Power meter circuit for measuring low power system

September 2016

MSc System on Chip
Abstract

This project designed a power meter for measuring low power system. Test the power consumption from a tested chip when it is not doing calculation or any other operations. The power meter has two channels to measure the current. The lower channel can measure lower current but it has higher accuracy. The higher channel can detect much higher current than lower channel but its accuracy is lower. A digital processor in this power meter to calculate the power based on the collected data and display the real time power value on a 16x2 LCD screen.
## Contents

Abstract ............................................................................................................................. 2

Acknowledgements ............................................................................................................ 5

1. Introduction .................................................................................................................... 6

2. Background .................................................................................................................... 7

   2.1. Electronic Noise ........................................................................................................ 7

       2.1.1. Spectral Density ................................................................................................. 7

       2.1.2. Thermal Noise .................................................................................................... 8

       2.1.3. Shot Noise ........................................................................................................ 9

       2.1.4. Signal-to-Noise Ratio ....................................................................................... 9

   2.2. Instrumentation Amplifier ....................................................................................... 10

       2.2.1. What is Instrumentation Amplifier ..................................................................... 10

       2.2.2. Signal Amplification and CMRR ........................................................................ 11

       2.2.3. Application ......................................................................................................... 11

       2.2.4. Characteristics of Instrumentation Amplifier ..................................................... 12

       2.2.5. The Principles of Instrumentation Amplifier ..................................................... 14

3. Component Description ................................................................................................ 16

   3.1. AD8221 .................................................................................................................. 16

   3.2. OPA211 ................................................................................................................. 20

   3.3. MBED NXP LPC1768 ............................................................................................ 21

4. Methodology ................................................................................................................ 23

   4.1. Virtual Ground Circuits ........................................................................................... 24

   4.2. Amplifier Design ..................................................................................................... 27

       4.2.1. Lower Current Channel ................................................................................... 28

       4.2.2. Higher Current Channel .................................................................................. 30

       4.2.3. Noise filtering ................................................................................................... 31

   4.3. Digital Processor and Display ................................................................................ 33

5. Testing and Discussion .................................................................................................. 37

   5.1. Virtual Ground Circuit Testing ............................................................................... 37

   5.2. Amplifying Gain Testing ....................................................................................... 38

       5.2.1. Instrumentation Amplifier ............................................................................... 39

       5.2.2. OPA211 Circuit ............................................................................................... 39

   5.3. Result ....................................................................................................................... 41
5.4. Discussion ........................................................................................................... 45

6. Project Management ................................................................................................ 46
   6.1. Gantt Chart ........................................................................................................ 46
   6.2. Risk Management ............................................................................................... 47
   6.3. Project Deliverable ............................................................................................ 48

7. Conclusion and Future Work .................................................................................. 48

Appendix .................................................................................................................... 50

Bibliography ............................................................................................................... 52
1. Introduction

Power meter is an instrument measuring the system power consumption at the real time. The embedded chip consumes high power when it calculation or transfer data, the input voltage and current are big enough to be measured when embedded chip working. However, when measuring the chip power consumption, the chip will consume power even it does not work. This power is infinitesimally small but it is still exist. This power is called ‘leakage power’ in this project. This leakage power consumption is down to 10uW which means measuring currents of the order of 1uA. This consumption of power is very small but it is still a problem when engineer design an embedded chip. Therefore, in this project, the power meter is built to measure this low system power and to display it.

This project builds a power meter circuit for measuring low power of embedded chip and display the real time power consumption on LCD. This power meter has two channel to measure the input current, the lower current channel range is from around 1uA to 1mA, its detecting range is small but has higher accuracy. The range of higher channel is from 0.9mA to 660mA, it can measure higher current but has lower accuracy. This information is based on ideal circuit design. The real result from the power meter may depend on lots of area like temperature and resistor precision is not enough.

This power meter aims to measure low power. It must consider about the low level signal amplification. The noise is a big problem for low level signal, because it will cover the signal. Reducing the influence from the electronic noise will be solved in this analogue design. The gain during the amplification also should be considered, the gain depends on current and ADC reference voltage and ADC bits. The amplifiers in this design require dual voltage supply but only one power source want be used, this project will solve this problem. The digital processor in this project is mbed NXP LPC7268, which is a microcontroller use the ARM Cortex M3 MCU in a DIP package.
This processor is used to collect the analogue inputs, detecting the power and driving the LCD to display the power.

2. Background

2.1. Electronic Noise

Noise is a random undulation in an electrical signal in electronics. All electronic circuits contain inherent noise sources; electronic devices generate noise varies greatly. The noise can be generated by numerous different effects. The sources of noise must investigate when building electronic circuit. This report talks about the thermal noise contributed by a resistor and the shot noise produced by the random emanation of electrons. [1]

2.1.1. Spectral Density

The spectral density $W_x(f)$ provides the best way of describing a random signal. If we know the system impulse response and the input signal spectral density, that is easy to compute the system output spectral density. The real function of frequency of the spectral density is defined as

$$W_x(f) = \lim_{t \to \infty} \frac{1}{t} |X_t(f)|^2$$

The $X_t(f)$ in this function is a truncated version Fourier transform of the random signal $x(t)$ and lasting for time $t$.

This Fourier transform is defined as

$$X_t(f) = \int_{-t/2}^{t/2} x(t)e^{-j2\pi ft} dt$$

The random signal square average value may be computed by spectral density based on all frequencies. Thus it can be defined as
\[ [x(t)]^2 = \int_{-\infty}^{\infty} W_x(f) df \]

In this function, it must first compute the spectral density \( W_y(f) \) of the output from when wish to find the average value of random signal square of the output whose input is a known spectral density random signal.

\[ W_y(f) = W_x(f) * |H(f)|^2 \]

The \( H(f) \) in this function is the system impulse response Fourier transform and it can obtain \( |y^2(t)| \) by integrated over all frequencies \( W_y(f) \). [1]

### 2.1.2. Thermal Noise

Thermal Noise also can be called Johnson-Nyquist noise. Thermal noise the electronic noise that is generated by electrical conductor which inside the charge carriers, these charge carriers are usually the electrons. This noise is the characteristic of the electronic conductor which will not be influenced by the change of the frequency.

This thermal noise is dissimilar from shot noise. In the thermal noise, consists of additional current variations is occurred when macroscopic current starts to flow and a voltage is applied meanwhile. In the universal case, the thermal noise not just be generated in the resistors, it can be find in any type of conducting medium like ions in an electrolyte.

