

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.

EXPEYES-17 user's manual is distributed under GNU FREE DOCUMENTATION LICENSE.

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

V V V Satyanarayana

Contents

1	Introduction	1
2	The equipment	3
2.1	External connections	4
2.2	1.2.2 Accessory Set	7
3	Software Installation	9
3.1	On Debian-10 , Ubuntu-18.04 and above	9
3.2	Installing on MS Windows	10
4	The main GUI program	11
5	Getting Familiar with ExpEYES17	13
6	School Level Experiments	15
6.1	Measuring Voltage	15
6.2	Measuring Resistance	16
6.3	Measuring Resistance series combination	16
6.4	Measuring Resistance parallel combination	17
6.5	Measuring Capacitance	18
6.6	Measuring Capacitance in series combination	18
6.7	Measuring Capacitance of a parallel combination	19
6.8	Measure resistance by comparison	20
6.9	Direct and Alternating Currents	21
6.10	AC mains pickup	23
6.11	Separating DC & AC components	24
6.12	Human body as a conductor	25
6.13	Resistance of human body	27
6.14	Light dependent resistors	28
6.15	Voltage of a lemon cell	28
6.16	A simple AC generator	29
6.17	AC Transformer	30
6.18	Resistance of water, using AC	31

6.19	Generating sound	33
6.20	Digitizing sound	33
6.21	Stroboscope	34
7	Electronics	37
7.1	Oscilloscope and other Equipment	37
7.2	Half wave rectifier using PN junction	41
7.3	Fullwave rectifier using PN junctions	44
7.4	Clipping using PN junction diode	45
7.5	Clamping using PN junction diode	46
7.6	IC555 Oscillator	47
7.7	Transistor Amplifier CE	49
7.8	Inverting Amplifier	50
7.9	Non-inverting Amplifier	52
7.10	Summing Amplifier	53
7.11	Logic gates	54
7.12	Clock Divider	56
7.13	Diode I-V characteristics	58
7.14	NPN Transistor Output characteristics (CE)	60
7.15	PNP Transistor Output characteristics (CE)	61
8	Electricity and Magnetism	63
8.1	Plot I-V Curve	63
8.2	XY plotting	64
8.3	RLC circuits, steady state response	65
8.4	LCR and Series Resonance	67
8.5	Transient Response of RC circuits	69
8.6	Transient Response of RL circuits	70
8.7	Transient response of LCR circuits	71
8.8	Frequency response of filter circuit	73
8.9	Electromagnetic induction	73
9	Sound	75
9.1	Frequency response of Piezo	75
9.2	Velocity of sound	76
9.3	Sound beats	78
10	Mechanics	81
10.1	Acceleration due to gravity using Rod pendulum	81
10.2	Digitizing Pendulum Oscillations	82
10.3	Resonance of a driven pendulum	84
10.4	Distance Measurement, Echo module	85
11	Other experiments	87
11.1	Temperature measurement using PT100	87
11.2	Data Logger	88
11.3	Adanced Data Logger	88
12	I2C Modules	91

12.1	B-H Curve	91
12.2	Light Sensor Logger	92
12.3	MPU6050	93
12.4	I2C Logger	93
13	Coding expEYES-17 in Python	95
13.1	Programming ExpEYES using Python	95

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.

