expEYES-17

User Manual

Experiments for
Young Engineers and Scientists

http://expeyes.in

from

Projet PHOENIX
Inter-University Accelerator Centre
(A Research Centre of UGC)
New Delhi 110 067
www.iuac.res.in

Preface

The PHOENIX (PHYSICS WITH HOME-MADE EQUIPMENT & INNOVATIVE EXPERIMENTS) project was started in 2004 by INTER- UNIVERSITY ACCELERATOR CENTRE with the objective of improving the science education at Indian Universities. Development of low cost laboratory equipment and training teachers are the two major activities under this project.

EXPEYES-17 is an advanced version of EXPEYES released earlier. It is meant to be a tool for learning by exploration, suitable for high school classes and above. We have tried optimizing the design to be simple, flexible, rugged and low cost. The low price makes it affordable to individuals and we hope to see students performing experiments outside the four walls of the laboratory, that closes when the bell rings.

The software is released under GNU GENERAL PUBLIC LICENSE and the hardware under CERN OPEN HARDWARE LICENCE. The project has progressed due to the active participation and contributions from the user community and many other persons outside IUAC. We are thankful to Dr D Kanjilal for taking necessary steps to obtain this new design from its developer Jithin B P, CSpark Research.

Ajith Kumar B.P. (ajith@iuac.res.in) http://expeyes.in

V V V Satyanarayana

Contents

CHAPTER 1

Introduction

Science is the study of the physical world by systematic observations and experiments. Proper science education is essential for cultivating a society where reasoning and logical thinking prevails and not superstition and irrational beliefs. Science education is also essential for training enough technicians, engineers and scientists for the economy of the modern world. It is widely accepted that personal experience in the form of experiments and observations, either carried out by students or performed as demonstrations by teachers, are essential to the pedagogy of science. However, almost everywhere science is mostly taught from the text books without giving importance to experiments, partly due to lack of equipment. As a result, most of the students fail to correlate their classroom experience to problems encountered in daily life. To some extent this can be corrected by learning science based on exploration and experimenting.

The advent of personal computers and their easy availability has opened up a new path for making laboratory equipment. Addition of some hardware to an ordinary computer can convert it in to a science laboratory. Performing quick measurements with good accuracy enables one to study a wide range of phenomena. Science experiments generally involve measuring/controlling physical parameters like temperature, pressure, velocity, acceleration, force, voltage, current etc. If the measured physical property is changing rapidly, the measurements need to be automated and a computer becomes a useful tool. For example, understanding the variation of AC mains voltage with time requires measuring it after every millisecond.

The ability to perform experiments with reasonable accuracy also opens up the possibility of research oriented science education. Students can compare the experimental data with mathematical models and examine the fundamental laws governing various phenomena. Research scientists do the same with highly sophisticated equipment. The expEYES (expEriments for Young Engineers & Scientists) kit is designed to support a wide range of experiments, from school to post graduate level. It also acts as a test equipment for electronics engineers and hobbyists. The simple and open architecture of expEYES allows the users to *develop new experiments, without getting into the details of electronics or computer programming*. This User's manual describes *expEYES-17* along with several experiments, there is also a Programmer's manual available.

CHAPTER 2

The equipment

ExpEYES-17 is interfaced and powered by the USB port of the computer, and it is programmable in Python. It can function as a low frequency oscilloscope, function generator, programmable voltage source, frequency counter and data logger. For connecting external signals, it has two spring loaded terminals blocks, one for output signals and another for inputs, as shown in figure *1.1*. The software can monitor and control the voltages at these terminals. In order to measure other parameters (like temperature, pressure etc.), we need to convert them in to electrical signals by using appropriate sensor elements. The accuracy of the voltage measurements is decided by the stability of the 3.3V reference used, it is 50ppm per degree celcius. The gain and offset errors are eliminated by initial calibration, using a 16bit ADC. Even though our primary objective is to do experiments, you are advised to read through the brief description of the equipment given below. The device can be also used as a test equipment for electrical and electronics engineering experiments.

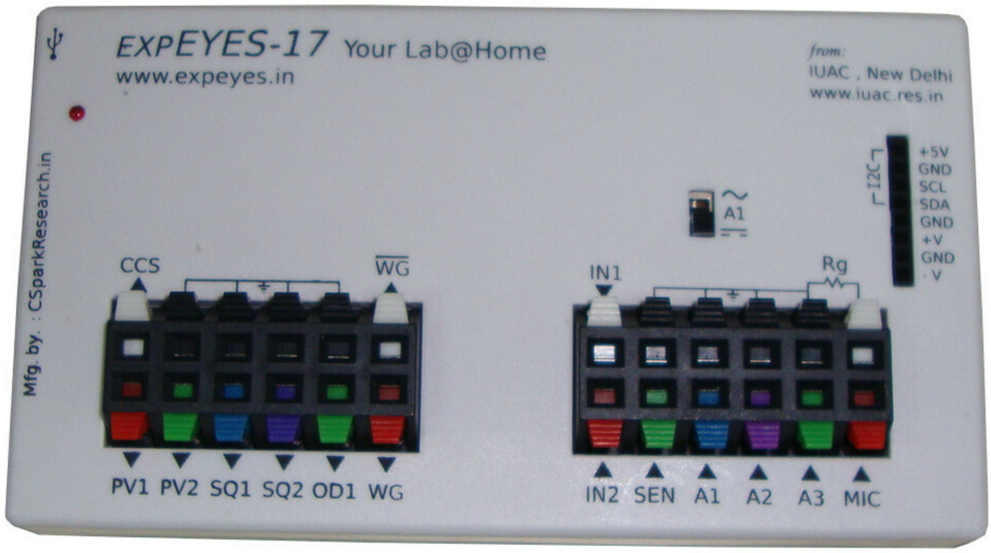
IMPORTANT :

The external voltages connected to ExpEYES17 must be within the allowed limits. Inputs A1 and A2 must be within $\pm$*16 volts range and Inputs IN1 and IN2 must be in 0 to 3.3V range. Exceeding these limits may result in damage to the equipment. To measure higher voltages, scale them down using resistive potential divider networks.*

Figure 1.1 The ExpEYES17 top panel showing the external connections.

2.1 External connections

The functions of the external connections briefly explained below. All the black coulored terminals are at ground potential, all other voltages are measured with respect to it.



2.1.1 Outputs:

Constant Current Source (CCS) :

The constant current source can be switched ON and OFF under software control. The nominal value is 1.1 mA but may vary from unit to unit, due to component tolerances. To measure the exact value, connect an ammeter from CCS to GND. Another method is to connect a known resistance (~1k) and measure the voltage drop across it. The load resistor should be less than 3k for this current source.

Programmable Voltage (PV1) :

Can be set, from software, to any value in the -5V to +5V range. The resolution is 12 bits, implies a minimum voltage step of around 2.5 millivolts.

Programmable Voltage (PV2) :

Can be set, from software, to any value in the -3.3V to +3.3V range. The resolution is 12 bits.

Square Wave SQ1:

Output swings from 0 to 5 volts and frequency can be varied 4Hz to 100kHz. All intermediate values of frequency are not possible. The duty cycle of the output is programmable. Setting frequency to 0Hz will make the output HIGH and setting it to - 1 will make it LOW, in both cases the wave generation is disabled. SQR1 output has a $100\ \Omega$ **series resistor** inside so that it can drive LEDs directly.

Square Wave SQ2:

Output swings from 0 to 5 volts and frequency can be varied 4Hz to 100kHz. All intermediate values of frequency are not possible. The duty cycle of the output is programmable. SQR2 is not available when WG is active.

Digital Output (OD1) :

The voltage at OD1 can be set to 0 or 5 volts, using software.

Sine/Triangular Wave WG:

Frequency can be varied from 5Hz to 5kHz. The peak value of the amplitude can be set to 3 volts, 1.0 volt or 80 mV. Shape of the output waveform is programmable. Using the GUI sine or triangular can be selected. WG bar is inverted WG.

2.1.2 Inputs:

Capacitance meter IN1:

Capacitance connected between IN1 and Ground can be measured. It works better for lower capacitance values, upto 10 nanoFarads, results may not be very accurate beyond that.

Frequency Counter IN2:

Capable of measuring frequencies upto several MHz.

Resistive Sensor Input (SEN):

This is mainly meant for sensors like Light Dependent Resistor, Thermistor, Photo-transistor etc. SEN is internally connected to 3.3 volts through a 5.1kΩ resistor.

$\pm 16\ V$ Analog Inputs, A1 & A2:

Can measure voltage within the $\pm$16 volts range. The input voltage range can be selected from .5V to 16V fullscale. Voltage at these terminals can be displayed as a function of time, giving the functionality of a low frequency oscilloscope. The maximum sampling rate is 1 Msps /channel. Both have an input impedance of 1MΩ .

$\pm 3.3\ V$ Analog Input A3:

Can measure voltage within the ±3.3 volts range. The input can be amplified by connecting a resistor from Rg to Ground, gain =1 + (Rg)/(10000). This enables displaying very small amplitude signals. The input impedance of A3 is 10MΩ.

Microphone input MIC:

A condenser microphone can be connected to this terminal and the output can be captured.

2.1.3 I2C Sensor Interface:

The four connections (+5V, Ground, SCL and SDA) of the 8 terminal berg strip supports I2C sensors. The software is capable of recognizing a large number of commercially available I2C sensors.

2.1.4 $\pm\ 6\ V/10\ mA$ Power supply:

The VR+ and VR- are regulated power outputs. They can supply very little current, but good enough to power an Op-Amp.

2.2 1.2.2 Accessory Set

Some accessories are provided with expEYES.

- Pieces of wires, with pin and with crocodile clip.
- Condenser microphone with leads.
- Inductor Coil (2) : 44SWG wire on 1cm dia bobbin. Around 3000 Turns (some may have more turns). These coils can be used for studying inductance, electromagnetic induction etc.
- Piezo Electric Discs (2) : Resonant frequency is around 3500 Hz. Can be energized by WG output or SQR1. Discs are enclosed in a plastic shell that forms a cavity, that enhances the amplitude of sound produced.
- DC Motor : Should be powered by a DC voltage less than 3 volts.
- Permanent Magnets : (a) 10mm dia & length (b) 5 mm dia & 10 mm length (c) Button size magnets(2)
- 5mm LEDS : RED, BLUE, GREEN, WHITE
- Capacitors : 100pF, 0.1uF , 1 uF & 22uF
- Inductor : 10 mH / 20Ω,
- Resistors : 560Ω, 1kΩ, 2.2kΩ , 10kΩ , 51kΩ and 100 kΩ

- LDR
- Two silicon diodes (1N4148) and one 3.3 volts zener diode
- NPN Transistor(2N2222)

CHAPTER 3

Software Installation

ExpEYES can run on any computer having a Python Interpreter and required modules. The USB interface is handled by a device driver program that presents the USB port as a Serial port to the Python programs. The communication the expEYES is done using a library written in Python. Programs with GUI have been written for many experiments. Eyes17 software require the following packages

- python-serial
- python-numpy
- python-scipy
- python-qt4
- python-pyqtgraph

3.1 Any GNU/Linux distributions

Download **eyes17-x.x.x.zip** (the latest version) from **http://expeyes.in** and upzip it, and change to the newly created folder. Issue the command

$ sudo sh postinst # set user write permission

$ python main.py

You will get error messages for any missing packages that are required for expeyes. Install them one by one and try again. Python programs required for several experiments are in the same directory, they are called by 'main.py'.

3.2 Debian or Ubuntu GNU/Linux distributions

Download **eyes17-x.x.x.deb** (the latest version) from the software section of **http://expeyes.in** and install it using the command;

> $ sudo gdebi eyes17-x.x.x.deb

while connected to Internet

The package 'eyes17' (later than version 3) does not depend on the earlier versions of ExpEYES, like expeyes junior. During installation gdebi will automatically dowload and install all the required packages.

3.3 The expEYES Live CD / USB pendrive

The ISO image containing support for eyes17 can be downloaded from HERE. Make a DVD or USB memory stick bootable using this ISO image (Download rufus from https://rufus.akeo.ie to do this under MSWindows)

Switch off the PC and insert the liveCD/Pendrive and switch it on. Enter the BIOS while booting, make the CDdrive/USB hard disk as the first boot device. A desktop will appear and you can start expEYES-17 from the menu **Applications->Education**->ExpEYES-17. You can also start it from a Terminal using the command:

> $ python /usr/share/expeyes/eyes17/main.py

3.4 On MSWindows

The first thing to do is to install the driver software for the USB to serial converter IC MCP2200, available on Microchip website (also given on expeyes website). After installing this the device will appear as a COM port, that can be verified from the device manager of MSWindows. After this there are two options.

A zip file containing all the necessary things for ExpEYES is available on the website, named eyes17win.zip. Unzip this file and run main.py from that. By using this method you will not able to write your own Python code to access expeyes, for that you need to install the following

1. Python-2.x version
2. python-serial
3. python-qt4
4. python-pyqtgraph,
5. python-numpy
6. python-scipy

Download the eyes17-x.x.x.zip (take latest version) from the website. Unzipping the files will create a directory **named eyes17**, run **main.py** from that.

CHAPTER 4

The main GUI program

Start Applications->Education->ExpEYES-17 from the menu. A four channel oscilloscope screen with several extra features will open as shown in figure *The scope17 screen showing two traces*. Various experiments can be selected from the menu.

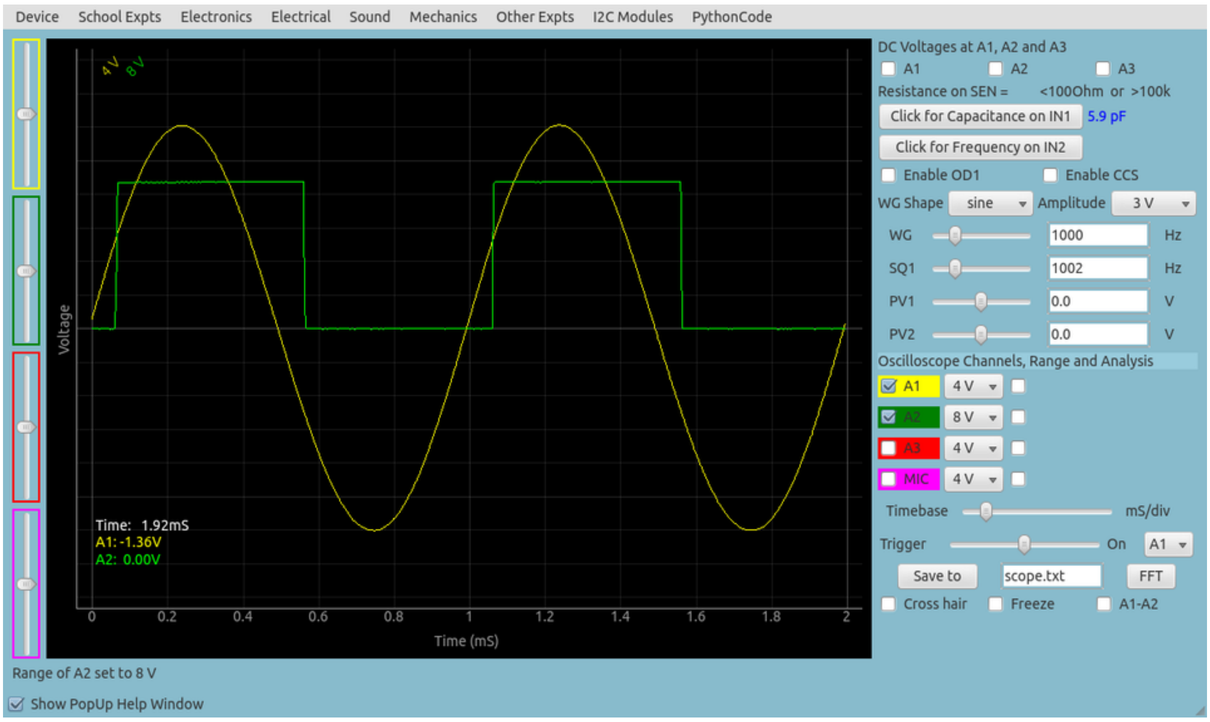


Fig. 1: The scope17 screen showing two traces

The main window looks like a low frequency four channel oscilloscope, with some extra features, on the right side panel. Applications for various experiments can be selected from the pulldown menu. A brief description of the oscilloscope program is given below.

- Any of the four inputs (A1, A2, A3 or MIC) can be enabled using the corresponding checkbox. The input range can be selected by clicking on the menubutton on the right side of the checkbox. Select the desired input range from the popup menu.
- There is another checkbox, to enable mathematical fitting of the data using V = V0sin(2πft + θ) + C to show the amplitude and frequency.
- The horizontal scale (time base) can be changed by a slider, from .5 mS fullscale to 500 mS full scale.
- The Checkbutton **Freeze**, allows to pause and resume the oscilloscope operation.
- The Trigger level can be set by a slider, and there is a menubutton to select the trigger source.
- To save the traces to a file, edit the filename and click on the **SaveTo** button.
- Clicking on **FFT** shows the frequency spectrum of all the eneabled channels, appears on popup windows.

In addition to the Oscilloscope, there are several measurement/control options available on the GUI, they are explained below.