When the temperature is higher than absolute temperature, every resistor has thermal noise because the thermal motion of the electrons. The computational formula of the thermal noise is defined as

\[ V_n = 4K_B TR \]

Where the \( K_B \) in the formula is Boltzmann’s constant in joules per kelvin (1.28*10^-}
23J/K). \( T \) is the absolute temperature and \( R \) is the resistance value in ohms. Thus, the thermal noise is proportional to temperature. [1]

2.1.3. Shot Noise

Shot noise is a type of electronic noise. The origin of shot noise in the electronics is from the discrete nature of electric charge. The shot noise also can find from the photon counting which is in optical devices, the shot noise is related with the particle quality of light. The electronic circuit shot noise is consisted by random electric current fluctuations in a DC current, in fact a flow of discrete charges like electrons consist the current. The electrons are tiny charge but the shot noise is even more negligible than electrons in most cases of electrical conduction. There are 6.24x10e18 electrons per second in 1 ampere of current, even this number times randomly by several billion in each second, the fluctuation is still infinitesimal when compared with the current. The shot noise is often less than Johnson Nyquist noise and flicker noise but it is independent with frequency and temperature. Johnson Nyquist noise is influenced with temperature and the flicker noise is proportional to frequency. Thus in this case, the shot noise may become the main noise source when the circuit works at high frequencies and low temperatures. [1]

2.1.4. Signal-to-Noise Ratio

Signal-to-Noise ratio (SNR) is a measure used in science and engineering that is the ratio of the level of an anticipated signal with the level of background noise. The SNR is defined as the ratio of signal to the noise power and it is often expressed in decibels. When the SNR is higher than 1:1 signifying that more signal than noise.

SNR is ratio of the ratio power to the noise power, thus it is also the ratio of the ratio power amplitude square to the noise power amplitude square:

\[
\text{SNR} = \frac{P_{\text{signal}}}{P_{\text{noise}}} = \frac{A_{\text{signal}}^2}{A_{\text{noise}}^2}
\]
The SNR is defined in decibels; its value is the ten times logarithm of the ratio of the signal power to the noise power: [2]

\[
\text{SNR(dB)} = 10 \log_{10} \left( \frac{P_{\text{signal}}}{P_{\text{noise}}} \right) = 20 \log_{10} \left( \frac{A_{\text{signal}}}{A_{\text{noise}}} \right)
\]

2.2. Instrumentation Amplifier

2.2.1. What is Instrumentation Amplifier

Instrumentation amplifier can be used in wide area rather than only in instrumental construct. From motor controlling to vehicle systems all need instrumentation amplifier to operate analog signal which within a noisy environment or has large common mode signals.

Instrumentation amplifier is a kind of amplifier but it has some very important ways which are unlike common amplifier. Instrumentation amplifier has differential inputs and single ended output with close gain unit relative to reference point. Two input independences of instrumentation amplifier are balanced and the typical value is bigger than \(10^9\Omega\). Because instrumentation amplifier has high input independences, thus it has lower bias current commonly and the value always low to 50nA to 1nA. Same like other kinds of amplifiers, instrumentation amplifier has low output independence, only milliohm when output signal has low frequency.

The close gain in instrumentation amplifier is based innumerable ways using external feedback components like R, C and L sometimes. The difference with common amplifier is that the instrumentation amplifier has an internal feedback resistances network which is isolation with the input signal. Inputting the input signals to differential input ports, the amplifier gain can be set by internal preset resistors or user can connect an external gain resistor and this gain resistor is also quarantine with the input signal. [3]
2.2.2. Signal Amplification and CMRR

Instrumentation amplifier is an amplifier which magnify the voltage difference between two differential voltage inputs to restrain the common input signals. Therefore, instrumentation amplifier plays a very important role when sample the week signals. [3]

CMRR (common-mode rejection ratio) is an important parameter used in differential amplifier in analogue circuit. It is used to express the suppression of common inputs of differential inputs. In practical application, CMRR is very significant when consider about the dwarf signals or the differential inputs.

When two differential inputs are $V_+$ and $V_-$, the output should be $V_o = A_d(V_+ - V_-)$ under ideal conditions. However, the output should be defined as:

$$V_o = A_d(V_+ - V_-) + \frac{1}{2}A_{cm}(V_+ + V_-)$$

$A_{cm}$ is the common gain in the amplifier, it is much less than differential gain in the most cases.

The definition of CMRR is expression as decibel when use logarithm:

$$CMRR = 10\log_{10}(\frac{A_d}{A_{cm}})^2 = 20\log_{10}(\frac{A_d}{|A_{cm}|})$$

Because the differential gain is much higher than common gain, the CMRR is a positive number. The CMRR is an important product parameter, it expresses the restrain and the attenuation of the common signal of differential amplifier. Restrain the common signal is very important when consider about the noise reducing. [2]

2.2.3. Application

The main application of instrumentation amplifier is to amplifier the week signal from
sensor under the noise environment. The signal of pressure sensor or temperature sensor amplification is common instrumentation amplifier application.

Instrumentation amplifier is widely used in medical equipment, like electrocardiograph, electroencephalograph and sphygmomanometer. Instrumentation amplifier is also used in monitoring and controlling electronic device, it can monitor the voltage and current in the device and trigger the alarm system when the value of voltage and current beyond the threshold value.

Instrumentation amplifier has high CMRR, thus it is used widely in audio processing. Using instrumentation amplifier as the first amplifier to extract the week signal and avoid the interference from the noise and offset voltage of ground loop. [4]

2.2.4. Characteristics of Instrumentation Amplifier

The high performance instrumentation amplifier should have these characteristics:

High AC and DC CMRR
Instrumentation amplifier should have high CMRR at least between the input frequencies. It means high CMRR in power transmission line frequency and its second harmonic frequency range.

Low Offset Voltage
Instrumentation amplifier must have low voltage drift like operation amplifier. There are two parts consist the instrumentation: input stage and output stage. The totally offset voltage equal input offset multiply the voltage gain and add the output amplifier offset voltage. Offset voltage cannot be removed through circuit adjustment. This is the main error when input signal is small and offset voltage is big, it will cover the useful signal.
High Input Impedance
The input impedance of two inputs port of amplifier must be very high and match closely in order to avoid the decreasing of loading ability and input signal voltage. The typical input impedance is from $10^9\Omega$ to $10^{12}\Omega$. Some differential amplifier has lower input impedance like AD629, but it has highly effective when process high common voltage.

