cable.
6. Ensure that the device is selected under Tools->Port. If it hasn't, just click on it to select it.



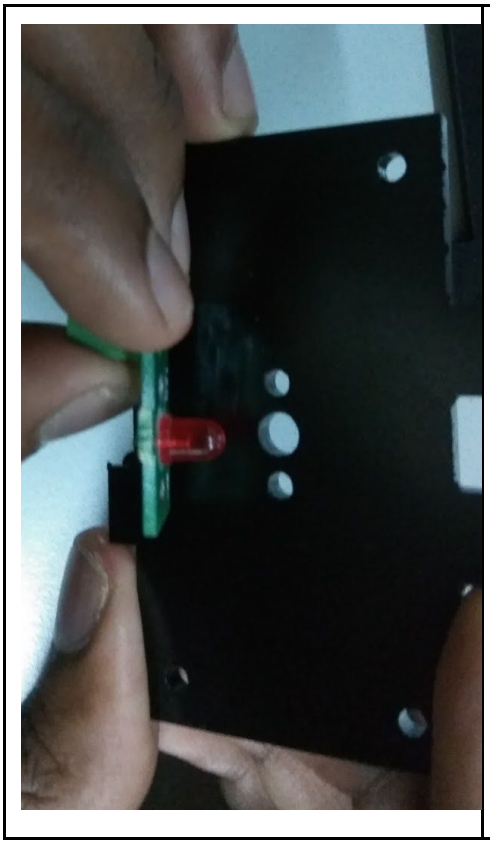
7. Hit the upload button .
8. When it's done uploading, you can disconnect your Arduino.

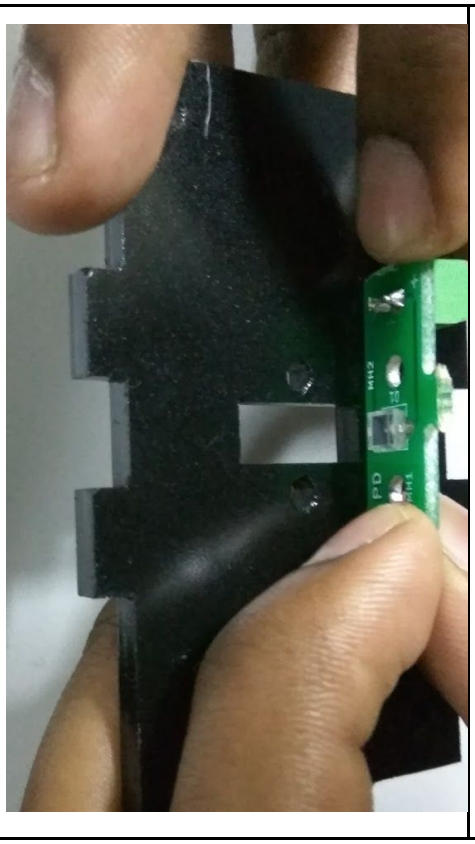
Physical Assembly

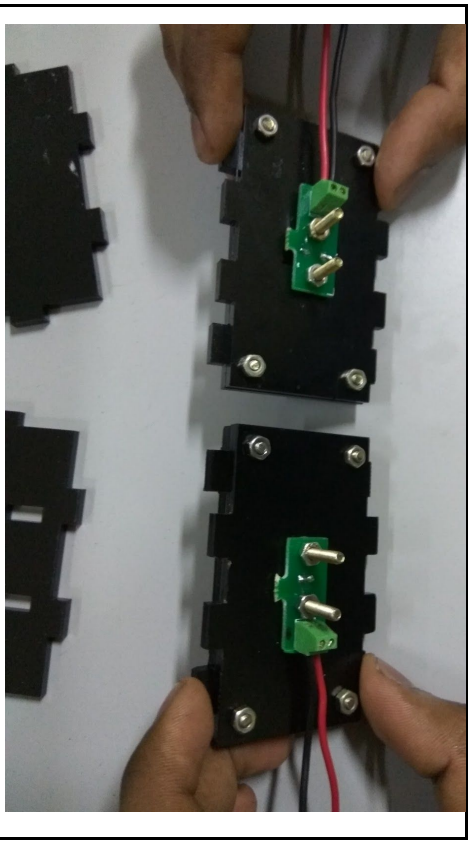
1. Break the PCB and solder the parts. Pay attention to the polarity of the LED and photodiode. It's been marked out in the silk screen (white) and in copper (raised and kinda silvery)
2. Lay your parts out



3. Attach the LED and photodiode PCBs here. They will not sit flush against the plastic. Don't worry; this is normal. Tighten it just enough that it sits properly. Not too much so that the PCB bends.