- If selected, the voltages at the inputs A1, A2 and A3 are sampled every second and displayed.
- The resistance connected between SEN and Ground is measured and displayed every second.
- Clicking **Capacitance on IN1**, measures the value of the capacitor connected between IN1 and GND.
- Clicking **Frequency on IN2**, measures the frequency of an external digital (TTL standard) pulse connected to IN2
- The shape of the waveform can be selected using the menubutton, default shape is sine. It can be changed to triangular. When the square wave option is selected, the output is shifted to SQ2. You cannot have sine/triangular and SQ2 at the same time.
- Frequency of the Waveform generator WG can be set using the slider or the text entry window. The two input methods follow each other, changing the slider will change the text field and entering data using text field will set the slider to that value. The frequency will be set to the nearest possible value and it will be displayed in the message window at the bottom. The amplitude of WG output can be set to 3 volts, 1 volt or 80 mV.
- SQ1 can be set using the same method as explained above. The duty cycle can be set between 1% to 99%, default is 50%.
- The programmable volages PV1 and PV2 are also set in a similar manner.
- Checkbuttons are provided to control OD1 and CCS.

CHAPTER 5

Getting Familiar with ExpEYES17

Before proceeding with the experiments, let us do some simple exercises to become familiar with expEYES-17. Connect the device a USB port and start the ExpEYES-17 program from the menu 'Applications->Education'. Enable the 'Popup Help' option and select the first few items from the school menu.

The following chapters are organized according to the pulldown menus of the eyes17 program, each chapter containing the experiments under the corresponding menu; like School level, Eelectronics, Eelectrical etc. To perform the expeiment, select it from the menu. Online help is available for every experiment, making this manual almost redundant.

The screen shots given in this document are not from the GUI program, because the black background images are difficult to print. The plots are generated by separate code.

CHAPTER 6

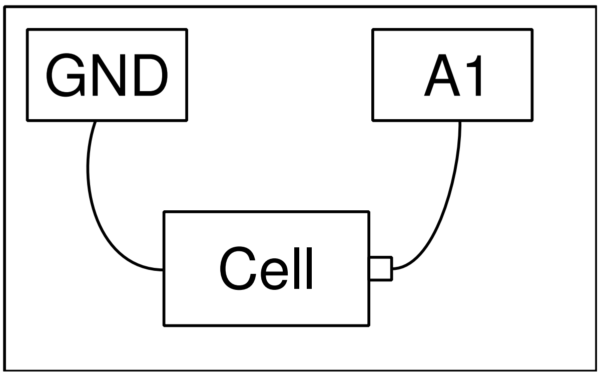
School Level Experiments

This chapter will discuss the experiments and demonstrations without much data analysis, experiments given in the menu SchoolExpts. Simple tasks like measuring voltage, resistance, capacitance etc. will be done followed by resistances changing with temperature or light. The concept of Alternating Current is introduced by plotting the voltage as a function of time. Generating and digitizing sound will be covered. When an experiment is selected, the corresponding help window will popup, if enabled.

6.1 Measuring Voltage

Objective

Learn to measure voltage using expEYES and get some idea about the concept of Electrical Ground. A dry-cell and two wires are required.



Procedure

- Observe the voltage at A1 displayed.
- Repeat by reversing the cell connections.

Discussion

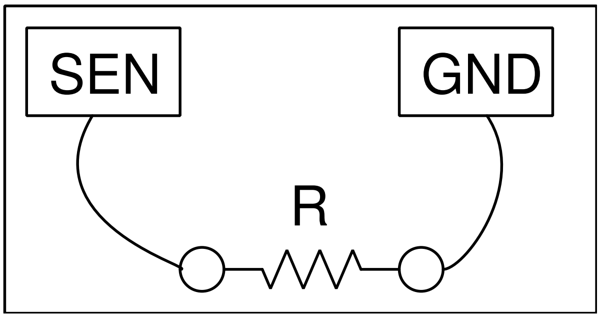
Voltages measured value is +1.5 volts and it becomes -1.5 after reversing the connections.

We are measuring the potential difference between two points. One of them can be treated as at zero volts, or Ground potential. The voltage measuring points of expEYES measure the voltage with respect to the terminals marked GND. We have connected the negative terminal of the cell to Ground. The positive terminal is at +1.5 volts with respect to the negative terminal. *Will it show correct voltage if GND is not connected ?*

6.2 Measuring Resistance

Objective

ExpEYES has a terminal marked **SEN**, that can be used for measuring resistances in the range of $100\ \Omega$ to $100\ k\Omega$. You can also study the series and parallel combination of resistors.



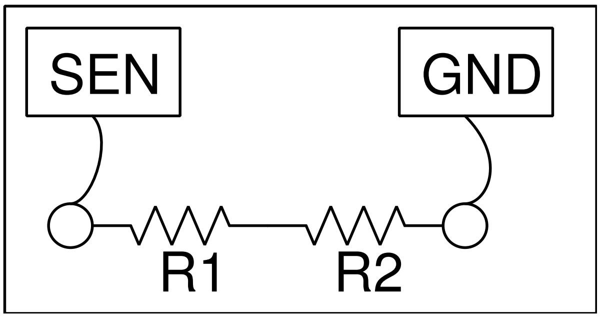
Procedure

- Connect the resistor between SEN and any Ground terminal
- Observe the value shown on the right side panel

6.3 Measuring Resistance series combination

Objective

The effective resistance of a series combination of resistors is $R = R_1 + R_2 + \ldots$.



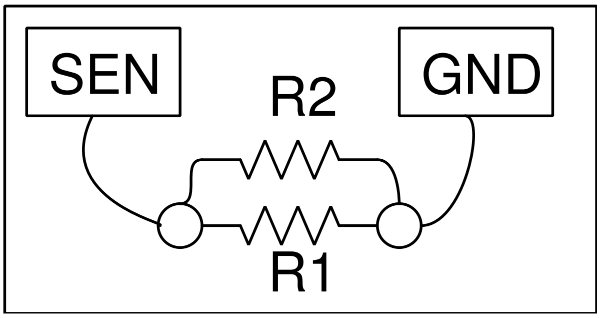
Procedure for two resitors

- Connect one resistor in SEN and the other one in Ground terminal. Connect opposite ends of the resistors together.
- Observe the value shown on the right side panel

6.4 Measuring Resistance parallel combination

Objective

For parallel combination of resistors, this relation exists between the effective resistance R and the components: $\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2} + \ldots$



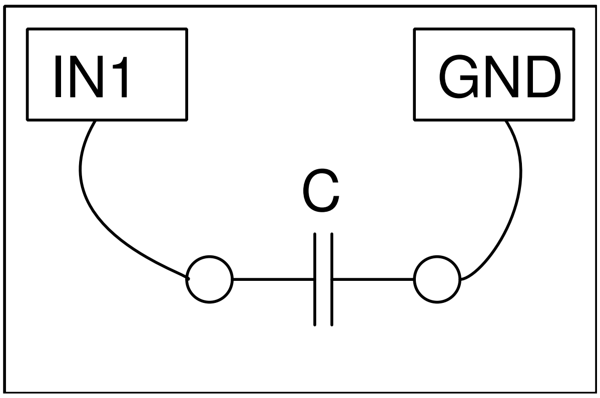
Procedure for two resistors

- Connect both resistors between SEN and any Ground terminal
- Observe the value shown on the right side panel

6.5 Measuring Capacitance

Objective

Measuring a capacitance.



Procedure

- Connect the capacitor between IN1 and Ground.
- Click on “Capacitance on IN1” . Should not touch the capacitor while measuring

Discussion

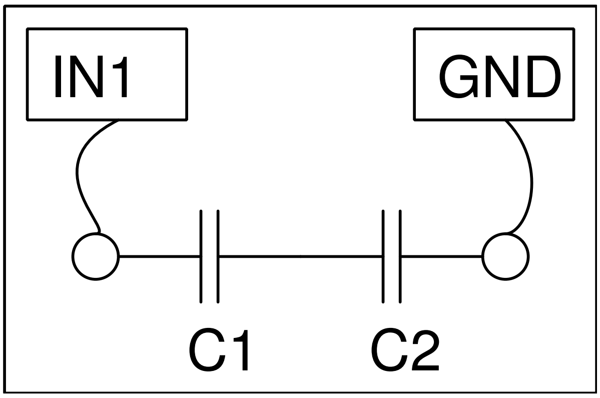
We have used $100\ pF$ capacitors in this activity.

You can make the capacitors by pasting thin metal foils on both sides of insulators like paper, polythene or glass.

6.6 Measuring Capacitance in series combination

Objective

Measuring the capacitance of series combination of capacitors.



Procedure for two capacitors

- Connect one capacitor in IN1 and the second one in Ground. Connect the opposite ends of both capacitors together.
- Click on "Capacitance on IN1" . Should not touch the capacitor while measuring

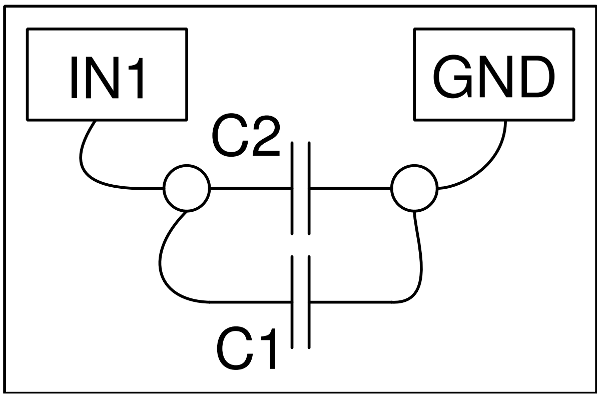
Discussion

For a series combination of capacitors, the effective capacitance is given by the relation $\frac{1}{C} = \frac{1}{C_1} + \frac{1}{C_2} + \ldots$.

6.7 Measuring Capacitance of a parallel combination

Objective

Measuring the capacitance of a parallel combination of capacitors.



Procedure for two capacitors

- Connect both capacitors between IN1 and Ground.
- Click on "Capacitance on IN1" . Should not touch the capacitor while measuring

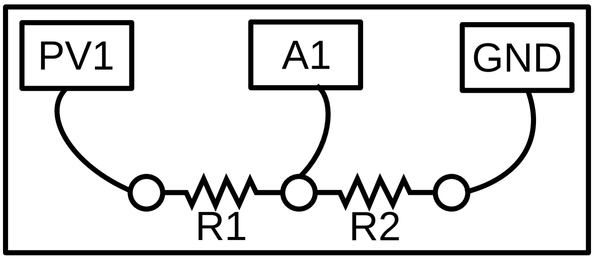
Discussion

For parallel combination, the effective capacitance is given by $C = C_1 + C_2 + \ldots$.

6.8 Measure resistance by comparison

Objective

Learn to apply Ohm's law to find the value of an unknown resistance by comparing it with a known one. Voltage across a resistor is given by V = IR . If same amount of current is flowing through two different resistors, the ratio of voltages will be the same as the ratio of resistances, $I = U_{A1}/R_2 = (U_{PV1} - U_{A1})/R_1$.



Procedure

- Connect the unknown resistor R from PV1 to A1.
- Connect $1\ k\Omega$ (R_2) from A1 to Ground.
- Set PV1 to 4 volts.
- Measure voltage at A1. Calculate the current $I = U_{A1}/R_2$. Value of $R_1 = (U_{PV1} - U_{A1})/I$.
- Select Electrical->Plot I-V curve from the menu to get an I-V plot

Discussion

What is the limitation of this method ? How do we choose the reference resistor ? suppose the unknown value is in $M\Omega$, what will be the voltage drop across a $1\ k\Omega$ reference resistor ? Our voltage measurement is having a resolution of $1/4096$.

We will use this method later to measure the resistance of solutions, using AC.

6.9 Direct and Alternating Currents

Objective

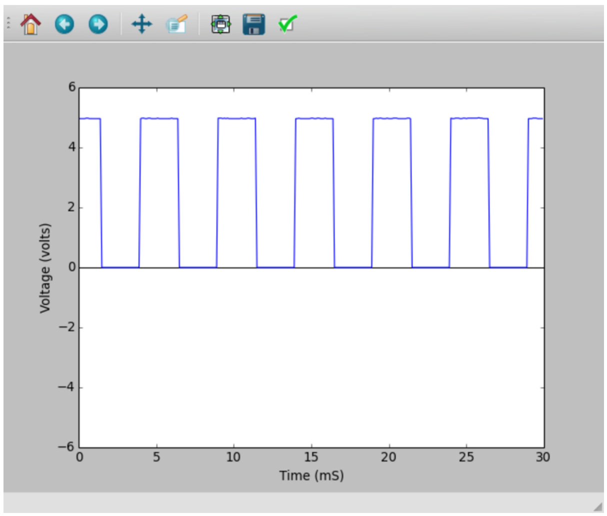
Introduce the concept of time dependent voltages, using a V(t) graph. Compare the graph of DC and AC.

Procedure

- Set PV1 to 2 volts and Set WG to 200 Hz
- Enable analyse on A1, to measure amplitude and frequency.
- Enable A2

Discussion

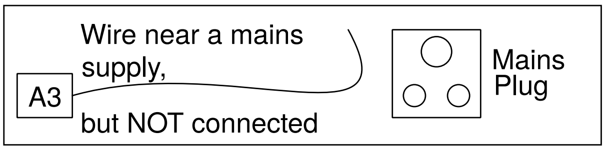
In the plot if voltage is not changing, it is pure DC. If the voltage is changing with time, it has an AC component. if the average voltage is zero, it is pure DC. In the second plot, the voltage is changing from zero to file volts, is it AC, DC or something else ???



6.10 AC mains pickup

Objective

Learn about the AC mains supply. Explore the phenomenon of propagation of AC through free space.

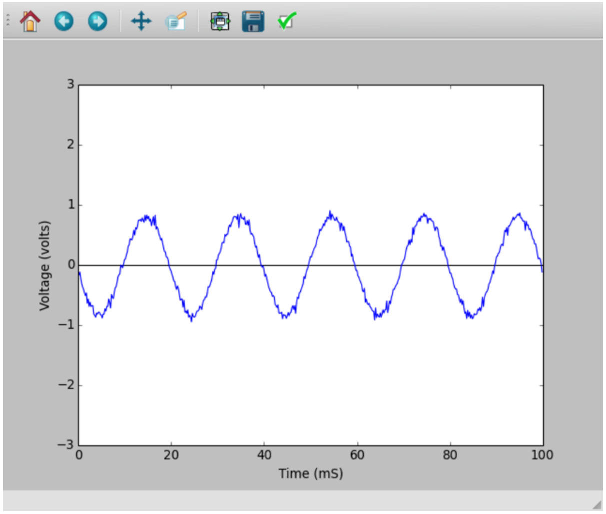


Procedure

- Connect a long wire to A3
- Take one end of the wire near the AC mains line, without touching any mains supply.
- Enable A3, and it's analysis.

Discussion

The power line pickup is shown below, there are five cycles in 100 milliseconds. Without making any connection, how are we getting the AC voltage from the mains supply ? Why the voltage increaes when you touch the end of the wire connected to A1 by hand.

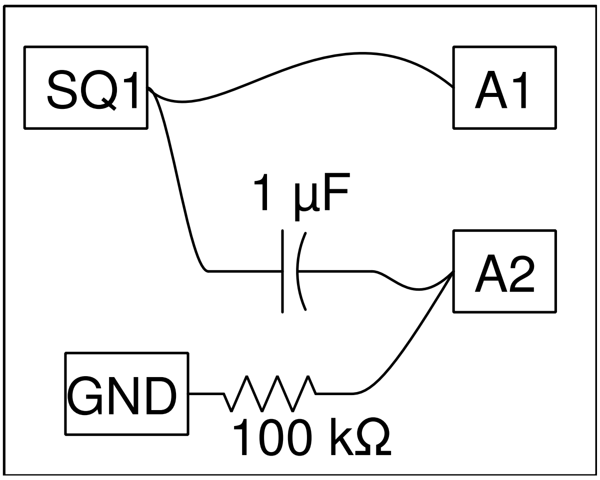


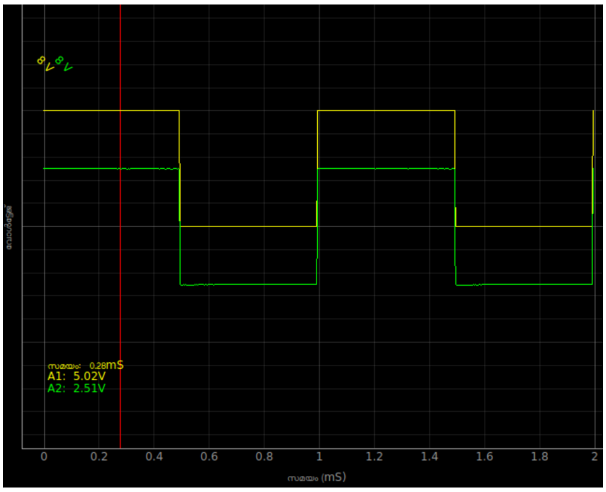
6.11 Separating DC & AC components

Objective

Separating AC and DC components of a voltage waveform using a capacitor.

Procedure





- Set SQR1 to 500 Hz
- Enable A1 and A2
- Adjust the horizontal scale to see several cycles.

Discussion

The observed waveforms with and without the series capacitor are shown in figure. The voltage is swinging between 0 and 5 volts. After passing through the capacitor the voltage swings from -2.5 volts to +2.5 volts.

