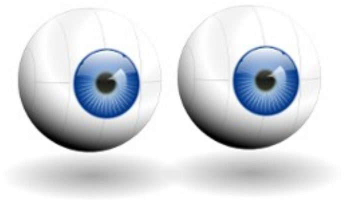
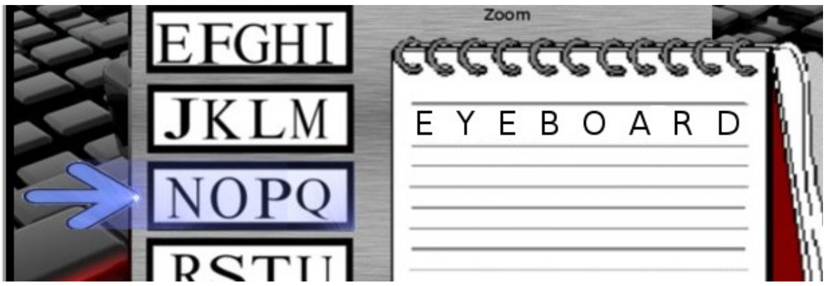
INTELSATH

The Eyeboard: Guide





http://www.intelsath.com

The Eyeboard guide: DIY Kit
by Luis Cruz rev 2

Table of contents

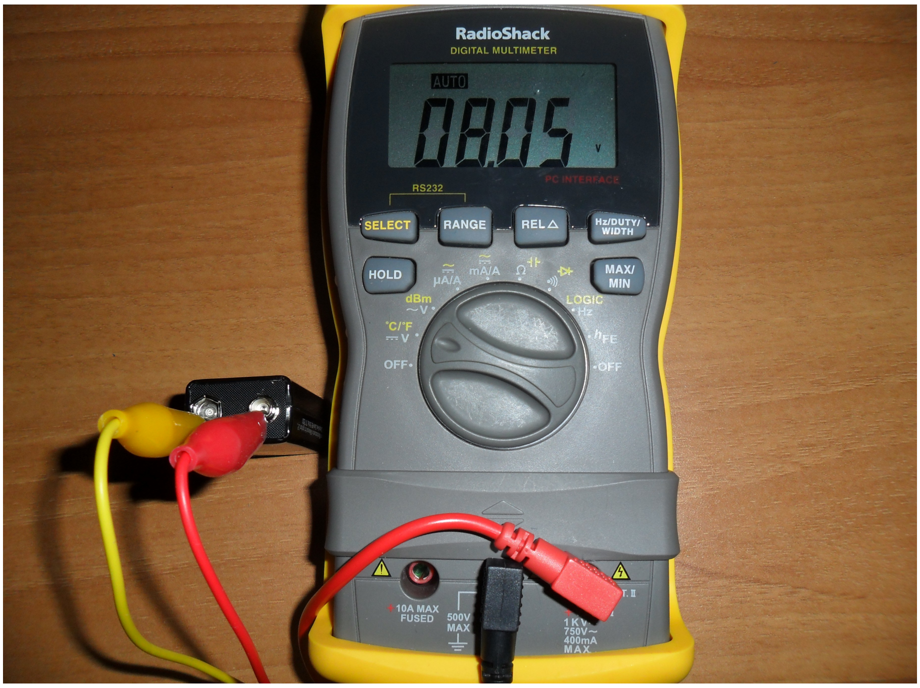
Introduction

The Eyeboard is an inexpensive eye tracking system originally developed to aid people with disabilities. The eyeboard is an interface that lets the user communicate with the computer with only eye movements. On this tutorial we'll show how you can combine some electronics and software to make this interface that will let the user write on the computer with their eyes. The eyeboard, however, is not limited to only forming words with eye movements, the applications of the eyeboard are limitless. Hopefully you will be able to apply the concepts you learn on this tutorial to many other applications with the eyeboard!

This tutorial guides you through the building process of the eyeboard. We tried to keep this guide as simple as possible so anyone with a limited electronics and software programming background can build the eyebord themselves. However, we'll review most of the important technical concepts you will need to understand how this project works. The goal of this tutorial is not only to teach you how to build the eyeboard from scratch but guide you through the process of building a real-world application using digital electronics and software programming. Thus we hope that you get to learn by doing, so you can apply the concepts you learn on this tutorial on some other cool projects! Some electronics and C programming experience is recommended but not required.

Electrooculography

First we need to understand how the eyeboard works. The basic idea behind the eyeboard is the use of this technique called Electrooulography (EOG). Aside from this complicated name, the idea is pretty simple. There is a small electrical potential difference in the human eye, between the retina and the cornea. Putting it in more simple terms, the eye acts like a battery.



Just as we can measure the voltage between the positive and negative terminals of a battery with a tester, we can measure the voltage between the retina and the cornea of the human eye. And essentially the eyeboard is just like a tester, but instead of measuring the voltage of a battery, we'll be measuring the voltage of the eyeball.

The potential difference of the eyeball depend on many things, but for simplicity's sake we will assume that that voltage is fixed and that it doesn't change over time. So you might be wondering how does measuring the voltage of the eyeball can help us determine the movement of the eyes. Well to understand this we need to review the concept of voltage and an electric dipole.

Introduction to Electricity

Before we keep going any further with the design of the eyeboard, we need to understand a little bit more about electricity and voltage. As you have seen so far, the terms *voltage* and *electrical potential (or potential difference)* are interchangeable. Essentially, voltage is the difference in electric potential energy between two points. Voltage is always relative, so we measure voltage with respect to one point. Electricity is just really charged particles. Everything in the universe is made up of atoms, which are made up of some particles called electrons, protons and neutrons. The protons are positively charged (+), and the electrons are negatively charged (-). Equal charges repel, and appositive charges attract. In order to have a voltage, we need electrons to move towards the protons, which they travel inside a conductor. When you move electrons, you have a current, which means you have electricity.

One of the most fundamentals laws of electricity, is ohm's law and it's worth mentioning it. It basically says that the more voltage you have, the higher current you have and current is decreased by resistance. Ohm's law states that $V = IR$. Where **V** stands for voltage and it's measured in volts (V), **I** stands for current and it's measured in amperes (A), and **R** stands for resistance and it's measure in ohms (Ω).

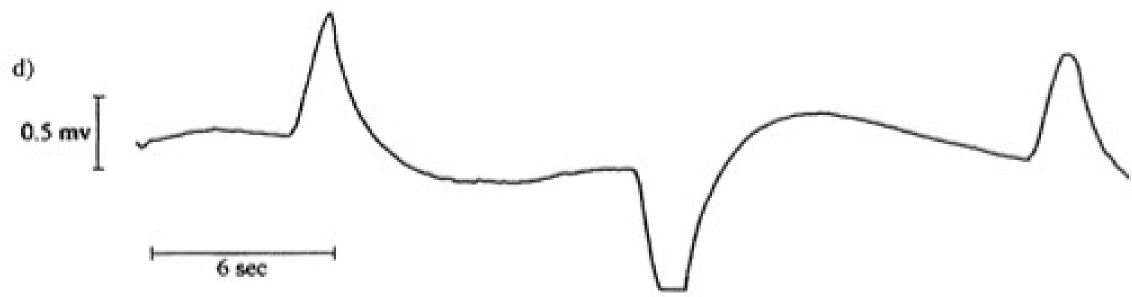
The eyeball as a dipole

A electrical dipole is just the separation of positive and negative charges. If we have two charges in space, a negative charge (an electron) and a positive charge (a proton), separated by a distance, we have a dipole. Basically there is a small voltage between those two charges, which is called the electrical dipole potential. That electrical potential is inversely proportional to the distance. So the larger the distance, the smaller the voltage.

Unfortunately the physics of electricity and magnetism is more complicated, however now that you have the basic understanding of how electricity works you will have a better idea of how the electrooculography works.

So in short, the eyeball acts as an electrical dipole, with the negative pole at the retina and the positive pole at the cornea. We can measure the potential difference in the eyeball with a pair of electrodes. An electrode is a conductor that is placed on a non-metallic material, such as the skin. When the eyes are moved from the center to one side, one electrode "sees" the positive voltage and the other electrode "sees" the negative voltage with respect to the reference electrode, which can be placed on other part of the skin, such as the hand. Then we can monitor the change in the voltage differences, so that way we can know when the eyes are moved.

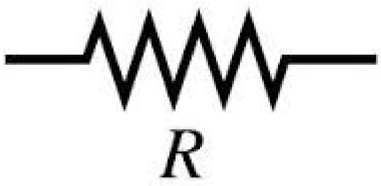
The EOG signal is just the change in voltage difference between the electrodes. If we saw the signal on an oscilloscope (a tool that let us observe varying signal voltages as a graph), we would see the changes in the slope of the signal. Depending on what we choose as our reference and how we connect the electrodes, we would have a negative and positive slopes when moving the eyes to the right or to the left.



Electronic components and diagrams

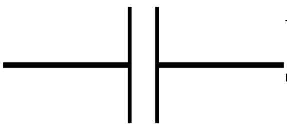
We will next discuss how to build the actual electrical circuit of the eyeboard. We will often show the connections you need to do with a schematic diagram. A schematic diagram of an electrical circuit is simply a representation of all the electronic components of the circuit and their connections. At first it might be a bit confusing, but after you have seen lots of diagrams you will get used to it. So here is a review of the symbols that we will use to represent each component and their description.

Resistors



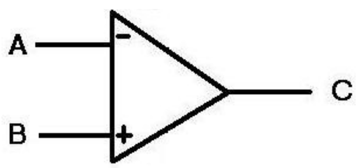
A resistor is an electrical component that implements electrical resistance in a circuit element.