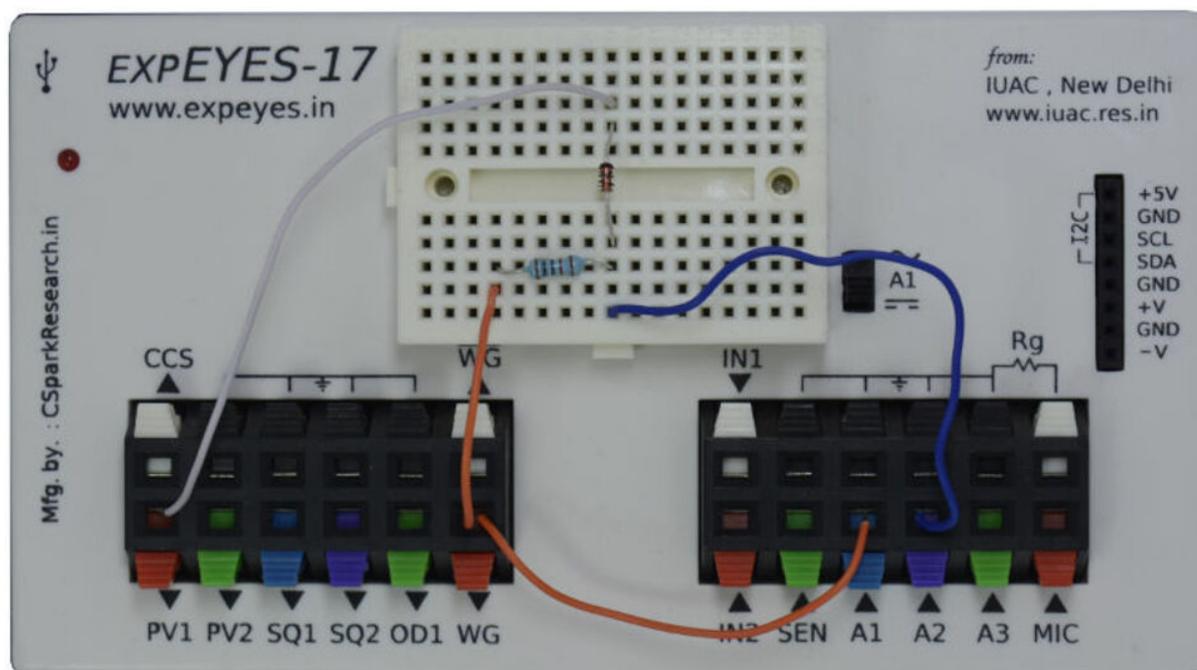
CHAPTER 2

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 connectors on the top panel. as shown in the figure below. 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 ± 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.



2.1 External connections

The functions of the external connections briefly explained below. All the black colored 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 Ω **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 5.1k Ω resistor.

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

Can measure voltage within the ± 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 $1\text{M}\Omega$.

$\pm 3.3\text{ V}$ 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 = $1 + (R_g)/(10000)$. This enables displaying very small amplitude signals. The input impedance of A3 is $10\text{M}\Omega$.

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 $\pm 6\text{ V}/10\text{ mA}$ 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



Missing image
(bitmap format)

To get started, a set of 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 / 20 Ω ,
- Resistors : 560 Ω , 1k Ω , 2.2k Ω , 10k Ω , 51k Ω and 100 k Ω

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

Software Installation

ExpEYES can run on any computer having a USB port and a Python Interpreter along with the modules required for accessing them. The USB interface is handled by a device driver program that presents the USB port as a Serial port to the Python module. The communication the expEYES is done using a library written in Python. Programs with GUI have been written for many experiments. ExpEYES-17 communication library requires the 'numpy' module. The GUI depends on packages like PyQtgraph and scipy. The code has been ported to python3 and installation can be done in several ways.

3.1 On Debian-10 , Ubuntu-18.04 and above

```
$ sudo apt update $ sudo apt install eyes17
```

This will install the versions available in the repository of Ubuntu or Debian. To install the latest versions download the .deb packages from <https://expeyes.in/software.html> and install using the commands.

```
$ sudo apt-remove eyes17 # to remove any old version $ sudo gdebi down-  
loaded_deb_file
```

while connected to the Internet

3.1.1 Any GNU/Linux distributions

Download **eyes17-x.x.x.zip** (the latest version) from <https://expeyes.in/software.html> unzip it, and change to the newly created folder named 'eyes17'. 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'.