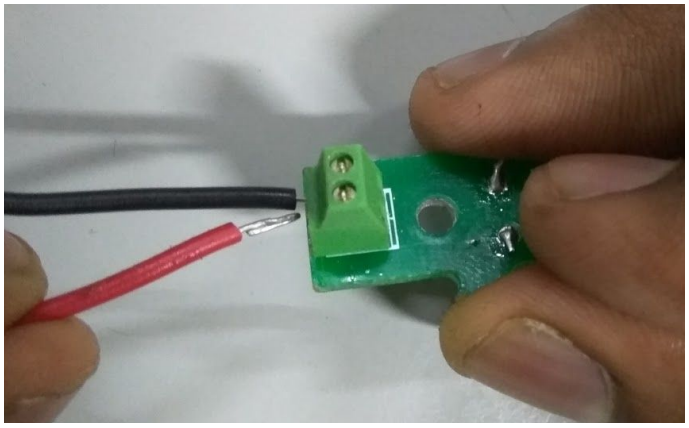


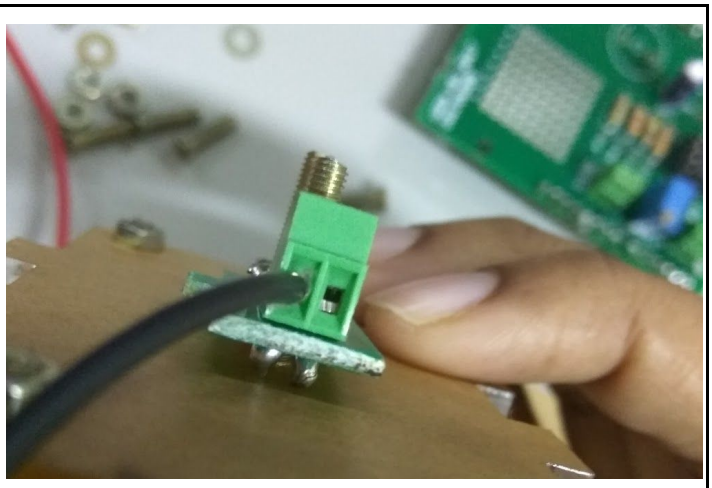




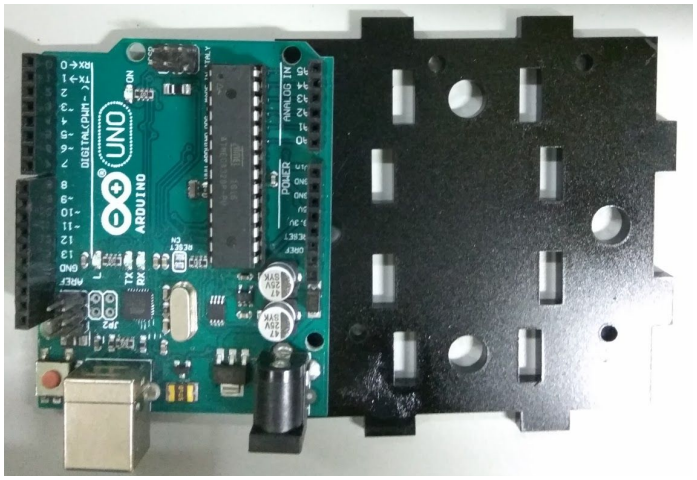
(don't attach the wires yet - we will attach them at a later step)