Capacitors



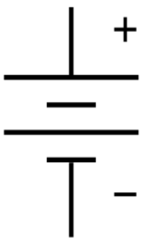
A capacitor is an electrical component used to store electrical energy in it.

Operational Amplifiers



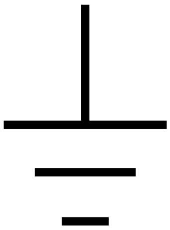
An operational amplifier is a powerful tool that let us do mathematical operations in circuits.

Power supply

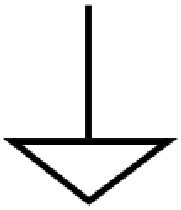


This is the symbol for the power supply, which can be a battery for instance.

Ground



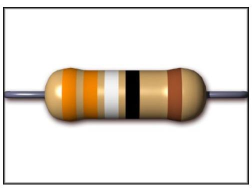
Ground is usually the reference point in an electrical circuit from which voltages are measured. We commonly use the negative terminal of the power supply as ground (GND).

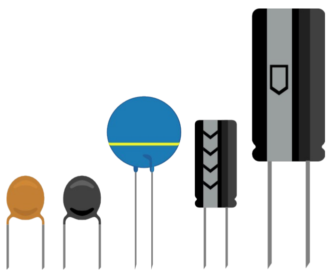


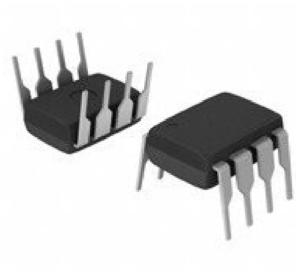
Another symbol for ground. We will be using this symbol to differentiate between ground (zero volts) with virtual ground.

Note: If we require to connect a component to the negative terminal we usually denote it as GND. If we require to connect a component to the positive terminal we denote it as VCC, VDD, V+ or Vs+

Below is a picture of a resistor, some capacitors, and operational amplifiers ICs (Integrated Circuits, or chips) respectively.







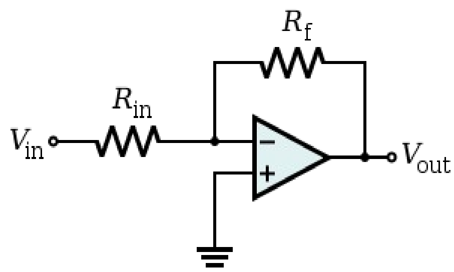
Getting started

So now that we understand the fundamentals of electricity and we know how we can approach approach the development of the eyeboard, we are ready to start building it! BUT, before starting the circuit we need to understand one last thing. It is probably the most important component of the eyeboard, it's the operational amplifier, it's the basic building block that we will use to make the eyeboard.

The operational amplifier

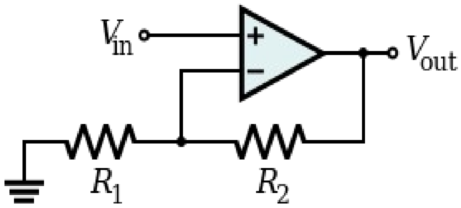
An operational amplifier, or op-amp for short, is a very powerful tool that not only has a high gain that let us amplify electrical signals, but also it let us do many mathematical operations. There are many applications of the operational amplifier, we will be using this versatile electronic component for a couple of things in our circuit. Some of the most important applications of the operational amplifier are described below.

Inverting amplifier



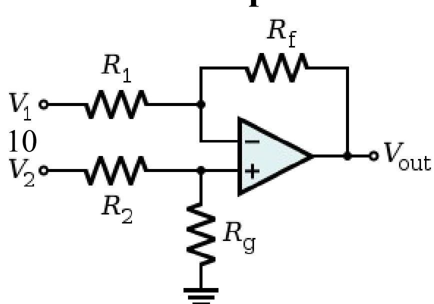
The inverting amplifier is a common application of an operational amplifier. It is a circuit that amplifies the signal and inverts it.

Non-inverting amplifier



The non inverting amplifier is another common application of the operational amplifier. Unlike the inverting amplifier, the signal is not inverted just amplified.

Differential amplifier



The differential amplifier is a very important application of the op amp that we will use to build the eyeboard. The differential amplifier amplifies the difference of both input voltages.

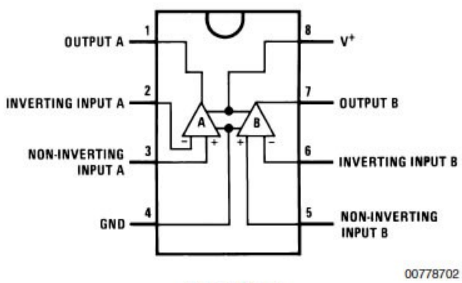
The theory behind the functionality of operational amplifiers is fairly complicated. However, now that we know the basics of the op amps, implementing any of the applications is pretty straightforward. To get a better understanding of how an operational amplifier work, let's learn by doing!

Implementing a non-inverting amplifier: Example

We'll use the LM358N to amplify an input signal by some constant. By doing so, you will get to see how to implement an amplifier using an op-amp, what better way to learn more about op-amps than by actually using them?

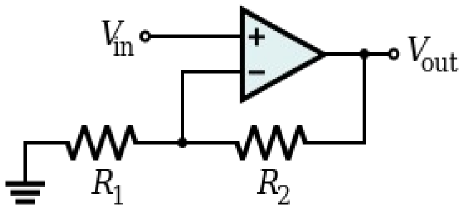
The LM358N is an IC that has two operational amplifiers in it. From the datasheet, we can see each pin description.

DIP/SO Package

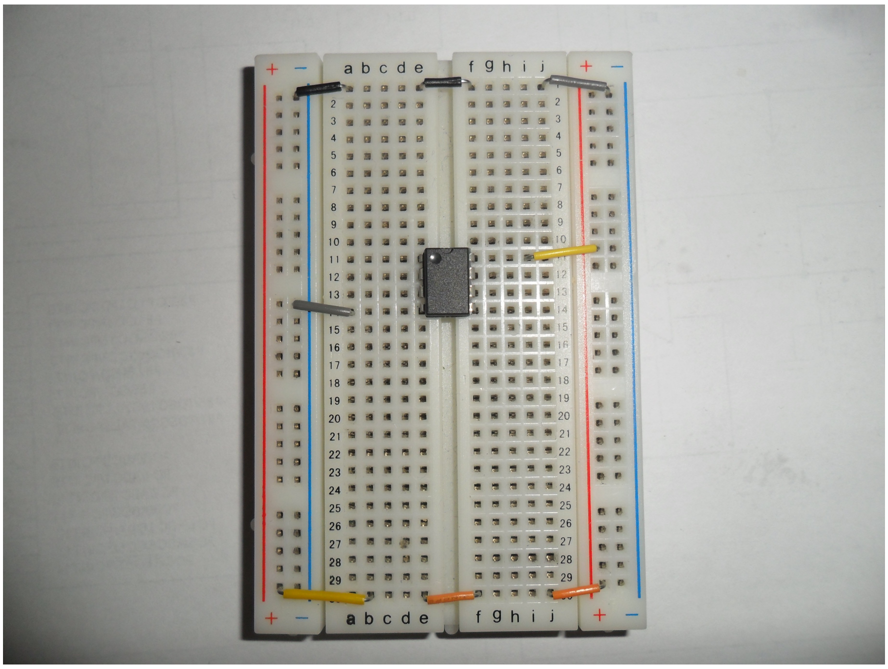


Top View

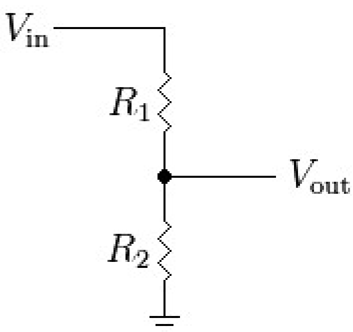
The diagram below shows the circuit we need to implement for a non-inverting amplifier. Refer to this diagram when following the steps shown below.



1) Start by connecting the LM358N to the power supply and to ground. For simplicity's sake we'll use the arduino as the power supply (we'll connect 5V pin of the Arduino to the red power rail of the breadboard as VCC, and GND to the blue rail of the breadboard).



2) Then we will use a voltage divider circuit to get a smaller voltage so we can amplify it. A voltage divider basically consists of two resistors that will give us a lower voltage. The following diagram shows how to implement a voltage divider circuit with two resistors.