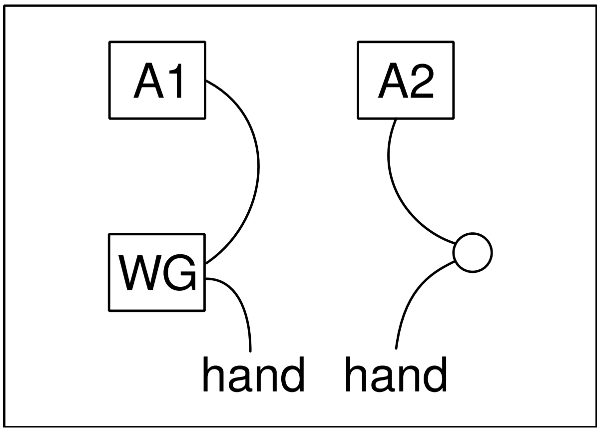
What will you get if you subtract a 2.5 from the y-coordinate of every point of the first graph? That is what the capacitor did. It did not allow the DC part to pass through. This original square wave can be considered as a 2.5V AC superimposed on a 2.5V DC.

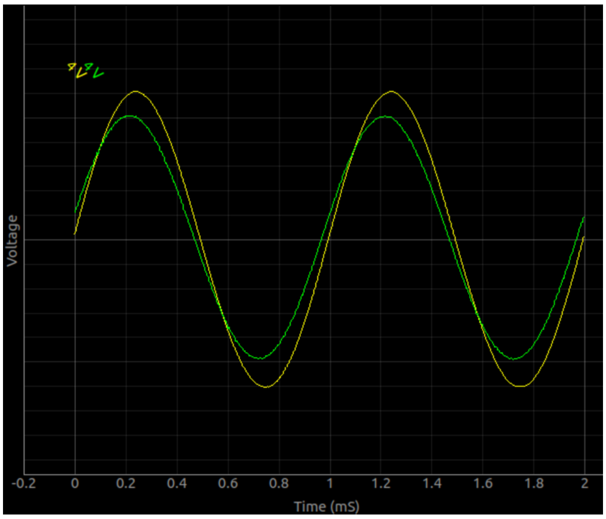
You may need to connect a resistor from A2 to GND to see a waveform swinging between -2.5 to +2.5 volts. Remove the resistor and observe the result.

6.12 Human body as a conductor

Objective

Touching the AC mains is fatal to us because our body is a conductor. We can explore this using low voltage signals.





Procedure

- Set WG to 200 Hz.
- Enable A1, A2 with analysis enabled.
- Conenct WG and A1, with a wire
- Connect WG and A2 through your body and note voltages
- Repeat it using a 3 volt DC signal from PV1.

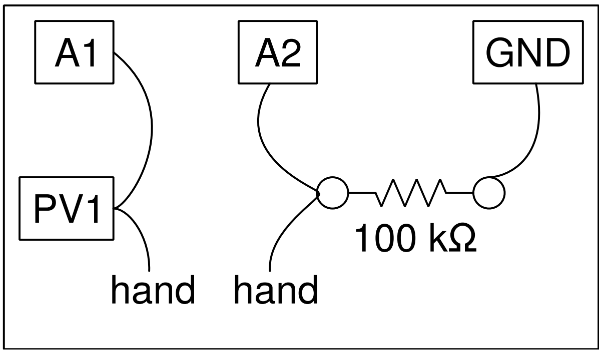
Discussion

The observed peak voltage will be less than 3volts, due to the resistance of the body. There could be some ripple due to the 50Hz AC pickup. This can be eliminated by performing the experiment far away from power lines, using a laptop.

6.13 Resistance of human body

Objective

Measure the resistance of human body by comparing it with a known resistor. We start with a DC input from PV1 and then using the AC signal from WG.



Procedure

- Set PV1 to 3 volts

- Join PV1 and A2, through your body and measure voltage at A2
- Calculate your body's resistance, as given in section *2.4*
- Repeat using SINE instead of PV1. Enable analysis on A1 and A2 to measure voltage.

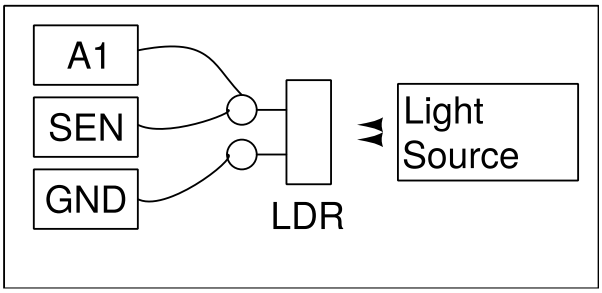
Discussion

The DC measurements are affected more by the electrical noise. The AC resistance should be less than the DC resistance. The resistance is due to our skin and AC can pass through this, like it passes through a capacitor.

6.14 Light dependent resistors

Objective

Learn about LDR. Measure intensity of light and its variation with distance from the source.



Procedure

- Measure the LDR's resistance, for different light intensities.
- Iluminate LDR using a fluorescent lamp, A1 should show ripples
- Put A1 in AC mode and measure ripple frequency

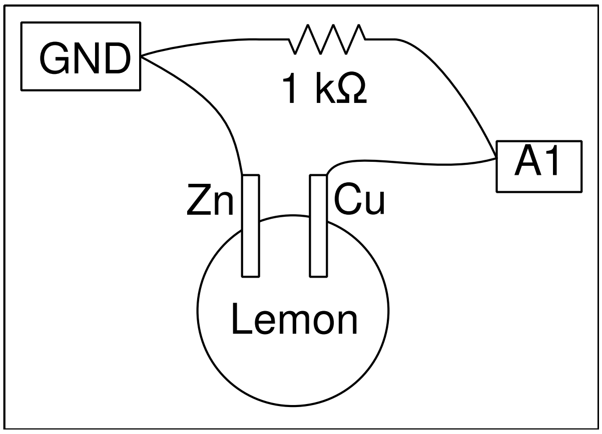
Discussion

The resistance vary from 1kΩ to around 100 kΩ depending on the intensity of light falling on it. The voltage is proportional to the resistance. The resistance decreases with intensity of light. If you use a point source of light, the resistance should increase as the square of the distance between the LDR and the light source.

6.15 Voltage of a lemon cell

Objective

Make a voltage source by inserting Zinc and Copper plates into a lemon. Explore the current driving capability and internal resistance.



Procedure

- Click on A1 to measure voltage
- Measure the voltage with and without the 1k resistor

Discussion

Voltage across the Copper and Zinc terminals is nearly .9 volts. Connecting the resistor reduces it to 0.33 volts. When connected, current will start flowing through the resistor. But why is the voltage going down ?

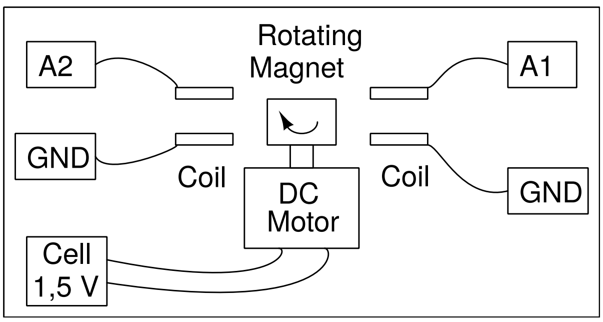
What is the internal resistance of the cell ?

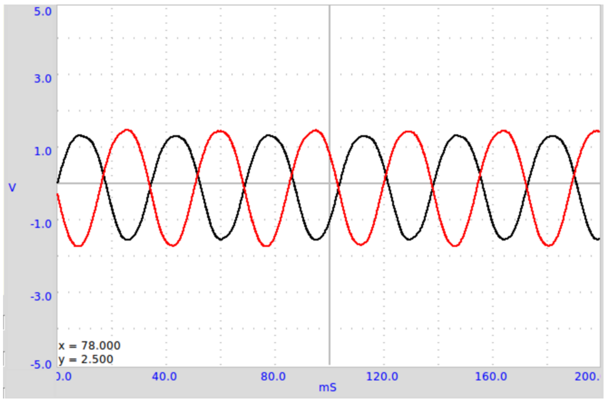
Current is the flow of charges and it has to complete the path. That means, current has to flow through the cell also. Depending on the internal resistance of the cell, part of the voltage gets dropped inside the cell itself. Does the same happen with a new dry-cell ?

6.16 A simple AC generator

Objective

Measure the frequency and amplitude of the voltage induced across a solenoid coil by a rotating magnet. Use the 10 mm x 10 mm magnet and the 3000T coils that comes with the kit.





Procedure

- Mount the magnet horizontally and power the DC motor from a 1.5 volts cell

- Enable A1 and A2, with analysis option
- Set timebase to 100 mS full scale
- Bring the coil near the magnet (not to touch it), watch the induced voltage
- Repeat the experiment using 2 coils.

Discussion

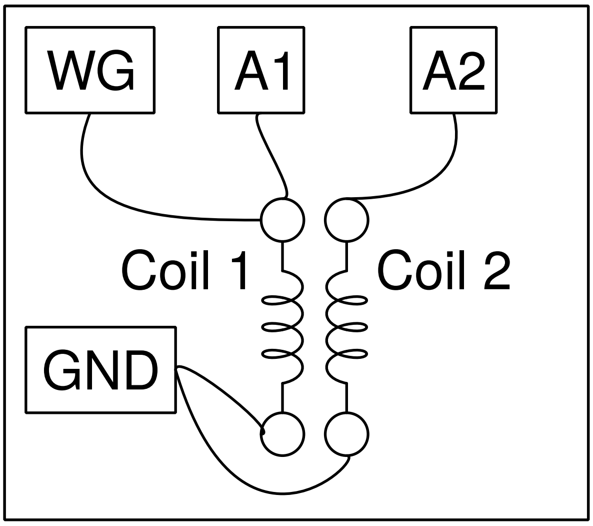
The voltage output is shown in figure. The phase difference between the two voltages depends on the angle between the axes of the two coils.

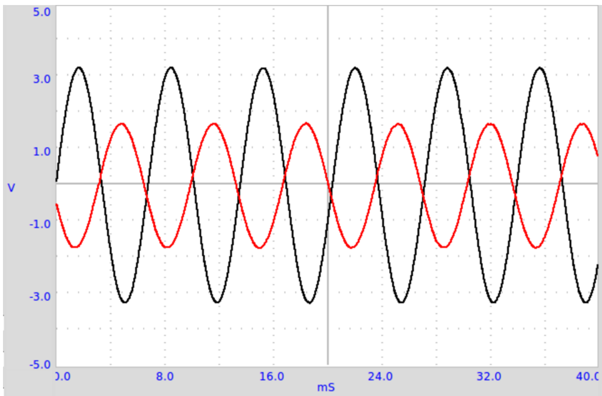
Bring a shorted coil near the magnet to observe the change in frequency. The shorted coil is drawing energy from the generator and the speed get reduced. The magnetic field in this generator is very weak. The resistance of the coil is very high and trying to draw any current from it will drop most of the voltage across the coil itself.

6.17 AC Transformer

Objective

Demonstrate mutual induction using two coils, supplied with ExpEYES. One coil, the primary, is connected between WG and Ground. The axes of the coils are aligned and a ferrite core is inserted.begin_inset Separator latexparend_inset





Procedure

- Make connections as shown in the figure
- Enable A1 and A2
- Set WG to 500 Hz
- Bring the coils close and watch the voltage on A2.
- Try inserting an ion core

Discussion

The applied waveform and the induced waveform are shown in figure. A changing magnetic filed is causing the induced voltage. In the previous two experiments, the changing magnetic field was created by the movement of permanent magnets. In the present case the changing magnetic field is created by a time varying current.

Try doing this experiment using a squarewave. Connect a $1k\Omega$ resistor across secondary coil to reduce ringing.

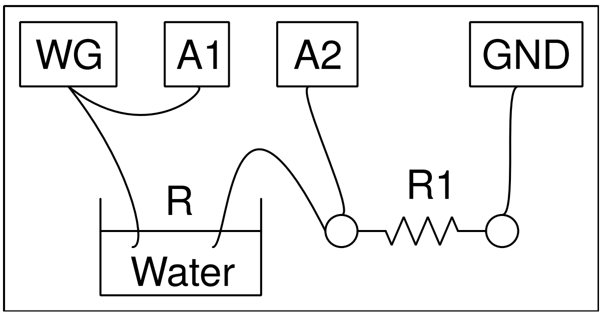
The concept of Alternating Current is introduced by plotting the voltage as a function of time. The behavior of circuits elements like capacitors and inductors in AC and DC circuits are explored, by measuring parameters like amplitude, frequency and phase. Converting electrical signals into sound and back is demonstrated.

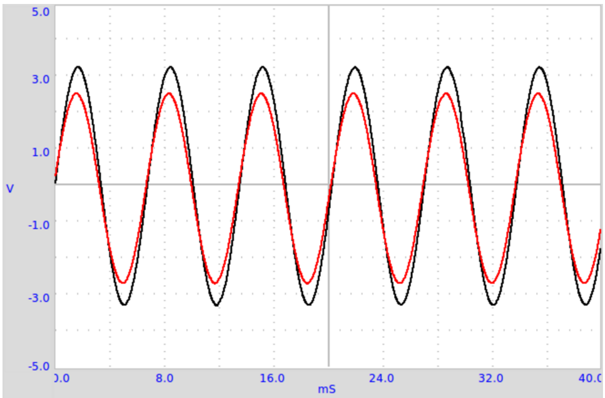
For each experiment, make connections as per the diagram given.

6.18 Resistance of water, using AC

Objective

Measure the resistance of ionic solutions, using both DC and AC voltages. We have used normal tap water. Try measuring the resistance using a multimeter first.begin_inset Separator latexparend_inset





Procedure

- R1 should be comparable to R, start with 10k.
- Enable A1 and A2
- Calculate the resistance as explained in section *2.4*

Discussion

Observed values are shown in the table. The DC and AC resistances seems to be very different. With DC, the resistance of the liquid changes with time, due to electrolysis and bubble formation. The resistance does not depend much on the distance between the electrodes, the area of the electrode is having some effect. The resistance depends on the ion concentration and presence of impurities in the water used.

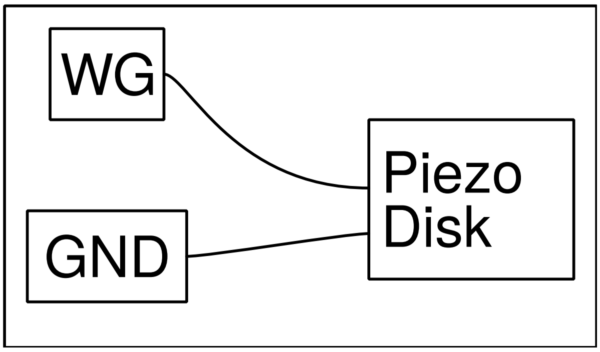
Try changing the distance between electrodes. Try adding some common salt and repeat the measurements. Why is the behavior different for AC and DC ? What are the charge carriers responsible for the flow of electricity through solutions ? Is there any chemical reaction taking place ?

6.19 Generating sound

Objective

Generate sound from electrical signals, using a Piezo-electric buzzer. Digitize sound and measure its frequency. Use the Piezo buzzer or any other source of sound like a tuning fork.

Procedure



- Enable A1, and its analysis
- Set WG to 1000Hz, change it and listen to the sound.

Discussion

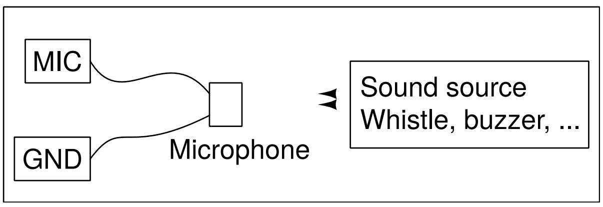
When you change the frequency of the voltage that excites the Piezo, both the frequency and the intesity of the sound changes. The intensity is maximum near 3500 Hz, due to resonance. The resonant frequency of the Piezo buzzer is decided by its size and mechanical properties.

6.20 Digtizing sound

Objective

Digitize sound signals from a microphone, and measure its frequency. Use the Piezo buzzer or any other source of sound like a tuning fork.

Procedure



- Enable A1 and MIC , with analysis

- Position the buzzer facing the microphone
- Set WG to 1000Hz, change it and watch the MIC output
- Use a whistle instead of the buzzer and find out the frequency of MIC output.

Discussion

The driving signal and the microphone output is shown in figure

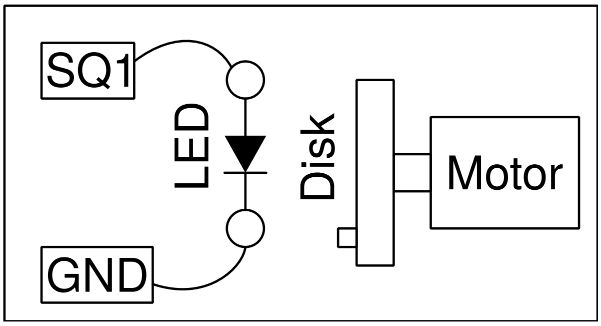
Sound waves create pressure variations in the medium through which it travel. The microphone generates a voltage proportional to the pressure. The voltage variations are in tune with the pressure variations. You can consider the microphone as a pressure sensor, but working only for time varying pressures.

6.21 Stroboscope

Objective

Observation of a periodic phenomenon with a periodic flashed light.