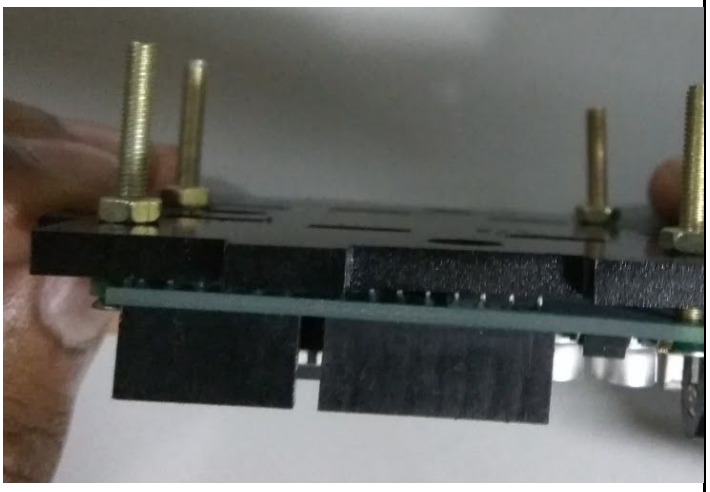
4. Attach 20 cm of wire to each terminal: red to + and black to -. Make sure to bend the end as shown in the image. Also, you may have to turn the screw so as to expose the cavity in which the wire is supposed to go. In the figure on the right below, the wire goes in the visible cavity, above the metal. Protip: Pass the wires through the circular holes (the ones on the outside of the 4 rectangular holes) in the faceplate shown below right now, before attaching the Arduino. It'll make life easier.



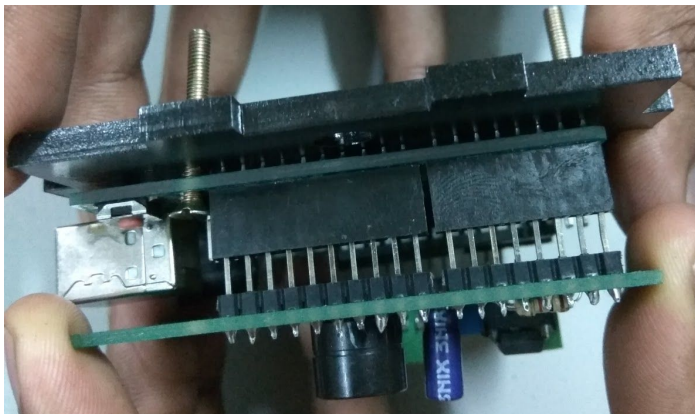


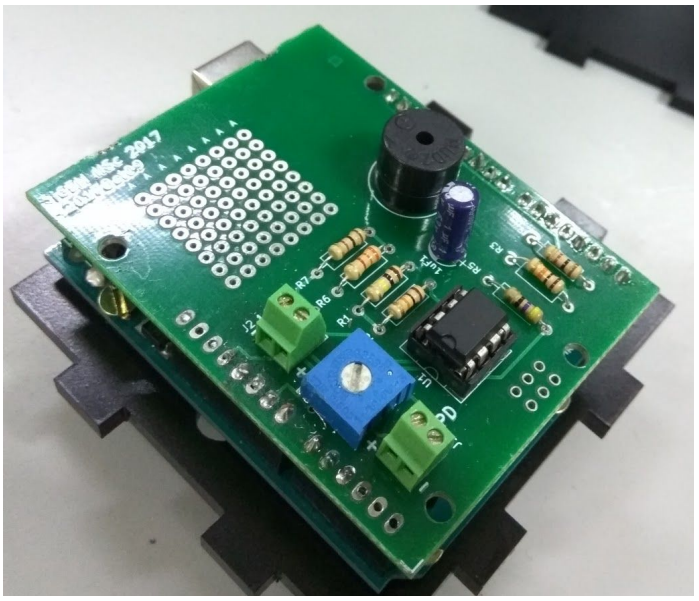
5. Attach the Arduino Uno to this piece using the 3mm nuts and bolts. If it doesn't seem to align, try flipping the black plate around.