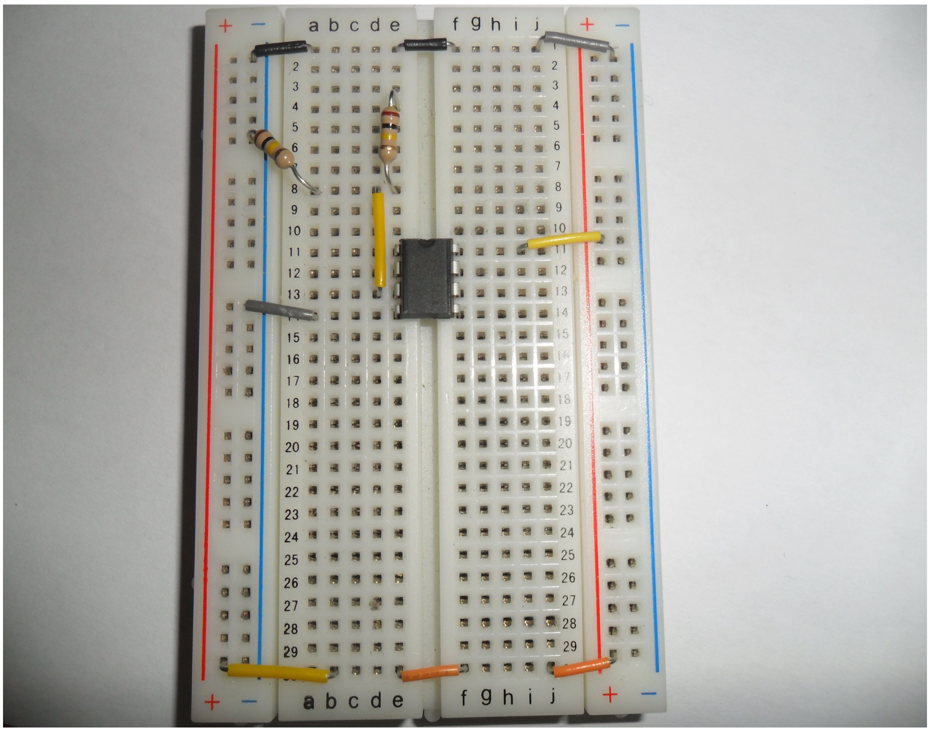
$$Vout = \frac{R2}{R1 + R2} * Vin$$

We'll use the above formula to calculate the output voltage.

So if we wanted to get ½ of the input voltage, we would need two resistors of the same value. We will use the 3.3V output from the Arduino as the input for our voltage divider circuit, then we will divide that voltage by half to get a voltage of about 1.65V. You can use two 100k ohm resistors which are included in the kit.

$$Vout = \frac{100000}{100000+100000} * 3.3V \quad = \quad \frac{100000}{200000} * 3.3V \quad = \quad \frac{1}{2} * 3.3V \quad = \quad 1.65V$$

So now that we have confirmed that with two 100k ohm resistors we can get half of the input voltage, let's connect the output to the positive terminal of the op-amp (refer to the diagram of the non-inverting amplifier shown above).

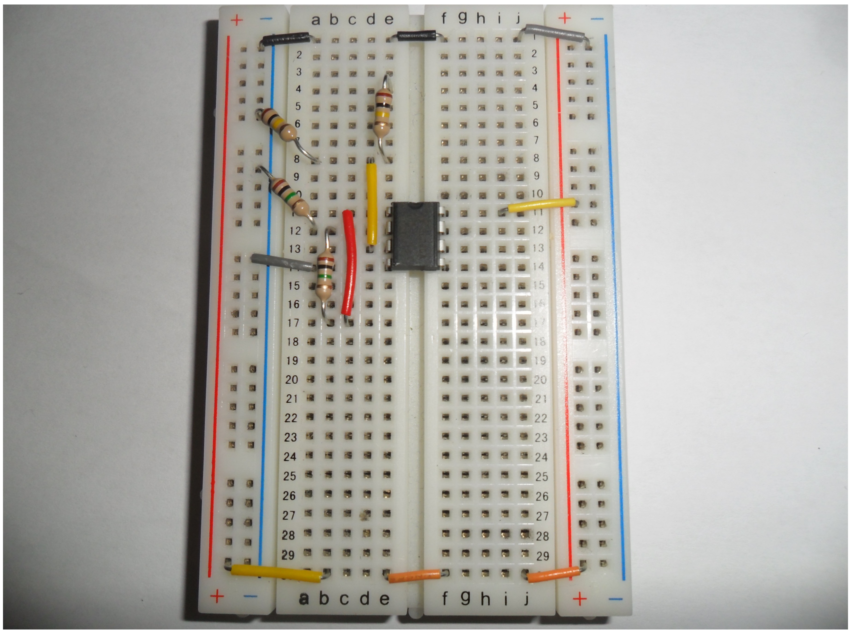


We'll then connect row 2 of the breadboard, to the 3.3V output of the Arduino, that way we'll get the 1.65V on the non-inverting (+) input of the op-amp.

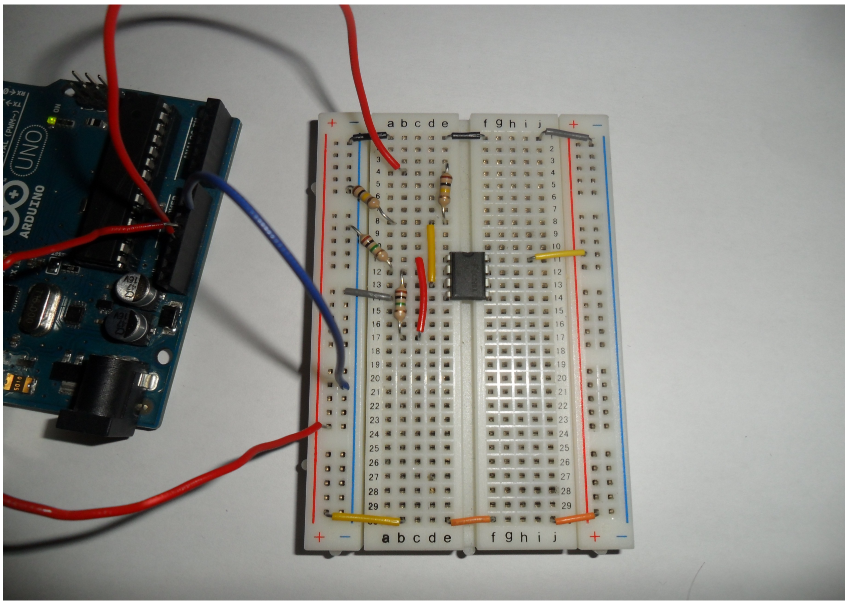
3) Now we have to connect the two gain resistors. R1 and R2. In order to calculate the gain (how much we want the input signal to be amplified), we need to use the following formula:

$Vout = Vin * (1 + \frac{R2}{R1})$ So if we wanted a gain of 2 (the output voltage is twice the input voltage), we need two equal resistors. Because our input is around 1.65V, if we wanted to amplify that by 2, we would get about 3.3V in the output. The eyeboard kit includes two 10Mohm resistors, you can use those.

$$Vout = Vin * (1 + \frac{R2}{R1}) = 1.65 * (1 + \frac{1x10^6}{1x10^6}) = 1.65V * 2 = 3.3V$$

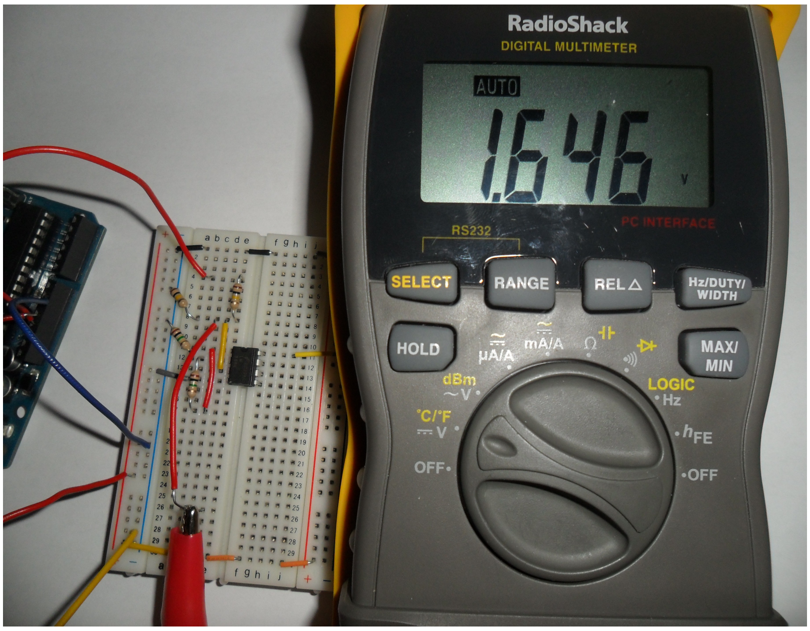


4) Now that the circuit is complete, let's connect the Arduino's 5V output to the red power rail of the breadboard, GND to the blue rail of the breadboard, and the 3.3V to the input of the voltage divider circuit (on the above picture is row 3 of the breadboard).

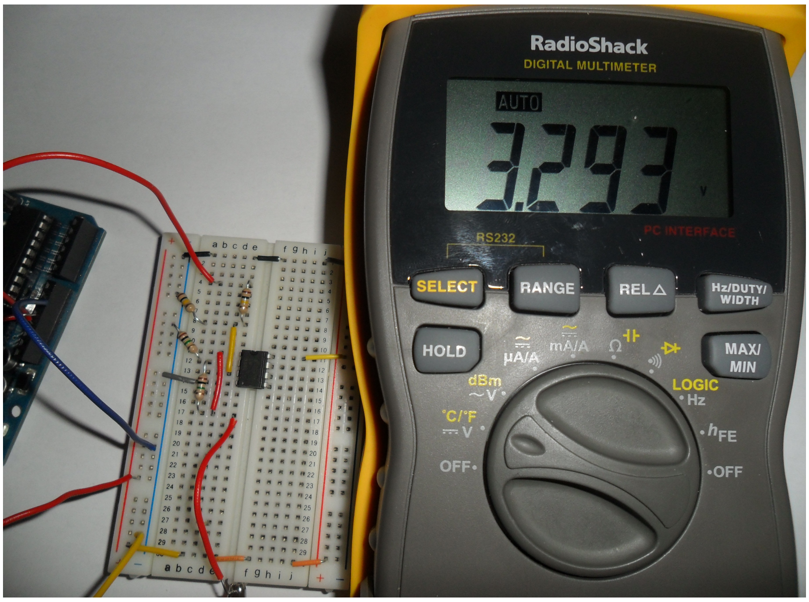


5) Now let's test the circuit. If you have a tester you can measure the voltage of the input of the op-amp (pin 3), and then measure the output of the op-amp (pin 1). The output should be the double of the input voltage.