Procedure



- The disk is rotated by powering the motor by a 1.5 V cell.
- The disk is illuminated with light from the LED only, no other light should be present.
- Adjust the frequency of SQ1, the disk will appear stationary when it is equal to the frequency of rotation of the disk.

Discussion

When the frequency of the phenomenon under observation and the frequency of the flashing light are matching, one can see a still image.

What happens when the frequency of the light is slightly increased, or slightly decreased?

What happens when the frequency of the flasing light is twice the frequency of the phenomenon? and when it is the half of its value?

CHAPTER 7

Electronics

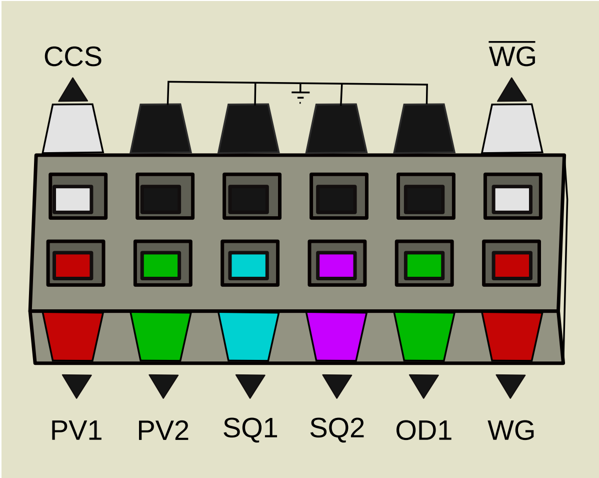
This chapter explains several electronics experiments. Most of them are done using the oscilloscope GUI. Some of them like Diode and Transistor characteristics have a dedicated GUI.

7.1 Four-channel oscilloscope, and much more

Eyes17 comes with an application whose default User Interface is an enhanced four-channel oscilloscope.

- Link to YouTube videos
- The Oscilloscope program mainly functions as a four channel oscilloscope, with inputs A1, A2, A3 and MIC.
- Adjust the x-axis limit of the graph, using the Timebase Slider, generally to view several cycles of the waveform.
- If the waveform is not stable, select the proper trigger source. If needed adjust the Trigger level.
- The traces can be saved to a file, in text format. It is possible to take the Fourier transform and view the frequency spectrum of the input waveform.
- The oscilloscope program also has control/monitor widgets on the right side panel to access most of the ExpEYES features.
- The inputs A1, A2, A3 and the resistance connected to SEN are measured and displayed every second. But these readings are meaningless when AC inputs are connected.
- For sinusoidal AC inputs, enable the Check-Button in front of the channel widget to view the Peak voltage and frequency.

- The ExpEYES Input/Output terminals are briefly described below.

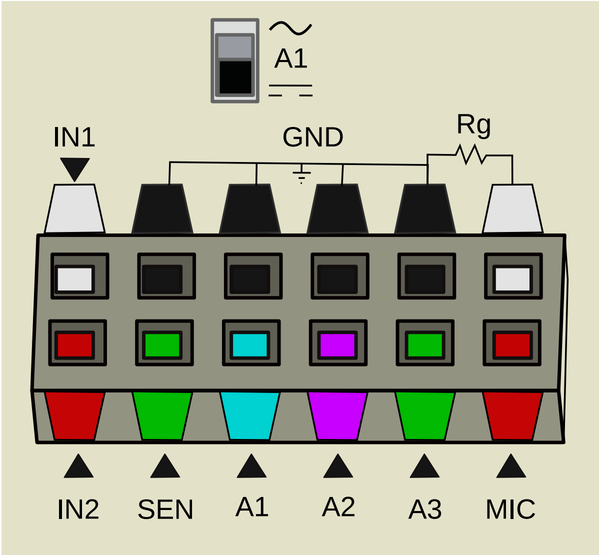


7.1.1 Output Terminals

- **CCS:** $1.1\ mA$ Constant Current Source. On/Off using Check-Button Enable CCS.
- **PV1:** Programmable Voltage, $\pm 5\ V$ range. Can be set using the Slider or Text-Entry widget
- **PV2:** Similar to PV1, but ranges from $-3.3\ V$ to $+3.3\ V$
- **SQ1:** Square Wave Generator, swings from 0 to $5\ V$. Frequency can be set from $1\ Hz$ to $5\ kHz$.
- **SQ2:** Same as SQ1, but available as an option of WG.
- **OD1:** Digital Output, voltage can be set to 0 or $5\ V$.
- **WG:** Waveform Generator. Frequency from $1\ Hz$ to $5\ kHz$. Amplitude can be set to $3\ V$, $1\ V$ or $80\ mV$. Can be set to Sine, Triangular or Square. In Square mode the output is on SQ2, with 0 to $5\ V$ swing.
- **-WG:** Inverted output of WG

7.1.2 Input Terminals

- **IN1:** Input for measuring Capacitance. Push-Button provided for measurement.
- **IN2:** Input for measuring frequency of digital signals, swinging between 0 and 3 to $5\ V$. Push-Button provided for measurement.
- **SEN:** Input for measuring resistance. This point is internally connected to $3.3\ V$ via a $5.1\ k\Omega$ resistor
- **A1:** Voltage measurement point, functions as voltmeter and oscilloscope. Maximum Input range $\pm\ 16\ V$, range is selectable from pull down menu. AC/DC mode selection by slider switch on the box.

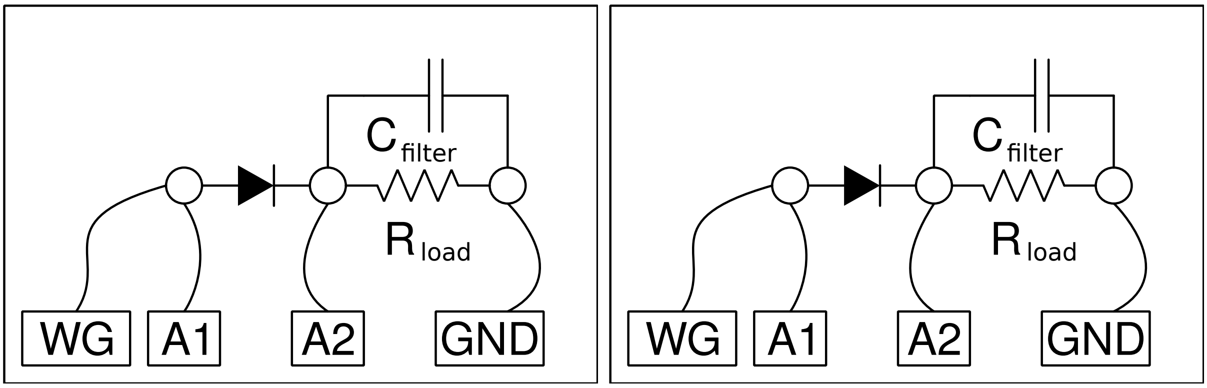


- **A2:** Same as A1, but no AC coupled mode
- **A3:** Voltage measurement in $\pm\, 3.3\ V$. Small signals can be amplified by connecting a resistance from Rg to Ground
- **MIC:** Condenser microphone input, output appears as the fourth channel of the oscilloscope
- **Rg:** Gain resistor for A3. $Gain = 1 + \frac{R_g}{100}$. For example connecting a $1\ k\Omega$ resistor gives a gain of 11.

7.2 Half wave rectifier using PN junction

Objective

Learn the working of a PN junction diode as a rectifier. RC filtering to reduce the ripple (the AC component).



Procedure

- Make connections and observe the output
- Connect a $1\ k\Omega$ load resistor, note the difference in amplitude

- Connect a $1\mu F$ capacitor, and see the filtering effect.
- Try different values load resistors and filter capacitors

Discussion

The negative half is removed by the diode as shown in figure. Also notice that the voltage in the positive half is reduced by around 0.7 volts, the voltage drop across a silicon diode. A load resistor is required for the proper operation of the circuit, it could be more than 1kΩ but do NOT use very low values since our AC source can drive only up to 5 mA current.

We can see that the capacitor charges up and then during the missing cycle it maintains the voltage. The remaining AC component is called the ripple in the DC.

Can we use very large capacitance to reduce the ripple ?

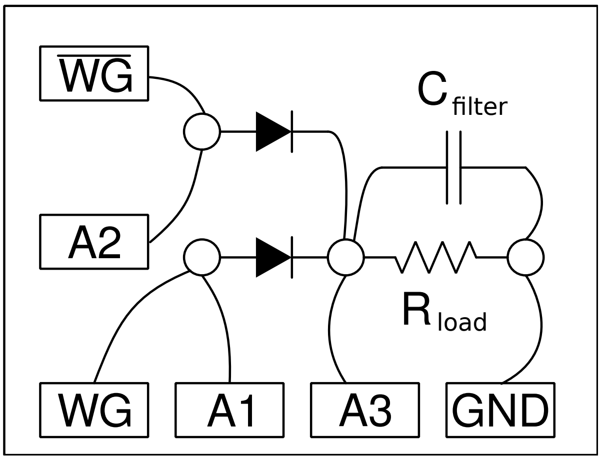
During what part of the cycle does current flow through the diode ?

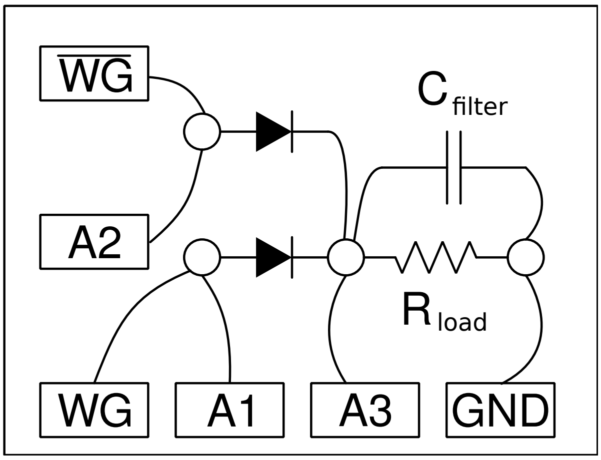
Amount of peak current is decided by what ?

7.3 Fullwave rectifier using PN junctions

Objective

Make a full wave rectifier, using two diodes. Two AC waveforms, differing by 180 degree in phase as required. WG and WG bar provide the same.





Procedure

- Make connections
- Enable A1, A2 and A3
- Set WG to 200Hz and adjust timebase to view 4 to 5 cycles

Discussion

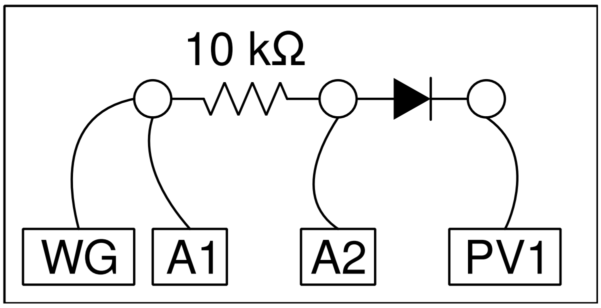
Adding capacitors to reduce the ripple is left as an exercise to the user. This experiment is only to demonstrate the working of a full wave rectifier, it cannot provide more than few milli amperes of current.

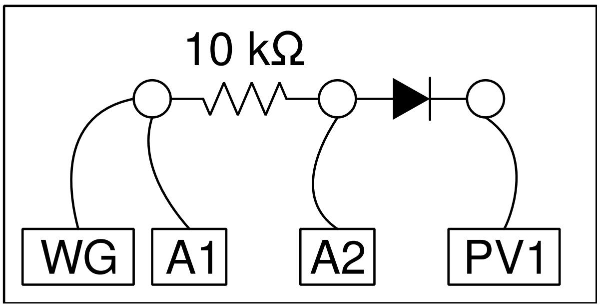
Why full-wave rectifier is superior to half-wave rectifier ?

7.4 Clipping using PN junction diode

Objective

Demostrate clipping of an AC signal at different levels, using a PN junction diode





Procedure

- Make connections and observe the output
- Change PV1 and note the difference in the output

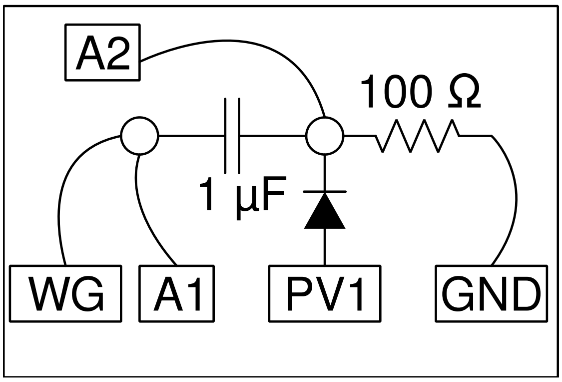
Discussion

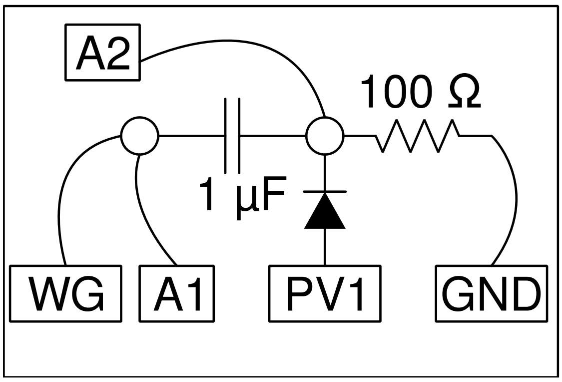
The clipping level is decided by the applied DC voltage and the diode drop.

7.5 Clamping using PN junction diode

Objective

Demostrate clamping of an AC signal at different levels, using a PN junction diode





Procedure

- Make connections and observe the output
- Change PV1 and note the difference in the output

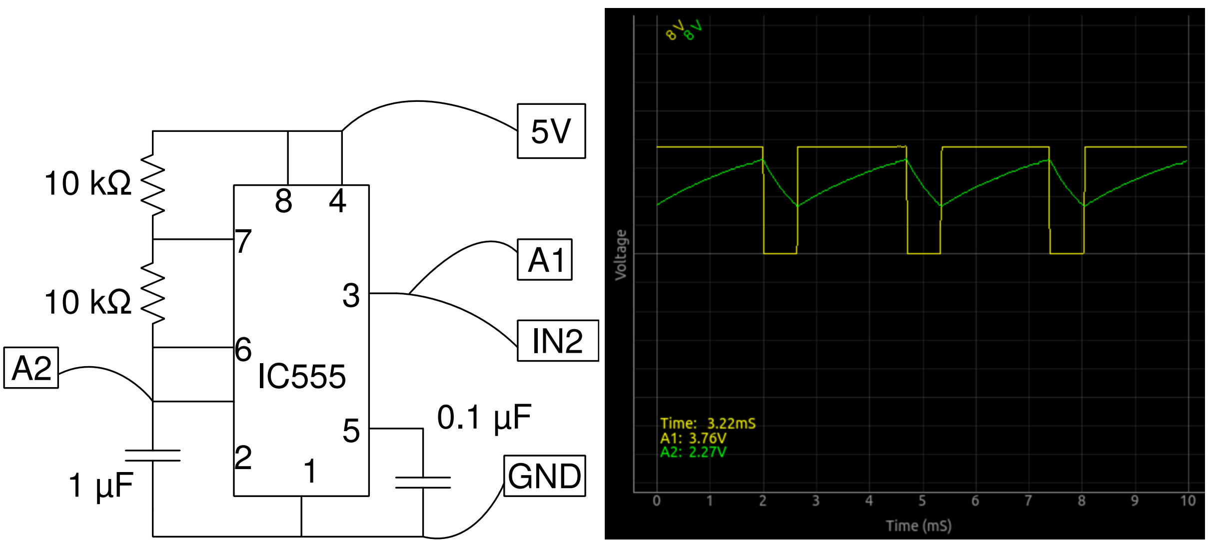
Discussion

The clamping level is decided by the applied DC voltage and the diode drop.

7.6 IC555 Oscillator

Objective

Wire an astable multivibrator circuit using IC555, measure the frequency and duty cycle of the output.



Circuit is shown in figure. The frequency is given by $f = 1/(\ln 2 \times C \times (R_1 + 2R_2)$. The HIGH time is given by $\ln 2 \times C \times (R_1 + R_2)$ and LOW time by $\ln 2 \times C \times R_2$.