3.1.2 The expEYES Live CD / USB pendrive

The ISO image containing support for eyes17 can be downloaded from the web page <http://expeyes.in/software.html> 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
```

3.2 Installing on MS Windows

Bundled Installer: Download and install eyes17-setup installer, from the link provided on <https://expeyes.in/software.html>

Run the installer and launch eyes17 from the icon created on the desktop. This has been tested on Windows10.

Another method is to Install Python3 and the required modules to run the ExpEYES17 code. Download the latest eyes17 zip file. Since the programs are written in Python, the same source code works on GNU/Linux and Windows.

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 the figure. Various experiments can be selected from the menu.

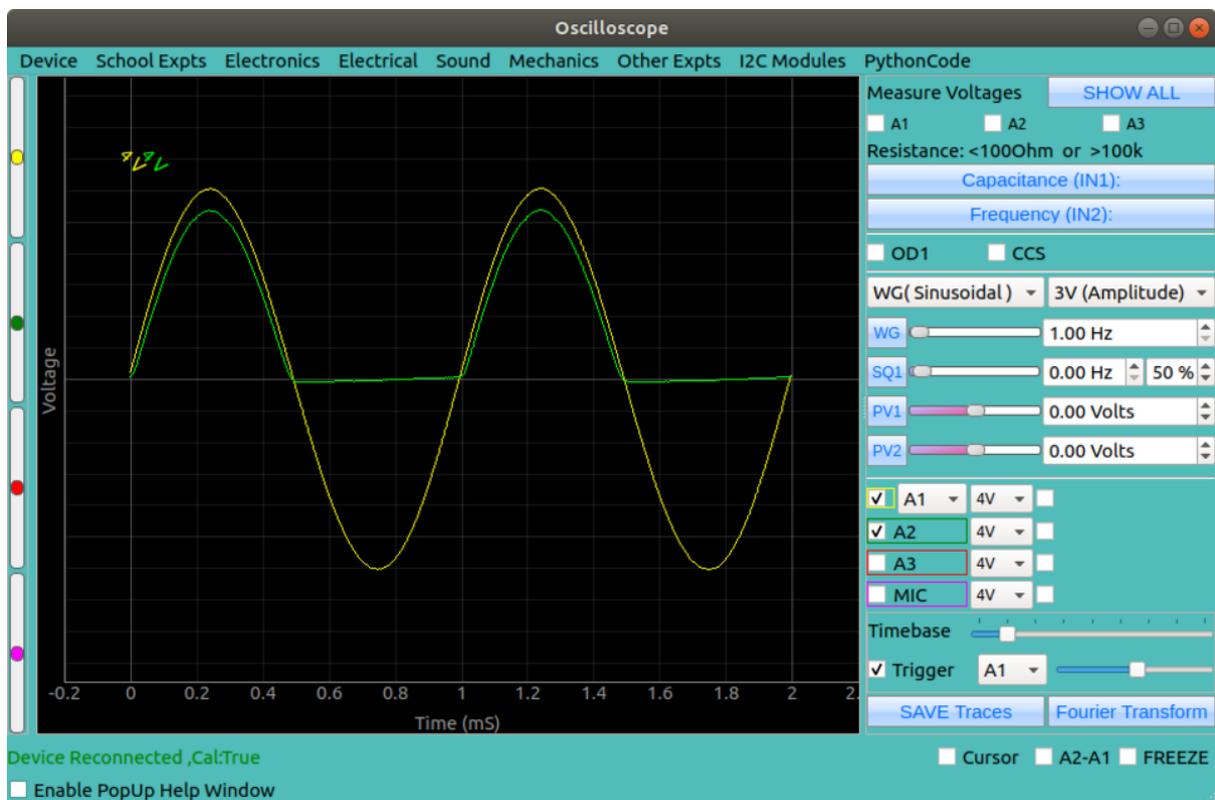


Fig. 1: The ExpEYES-17 main GUI

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

pull-down 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 menu button 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 $V = V_0 \sin(2\pi ft + \theta) + C$ 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 menu button 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 enabled 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 menu button, 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 experiment, select it from the menu. Online help is available for every experiment, making this manual almost redundant.

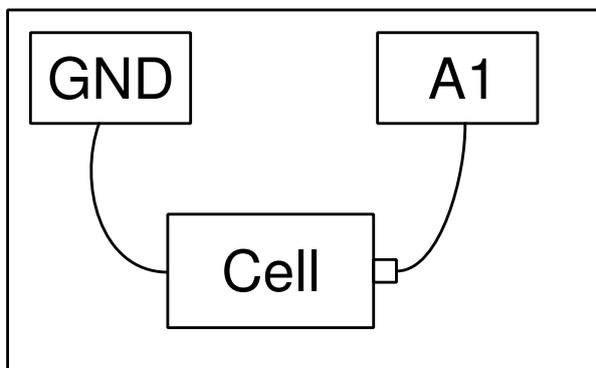
School Level Experiments

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.

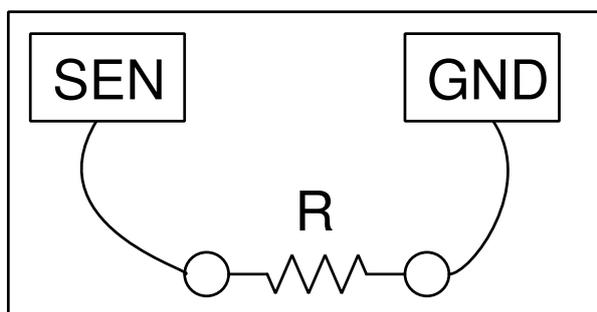
We are measuring the potential difference between two points. One of them can be treated as at zero volts, or Ground potential. The voltage measuring points of expEYES measure the voltage with respect to the terminals marked GND. We have connected the negative terminal of the cell to Ground. The positive 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