Op-amp input:



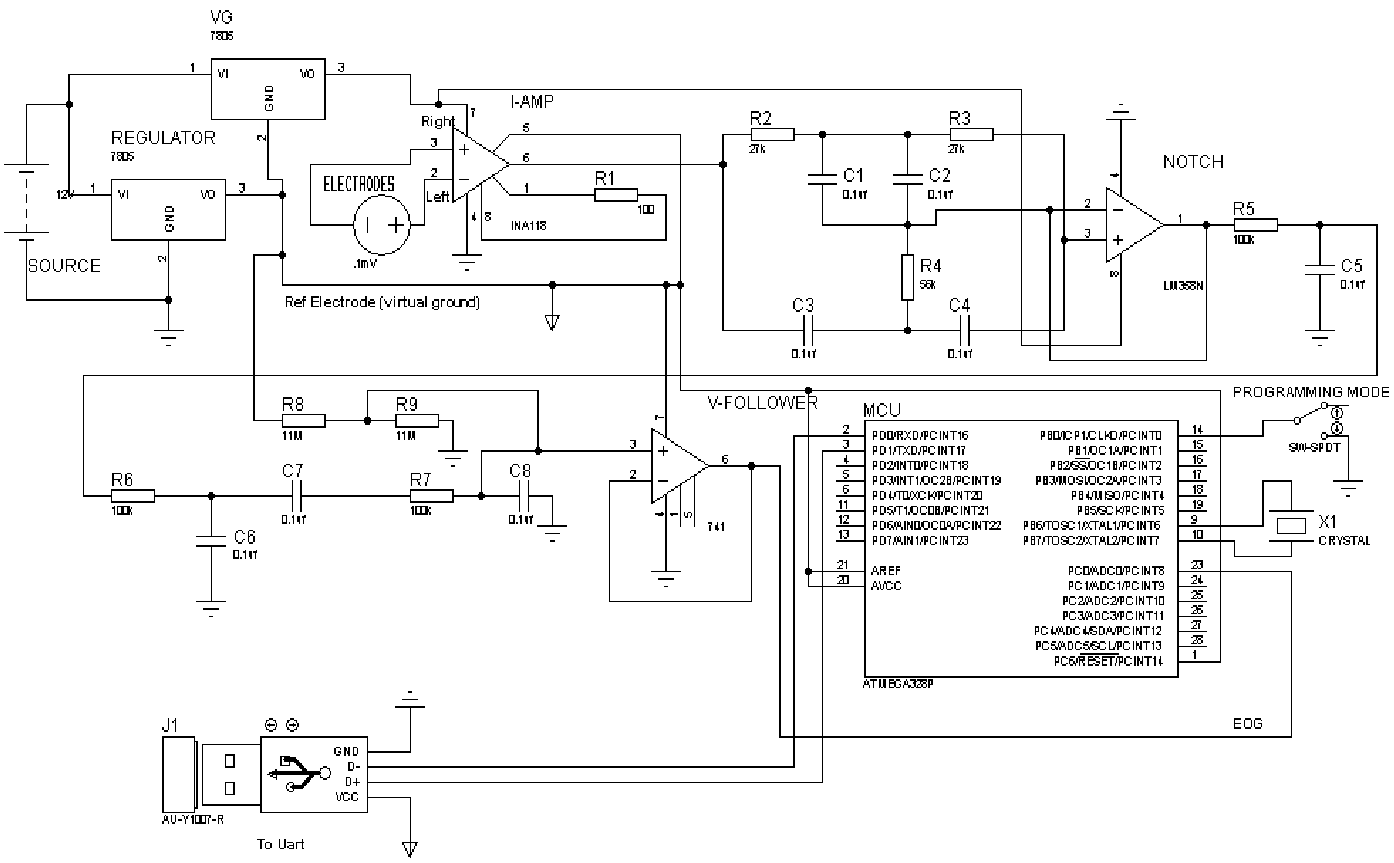
Op-amp output:



So the circuit works! We have doubled the input voltage with an op-amp. We got the same 3.3V that we used as the input for the voltage divider. This is correct, since we first divided the voltage by 2, and then multiplied that voltage by 2 again, we got the same result. We decided to do this to make it simpler, but we can amplify any voltage by any gain we want by using certain values of resistors on the op-amp. Now that you have seen it yourself, the op amp will be the basic building block of the eyeboard. We will use it to amplify the very low voltages that we will read from the electrodes sensing the low voltage differences from the eyeball.

The general diagram of the eyeboard

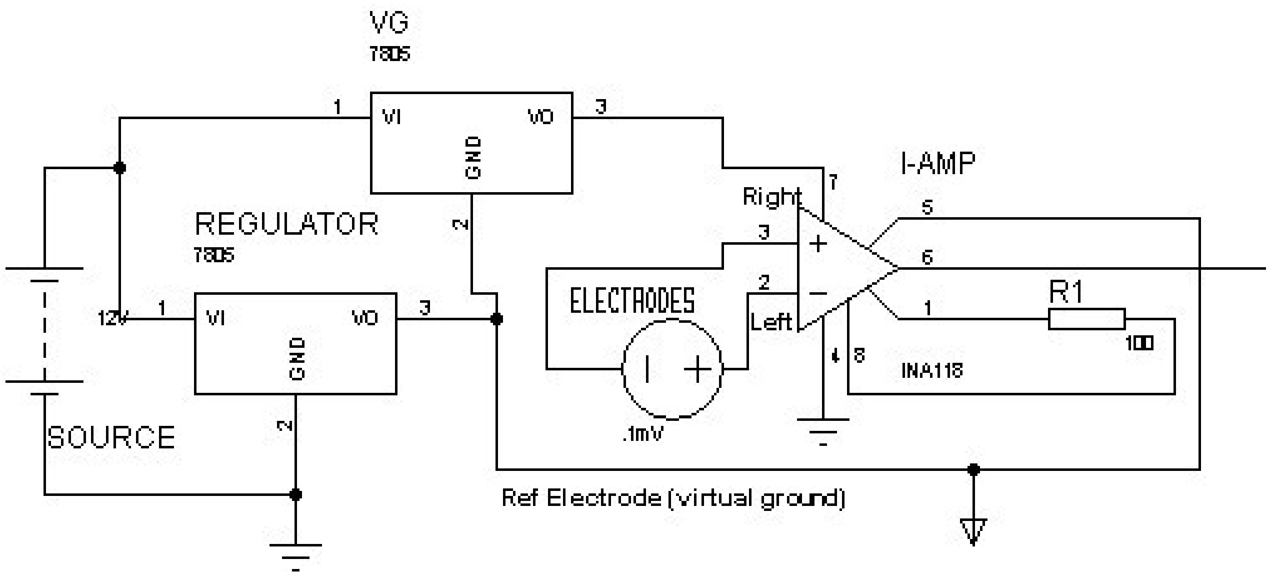
Let's start by taking a look at the general schematic diagram of the completed eyeboard circuit.



It might look a bit complicated, but let's divide the circuit into different sections. The whole circuit is composed of three different stages: the amplifier section, the filtering section and the reading section where we read the signal and use it.

We will start with the **The amplifier stage.** The voltage that we will be measuring across the eyeball is pretty small. It is usually in the order of microvolts, or 10^{-6} volts. Therefore, we will need to amplify that signal to be able to read it with a computer such as a microcontroller. To accomplished this, you guessed it, we will use operational amplifiers. Since we need to read the changes in the voltage differences, we will use a different type of amplifier than the one we implemented before, it's called an instrumentation amplifier. An instrumentation amplifier is basically a differential amplifier, but usually it has a higher gain and a very high CMRR (Common mode Rejection Ratio) which is just a measure of how noise that is common in both inputs is reduced. When we are working at such low voltages, we need to eliminate the noise as much as possible because we don't to amplify that unwanted signal!

First let's implement the following circuit, which is the first stage of the eyeboard.



Refer to the above circuit diagram for the following steps of the building process.

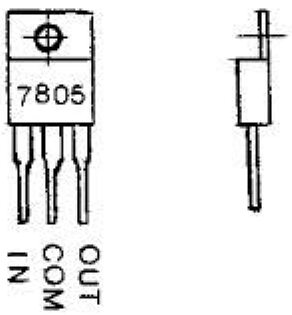
The 7805 voltage regulator

A 7805 is a voltage regulator that makes sure that the voltage we use in the circuit is always 5V. The eyeboard includes two of these voltage regulators. Usually operational amplifiers work with dual power supplies. That is, two power supples, so we can have +VCC and -VCC (for instance +5V, and -5V). However, the INA118P and the LM358N are both single power supply op-amps. Because the voltage difference of both electrodes is not zero (there is always going to be an offset), that offset gets amplified too. This could lead to amplifying the offset and saturating the op-amp so that the output voltage is just VCC, and not the EOG signal we want. This is where having a dual supply would help. Since the eyeboard works with only one power supply (the eyeboard kits come with a 12V DC power supply), we will use two voltage regulators to "simulate" the dual power supplies. Here we introduce a new concept called, virtual ground. A **virtual ground** is just another reference point we will use, it will be the "middle" of our voltage range.

So finally! Let's start building the circuit!