Procedure

- Make connections
- measure frequency and duty cycle.
- Repeat by changing the value of R1

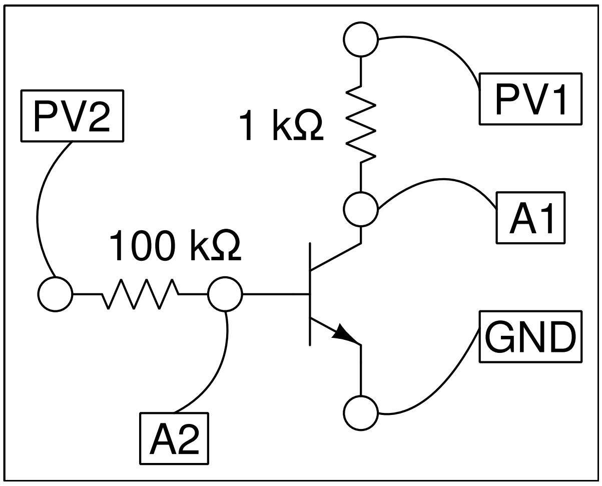
Discussion

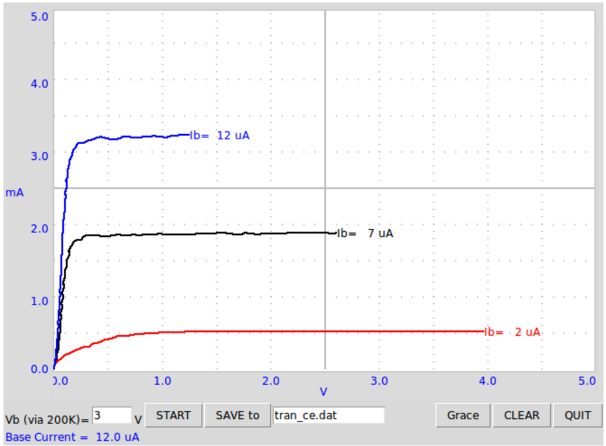
The output waveform is shown in figure. Change the value of resistors or the capacitor, and compare the frequency and duty cycle with the calculated values.

7.7 Transistor Output characteristics (CE)

Objective

Plot the output characteristic curve of a transistor. Collector is connected to PV1 through a 1K resistor.





Procedure

- Set base voltage to the 1 volt and START.
- Repeat for different base currents.

Discussion

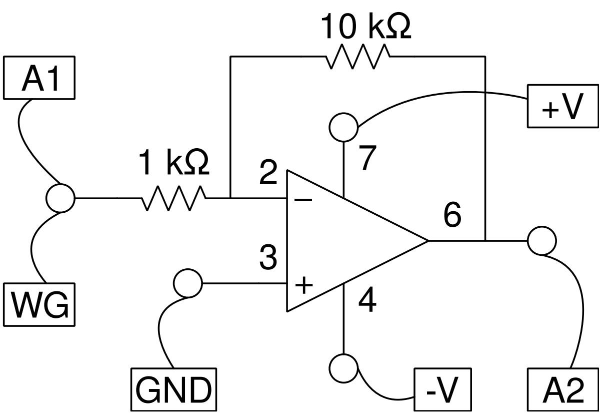
The characteristic curves for different base currents are shown in figure. The collector current is obtained from the voltage difference across the 1 $k\Omega$ resistor.

The base current is set by setting the voltage at one end of the 100 $k\Omega$ resistor, the other end is connected to the transistor base. The value of base current is calculated by, $I_b = (U_{PV2} - U_{A2})/(100 \times 10^3) \times 10^6 \ \mu A$. If A2 is not connected, the code assumes 0.6 volts at the base to calculate the base current.

7.8 Inverting Amplifier

Objective

Wire an Inverting amplifier using an Op-Amp and test it.



Procedure

- Set WG amplitude to 80 mV and frequency to 1000 Hz

- Enable A1 and A2 and their analysis option
- Select 1V range for both A1 and A2
- Make connections and observe the output
- Change gain by changing the resistor values.

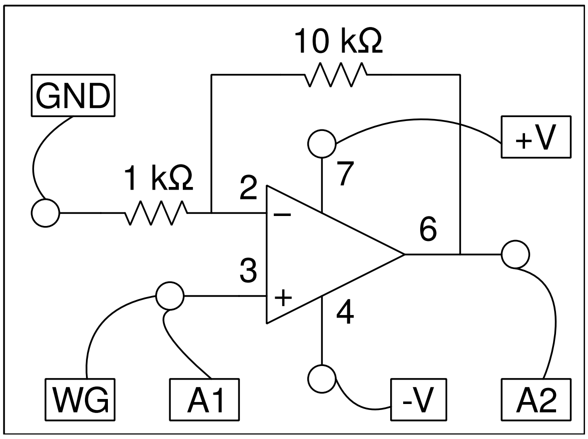
Discussion

The amplitude gain and and the phase difference can be observed from the results.

7.9 Non-Inverting Amplifier

Objective

Wire an Inverting amplifier using an Op-Amp and test it.



Procedure

- Set WG amplitude to 80 mV and frequency to 1000 Hz
- Enable A1 and A2 and their analysis option
- Select 1V range for both A1 and A2
- Make connections and observe the output
- Change gain by changing the resistor values.

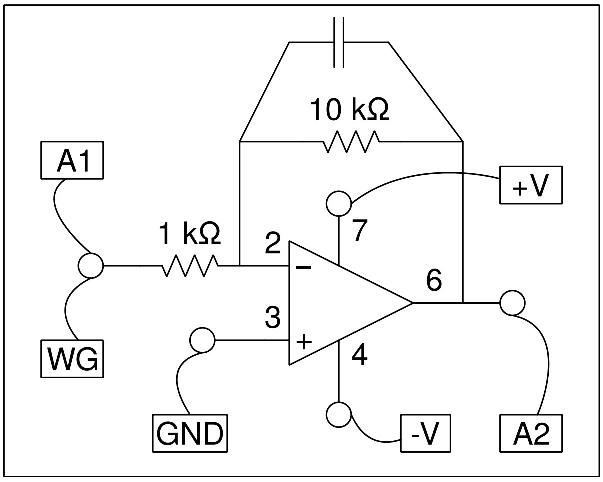
Discussion

The amplitude gain and and the phase correlation can be observed from the results.

7.10 Op-Amp Ingrator

Objective

Wire an Inverting amplifier using an Op-Amp and test it.



Procedure

- Set WG amplitude to 80 mV and frequency to 1000 Hz
- Enable A1 and A2 and their analysis option
- Select 1V range for both A1 and A2
- Make connections and observe the output
- Change gain by changing the resistor values.

Discussion

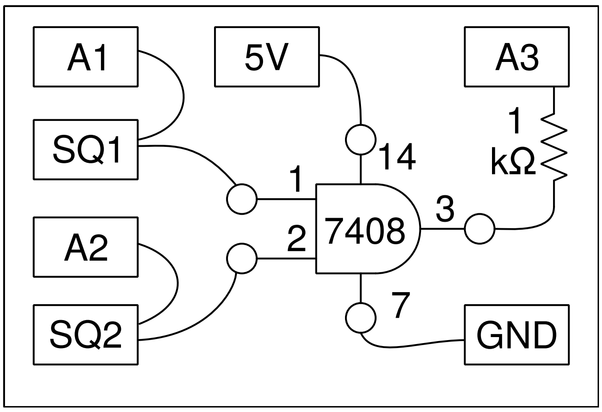
The amplitude gain and and the phase correlation can be observed from the results.

7.11 Logic gates

Objective

Study of logic gates using SQ1 and PV1 as inputs, using TTL logic ICs 7408 and 7432.

Procedure



- Enable A1, A2 ands A3. Set input range on A1 and A2 to 8V
- Set SQ1 to 200 Hz and adjust timebase to view several cycles

- Select SQ2 from the WG wave shape, set WG to 200 Hz
- Repeat using the OR gate, 7432
- The 1k resistor is required to give the 5 volt signal to A3 input

Discussion

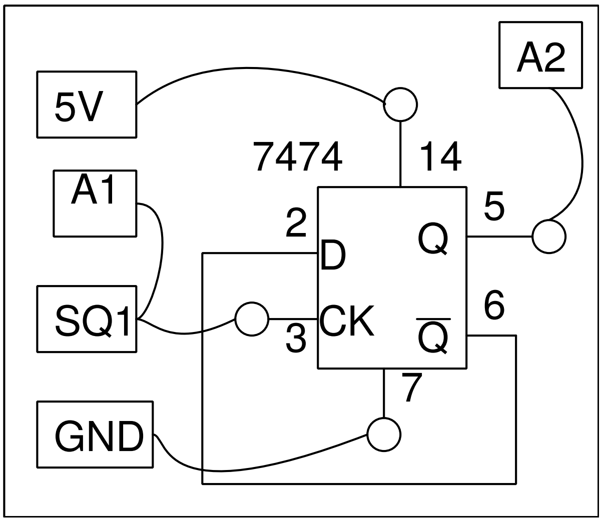
The working of the logic gate will be evident from the 3 waveforms. You may shift traces vertically to separate them for clarity.

7.12 Clock Divider

Objective

Study of a clock divider, using a D flip-flop (TTL family, 7474).

Procedure



- Enable A1 and A2, set range to 8 volts fullscale
- Set SQ1 to 500 Hz

Discussion

The output toggles at every rising edge of the input, resulting in a division of frequency by two. The output is a symmetric squarewave, irrespective of the duty cycle of the input pulse. The HIGH output of the TTL IC is around 4 volts only.

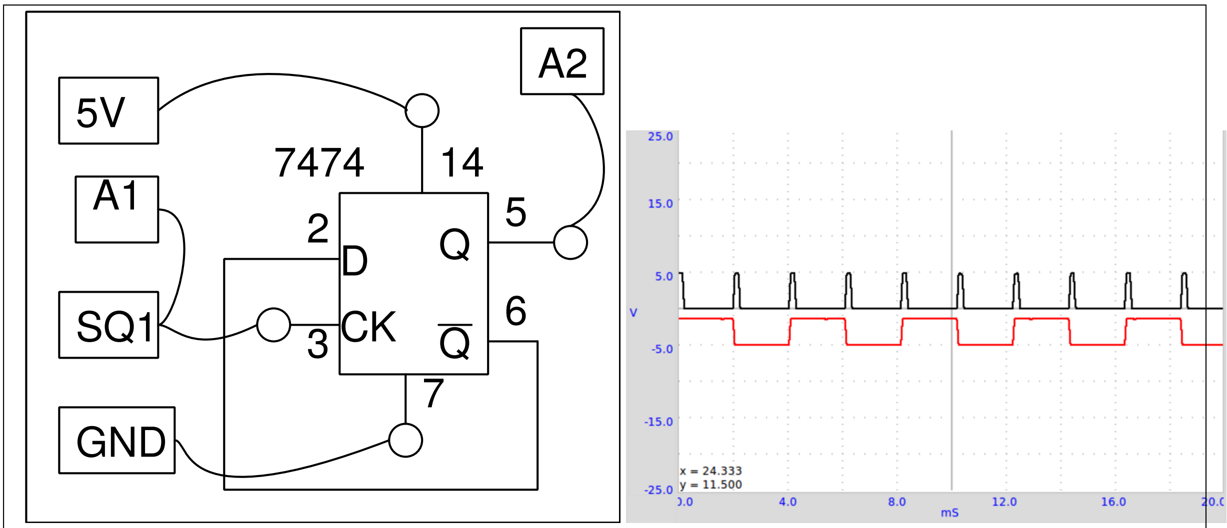


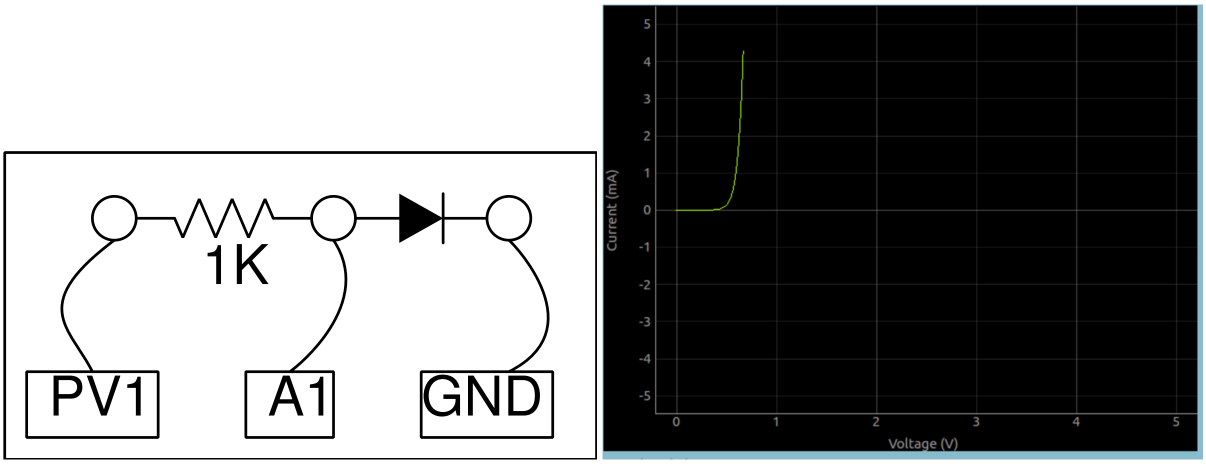
Figure 3.1 A clock divider circuit, using a D-flipflop. Outputs for two different types of input are shown

7.13 Diode I-V characteristics

Objective

Draw the I-V Characteristic of diode and compare the result with the theory.

Procedure



- Make connections
- Click on START to draw the characteristic curve.
- Analyse the data
- Plot the IV of LEDs

Discussion

The IV characteristic of an ideal PN junction diode is given by equation $I = I_0 \times e^{(qU/kT) - 1}$, where I_0 is the reverse saturation current, q the charge of electron, k the Boltzmann constant, T

the temperature in Kelvin. For a practical, non-ideal, diode, the equation is $I = I_0 \times e^{(qU/nkT)-1}$, where n is the ideality factor, that is 1 for an ideal diode. For practical diodes it varies from 1 to 2. We have used a IN4148 silicon diode. The value of *n* for 1N4148 is around 2. We have calculated the value of n by fitting the experimental data with the equation.

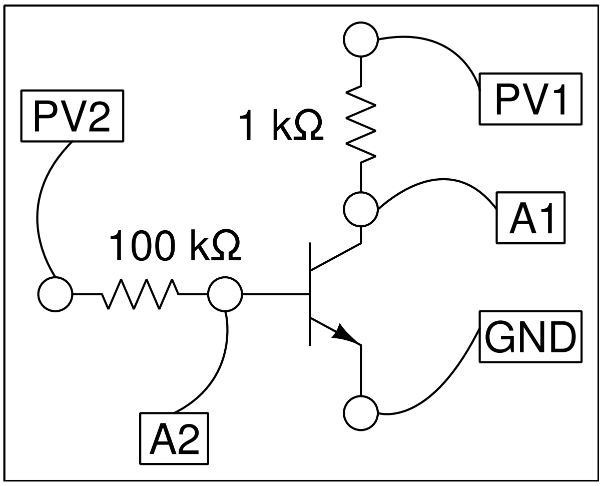
The voltage at which LED starts emitting light depends on its wavelength and Planck's constant. Energy of a photon is given by $E = h\nu = hc/\lambda$. This energy is equal to the energy of an electron that overcomes the junction barrier and is given by $E = eV_0$. So Planck's constant $h = eV_0 \times \lambda/c$, where λ is the wavelength of light from the LED, e the charge of electron and c the velocity of light.

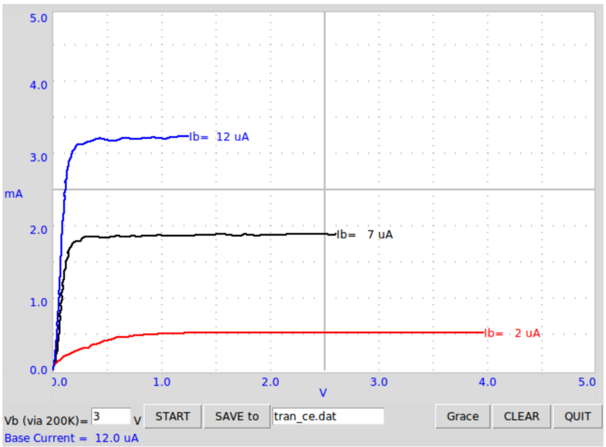
Repeat the experiment by heating the diode to different temperatures.

7.14 Transistor Output characteristics (CE)

Objective

Plot the output characteristic curve of a transistor. Collector is connected to PV1 through a 1K resistor.





Procedure

- Set base voltage to the 1 volt and START.
- Repeat for different base currents.

Discussion

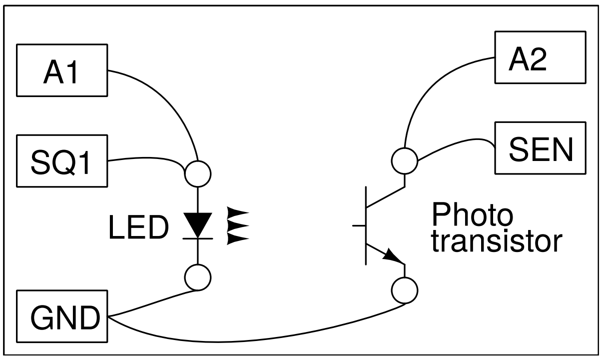
The characteristic curves for different base currents are shown in figure. The collector current is obtained from the voltage difference across the 1 $k\Omega$ resistor.

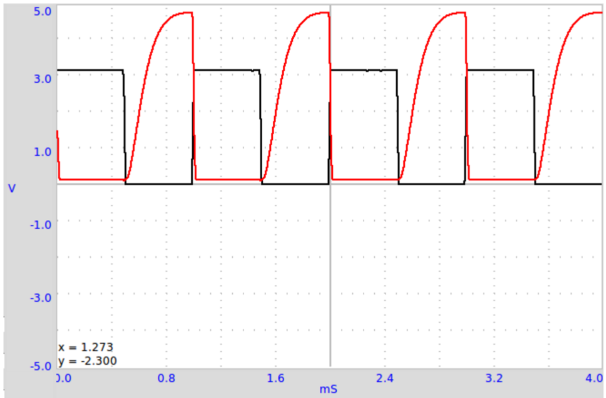
The base current is set by setting the voltage at one end of the 100 $k\Omega$ resistor, the other end is connected to the transistor base. The value of base current is calculated by, $I_b = (U_{PV2} - U_{A2})/(100 \times 10^3) \times 10^6 \ \mu A$. If A2 is not connected, the code assumes 0.6 volts at the base to calculate the base current.