Low Offset Current and Bios Current
Like operation amplifier, instrumentation amplifier has bios current input and output its two input ports. For bipolar input instrumentation amplifier, it is base current and it is gate leakage current for FET input instrumentation amplifier. These input bios current will generate offset error when crossing unbalanced input impedance. The error of input offset current is defined as offset degree between two input bios currents. The typically bios current value of bipolar instrumentation amplifier is from 1nA to 50nA and for FET input is from 1pA to 50pA.

Low Noise
The instrumentation amplifier must process tiny input signal voltage, so it should have very low input noise. Under the 1kHz frequency input signal and the gain is bigger than 100, the 10nV input RTL minimum noise is permitted. Micro power consumption instrumentation amplifier is used to sample tiny input current, so it may have higher input noise than other instrumentation amplifiers which have higher input current.

Gain Choice Convenient
A good instrumentation amplifier should easy to choice the amplify gain. Amplifier use an external resistor to set up the gain generally. However, the external resistor will influence the accuracy and its value will be change when temperature drift. Some instrumentation amplifiers offer an another choice that connecting the ports of amplifier to change different gain, but only few gain can be choice when use this way to set the amplify gain. [3]
2.2.5. The Principles of Instrumentation Amplifier

One Operation Amplifier Differential Circuit

This is the simplest circuit to realize the differential gain circuit but it is very useful. The Circuit is shown in Figure 2-1.

When R1=R3 and R2=R4, the output voltage is $V_{out} = (V_{in2} - V_{in1}) \left( \frac{R1}{R2} \right)$. This circuit can realize differential input and restrain the common signal but it also has some defects. Input impedance in this circuit is low and unbalanced. The impedance at non-inverting input is 200kΩ but only 100kΩ at inverting input, this will reduce the CMRR in the circuit. This circuit requires very accurate resistors otherwise the gain will have big discrepancy.

TRI Operation Instrumentation Amplifier

In order to fix the problem of the simplest differential inputs circuit, improve the simple circuit to figure 2-2 which add a buffer before the original circuit.
This circuit offers high input impedance to the amplifier to restrain the common signal influence. Using two operation amplifiers as the buffer is a good way to fix the problem from the simplest circuit.

Figure 2-3 is the farther improvement of this circuit. Improving this input buffer to amplifier and has gain. This is a more flexible circuit. When R5=R7, the output voltage is defined as:

\[ V_{out} = (V_{in2} - V_{in1}) \left(\frac{R_1}{R_2}\right) \left(1 + \frac{2R_5}{R_6}\right) \]

When the buffer has gain, this gain will work on differential signal and common signal. The \( R_6 \) in this circuit also can be called \( RG \). Customers can change the value of \( RG \) to control the differential gain.

(Figure 2-3 Three operation amplifier instrumentation amplifier)
Monolithic Instrumentation Amplifier

In order to improve the practicability of instrumentation amplifier, ADI company developed the monolithic IC instrumentation amplifier. These ICs include the improvement of the instrumentation amplifier which are based on three operation amplifiers or two operation amplifiers and some ICs offer the laser trimming resistor to insure the accurate of the gain. Active devices and passive devices are integrated together, so they matching precisely, this accurate matching make the devices have high CMRR. IC instrumentation amplifier size is tiny because it can be integrated into SOIC or MSOP which are very small packages, so mass production of IC instrumentation amplifier is easy achieved. [3]

3. Component Description

This project aims measuring low power system, therefore it must reduce the resolution of noise. Noise will mask the aimed signal, so noise will be important in this system. Therefore, noise reduction is the most important problem in this amplifying circuit design. Digital part is also an important part in this project, it is used to convert the analog signal to digital data and processing the data and display the result. Thus choosing a suitable digital processor is also important.

3.1. AD8221

The AD8221 is a high performance instrumentation amplifier that has the highest CMRR over frequency of industry in its class. The minimum CMRR of the AD8221 is 80dB to 10kHz for G=1. The AD8221 IC instrumentation amplifier has low offset voltage, low gain drift, high gain accuracy and high CMRR. This chip is an excellent choice in data acquisition, aerospace instrumentation and biomedical analysis.
The programmable gain of AD8221 affords the design of user flexibility. User uses a single resistor which is called \( R_G \) to set the gain from 1 to 1000. The AD8221 can operate on one signal or dual supplies and it is suitable for up to \( \pm 10V \) input voltages applications.

The AD8221 is packaged in an 8-lead MSOP or 8-lead SOIC (Figure 3-1). Both of them offer the best performance of industry.

![Top view of the AD8221](image)

(Figure 3-1 Top view of the AD8221)

The performance of AD8221 is quantified over the entire industrial temperature which from -40\(^\circ\)C to +85\(^\circ\)C and AD8221 can be operated from -40\(^\circ\)C to +125\(^\circ\)C.

**CMRR**

AD8221 has high CMRR. The CMRR of instrumentation amplifiers will fall off at 200Hz that on the market now. By contrast, when the gain is 1 the minimum CMRR of AD8221 is 80dB. High CMRR of instrumentation amplifier allows it working in wideband distraction and line harmonics. As shown in figure 3-2, when gain is 1 the CMRR is 100dB and it will fall off at around 5 kHz.
This table shows the minimum CMRR of different gain of the AD8221 when CMRR DC to 60Hz with 1 kΩ and the reference voltage is 0V and temperature is 25°C.

<table>
<thead>
<tr>
<th>Gain</th>
<th>Minimum CMRR</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>G=1</td>
<td>80</td>
<td>dB</td>
</tr>
<tr>
<td>G=10</td>
<td>100</td>
<td>dB</td>
</tr>
<tr>
<td>G=100</td>
<td>120</td>
<td>dB</td>
</tr>
<tr>
<td>G=1000</td>
<td>130</td>
<td>dB</td>
</tr>
</tbody>
</table>

Maximum Voltage and Current Offset

AD8221 instrumentation amplifier has low offset voltage, offset current and bias current. The maximum offset voltage of AD8221 is 25uV p-p input noise (0.1Hz to 10Hz) when the source voltage is from ±5V to ±15V and the temperature is from -40°C to +85°C.

Under industrial temperature, the maximum input bias current is 0.4nA and the maximum input offset current is also 0.4nA.

The AD8221 is a monolithic instrumentation amplifier which is based on classic tri...
operation amplifiers topology. The theory of operation of AD8221 as shown as figure 3-3. Input signal cross the input and creates a current that through $R_1$, $R_2$ and $R_G$ (which is connected between pin2 and pin3). Based on the tri operation amplifiers topology, the output of AD8221 is $V_{out} = V_{in}(1 + \frac{2+24.7k\Omega}{R_G})\left(\frac{10k\Omega}{10k\Omega}\right)$.