Step 1: Connect the two 7805's

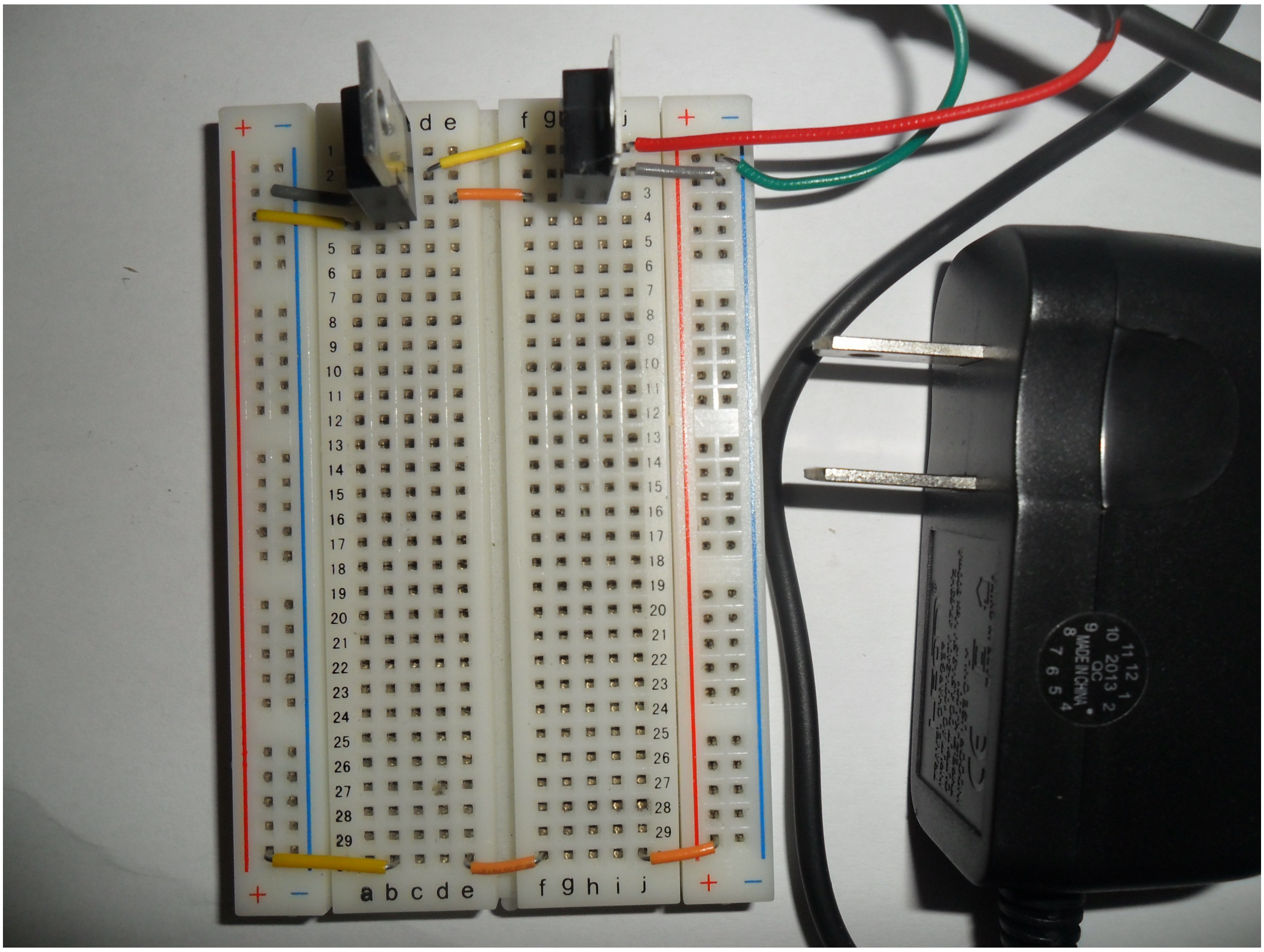
Start by connecting the two 7805s as it is shown in the diagram shown above.



The left pin of the 7805 is the input, the middle pin is ground, and the right pin of the 7805 is the output.

So connect your power supply to the input of the first 7805, the negative terminal of the power supply to the gnd of the first 7805. If you don't have an adapter for the power supply, you can just solder some cables to it like I did (see picture below). The output of the first 7805 should go the gnd terminal of the second 7805 (this is virtual ground), both have common inputs and the output of the second 7805 is our VCC.

Your circuit should look like the picture below:

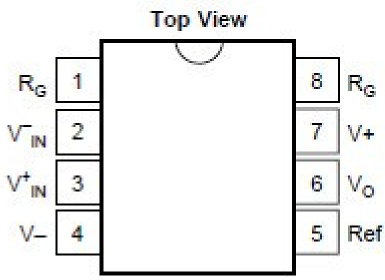


I am using the right blue rail as ground (0V), and the left blue rail as virtual ground. You can do something similar with your circuit to differentiate virtual ground from ground, just make sure to not mix them up. Both the red rails of the breadboard are VCC (+5V with respect to real GND).

So if you analyze the above circuit, we have simulated the dual power supply we mentioned before. If you measure the potential difference between VCC and virtual ground, we have a voltage of (+5V), if we measure the potential difference between GND and virtual ground, we have (-5V). You can try this yourself if you have a tester.

Step 2: Connect the INA118P instrumentation amplifier

From the datasheet, the following is the pin configuration of the INA118P.



The gain is calculated with the following formula:

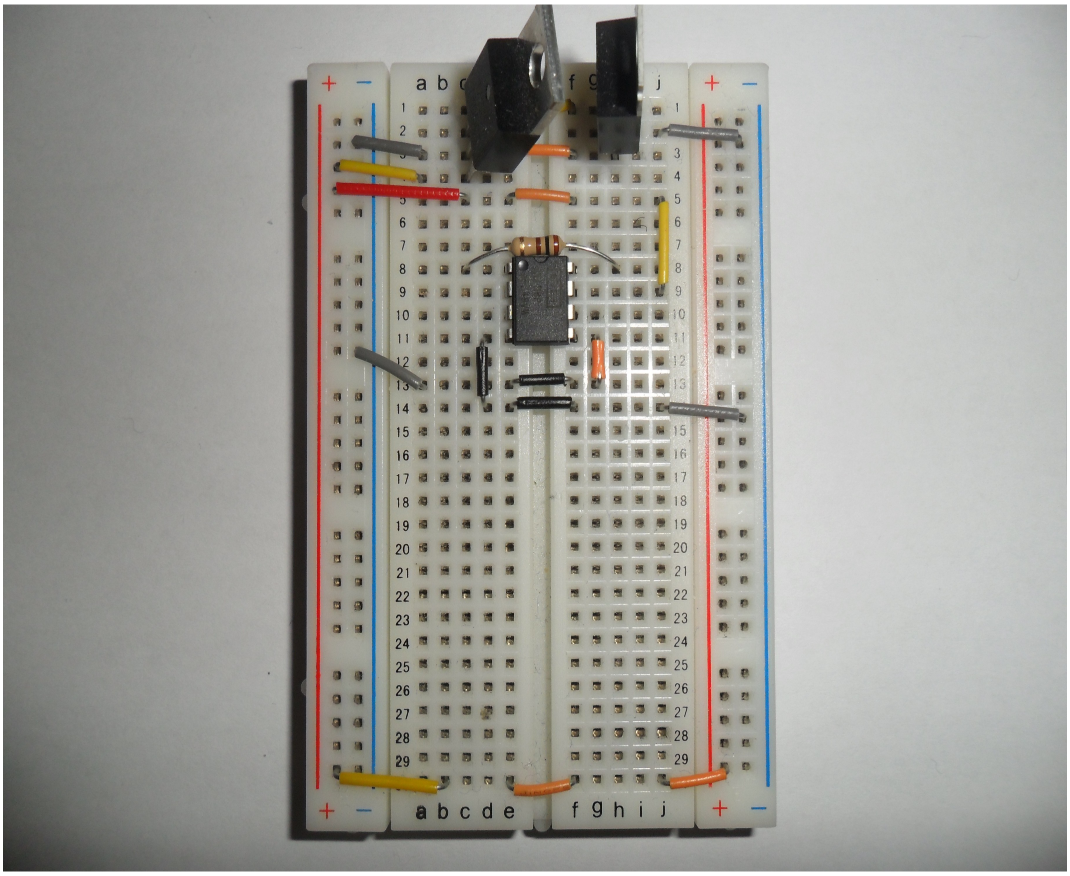
$$G=1+\frac{50k\Omega}{Rg}$$

We will need a really high gain to amplify the really low potential difference sensed by the electrodes. We will be using a high gain to get a signal that we could use. Let's try with a 100Ω resistor.

$G=1+\frac{50k\Omega}{100\Omega}=1+\frac{50000}{100}=1+500=501$ So with a 100Ω resistor, we get a gain of 501. That'll be enough to get a signal that we could read with a microcontroller. So we will amplify the input signal 501 times. So let's connect the op-amp. Connect the gain resistor between pin 1 and pin 8. V+ is VCC (+5V), pin 2 and pin 3 are the where we will be connecting the electrodes and pin 5 is virtual gnd, and there we will connect the reference electrode.