7.15 Opto-electric signal transmission

Objective

Demonstrate the transmission of signals using light. An LED is powered by a 1kHz signal and the light is made to fall on a photo-transistor.





Procedure

- Keep the LED facing the photo-transistor and set SQR1 to 1000Hz
- Repeat the experiment by changing the frequency.

Discussion

The SEN input is internally connected to 5 volts through a $5,1\ k\Omega$ resistor. The output of the photo-transistor at $1\ kHz$ is shown in figure. The square trace is the voltage across the LED. When the LED is ON, photo-transistor conducts and the voltage across the collector drops to $0,2\ V$. When the LED is OFF the photo-transistor goes into cut off mode and the collector shows almost the supply voltage. The rise and fall times of the photo-transistor seem to be different. Find the upper limit of the frequency that the given photo-transistor can respond.

Repeat this experiment with a Fiber Optic cable to guide the light from LED to the photo-transistor.

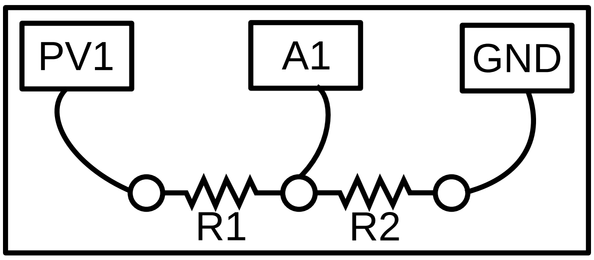
CHAPTER 8

Electricity and Magnetism

This section mainly contains experiments on the steady state and transient response of LCR circuit elements. the experimental results with the theory. It also gives an experiment of electromagnetic induction.

8.1 Plot I-V Curve

8.1.1 Schematic

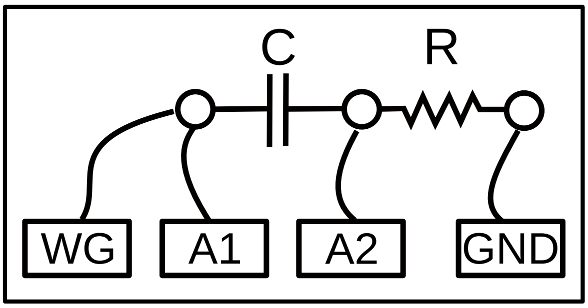


8.1.2 Instructions

- Connect the resistors as shown in the figure.
- R2 is used for measuring current, generally 1000 Ohms
- Current through the circuits is Voltage at A1 / R2
- PV1 is varied is steps. Voltage across R1 with current is plotted

8.2 XY plotting

This program plots A1 vs A2 or (A1-A2) vs A2. It is useful for plotting Lissajous figures and stdying RC circuits



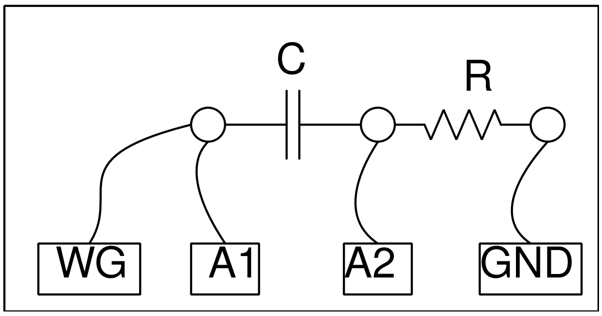
For the circuit shown above, we can plot the voltage across the capacitor against the voltage across the resistor. It will form a circle when they are equal.

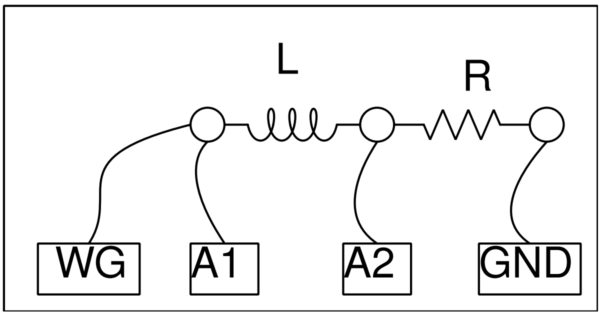
- Select C = 1uF, R = 1 kOhm and plot (A1-A2) vs A2. Adjust the frequency to get a circle

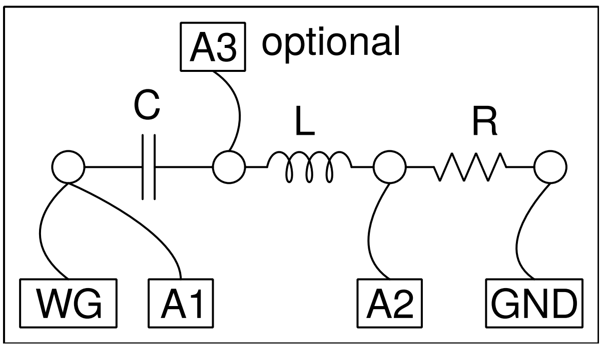
8.3 RLC circuits, steady state response

Objective

Study the effect of series LCR elements in an AC circuit. Three different combinations can be studied.







Procedure

- Make connections one by one, as per the drawing

- Note down the amplitude and phase measurements, in each case
- Repeat the measurements by changing the frequency.
- For RLC series circuit, the junction of L and C is monitored by A3
- For resonance select $C = 1\ \mu F$, $L = 10\ mH$ and $f = 1600\ Hz$, adjust f to make phase shift zero
- The total voltage across L and C together goes almost to zero, the voltage across them are out of phase at resonance

Discussion

The applied AC voltage is measured on A1 and the voltage across the resistor on A2. Subtracting the instantaneous values of A2 from A1 gives the combined voltage across the inductor and capacitor. We need to use an inductor with negligible resistance for good results. The phase difference between current and voltage is given by $\Delta\Phi = \arctan((X_C - X_L)/X_R)$.

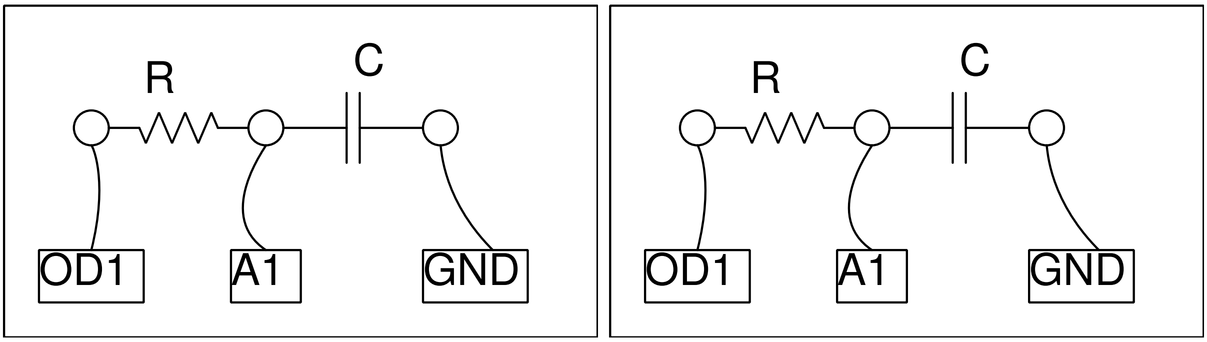
The total voltage, voltage across R and the voltage across LC are shown in figure. The phasor diagram shows the phase angle between the current and the voltage. The inductance used in this experiment is around $10\ mH$, having a resistance of 20 Ω.

At $1600\ Hz$, $X_C \simeq X_L$ and the voltage across LC is decided by the resistance of the inductor. At the resonant frequency, the voltage drop across LC will be minimum, decided by the resistance of the inductor. The input A3 is connected between L and C, so that the individual voltage drop across L and C can be displayed.

8.4 Transient Response of RC circuits

Objective

Plot the voltage across a capacitor, when it is charged by applying a voltage step through a resistor. Calculate the value of the capacitance from the graph.



Procedure

- From **\emphElectrical** , select **\emphRCTransientresponse**
- Click on *0->5V STEP* and *5->0V step* Buttons to plot the graphs
- Adjust the horizontal scale, if required, and repeat.
- Calculate RC time constant.

Discussion

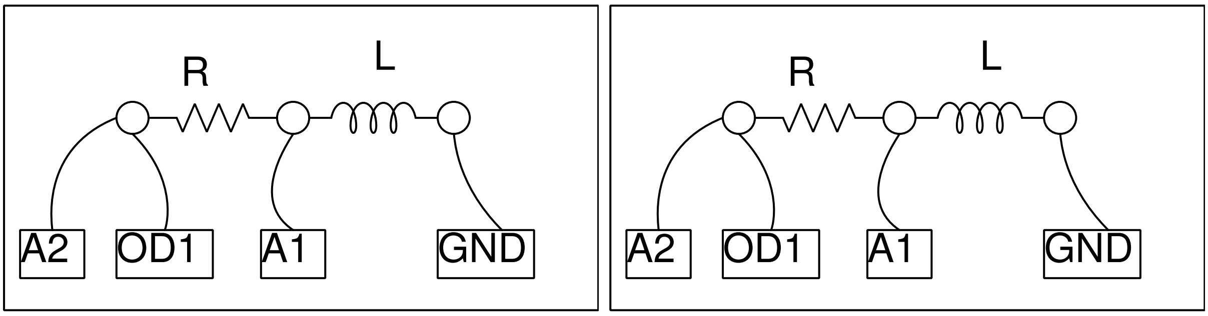
Applying a 0 to 5V step makes the voltage across the capacitor to rise exponentially as shown in the figure. By fitting the discharge curve with $U(t) = U_0 \times e^{-t/RC}$, we can extract the RC time constant and find the values of capacitance from it.

The voltage across a capacitor is exponential only when it is charged trough a linear element, a resistor for example. When charged from a constant current source, the voltage shows linear increase, because $Q = It = CV$, and voltage increases linearly with time as $V = (I/C) \times t$.

8.5 Transient Response of RL circuits

Objective

Explore the nature of current and voltage when a voltage step is applied to resistor and inductor in series. By measuring the voltage across the inductor as a function of time, we can calculate its inductance.



In an RL circuit $V = RI + L(dI/dt)$ and solving this will give $I = I_0 \times e^{-(R/L)t}$. The coefficient of the exponential term R/L can be extracted from the graph of voltage across the inductor. The resistance of the inductor coil should be included in the calculations, $R = R_{ext} + R*_L$.

Procedure

- Inductor is the 3000 Turn coil
- Click on *0->5V STEP* and *5->0V step* Buttons to plot the graphs
- Adjust the horizontal scale, if required, and repeat.
- Calculate the value of inductance
- Insert an iron core into the inductor and repeat

Discussion

The transient response of the RL circuit is shown in figure. The exponential curve is fitted to extract the L/R value. The resistance of the coil is measured by comparing it with the known external resistance under DC conditions. A2 is connected to OD1 for a more accurate measurement of the coil resistance.

The applied voltages are above zero, but the graph went to negative voltages. Why ?

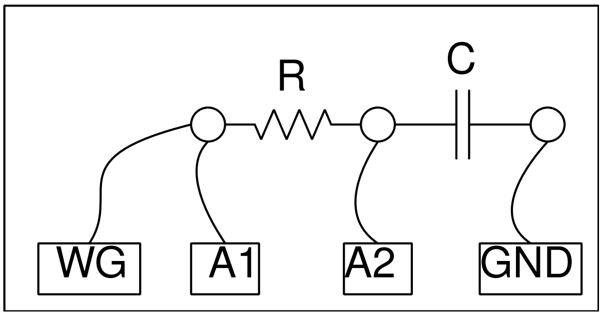
What was the current before doing the 5->0 step ? What is back EMF ?

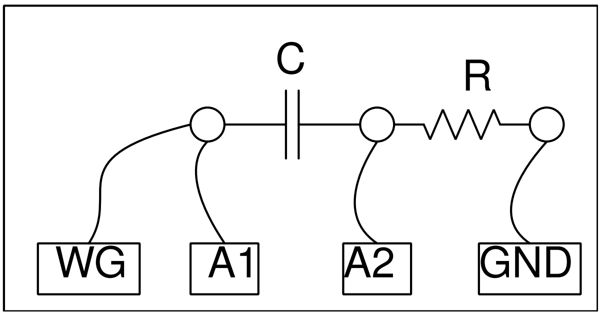
Repeat with two coils in series, by (a) placing them far away (b) placing one over the other and (c) after changing the orientation. The effect of mutual inductance can be seen.

8.6 RC Integration & Differentiation

Objective

RC circuits can integrate or differentiate a voltage waveform with respect to time. A square wave is integrated to get a triangular wave and differentiated to get spikes at the transitions.begin_inset Separator latexparend_inset



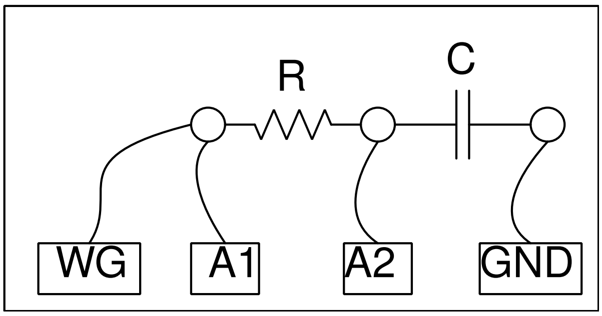


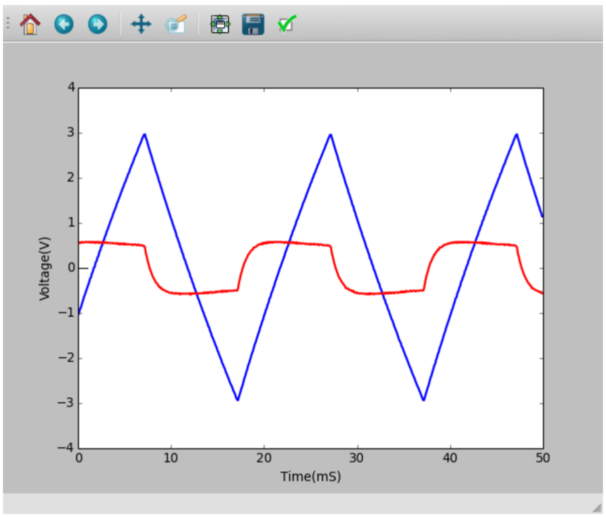
Procedure

- Select WG triangular wave option
- Set WG to 500Hz ($T = 2\ ms$), $R = 1\ k\Omega$ and $C = 1\ \mu F$
- Adjust the horizontal scale to view more than 4 cycles.
- Repeat the same for RC differentiator, at $50\ Hz$.

Discussion

Integration of a triangular waveform gives parabolic shape and differentiation gives a square shape. The differentiation can only be shown at lower frequency. Try these for other wave shapes, for example a squarewave. Integrating a square wave should give a triangular wave.





8.7 Fourier Analysis

Objective

Learn about Fourier Transform of a signal. Time and Frequency domain representations.

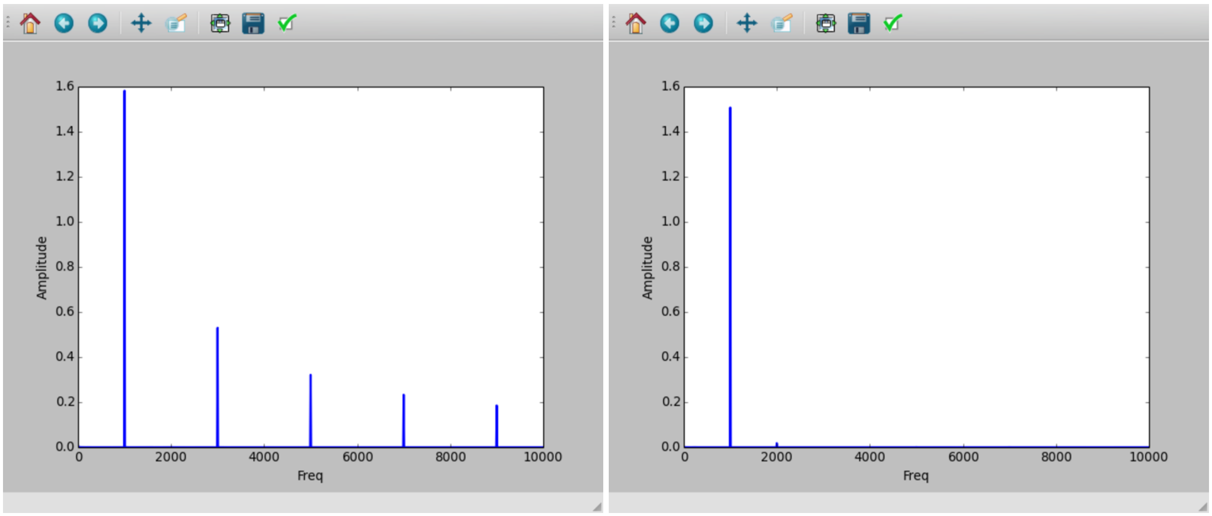
Procedure

- Connect SQ1 to A1 and WG to A2. Put A1 in AC coupled mode (slide switch on the box)
- Enable A1 and A2, select 4 volt scale
- Set both WG and SQ1 to 500Hz
- Press the FFT button

Discussion

In the Fourier transform plot, frequency is on the x-axis and the y-axis shows the relative strength of the frequency components of the signal. This is called the frequency domain representation (http://en.wikipedia.org/wiki/Fourier_transform). For the sine wave there is only one dominant peak, the smaller ones are a measure of distortion of the sine wave.