![Figure 3-3 Theory of AD8221 [5]](image)

Because the output voltage is $V_{out} = V_{in}\left(1 + \frac{2+24.7k\Omega}{R_G}\right)\left(\frac{10k\Omega}{10k\Omega}\right)$, the gain of the AD8221 can be defined as:

$$G = 1 + \frac{49.4k\Omega}{R_G}$$

In order to change the gain of AD8221, user only need to change the resistance connected between pin2 and pin3.

$$R_G = \frac{49.4k\Omega}{G - 1}$$

This table shows the relationship of $R_G$ and calculated gain.
When there has no resistor connected between pin2 and pin3, the gain of AD8221 equals 1. [5]

### 3.2. OPA211

OPA211 is a 1.1-nv/VHz Noise, low power, precision operational amplifier. The series amplifier OPA211 achieves very low noise density which can low to 1.1-nv/VHz. Rail to rail output swing that maximizes dynamic range also offered by this series. The maximum offset voltage in OPA211 is 30uV.

OPA211 has extremely low voltage and current noise. It also has high speed and wide output swing. In this case, the OPA211 series amplifier is suitable in loop filter amplifier design.

In the application of data acquisition, the series amplifier OPA211 provides 700ns settling time to 16bits precise throughout 10V output swings. The maximum offset voltage is 125uV and the typically offset voltage is around 30uV. It has 0.35uV/°C voltage drift over temperature. These parameters make OPA211 is good for driving high accurately analog to digital converters or as an output buffer of high resolution digital to analog converters.

The OPA211 series amplifiers can work under dual power supply that the range is from
±2.25V to ±18V and also can work for single power supply from 4.5V to 36V. The temperature of this series is specified from -40℃ to +125℃.

The top view of OPA211 as shown in figure 3-4.

(Figure 3-4 OPA211 D package 8-pin SOIC top view [6])

When the OPA211 works at 25℃ and the source voltage is ±18V and \( R_L = 10k\Omega \), the relationship between CMRR and frequency as shown in figure 3-5. The CMRR is stabilization at 120dB until frequency higher than around 70kHz. [6]

(Figure 3-5 OPA211 CMRR vs frequency [6])

### 3.3. MBED NXP LPC1768

The mbed Microcontrollers are designed for rapid prototyping. These series microcontrollers belong to ARM microcontroller development boards.

The mbed NXP LPC1768 microcontroller is order to design for prototyping all sorts of
devices. Those devices include USB, Ethernet, flash memory and lots of flexibility peripheral interfaces. This microcontroller is packaged as a small DIP for prototyping with through hole PCBs, breadboard or stripboard and it has a built in USB flash programmer.

(Figure 3-6 mbed LPC1768 interfaces [7])

This mbed LPC1768 interfaces figure is based on the NXP LPC1768 which has a 32bit ARM Cortex M3 core running at 96MHz. This microcontroller has 32KB RAM and 512KB flash memory. Its interfaces include USB host, built in Ethernet, CAN, SPI, ADC, DAC, I2C, PWM and other generate I/O interfaces. This figure shows the commonly interfaces that used in LPC1768 and their locations. Those numbered pins p5 to p30 can also be used as normal digital in and digital out ports.

This microcontroller has 5V power source from USB or uses power supply from 4.5V to 9V.

The compiler of the mbed offers an easy operation online C or C++ IDE that is configured to let consumers to write programs, compile and download the programs to mbed microcontroller quickly. Consumers can download the compiled programs and run them in their mbed microcontrollers directly. Users need not install or set up any software or compile environment to get running with microcontroller, because the mbed compiler is a web app. Consumers can log in and carry on where you left on and
it is free on PC platforms.

4. Methodology

The calculated power in this power meter system is active power which is the average power consumption in a clock cycle also can be called ‘real power’. Assuming the instantaneous voltage and current is $u(t)$ and $i(t)$, the clock cycle is $T$. Then the function of active power is:

$$P = \frac{1}{T} \int_{-\frac{T}{2}}^{\frac{T}{2}} u(t)i(t)dt$$

This power meter is used to measure low power system. The low power system measures down to 10uW of power (or better) which means measuring currents of the order of 1uA. This power meter includes two parts: analogue part and digital part. In analogue part, the circuit collect and process the current or voltage from the low power system. In digital part, a digital processor is used to handle and display the data that received from the analogue circuit. The data will display on LCD and computer in real time. In analogue, it should make a very low noise differential amplifier and sample the input current with a very small value resistor.

The top level design of this project is shown in figure 4-1. It is consisted by sample resistor, first level amplifying circuit and last level amplifying circuit, digital processor and display part. The microcontroller mbed LPC1768 has internal analogue input ports, so there has no extra ADC in this design. The filter will be considered in the amplifying circuit design, that is a very important problem when design amplifier.
4.1. Virtual Ground Circuits

A common problem in analogue design is that circuit requires for a dual voltage supply like ±5V but designer only have or just want use a single supply like a battery to fix this problem. There are many ways to separate a single supply to a dual voltage supply.

The simplest way to fix this problem is to simply use two power source in this configuration:

![Virtual Ground Circuits Diagram](image)

(Figure 4-2 two power source dual voltage supply)

This circuit problem is that if one voltage source drains faster than other source like one source down to 1V and this changing is before the other source gets low, in this case, the DC offset at the voltage output will rise for some operation amplifiers. These two battery are unbalanced so their DC offset will always occurred and when this DC
offset is higher than a danger point, it will damage the amplifying circuit.

So designer try to use a single voltage source by various virtual ground schemes to offer this dual voltage supply.

The simple way to use virtual ground schemes to have a dual supply is resistor divider:

The virtual ground is created by two resistors. When the power source is 10V and R1 equal R2 than the resistor are half resistive divider which divide the source to ±5V. The absolutely value of two outputs are same and there only one power source is used. However, this circuit is still unbalance because consumers cannot find two absolutely same resistors. This problem lead this resistor divider is not a perfect voltage power source for amplifiers.

The best way to fix the problem is to increase the voltage source. However, this way requires a more expensive and bigger supply, the useless power is wasted. Another way is that use lower resistors to increase the current of the divider circuit. The power source can annihilate the run time differential from lower voltage source when it has the extra current through the circuit.