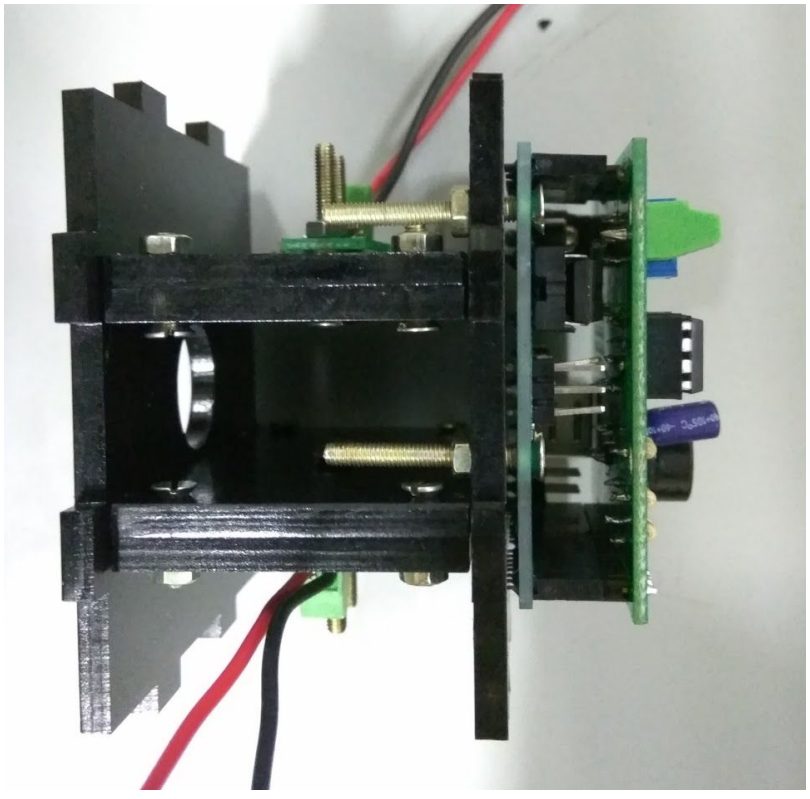


6. Attach the PCB to the Uno. Don’t be misled by the image on the left - the PCB is intended to go all the way in. Some force may be required.