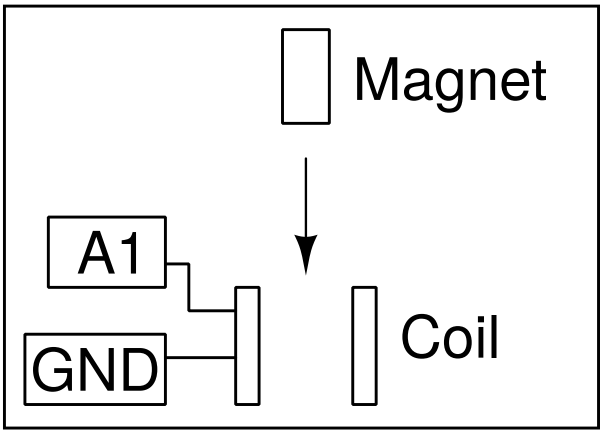
A square wave function can be represented as $f(\theta) = \sin(\theta) + \sin(3\theta)/3 + \sin(5\theta)/5 + \ldots$. In the Fourier transform of a square wave of frequency f , there will be a 3f component (having an amplitude of one third of f), 5f component (amplitude one fifth) etc. as shown in the figure.

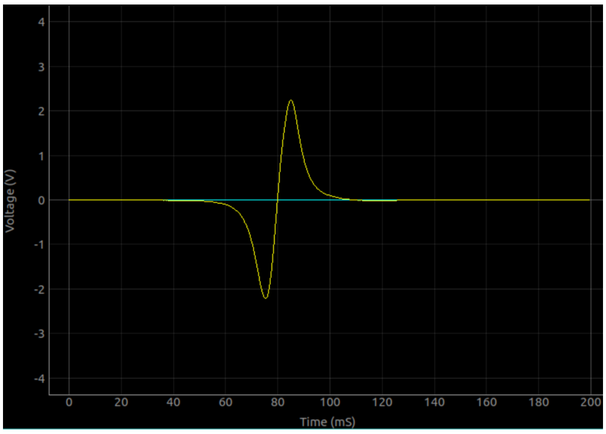


8.8 Electromagnetic induction

Objective

Explore the voltage induced across a coil by a changing magnetic field, by dropping a small cylindrical magnet into a coil. Use a tube to guide the magnet through the coil.





Procedure

- Click on Start Scanning. A horizontal trace should appear
- Drop the magnet through the coil, until a trace is caught.
- Repeat the process by changing the parameters like magnet strength, speed etc.

Discussion

The result is shown in figure. The amplitude increases with the speed of the magnet. From the graph, we can find the time taken by the magnet to travel through the coil.

The second peak is bigger than the first peak. Why ? Where will be the magnet at the zero crossing of the induced voltage? Drop the magnet from different heights and plot the voltage vs square root of the height.

CHAPTER 9

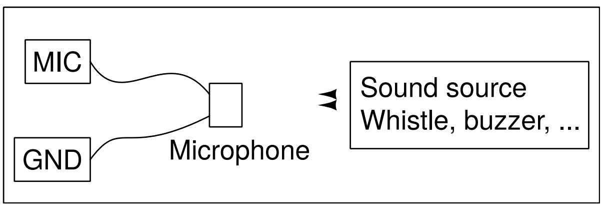
Sound

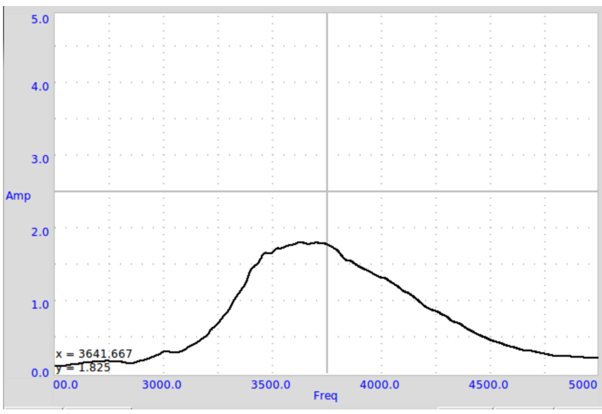
Pressure variations, about an equilibrium pressure, transmitted through a medium is called sound. They are longitudinal waves. Moving a sheet of paper back and forth in air can generate these kind of pressure waves, like the paper cone of a loudspeaker. When the frequency is within 20 to 20000Hz range, we can hear the sound. In this chapter, we will generate sound from electrical signals, detect them using the built-in microphone (a pressure sensor) and study the properties like amplitude and frequency. Velocity of sound is measured by observing the phase shift of digitized sound with distance.

9.1 Frequency response of Piezo

Objective

Plot the frequency response curve of the Piezo disk by scanning through the frequency and measuring the amplitude of the microphone output.





Procedure

- Make the connections and keep the Mic and the Buzzer facing each other
- Press START button

Discussion

The Frequency Vs Amplitude plot is shown in figure. The amplitude is maximum around 3500 Hz.

9.2 Velocity of sound

Objective

Calculate the velocity of sound by measuring the pressure variation with distance. Sound travels as a series of compressions and rarefactions. Figure (a) shows the High and Low pressure regions along the direction of travel, along with output of a pressure sensor at corresponding positions.

We can display the pressure variation at any point with respect to the variation at the starting point. The phase of the microphone output changes as you change its distance from the Piezo. Moving by one wavelength changes the phase by 360 degrees. If the phase changes by X degrees for ΔD cm change in distance, the wavelength is given by $\lambda = (360 \times \Delta D)/X$. The velocity of sound can be calculated by multiplying the frequency with this.

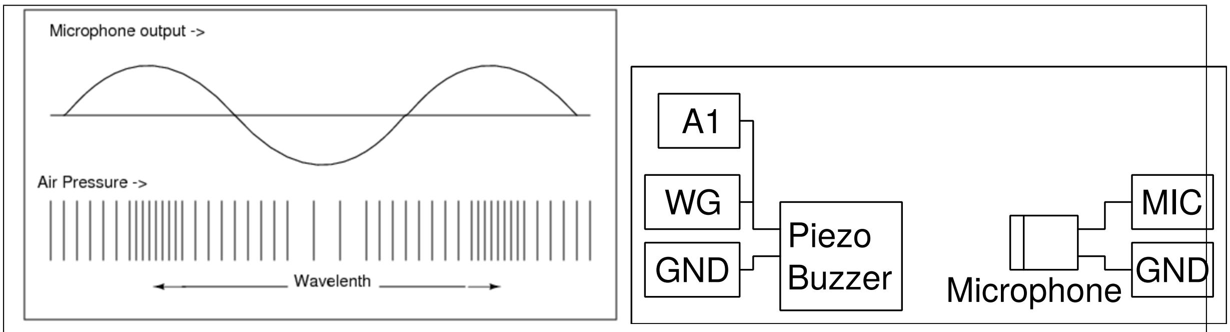


Figure 5.1 (a) compressions et expansions along the direction of sound (b) schematics

Procedure

- Set frequency to resonant maximum by measuring the frequency response *5.1*
- Keep the Piezo facing the microphone, on the same axis
- Enable measurement
- Adjust the distance to make both the traces in Phase
- Change the distance to make them 180 degree out of phase, that distance is half wave length.

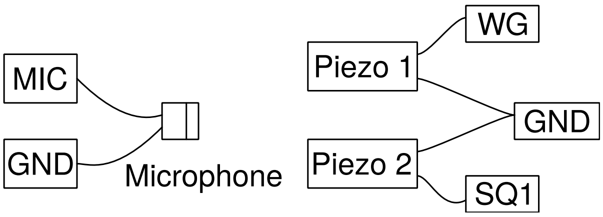
Discussion

At 3500 Hz, for a 2 cm change in distance the phase changed from 176 to 102. Using the equation, $v = f \times (360 \times \Delta D)/X$, $v = 3500 \times (360 \times 2)/(176 - 102) = 34054\ cm \cdot s^{-1}$. It is important to keep the mic and the Piezo disc on the same axis, for accurate results.

9.3 Sound beats

Objective

Study the interference of sound from two individual sources. Two Piezo buzzers are powered by two different sources, and the sound is directed towards the microphone.



Procedure

- Set WG to 3500 Hz and SQ1 to 3600 Hz
- Enable WG and SQ1 separately to check the MIC output
- Adjust positions of Piezo buzzers, from the mic, to get almost same amplitude with both
- Select both of them to get the beat pattern
- Press FFT to view the frequency spectrum

Discussion

From figure it can be seen how the low frequency envelope is created. Distance between two minimum pressure points., of the envelope, corresponds to the beat wavelength. The Fourier transform of the output is shown in figure .

CHAPTER 10

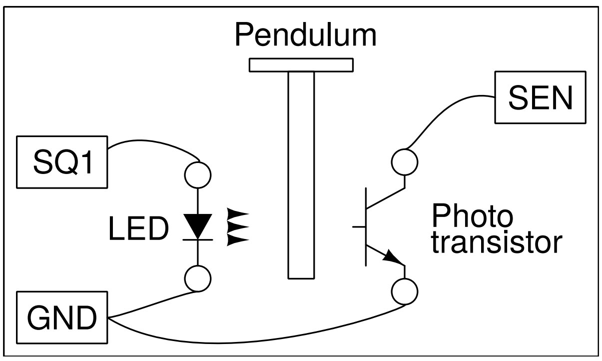
Mechanics

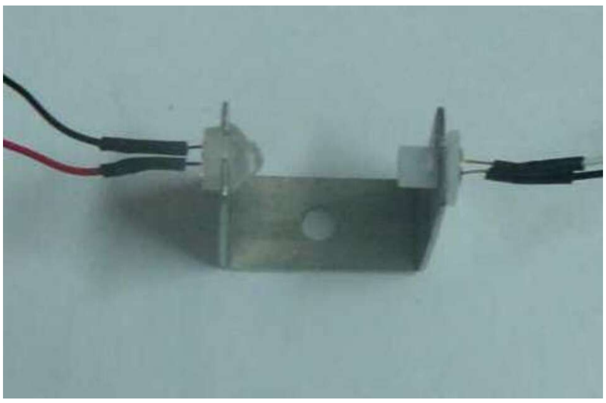
Resonance phenomena is studied using a driven pendulum. Value of acceleration due to gravity is measured using a pendulum.

10.1 Acceleration due to gravity using Rod pendulum

Objective

Measure the period of oscillations of a rod pendulum using a light barrier and calculate the value of acceleration due to gravity. Period of oscillation of a uniform rod about one end is given by $T = 2\pi\sqrt{2l/3g}$, where l is the length and g is the acceleration due to gravity. The pendulum (T-shaped, a knife edge attached to a 6mm dia rod) is made to swing between an LED and photo-transistor, connected to expEYES. The LED and photo-transistor are mounted on a U-shaped bracket as shown in figure.





Procedure

- Oscillate the pendulum and click on START

- Repeat with different pendulum lengths.

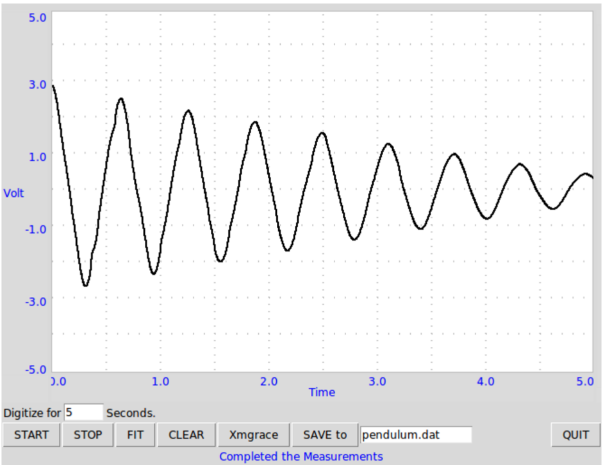
Discussion

The time period is measured 50 times, using a 14.6cm rod pendulum, and the average value is 0.627 seconds. The calculated value of 'g' is $977,4\ cm \cdot s^{-2}$, slightly different from the actual value due to the following reasons. The length is measured from the knife edge to the bottom and used in the formula. But there is a small mass projecting above the knife edge that is not included in the calculation. Another reason is that the pendulum may not be exactly vertical in the resting position.

10.2 Angular Velocity of Pendulum

Objective

To study the nature of oscillations of a pendulum. An angle encoder is required for measuring the angular displacement as a function of time. But using a DC motor as a sensor, we can measure the angular velocity as a function of time.begin_inset Separator latexparend_inset



Procedure

- Attach some sort of rigid pendulum to the axis of the motor.
- Connect the motor between A3 and GND
- Connect $100\ \Omega$ resistor from Rg to Ground
- Oscillate the pendulum and START digitizing

Discussion

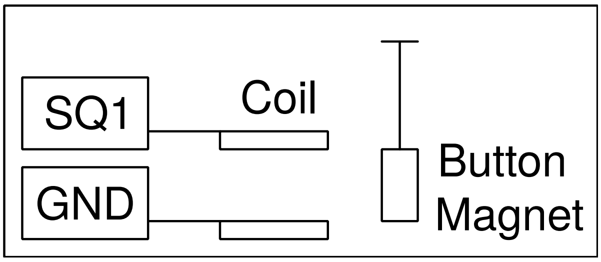
The observed waveform is shown in figure. Fitting it with equation $A = A_0 \sin(\omega t + \theta) \exp(-Dt) + C$, using Grace gave an angular frequency of $10\ Hz$.

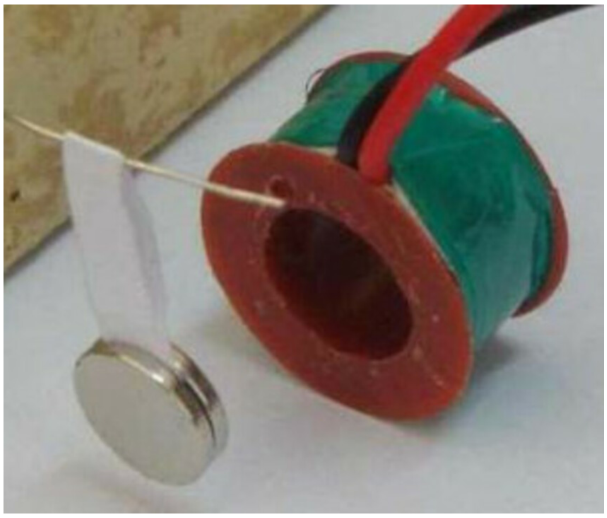
The pendulum should be made with a heavy bob and a light weight rod connecting it to the axis of the motor. In this case, the DC motor acts like a generator and the voltage is proportional to the instantaneous angular velocity.

10.3 Resonance of a driven pendulum

Objective

Demonstrate the resonance of a driven pendulum.





Procedure

Make a pendulum using two button magnets and a piece of paper. Suspend it and place the 3000T coil near that, as shown in figure.

- Connect the coil between SQ1 and ground
- Calculate the resonant frequency from the length of the pendulum
- Scan the frequency around the expected resonance frequency

Discussion

When SQ1 reaches the resonant frequency of the pendulum, the amplitude goes up due to resonance. A 4 cm (from the center of the magnet to the axis of oscillation) long pendulum resonated at around $2,5\ Hz$, almost tallying with its calculated natural frequency. The resonant frequency of the pendulum is given by $f = 1/(2\pi\sqrt{g/l})$, where l is the distance from the center of the magnet to the point of suspension and g is the acceleration due to gravity.

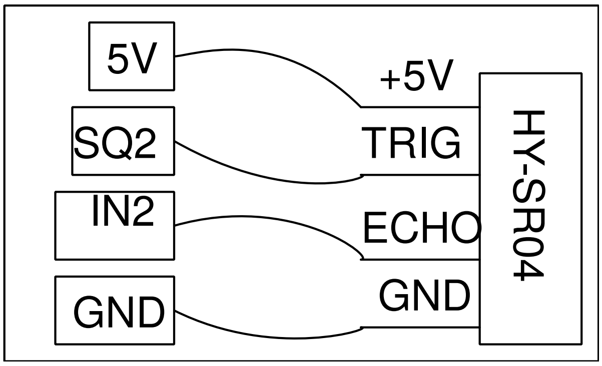
Repeat the experiment by changing the length of the pendulum.

10.4 Distance Measurement, by ultrasound echo

Objective

Measure distance by measuring the time taken by $40\ kHz$ pulse train to echo from a hard surface.

Procedure



- Keep a flat surface, like a cardboard sheet, around 10 cm from the echo module
- Press START
- Change the distance with time

Discussion

The distance is calculated from the time taken by a burst of sound to echo from the surface kept in front of the module. The distance can be measred as function of time, enabling to calculate velocity, acceleration etc.