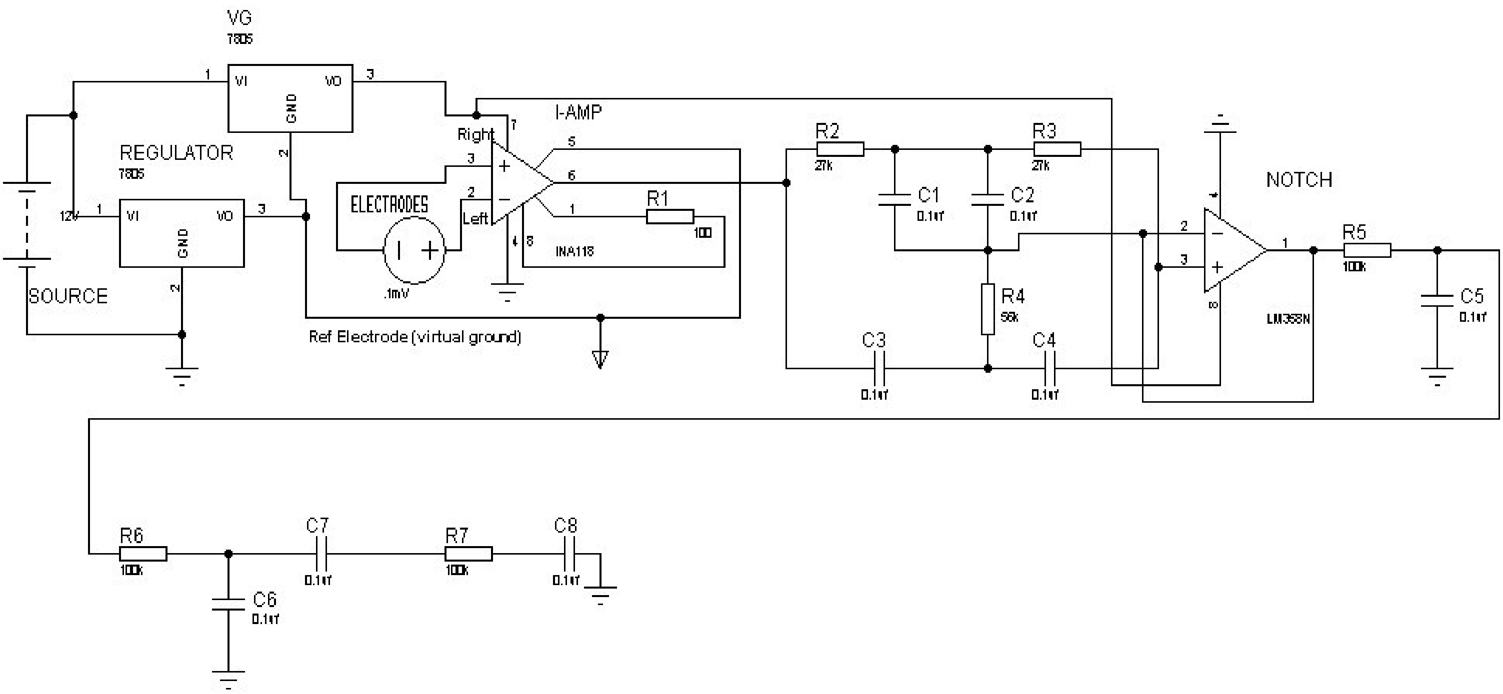
At this stage we could see the changes in slope of the EOG signal if we connected the electrodes to the INA118P. However, we still have to make sure that our signal is as reliable as possible, this is why we will use a next stage, a filter stage.

Your instrumentation amplifier circuit with the INA118P should look like the picture below.



We are now going to build the **filter stage**. Because we are dealing with such low voltages, amplifying them mean that we also amplify some noise. Even with a high CMRR that the INA118P has, we can still have noise from many sources, such as electromagnetic radiation from the AC electrical devices (60Hz in America, 50HZ in other places such as Europe). To block that unwanted signal we will use filters. A filter is mainly a circuit that “blocks” a range of frequencies that we don't need from our signal.

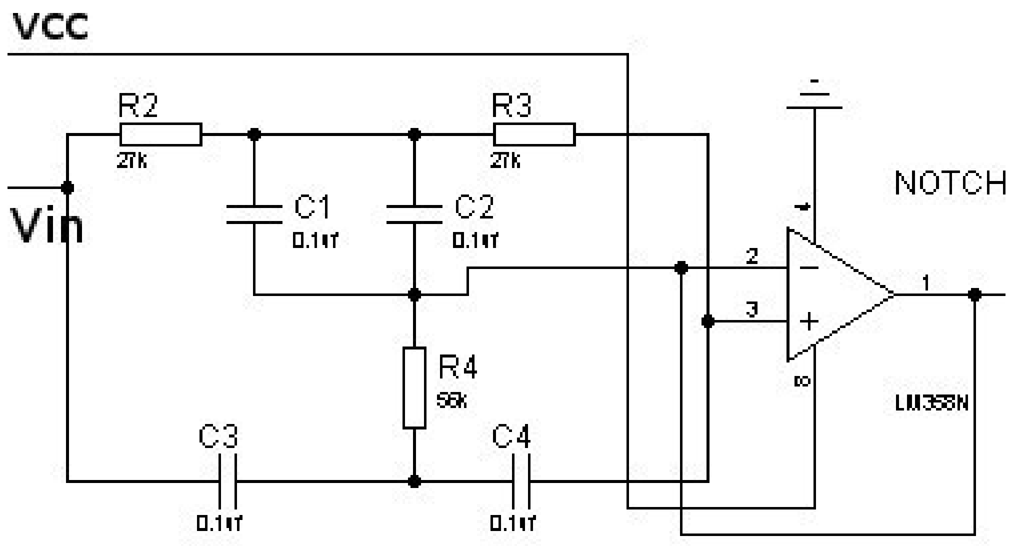
So that next step is to build the filter. We will need to connect the output of the INA118P to the filter stage. Basically we need to implement the following diagram.



Refer to the above circuit diagram for the following steps of the building process.

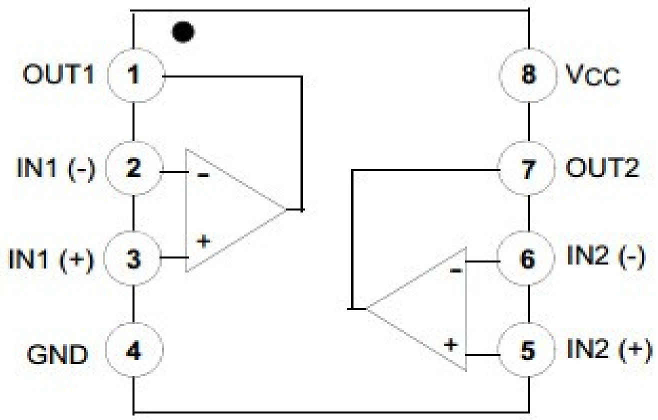
Step 3: Build the Notch filter

A notch filter is a type of band-stop filter, which a circuit that passes most frequencies but "blocks" the frequencies in a specified range. So in our EOG application, if we had an oscilloscope there is a chance that we might see a sine wave in the output of the INA118P, and we would notice that the sine wave has a very particular frequency. It is most likely electromagnetic noise from the AC line, which is 60Hz for some countries and 50Hz for others. We won't go on the details of why this circuit works, but it is really important to implement it on our circuit to block a range of frequencies that might be the AC electromagnetic interference. Although our next step involves in a low-pass filter which also blocks these frequencies (50Hz-60Hz), it is recommended to implement this stage as well. We will give more insight about filters on the next step. For now, we'll implement a notch filter using the LM358N op-amp. One of the applications of the op-amp is the filter, by adding a few capacitor and resistors we basically create a circuit that blocks certain frequencies that we do not want in our signal.

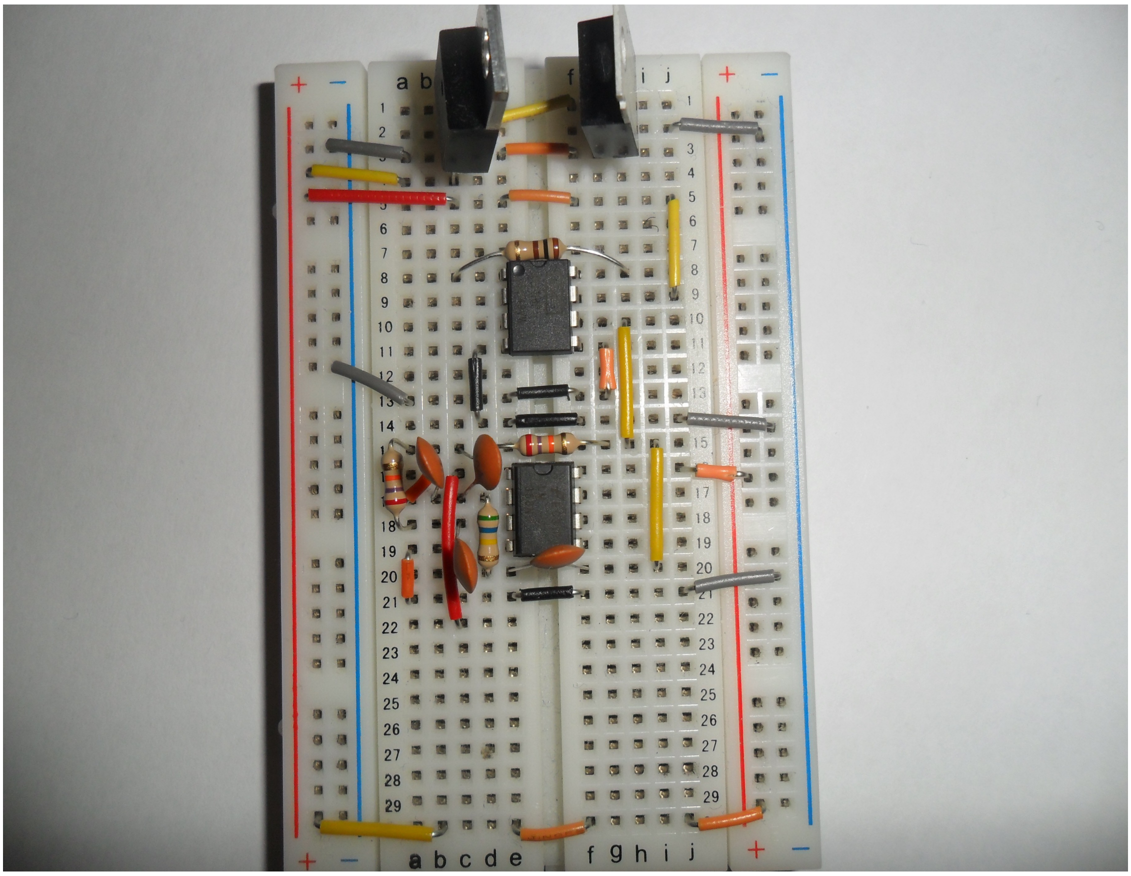


This circuit will "block" the frequencies of the AC electromagnetic interference.