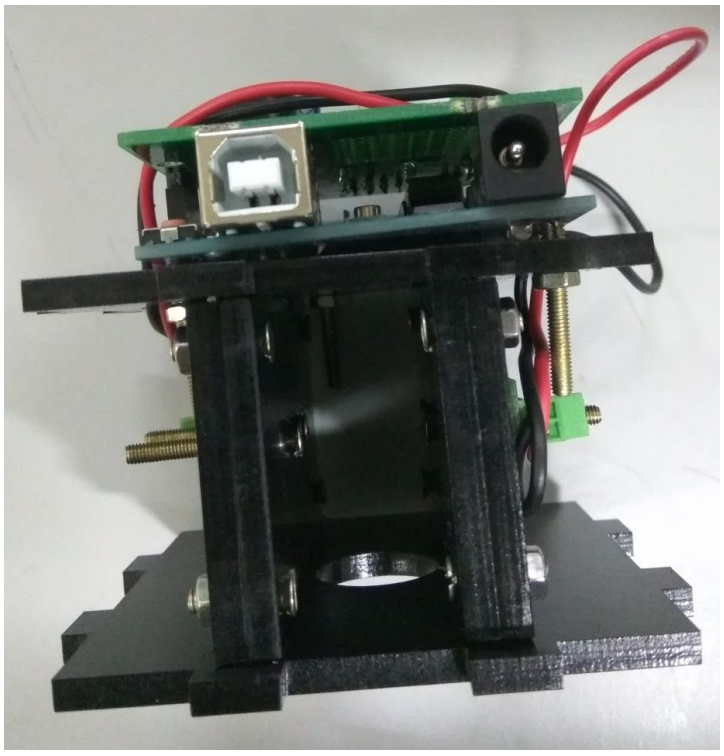




7. Assemble the optical chamber

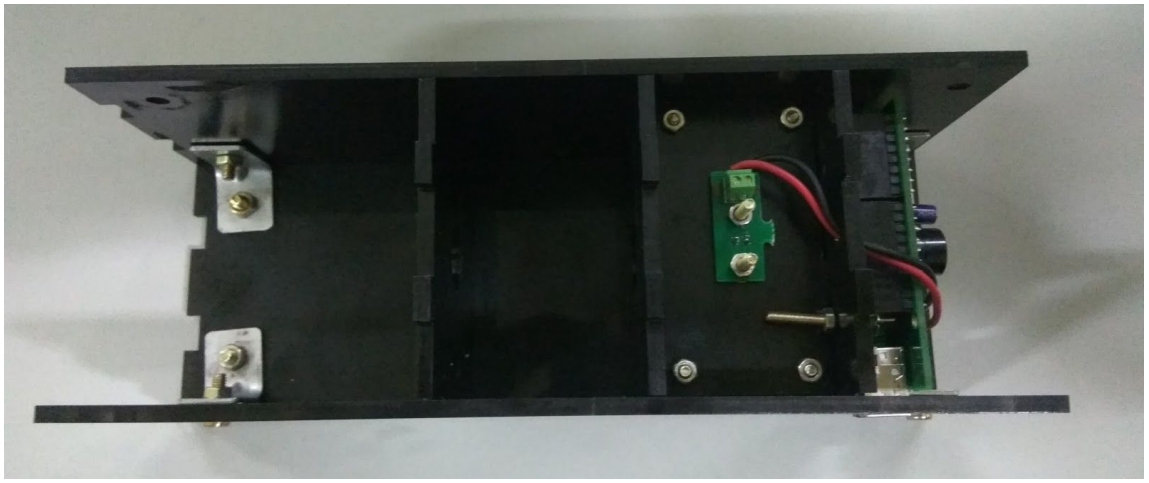


8. Connect the wires from the LED and photodiode to the screw terminals on the PCB marked “LED” and “PD” respectively, and make sure to get the polarity right here too. Red is + and black is -



9. Attach the optical chamber to the faceplate with a hole (in this orientation). Then attach the test-tube holder plate to it. Finally attach one of the wider plates to the right, with L-bends.

10. Attach the remaining faceplates

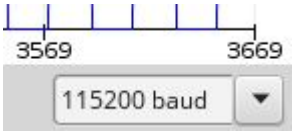


11. Put the lid on, and your GCode Mini is assembled!

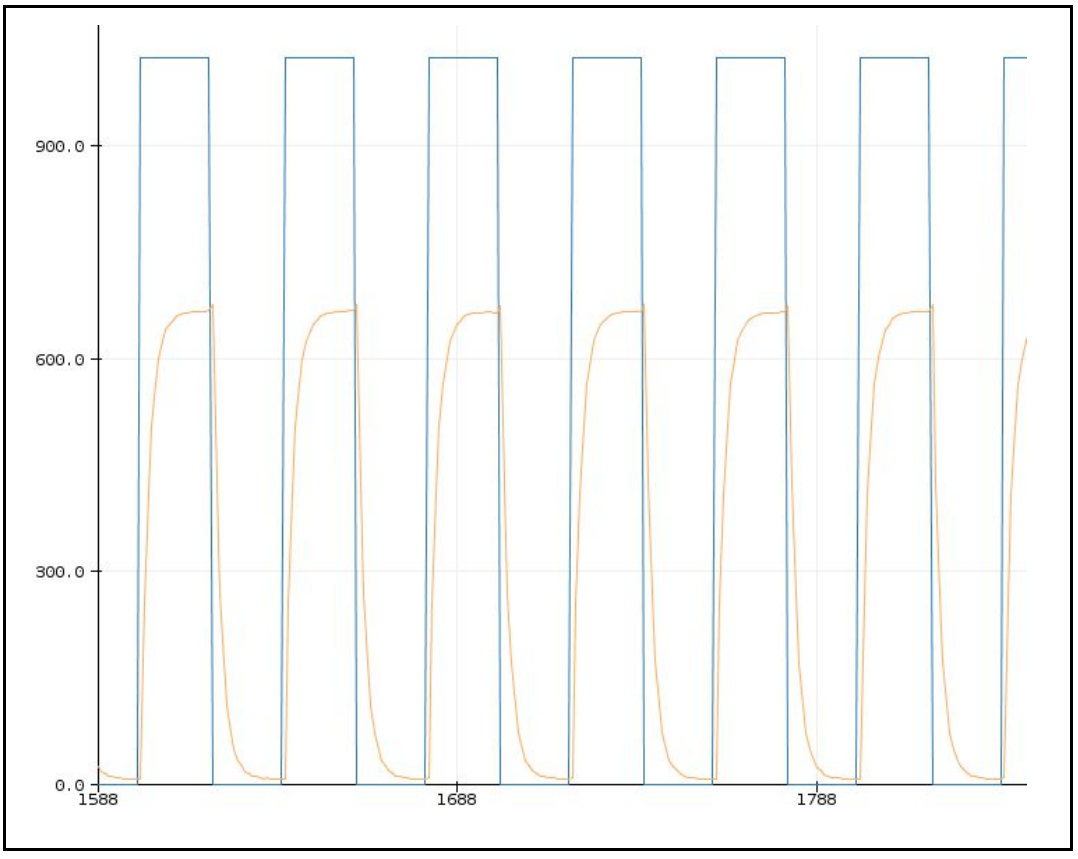
PS: If the lid doesn't fit, try taking the L-bends out and bending them out to make them 90 degrees or more obtuse.