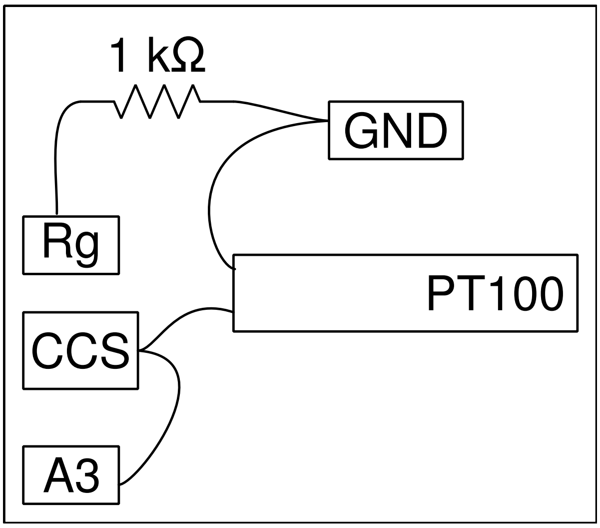
CHAPTER 11

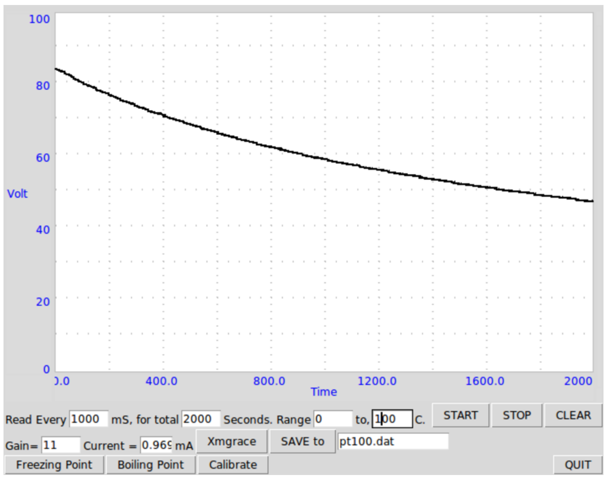
Other experiments

11.1 Temperature measurement using PT100

Objective

Record the temperature of a liquid by using a Platinum Resistance Thermometer. Resistance of a PT100 element is related to the temperature by the equation $R(T) = R_0(1 + AT + BT^2)$, where $A = 3,9083 \times 10^{-3}$ and $B = -5,775 \times 10^{-7}$.





Procedure

- Enter the Gain, Offset error and the Current from CCS
- Select the temperature range and time intervals
- Select the required parameters and press START

Discussion

Cooling curve of water is shown in figure

To measure the resistance of the PT100 element, we connect it from the CCS to ground and measure the voltage across it. The actual current of CCS should be measured using an ammeter or by measuring the voltage frop across an known resistor. The input to A3 is amplified 11 times by connecting $1\ k\Omega$ resistor from Rg to Ground.

The resistance of PT100 is $1000\ \Omega$ at $0°C$. It changes nearly $0,4\ \Omega/°C$, changing the voltage by $0,4\ mV$. The 12 bit ADC output changes by 1 LSB for $1,22\ mV$ change in input voltage, hence any temperature change less than 3 degrees will not be detected. Use an external non-inverting amplifier to increase the resolution. The gain of the amplifier should be such that the maximum temperature measured should give an output less than 3.3 volts. Change the gain field entry accordingly.

11.2 Data Logger

- Select Channels A1, A2, A3 or SEN
- Record them for the desired time interval

11.3 Adanced Data Logger

11.3.1 Introduction

The simple data logger you are already familiar with records any specified input voltage with respect to time, however, it is often desirable to vary one output parameter, and study the effect on some other aspect of the experiment.

In the advanced data logger, both X and Y can be chosen from the following list

- Inputs
  - Time
  - Voltmeter: A1,A2,A3,IN1,SEN,AN8,CCS
  - Capacitance
  - Resistance
  - Oscilloscope
    - Extracted frequency,phase,amplitude or offset using a sine fit
    - Difference in phase between A1(Any analog input) and A2. Also, ratio of amplitudes.
  - Frequency on IN2
  - Any connected I2C sensor (Magnetometer, accelerometer, temperature, gyro etc)

      * Select 1 parameter from any of the detected sensors added automatically to the list

  - SR04 distance sensor

- Outputs (Start and End must be specified)

  - WG Sine wave generator frequency

  - SQ1, SQ2 square wave generator

  - PV1, PV2 voltage outputs

Online Examples

CHAPTER 12

I2C Modules

12.1 B-H Curve

12.1.1 Schematic

- To Record Magnetic Hysterisis using a solenoid connected to PV1, and a MPU925x magnetometer connected to the I2C port.

12.1.2 Instructions

- Connect the coil between PV1 and ground
- Connect the Magnetometer(MPU925x) to the I2C port
- Place the solenoid on top of the sensor such that the magnetic axis is perpendicular to the sensor
- Acquire data : This changes the PV1 voltage from -3 to 3 volts in 100 steps, and returns from 3V to -3V . This results in a proportional magnetic field generated by the solenoid which is also measured by the sensor and plotted
- Add a ferromagnetic material such as an alligator clip, and repeat the acquisition. Observe the hysteris for various different materials.

12.2 Light Sensor Logger

12.2.1 Schematic

- To Record luminosity values from the TSL2561 sensor.

12.2.2 Instructions

- Connect the light sensor(TSL2561) to the I2C port
- Set the acquisition time, interval, and select the values to plot.
- Acquire data as a function of time

12.3 MPU6050

I2C module with Accelerometer, Gyroscope and Temperature

- Connect the module to the I2C port. Pins used are VCC, GND, SCL and SDA only
- Select the desired parameters to be measured.
- Select the Total Duration and the time between two consecutive readings.
- Press the start button. You can stop the measurement before it completes also
- Data is saved as text, in a two column (time, value) format, each set separated by an empty line.

12.4 I2C Logger

CHAPTER 13

Coding expEYES-17 in Python

The GUI programs described in the previous sections are meant for a fixed set of experiments. To develop new experiments, one should know how to access the features of expEYES from software. Important function calls used for communicating with the device is given below.

Note: The GUI programs described in the previous sections are meant for a fixed set of experiments. To develop new experiments, one should know how to access the features of expEYES from software. Important function calls used for communicating with the device is given below.

13.1 Establish Connection

To access the EYES17 hardware, the python modules for eyes17 must be installed. They should be inside a directory named eyes17, that could be in your home directory or on the Python PATH. **Every program should start with the following 2 lines**

```
import eyes17.eyes
p = eyes17.eyes.open()
```

The variable `p` is the software object, representing the hardware.

The following sections explains the Python function calls to access the eyes17 hardware. Each function call will be explained with an example usage.

13.2 set_pv1(v), set_pv2(v)

Sets the DC voltages at PV1 and PV2. PV1 range is -5 to 5. PV2 range is -3.3 to 3.3.

```
print p.set_pv1(4)
print p.set_pv2(2)
```

The value set is printed. Measure the voltages using a meter.

13.3 get_voltage(input)

Returns the voltage at the specified input.

```
print p.get_voltage('A1')
print p.get_voltage('A2')
print p.get_voltage('A3')
print p.get_voltage('MIC')
print p.get_voltage('SEN')
```

Connect PV1 to A1 and use the set_pv1() and get_voltage together. This function sets the input range by trial

and error, depending on the input signal.

13.4 get_voltage_time(input)

Returns a tuple, containing the computer's time stamp and the voltage at the specified input

```
print p.get_voltage_time('A1')
```

13.5 get_resistance()

Returns the value of resistance connected to SEN, it should be between 100 Ohm and 100k for reasonable accuracy.

```
print p.get_resistance()
```

13.6 get_capacitance()

Returns the value of capacitance connected to IN1 (works well in pF ranges)

```
print p.get_capacitance()
```

13.7 get_version()

Returns the version number of the Firmware

```
print p.get_version()
```

13.8 get_temperature()

Returns the temperature of the processor inside eyes17

```
print p.get_temperature()
```

13.9 set_state(OUPUT=value)

Sets the output of OD1, SQ1 etc. Connect OD1 to A1 and run

```
p.set_state(OD1=1)
print p.get_voltage('A1')
```

13.10 set_sine(frequency)

Generates the sinewave of requested frequency on WG (range from 5Hz to 5000Hz). All intermediate values are not possible, function returns the actual value set.

```
print p.set_sine(502)
```

502.00803

13.11 set_sine_amp(amplitude)

The amplitude can be set to 3 pre-defined values of the peak voltage (0-> 80mV, 1-> 1V, 2-> 3V)

```
p.set_sine_amp(2)
```

Sets the amplitude to 3 volts peak.

13.12 set_sqr1(frequency)

Sets the frequency of SQ1 output (range from 4Hz to 1 MHz).All intermediate values are not possible, function returns the actual value set.

```
print p.set_sqr1(15030)
```

15030.53

13.13 set_sqr1_slow(frequency)

Sets the frequency of SQ1 output (range from 0.1Hz to 1 MHz).All intermediate values are not possible, function returns the actual value set. Resolution is high but WG is disabled when SQ1 is operated in this mode.

```
print p.set_sqr1_slow(0.5)
```

13.14 set_sqr2(frequency)

Similar to set_sqr1() but SQ2 is not available along with WG, only one at a time.

13.15 set_sqr1(frequency, dutyCyle)

Sets the frequency of SQ1 output (range from 0.1Hz to 1 MHz).All intermediate values are not possible, function returns the actual value set.

```
print p.set_sqr1(1000, 30)          # 1000Hz with 30% duty cycle
```

13.16 get_freq(input)

Measures the frequency of a square wave on the input, IN2 or SEN. Connect SQ1 to IN2 and run the code

```
p.set_sqr1(1000)
print p.get_freq('IN2')
```

13.17 duty_cycle(input)

Measures the duty cycle a square wave on the input, IN2 or SEN. Connect SQ1 to IN2 and run the code

```
p.set_sqr1(1000, 30)
print p.duty_cycle('IN2')
```

13.18 r2ftime(input1, input2)

Measures the time interval between a rising edge on input1 to another one on input2, the inputs can be the same also. This can be tested using a square wave.

Connect SQ1 to IN2 and run

```
p.set_sqr1(1000, 30)
print p.r2ftime('IN2', 'IN2')
```

0.0003

The 1kHz square wave with 30% duty cycle has a Period of one millisecond and stays HIGH for .3 milliseconds.

13.19 multi_r2rtime(input, numCycles)

Measures the time interval between rising edges on input1. Time between 2 edges is one cycle. Number of cycles to be measured also can be specified, default value is 1. The allowed values are 1,2,4,8,12,16,32 and 48. This can be tested using a square wave.

Connect SQ1 to IN2 and run

```
p.set_sqr1(1000)
print p.multi_r2rtime('IN2', 8)
```

0.008

13.20 select_range(channel, range)

The input range of A1 and A2 can be set from ±0.5V to ±16V fullscale, using the programmable gain amplifiers.

```
p.select_range('A1', 4)          # 4volt maximum
p.select_range('A1', 8)          # 8 volt maximum
```

13.21 select_range(channel, range)

The input range of A1 and A2 can be set from ±0.5V to ±16V fullscale, using the programmable gain amplifiers.

```
p.select_range('A1', 4)          # 4volt maximum
p.select_range('A1', 8)          # 8 volt maximum
```

13.22 capture1(Input, Number of samples, time interval)

Digitizes the specified input. The number of samples could be upto 10000. The time gap between two consecutive samples id given in microseconds (range 2 to 1000 usec).

```
print p.capture1('A1', 5, 5)
```

will print two arrays of time and voltage.

We need to plot the graph of the output for a better understanding. This can be done using the matplotlib module, imported using the pylab interface. Connect WG to A1 with a wire and run;

```
from pylab import *
p.set_sine_amp(2)
p.set_sine(1000)
p.select_range('A1', 4)
t,v = p.capture1('A1', 300, 10)
plot(t,v)
show()
```

The output of this code is given below.

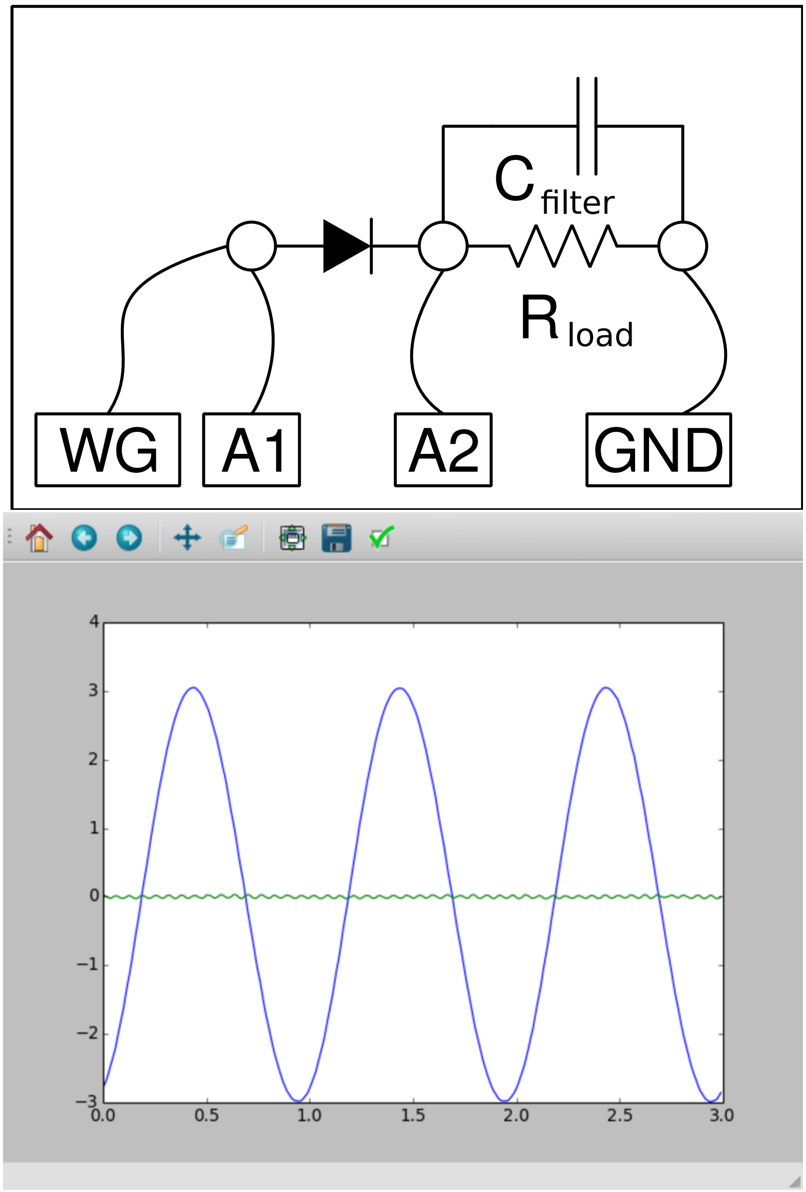
13.23 capture2(Number of samples, time interval)

Digitizes the inputs A1 and A2 together. The number of samples could be upto 10000. The time gap between two consecutive samples id given in microseconds (range 2 to 1000 usec).

Connect WG to A1 and a diode from A1 to A2. Run the code below

```
from pylab import *
p.set_sine_amp(2)
p.set_sine(1000)
p.select_range('A1', 4)
t,v,tt,vv = p.capture2(300, 10)
plot(t,v)
plot(tt,vv)
show()
```

The output of this code is given below.



13.24 capture4(Number of samples, time interval)

Digitizes the inputs A1,A2,A3 and MIC together. The number of samples could be upto 10000. The time gap between two consecutive samples id given in microseconds (range 2 to 1000 usec).

Connect WG to A3 and run the code given below. Result is shown above.

```
from pylab import *
p.set_sine_amp(2)
p.set_sine(1000)
p.select_range('A1', 4)
res = p.capture4(300, 10)
```

(continues on next page)

(continued from previous page)

```
plot(res[4],res[5])          # A3
plot(res[6],res[7])          # MIC
show()
```

13.25 set_wave(frequency, wavetype)

If wavetype is not specified, it generates the waveform using the existing wave table. If wavetype is specified ('sine' or 'tria')

corresponding wavetable is loaded.

```
from pylab import *
p.set_wave(1000, 'sine')
p.set_wave(100)        # Sets 100Hz using the existing table
x,y = p.capture1('A1', 500,50)
plot(x,y)
p.set_wave(100, 'tria')  # Sets triagular wave table and generates␣
↪100Hz
x,y = p.capture1('A1', 500,50)
plot(x,y)
show()
```

13.26 load_equation(function, span)

Makes the wave table using the quation. Connect WG to A1 and run the code below. The output also is shown below.

```
from pylab import *

def f1(x):
    return sin(x) + sin(3*x)/3

p.load_equation(f1, [-pi,pi])
p.set_wave(400)
x,y = p.capture1('A1', 500,10)
plot(x,y)
show()
```

13.27 load_table(function, span)

The wave table can be loaded with a 512 element array. Connect WG to A1 and run the code below. After taking the absolute value, the table starts with 256, goes down to zero and then goes upto 255, tracing a triagular wave. The tableoutput also is shown above.

```
from pylab import *
x = arange(-256, 256)
x = abs(x)
p.load_table(x)
p.set_wave(400)
x,y = p.capture1('A1', 500,10)
plot(x,y)
show()
```