Most useful way to solve this is buffering the virtual ground. This buffer makes the
divider of voltage has a very low impedance and its current is still low. This way makes the virtual ground nicely and allow designer to use a smaller power supply and increase the run time of the voltage source.

The normal buffering virtual ground is using a Texas Instruments’ TLE2426. It splits one power source to two, so customers can get two voltage rails. Applying a 10V between the IN and COM of TLE2426 and the differential voltage from IN and COM with OUT is 5V and -5V.

![Buffering virtual ground with TLE2426](Figure 4-4 buffering virtual ground with TLE2426)

In this project, it uses one power supply voltage to offer a dual voltage supply which is ±5V. It uses a TL071 which is a cheap and high performance operation amplifier as the buffer to build a buffering virtual ground. For using this two inputs operation amplifier, the output current is lower than normal buffer. This operation amplifier has feedback leading it having low output impedance unlike the open loop buffer. The biggest advantage of low output impedance is that it has lower crosstalk. The 1kΩ feedback resistor is arguably optional. The impact of this resistor is to keep the amplifier stable in the circuit which has heavy capacitive loads like a bypass capacitors powered for the circuit. In the real design, there has no that 1kΩ resistor. [8]
4.2. Amplifier Design

In the simulation, this project uses a load resistor to replace the tested board, so the tested circuit is a power source series with a load resistor.

In order to sample the current of the circuit, it needs a sample resistor. This sample resistor must negligibly small to avoid the voltage drop. This sample resistor is small and the load resistor is large so use inter connected to measure voltage and current to reduce the error from the voltage collection. The value of the sample resistor is around 10Ω.
4.2.1. Lower Current Channel

Collect the voltage from sample resistor and divide the value of the sample resistor to get the current of this circuit. The ADC bits that used in this design is 12 and the maximum value that mbed can get is 3.3V. Therefore, set the aimed current range of the lower measuring channel is from 1uA to 1mA which means the voltage of the sample resistor is from 10uV to 10mV. This means the amplify gain is 330.

In this design, it uses two steps amplifying circuit to realize this 330 times gain. For tiny signal amplifier, in multistep amplify circuit, the input impedance of the total circuit is the first step amplifier input impedance and the output impedance of the total circuit is the last step amplifier output impedance.

AD8221 is used to build first step instrumentation amplifying circuit. The first step amplifying circuit is used to amplify the input signal and filter input noise. Figure 4-7 shows the first step amplifying circuit. It is consisted by capacitor filter circuit and amplifier. Its input impedance is 100k. There have two capacitors that connected with ground. The high frequency noise will connect with ground through capacitor but the low frequency aimed signal cannot because the capacitor will pass the AC but stop the DC. In this case, the high frequency noise will be filter by these capacitors.

Because AD8221 is an instrumentation amplifier, the inputs of this circuit are differential signaling. Differential signaling is a method that uses two complementary signals to transmit information. The receiver responds to the two signals electrical difference. Based on differential input, it can avoid the common input influence and other noises. The instrumentation amplifier AD8221 is introduced in chapter 3. It is a high performance amplifier and costumers only need to set the gain resistance to change the gain. For first step amplifier, its gain should not higher because it will
amplify the noise that did not be eliminated. The gain resistance for AD8221 is set as 12.4kΩ and the gain should be 4.984. Using a 12Ω and a 390Ω resistor series connecting to realize this 12.4kΩ but the testing value of 12Ω resistor is 11.95, so the total gain resistor is 12.385kΩ. Therefore, the gain of this instrumentation amplifier is 5.003.

Because the total gain is 330, the gain of the last amplifier is 66 when the first amplifier gain is 5. There has a high performance operation amplifier OPA211 building the last amplifying circuit. For this last step circuit, there also has capacitors on two input ports to avoid the influence from the noise. The gain of this circuit is $A_V = \frac{R_{11}}{R_9}$ when $R_9 = R_{10}$. 
Testing the input impedance of OPA211 and the value of these two resistors are 998Ω. Because the input impedance of OPA211 is the load impedance of AD8221, this 998Ω resistances can offer enough low current input for OPA211 and will not lead too big output current from the AD8221. In order to have 66 times gain through this amplifier, set a resistor on the feedback and its value should be 66kΩ. Test a 66kΩ and its real value is 68kΩ. In this case, the real gain of the OPA211 is 68.136.

When designer get the gain of the first and the last amplifier, the total gain is the product of them. In this lower current channel, the gain is equal 340.886. The maximum voltage of mbed is 3.3V, thus the lower current channel collection range is from 0.968uA to 0.968mA.

4.2.2. Higher Current Channel

When the output from the lower current channel is bigger than 3V, the microcontroller decides to read the voltage value from the higher current channel.
For higher current channel, the gain is from the first step amplifier, so the gain should be 5.003. For 3.3V mbed analog input, the aimed current for higher current channel is from 0.9mA to 660mA. However, the output from the instrumentation amplifier is minus, it cannot be detected by the ADC from mbed. An active filter is used to solve this problem. This active filter is same like the last step amplifying circuit of the lower current channel, the only difference is that the gain of this filter is equal to 1. This filter also like an inverter which can invert the output signal from instrumentation amplifier to positive number and the absolute value is not be changed.

Because this higher current channel has a large current detecting range and the ADC is fixed at 12bits, the precision of this channel is low. For lower channel, its precision is very high. The sample unit for higher channel is equal \((660-0.9) \text{ mA}/2^{12}\) which is 0.161mA and the precision of low channel is around 0.00024mA.

### 4.2.3. Noise filtering

As shown above, noise caused great impact when building the analog circuit. In this design, the available signals from the inputs are tiny, they are close to 1uA. If the circuit has too much resolution, noise will mask the input signals.

In order to reduce the influence from the noise, bypass capacitor and coupling capacitor are used in this circuit design. They all can be seen as filter and the difference between bypass capacitor and coupling capacitor is their position on the circuit. Bypass capacitor aims to filter the high frequency signal in the input signal and decoupling capacitor reduce the distraction in the output signal. In this case, bypass capacitor is connected with inputs and decoupling capacitor usually connect with outputs and sources.

**Bypass capacitor**
Bypass capacitor is used to filter the common radio frequency signals from the components and electric cables. These signals are generated by components and cables unintentionally and they will cover the aimed signal. Bypass capacitor offers a low impedance pathway. This pathway dredges the useless high frequency signals to ground to avoid them influencing the real signal.

**Decoupling Capacitor**

Driving source and the load always exit in circuit. The driving circuit should charge and discharge to finish a capacitance jump when the load capacitor is big. Current in this process will increase during rising edge and the driving current absorb these increased current. Because of the inductance and resistance in the circuit, this current is the noise of the ideal circuit, it will influence the common work of the circuit. This situation is called coupling.