Gain Adjustment

1. Make sure your GCODe is connected to your PC.
2. Open up the Arduino IDE
3. Open the serial plotter (Ctrl+Shift+L). You may have to set the port again (under Tools->Port). You may also have to set the baud rate to 115200.



4. Rotate the variable resistor (the blue cuboid on the PCB) using a screwdriver until your graph looks like this (yellow line should saturate at about 700).
    - If the wiggly line only stays above 900, you probably flipped the polarity on the photodiode.
    - If the wiggly line basically stays at 0, you have probably misconnected the photodiode wires.
    - If it doesn't oscillate with the straight line, you have probably either flipped the polarity on the LED, or haven't connected the wires properly.



4. When you're done with the calibration above, open the GCODe_Mini_User_0.3.ino file in the folder of the same name
5. Hit the upload button

Test tube Preparation

1. You'll need a test tube to put your culture in. Unfortunately, not all test-tubes are the same - manufacturing tolerances are fairly large for our purpose.
2. Insert your favourite test tube, and wiggle it around to see if it wobbles. If it does, this is bad and needs to be corrected. Some tubes may not fit. You may need to find another one.
3. Wrap tape around the test tube at heights corresponding to the two test-tube holder pieces. (Carefully, and not too much - make sure you can take the tube out without damaging the tape). Avoid the light path between the LED and photodiode

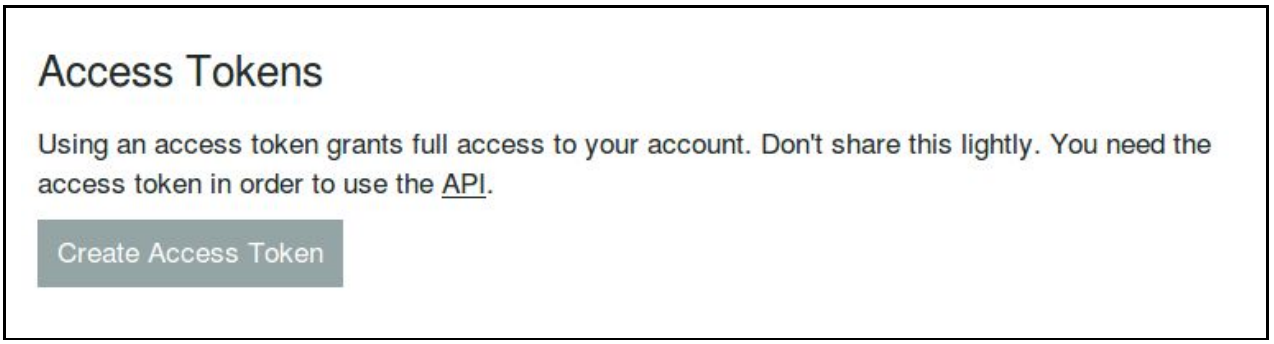
4. Insert the test tube again and make sure that it does not wobble. If it does, add more tape.

Software Installation

Get your Pushbullet access token

We use Pushbullet to send notifications to your device.