From the datasheet, the following image shows the internal block diagram of the LM358N. You can use any of the two internal op-amps, and you are still left with one in case you need an extra amplification or filter stage.



Your circuit so far should look like:

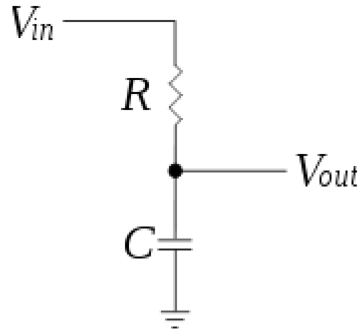


Step 4: Build the low-pass filter

So we have a basic understanding of what a filter is, and we have implemented a band-stop filter. But we haven't gone through the process of calculating what frequencies the filter is going to block. We also have assumed that the filter will magically stop those frequencies that we do not want. Unfortunately this doesn't happen in practice. What a filter *really* does is that it attenuates the frequencies that we wish to stop.

Some simple types of filters include the low-pass filter and the high pass filter. A low pass filter is a filter that attenuates high frequencies and passes low frequencies. A high pass filter on the other hand, is a filter that attenuates low frequencies and passes high frequencies. Because the EOG signal is mainly composed of low frequencies, we will want to stop high frequencies that may be noise.

Take a look at this simple analogue low pass filter:



If the input signal is mainly composed of low frequencies below 15Hz (like the EOG), we would like to attenuate frequencies above 15Hz. To do that, we implement the above low-pass filter with the calculated resistor and capacitor. To calculate the value of the resistors and capacitor, we use the following formula:

$$Fc=\frac{1}{2\pi \mathrm{RC}}$$

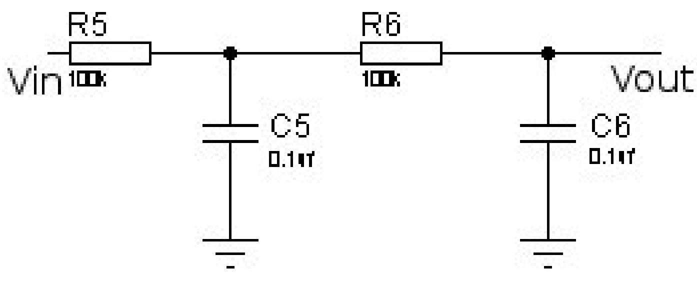
where Fc stands for "Cutoff frequency", which is the frequency where the filter will start attenuating our signal.

For a 15Hz low pass filter, we used a 100kΩ resistor and 0.1µF capacitor. This give us the 15Hz cutoff frequency we need.

$$Fc=\frac{1}{2\pi \mathrm{RC}}=\frac{1}{2\pi(100\mathrm{k}\Omega)(0.1\mu\mathrm{F})}=\frac{1}{2\pi(100000)(0.1/1000000)}=15.91\mathrm{Hz}$$

In general a low-pass filter has a rolloff of 20dB per decade, which just means that the frequency will attenuate by half everytime the frequency reaches twice the double frequency. So in our case, when the frequency of our signal reaches 31.8Hz, the amplitude of our signal will have been reduced by half.

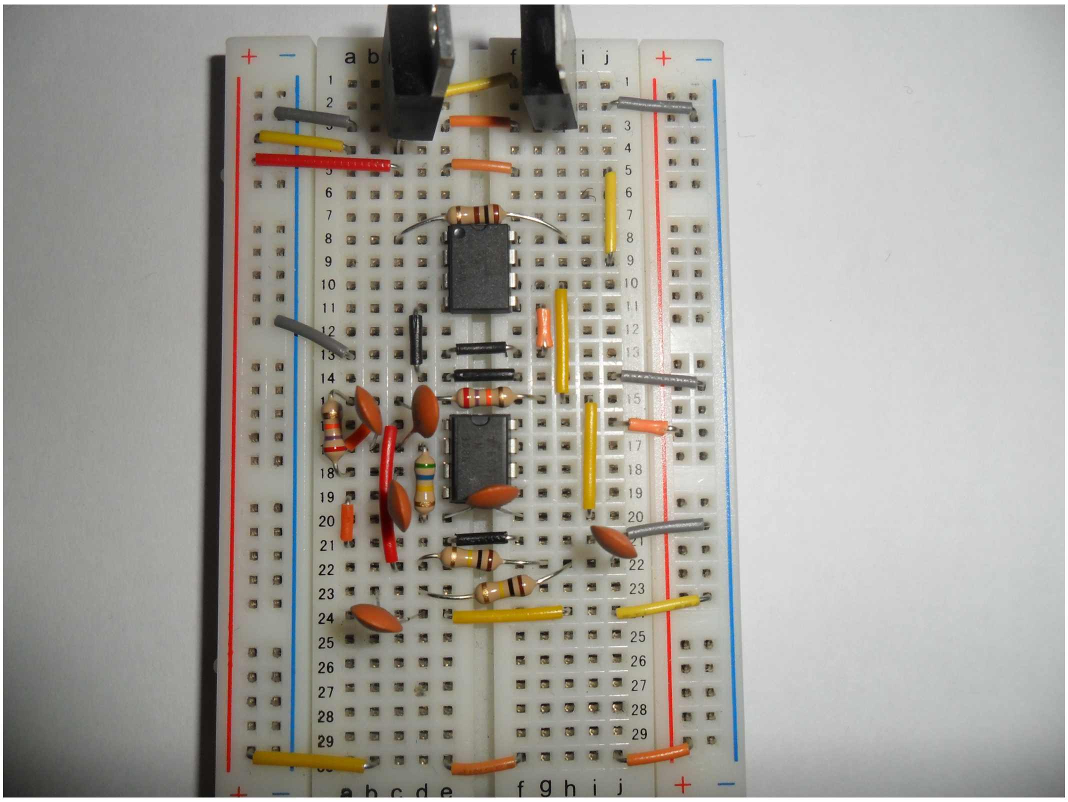
If we wanted to attenuate the noise even more, we could implement two low-pass filters in cascade (one after the other), and that will give us a a rolloff of 40dB per decade (our signal will be attenuated by ¼ when the frequency reaches 2Fc). This is what we did in our circuit of the eyeboard, we implemented the two low pass filters in cascade. So we will need to implement the following circuit:



Where Vin is the output of our notch fiter and Vout is our filter EOG signal.

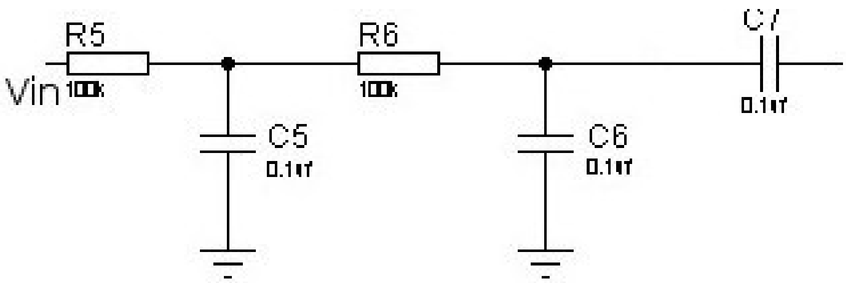
Now, we will build the **reading stage**. So now that we have filter our signal, we are now almost ready to use it. But we before sending the info to our Arduino, let's first implement two last stages to our circuit.

So your circuit so far should look like:



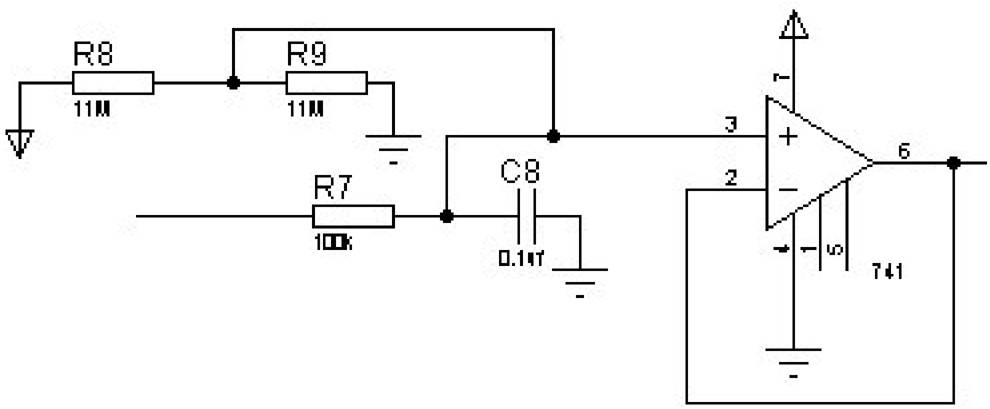
Step 5: Add a capacitor to the output of the filter

This step is pretty straightforward, just add a 0.1uf capacitor at the end of the two filters in cascade we just added. A capacitor in series removes the DC offset from the signal, so that means that will only read the slope changes, if there is no change at all, the voltage will just be 0.