Decoupling capacitor is used to avoid the influence from coupling. Decoupling capacitor dislodges the high frequency signal like RF signal which is generated from electromagnetic radiation. Decoupling capacitor can be parallel connected with the VCC to connect the AC signal from the source to ground. Meanwhile, it can decrease the noise in VCC.

Active device can generate high frequency switching noise when operate the switch, this noise is transit through power line. Another important function for decoupling capacitor is offer a local DC source to active device to avoid the switching noise transit on the board and guide this noise to ground.

The typically decoupling capacitor value is 0.1uF, it is good at processing the frequency which is lower than 10MHz but almost no effect to the higher than 40MHz frequency noise. The decoupling capacitor selecting is not strictness, it can base on $C = 1/F$ to select, it means 10MHz noise using 0.1uF capacitor and 100MHz noise using 0.01uF. In this design, 0.1uF is enough because it will not face the ultrahigh frequency noise.
Through these analyses, the completely analogue circuit design is finished. Sample resistor, virtual ground circuit, lower current channel circuit and higher current channel circuit with bypass and decoupling capacitors consist this analogue part design.

4.3. Digital Processor and Display

This digital processor is mbed NXP LPC1768 which is ARM Cortex M3 MCU in a DIP package. This microcontroller has eight analogue input ports and it has internal ADCs to convert the analogue signal to digital data. The ADC has 12bit and the maximum voltage is 3.3V. The data output range from ADC is from 0 to 1 which means if the input voltage is 3.3V the data that the microcontroller can detect is 1 and if detected number is 0.5 than the input voltage is 1.65V. The relationship between input and detected data is \( V_{in} = \text{digital} \times 3.3V \).
In order to reduce the error, the ADC samples ten times and calculates the average of them.

```c
for ( int i=0; i<10 ; i++){
    v[i] = Vol.read() * 3.3;
}
for ( int j=0; j<10 ; j++){
    c1[j] = ((CuL.read()++3.3/68.136-0.000030)/5.003-0.000025;
}
```

These ‘for’ loops sample the voltage from the buttery and the current that is collected from the lower current channel.

The ‘c1[j]’ is the voltage of the sample resistor. Microcontroller reads the input detected number from the ADC and multiply it with 3.3 to input voltage value. Divide the second amplifying circuit gain (68.136) and minus the offset voltage in the OPA211 (30uV). Still need to divide the gain of instrumentation amplifier (5.003) and minus the offset voltage of this instrumentation amplifier (25uV). Microcontroller samples 10 times of voltage and current and calculates the average both of them.

```c
```

Because the sample resistance is 15.5Ω, the voltage of the sample resistor should divide this resistance.

```c
if (CuL.read() <= 3) {
    current = currentLow;
} else {
    for ( int k=0; k<10 ; k++){
        c2[k] = (CuL.read()++3.3/0.000030)/5.003-0.000025;
    }
    current = currentHigh;
}
```
This program judges whether using the higher current channel. When the output from lower channel is higher than 3V which means when the detected number is higher than 0.9, the higher channel is opened. The microcontroller samples 10 times from the higher channel and calculate the current of the sample resistor, the difference between higher channel current and lower channel current calculation is that the last step amplifying gain is 1. Define a new parameter ‘current’, if input voltage is lower than 3V, the ‘current’ equal current value from the lower channel, on the contrary when input voltage higher than 3V, the ‘current’ value comes from the higher channel.

```plaintext
float power = voltage * current;
```

The last step, get the power by multiply voltage and current. The parameters are float type; this is the single precision which means the precision of this float type is 32bit.

In order to print the real time on the PC, an mbed windows serial port driver must be download. A terminal application also should be download, in this design, it uses TeraTerm as the terminal to print the data on PC.

```plaintext
pc.printf ("vol= %fV ", voltage);
pc.printf ("cinl= %f ", currentLow);
pc.printf ("cinh= %f ", currentHigh);
pc.printf ("Current = %fA ", current);
pc.printf ("Power = %fW \n", power);
```

Print the voltage, current from lower channel, current from higher channel, selected current and the real time power.

For LCD display, a ‘TextLCD.h’ library should be install in this programme project.

```plaintext
#include "mbed.h"
#include "TextLCD.h"
```

The LCD is the Hitachi HD44780 LCD controller. This is an LCD controller developed by Hitachi which was generally used for microcontroller. The character set of the controller covers ASCII characters. This device can display up to eighty characters. The common sizes of HD44780 LCD are one row of 8x1, 16x2, 20x2 and 20x4 formats. A
16X2 format is used in this design. [9]

(Figure 4-10 20x2 format HD44780 LCD)

The pin map of this LCD:

<table>
<thead>
<tr>
<th>NAME</th>
<th>IN/OUT/POWER</th>
<th>COLLECTION WITH MBED</th>
</tr>
</thead>
<tbody>
<tr>
<td>VSS</td>
<td>Ground</td>
<td>Power</td>
</tr>
<tr>
<td>VCC</td>
<td>LCD Controller Power (+3 to+5V)</td>
<td>Power</td>
</tr>
<tr>
<td>V0</td>
<td>LCD Display Bias (+5V to -5V)</td>
<td>Analog</td>
</tr>
<tr>
<td>RS</td>
<td>Register Select</td>
<td>Input</td>
</tr>
<tr>
<td>R/W</td>
<td>Read or Write</td>
<td>Input</td>
</tr>
<tr>
<td>E</td>
<td>Enable</td>
<td>Input</td>
</tr>
<tr>
<td>D0</td>
<td>Data LSB</td>
<td>I/O</td>
</tr>
<tr>
<td>D1</td>
<td>Data</td>
<td>I/O</td>
</tr>
<tr>
<td>D2</td>
<td>Data</td>
<td>I/O</td>
</tr>
<tr>
<td>D3</td>
<td>Data</td>
<td>I/O</td>
</tr>
<tr>
<td>D4</td>
<td>Data</td>
<td>I/O</td>
</tr>
<tr>
<td>D5</td>
<td>Data</td>
<td>I/O</td>
</tr>
<tr>
<td>D6</td>
<td>Data</td>
<td>I/O</td>
</tr>
<tr>
<td>D7</td>
<td>Data MSB</td>
<td>I/O</td>
</tr>
<tr>
<td>A</td>
<td>LED Backlight Anode</td>
<td>Power</td>
</tr>
<tr>
<td>K</td>
<td>LED Backlight Cathode</td>
<td>Power</td>
</tr>
</tbody>
</table>
In the mbed compiler, the LCD should be defined. Port 10 is ‘RS’, p11 is ‘enable’, p12 to p15 are ‘D4’ to ‘D7’.

```c
TextLCD lcd(p10, p11, p12, p13, p14, p15/*, TextLCD::LCD16x2*/);
```

The last step is to print the power value to LCD. Use a simple statement `lcd.printf` to achieve this operation. The first row of LCD will display ‘Power =’ and the second row display the real time power value.

```c
lcd.printf("Power = \n \n ", power);
```

5. Testing and Discussion

This chapter is for testing the designed circuit and discussing the accuracy and error in this design.