1. Sign up for an account at https://www.pushbullet.com/.
2. We need a Pushbullet access token; so log in and go to https://www.pushbullet.com/#settings/account, and scroll down until you see:



3. Hit “Create Access Token” and then copy the text in the black box somewhere.
4. You will need to have the Pushbullet app on the device you wish to receive notifications on, and will need to be signed in to your account on that device.

Installation on Ubuntu

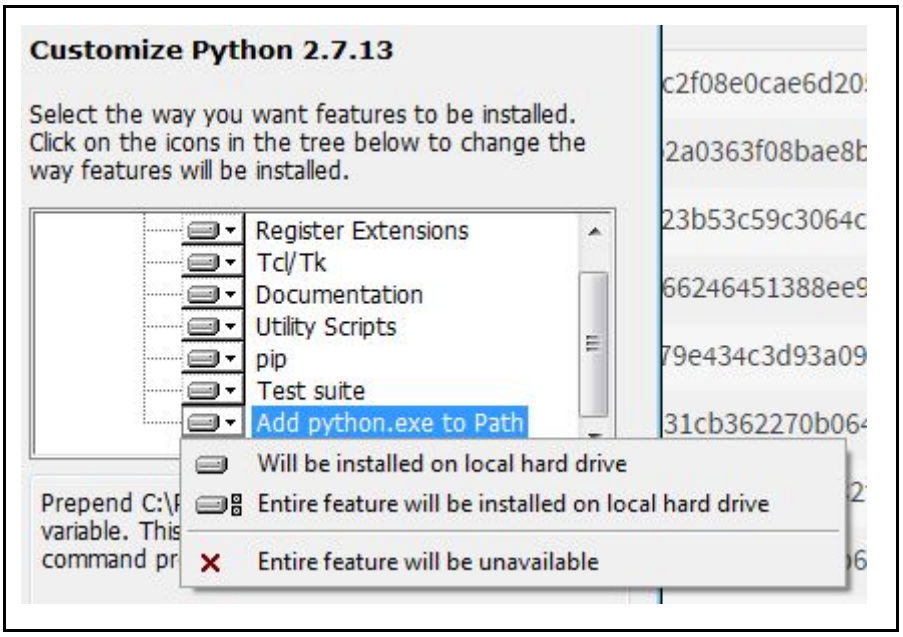
1. Open the Code folder, right click in a blank corner, and hit “Open in terminal” (Or you could navigate to that location using the terminal)
2. Run:

```
chmod u+x ./install_gcode_mini_ubuntu.sh
sudo -H ./install_gcode_mini_ubuntu.sh
```

3. Connect the mini to the PC
4. Double-click the “GCODe Mini GUI Ubuntu” file to run the GUI

Installation on Windows

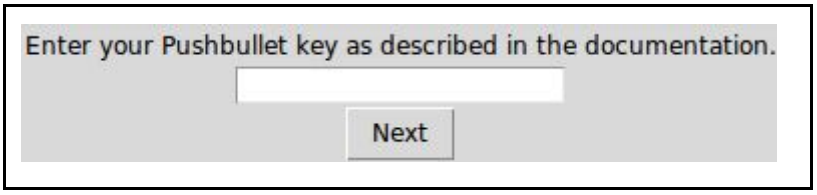
1. Get Python 2.7.13 from https://www.python.org/ftp/python/2.7.13/python-2.7.13.msi
2. Make sure to hit the “Will be installed on local hard drive” feature for “Add python.exe to Path”



3. Reboot your PC (or log out and log back in)
4. Go to your Code folder, and double click “install_gcode_mini_windows.bat” to install the required dependencies
5. Double click “abode_gui.py” to run the GUI

Setting up the software

Paste the access token you obtained in the previous section and hit next. It might take a while in the first run.



You may have to click on the device name in the device selection screen and hit next if you have multiple Minis connected. You can reset the access token by deleting the file named “pbkey.txt” in your Code folder.

Footnotes

1. The internal 3.3v is connected directly to the AREF pin on the PCB. This is to minimise the influence of voltage fluctuations on the USB port - USB isn’t supposed to give clean 5V to anyone. We elected not to connect it in a “safe” manner because that could cause variations on the AREF, which will make the readings bad.
2. We haven’t tested our software on Mac yet. No reason why the hardware shouldn’t work though.