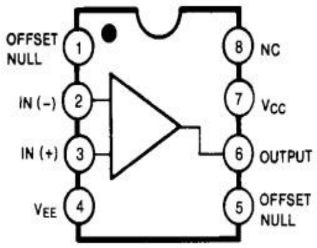


Step 6: Add a voltage follower circuit

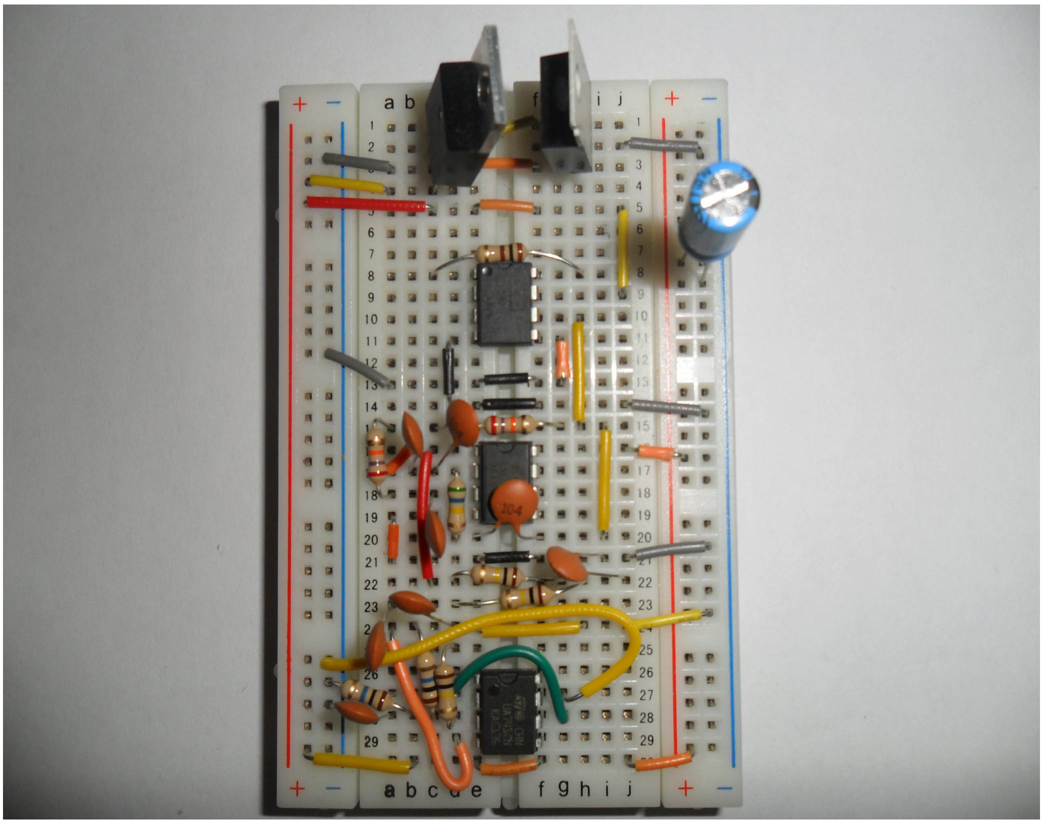
We have cleaned our signal, we have remove the DC component of the signal. Now if we saw the signal on an oscilloscope, we could affect our signal and therefore the readings of the oscilloscope or multimeter for that matter, wouldn't be reliable. This is because we are connecting a low source impedance to a device with high input impedance. Impedance is a measure of the “opposition” of current in a circuit. You can think of it as the resistance of a device. Well to solve this problem we add a voltage follower circuit with an op-amp. The voltage follower circuit eliminates these kind of problems because of the high input impedance of an op-amp, which in theory is infinite. So now we will implement the following circuit.



Notice that there is a voltage divider circuit above (R8 and R9), since we have removed the DC offset, we need to shift the signal upwards again so we can have about 2.5V when the voltage difference of the electrodes is zero (the user is looking at the center). Next notice that we have added one last low pass filter (R7 and C8) before the input of the voltage follower circuit. Now connect the LM741 as shown in the diagram of the voltage of follower above. From the datasheet, we know the pin configuration of the LM741:



So your finished circuit should look like:



Looks a little cluttered at the bottom, but hopefully you did the circuit on a larger breadboard (like a medium sized breadboard). You can also add the electrolytic bypass capacitor in parallel to the input power supply. This will make the DC voltage cleaner. Now we are ready to connect the EOG output to the Arduino and see analyze it with an ADC!

Step 7: Program the ADC of the Arduino

Now we are finally ready to read the EOG signal and send it to the computer. We will do it with the ADC (Analogue to Digital Converter) of the Arduino. The ADC is basically a way to convert analogue voltages into digital data. The ADC of the Arduino has a resolution of 10 bits. That means that there are 2^{10} numbers that we can use to represent these voltages, it ranges from 0 to 1023. So if the voltage is 2.5V (which is when the user has the eyes centered), the ADC value is about 512. We won't go through how to program the Arduino just because there is way too much of information out there. But the ADC is pretty straightforward.

From the Arduino tutorial (http://arduino.cc/en/Tutorial/ReadAnalogVoltage), the following code will read an analogue voltage and send it to the serial port:

```
/*
  ReadAnalogVoltage
  Reads an analog input on pin 0, converts it to voltage, and prints the result
to the serial monitor.
  Attach the center pin of a potentiometer to pin A0, and the outside pins to
+5V and ground.

 This example code is in the public domain.
 */

// the setup routine runs once when you press reset:
void setup() {
  // initialize serial communication at 9600 bits per second:
  Serial.begin(9600);
}

// the loop routine runs over and over again forever:
void loop() {
  // read the input on analog pin 0:
  int sensorValue = analogRead(A0);
  // Convert the analog reading (which goes from 0 - 1023) to a voltage (0 -
5V):
  float voltage = sensorValue * (5.0 / 1023.0);
  // print out the value you read:
```

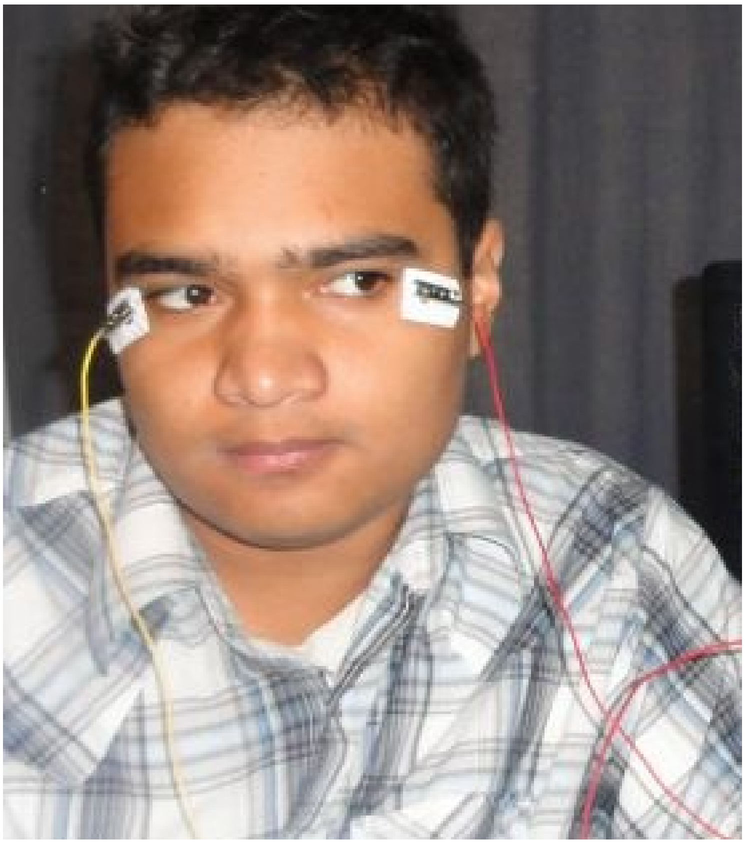
```
  Serial.println(voltage);
}
```

The first C function Setup(), is to initialize the serial port. We will connect the Arduino to the computer using the USB cable, so we can send the information from the ADC to the serial port. The second function, the loop is just a infinite loop that will keep going until the program is closed. We will keep reading the Analogue values of the EOG.

Step 8: Connect everything together, and try it out

Our program above reads the analogue input 0, so look for the analogue input 0 (A0) in your Arduino, and connect the output of the eyeboard to the A0 pin of the Arduino. Next, since the Arduino is connected to the computer, it is already connected to an external power supply. So connect the ground pin of the Arduino to GND of the eyeboard circuit (GND, NOT virtual gnd).

Next, take out a pair of electrodes from the included bag of the kit and paste them next to each eye. You can use a pair of alligator clips, or paste a couple of electrodes in some glasses. Be creative!



You can paste the third electrode (the reference electrode) in your hand, or wherever you feel more conformable. Now connect the two electrodes that you have next to the eyes on pin 3 and pin 2 of the INA118P (the order doesn't matter, it will just invert the slope, e.g if you move the eyes to the right the slope is negative, to the left the slope is positive), and connect the reference electrode to pin 5 of the INA118P.

After connecting it, you can go ahead and try it out! Open a terminal emulator program to read the serial port, turn on the Arduino and you can see how the ADC will change depending on how you move the eyes!