5.1. Virtual Ground Circuit Testing

Using oscilloscope test the virtual ground circuit. The circuit is built like this:

(Figure 5-1 virtual ground circuit with op-amp buffer)

Set the power source from the power supply is 10V and the result from the virtual ground circuit is 5V and -5V. These pictures show the result from the virtual ground circuit on oscilloscope, they prove that the virtual ground circuit designing is correct because output is ±5V.
5.2. Amplifying Gain Testing

The analogue circuit design was done in the chapter 4, place the probe to detect the current or voltage at the critical point and the result is show as below:

![Amplifying Gain Testing Diagram](Figure 5-3 simulate the circuit in the Multisim)

The gain in the simulation is matched the calculated result. However, for the practical circuit, there must have some errors in the circuit to lead the gain is not match the simulation, so the practical circuit must be tested and fix or reduce the errors in the circuit design or digital processor.
5.2.1. Instrumentation Amplifier

Testing the gain of the instrumentation amplifier in this chapter. In the simulation, the gain of the instrumentation amplifier is 5.003, however there must have error in the practical circuit. Therefore, test the real gain is important for analogue circuit. Connect the instrumentation amplifier circuit, the input impedances are 100kΩ and the gain resistance is 12.34kΩ. Set the load resistance is also 15.5Ω, the input voltage to the amplifier is half voltage from the power supply.

Connect the output with the cable of the multi meter (set at 20V) and the power supply is 1V. The oscilloscope can only detect to one decimal place which is same as multi meter, so in this testing, it uses multi meter. The input to the amplifier is 500mV. In this case, the output should be 500mV*5.003 = 2.501V. As shown as in figure 5-4, the output from the instrumentation amplifier is around -2.51V. There has no accuracy instrumentation to measure very precision output. Compare the output with the simulation and the practical gain is 5.022.

5.2.2. OPA211 Circuit

There has two OPA211 operation amplifier in this designs, they are used to build the last amplifying circuit in lower and higher channel. Test them separately. Firstly, connect the OPA211 which is used in lower channel, its resistance at feedback is 68kΩ. Set the power supply is 0.1V and change the input power supply of the virtual ground
circuit to 20V, in this case the voltage source of the OPA211 is ±10V. The result shows on the oscilloscope is -6.85V. Because the input impedance of OPA211 is 998Ω, the practical gain of this OPA211 amplifying circuit is \[\frac{6.85\text{V}}{0.1\text{V}} = 68.5\].

(figure 5-5 OPA211 testing result 1)

Then test the second OPA211 that the gain should be 1. Using 2V voltage from the power supply in this test, and the output from the OPA211 is -2.04V. Thus the gain of the second OPA211 circuit is \[\frac{2.04\text{V}}{2\text{V}} = 1.02\].
As these two figures show, the OPA211 is operation amplifier which has very low noise and when connect the filter capacitors with the feedback, its filter ability is good enough to output a smooth output.

5.3. Result

Based the gains of the instrumentation amplifier and two OPA211 operation amplifiers. The practical gain of the lower channel and the higher channel can be detected. The gain during the lower current channel is $5.022 \times 68.5 = 344.007$. For the higher one, the gain is $5.022 \times 1.02 = 5.12244$. Change the calculation function in the digital processor to solve the problem from the practical circuit.

\[
c1[j] = (\text{CuL.read()} \times 3.3/68.5-0.000030)/5.022-0.000025;
\]
\[
c2[k] = (\text{CuH.read()} \times 3.3/1.02-0.000030)/5.022-0.000025;
\]
Combine the circuit following the simulation in the Multisim and connect the lower channel output, the higher channel output and the voltage with the mbed microcontroller. This image shows the completely circuit which combine the aimed circuit, the analogue circuit, the microcontroller and the LCD.

(Figure 5-7 integrated circuit design)

When the input voltage from the power supply is 3V and the load resistance is 97kΩ, the voltage and the current of the load resistor are 3V and 30.9μA, the power is 92.7uW. For this power meter, the measuring results display on the computer as below:
The displayed current is 32μA and the voltage is 3.000436V. The power from the power meter is 96uW. For this set of data, the error rate of the power is 3.56%. This image shows the power value printed on the LCD at real time:

Change the load resistance from 97kΩ to 1kΩ. The ideal current should be changed to 2.95mA and the power is 8.85mW. Connect this circuit with the power meter and
display the results on the computer. The output from the lower channel is higher than 3V, thus close the lower channel and open the higher one and the current is 0.003072A. The measured power is 0.009217W. Compare this value with 8.85mW, the error rate is 4.147%.

And the LCD display:

![Figure 5-11 LCD display when power supply is 3V and RL = 1kΩ](image)

Change the load resistor to 200kΩ to test the lowest current that the power meter can
measure. Change the power supply and the minimum current can be measure is 10uA. When the current in the circuit is lower than 10uA, the current value that displayed on computer are not correctly.

5.4. Discussion

Analogue circuit design is a complicated process. Through these testing, the practical result is still not matching with the simulation result. The analogue circuit will be affected by various factors like temperature, resistance and noises. If anyone of them is changed or mismatched, the result from the analogue circuit will be changed.

Temperature may change the resistance of the resistors. This leads the resistances are not matched. The gain depends on resistance, when resistances are changed, the gain absolutely not accuracy.

Noise will also influence the accuracy of the circuit. Because the current which is lower than 1uA is so tiny and the noise cannot be filter clearly, the noise still can cover these signals. These noise may come from resistors, capacitors, power source, cables or even digital processor. These noises are difficult to be eliminated especially the platform of this project is breadboard, noises also generate from breadboard.

The error also may be generated because of mismatching amplifier voltage source. The mismatching voltage source will lead the reference voltage of the amplifier is not 0V. This will lead the error but not a big problem if the differential between the absolutely value of dual power supply.

The offset voltage is a big problem for the amplifier gain, but it will not be difficult to solve, it is already considered in the methodology. Test the offset of the AD8221 and OPA211 and the offset voltage are 25uV and 30uV. To solve the influence from the offset voltage, just minus them at calculation in the digital processor.
Unbalanced resistance also leads to the mismatching gain. In simulation, the resistance of resistor is ideal. However, in the practical circuit, the resistor is not accurate. The accuracy resistance can be measured by very high accuracy instrument and it can use high accuracy resistor to make sure the resistances match correctly.

6. Project Management

6.1. Gantt Chart

This Gantt chart was drawn before doing project. It lists the anticipate work duration for the project which includes from project title deciding to project deadline. It now appears that this Gantt chart is not suitable for the design.

(Figure 6-1 Gantt chart before project)

Figure 6-2 is the Gantt chart which drawn after project. As figure shown, the circuit schematic design spent more time in this project because it needs lots of reference support. After schematic design, simulated circuit making sure every submodule can be used and record the gain from different amplifying circuit and do optimization is the circuit not meet the specifications. Hardware connecting did not spend lots of times but for this project, it spent one week for waiting components. Testing the circuit.
costed many times however this project is lucky that the circuit did not meet big challengers when testing. The digital part design which includes calculation and display was finished in 1 week. This project also spent times on using Labview to display the waveform of from the circuit through NI myCAQ card. After design, the last thing is final report. It spent one month to finish this report.

<table>
<thead>
<tr>
<th>ID</th>
<th>Task Name</th>
<th>Start</th>
<th>Finish</th>
<th>Duration</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Circuit schematic design</td>
<td>26/06/2016</td>
<td>20/07/2016</td>
<td>46d</td>
</tr>
<tr>
<td>2</td>
<td>Circuit simulation and</td>
<td>21/07/2016</td>
<td>25/07/2016</td>
<td>16d</td>
</tr>
<tr>
<td>3</td>
<td>Hardware connection and</td>
<td>20/07/2016</td>
<td>31/07/2016</td>
<td>32d</td>
</tr>
<tr>
<td>4</td>
<td>Digital processor design</td>
<td>23/08/2016</td>
<td>27/08/2016</td>
<td>16d</td>
</tr>
<tr>
<td>5</td>
<td>Display modules</td>
<td>25/08/2016</td>
<td>02/09/2016</td>
<td>6d</td>
</tr>
<tr>
<td>6</td>
<td>Final report</td>
<td>01/09/2016</td>
<td>02/09/2016</td>
<td>33d</td>
</tr>
</tbody>
</table>

(Figure 6-1 timeline of the project)

6.2. Risk Management

Some risks come out during this project.

1. The instruments in the lab is not too accuracy to measure very high precision value of resistor this lead the low accuracy resistance leads the error in the gain of the amplifier.

2. The oscilloscope or the multi meter cannot measure the voltage low to micro volt, this leads the result from the circuit can only compare with the simulation result, this is hard to reduce the error which from the ADC of the digital processor.

3. The resistance from the cables will influence the result. If change them to better electricity conductivity cables may increase the accuracy of the result.
6.3. Project Deliverable

1. Dissertation
2. Mbed code
3. Circuit design document by Multisim

7. Conclusion and Future Work

A Power meter which is for measuring low power tested chip is designed in this project. This power meter can detect the low current of the tested chip and it has two channels to measure different levels of current. Before the designing, the aim specification was confirmed that the lower channel measures 1uA to 1mA and the higher channel measures 0.9mA to 660mA. However, lots of interferences may influence the practical circuit like environment, the resistor accuracy and power source. Comparing the result in the practical circuit and the result from simulation get the error rate and analyze the reasons are important in analog design. Not just analyze ability, the analogue circuit foundations like electric noise type, instrumentation amplifier, virtual ground circuit and filter are learned through this project. The microcontroller control is also learned during this project, especially LCD connection and control.

This project is a real challenger, the designing and discussion in the analogue circuit is the biggest part in this design. Designer must be careful when project, especially analogue circuit design. Through this analogue design, the author can consider about the circuit more comprehensive and independence analysis ability improved.

This project still has some problems that did not solved. Firstly, when design the analogue circuit, this design did not consider about the influence from the environment like temperature and humidity. The characteristic of the components may be changed because the environment. Secondly, the resistances in this design is not accuracy enough, it will generate a big error of the gain and the inaccuracy sample
resistance will cause getting wrong current result. Using an accuracy resistor or having a more accuracy resistance of resistor can solve this problem. Lastly, when the current is lower than 10uA, the result is not accuracy. There has three reasons about this problem, the filter in the circuit is not hard enough to filter noise, error of the resistances and noise from the digital part. These reasons lead the circuit cannot measure the current lower than 10uA. Hope can fix this problem in the further work.
Digital code:
#include "mbed.h"
#include "TextLCD.h"

AnalogIn CuL(p20);
AnalogIn CuH(p19);
AnalogIn Vol(p17);
Serial pc(USBTX, USBRX);

TextLCD lcd(p10, p11, p12, p13, p14, p15/*, TextLCD::LCD16x2*/);

int main()
{
    float c1[10];
    float c2[10];
    float v[10];
    float current;
    float currentLow;
    float currentHigh;
    float voltage;
    while (1)
    {

        for ( int i=0; i<10 ; i++){
            v[i] = Vol.read() * 3.3;
        }
        for ( int j=0; j<10 ; j++){
            c1[j] = ((CuL.read()*3.3/68.136 - 0.000030)/5.003 - 0.000025;
        }


        if (CuL.read()*3.3 <= 3) {
            current = currentLow;
            currentHigh = 0;
        }
        else {
            for ( int k=0; k<10 ; k++){
                c2[k] = (CuH.read()*3.3/1.05556-0.000030)/5.003-0.000025;
            }
currentLow = 0;
current = currentHigh;

float power = voltage * current;

pc.printf("L= %f ", CuL.read());
pc.printf("H= %f ", CuH.read());

pc.printf("vol= %f ", voltage);
pc.printf("cinL= %f ", currentLow);
pc.printf("cinH= %f ", currentHigh);
pc.printf("Current = %fA ", current);
pc.printf("Power = %fW \r", power);

lcd.printf("Power = \n %fW \n ", power);

wait (1);
}
Bibliography


[5] AD8221 Datasheet by ANALOG DEVICES

[6] OPA211 Datasheet by TEXAS INSTRUMENTS