Resistance is generally measured using a multimeter. ExpEYES provides a limited facility to do this, using the SEN terminal. SEN is actually a voltage measurement terminal, with a $5.1k\ \Omega$ resistor connected to $3.3V$ internally. The internal $5.1k\ \Omega$ and the external resistor connected to SEN forms a series combination. By knowing the voltage across both and the value of one resistance, we can calculate the other using Ohm's law. The drawback of this method is that the accuracy is poor when the external resistance differs widely from $5.1k\ \Omega$. A reasonable range is from $100\ \Omega$ to $100\ k\Omega$.

Objective

Measure resistance using the SEN terminal.



Procedure

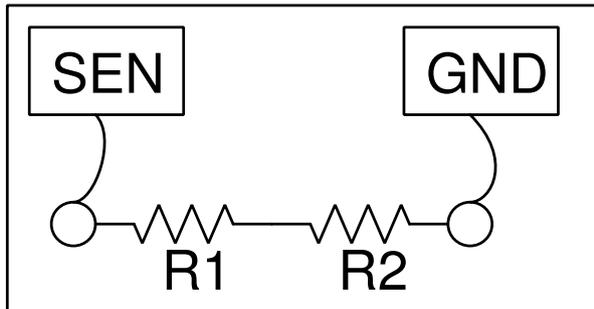
- Connect the resistor between SEN and any Ground terminal.
- Observe the value shown on the top right side of the GUI.
- Repeat with different resistors

6.3 Measuring Resistance series combination

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

Objective

Measure the resistance of series combination of resistors.



Procedure for two resistors

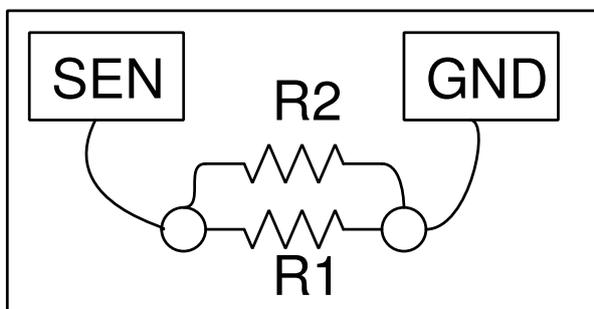
- Connect the resistors between SEN and ground as shown in the figure.
- Observe the value shown on the right side panel.
- Repeat with different resistors.

6.4 Measuring Resistance parallel combination

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

Objective

Measure the resistance of parallel combination of resistors. Values 560, 1000 and 2200 Ohms are available with the kit.



Procedure for two resistors

- Measure the resistances separately.
- Connect them as shown in the figure.
- Observe the value shown on the right side panel.
- Compare the values with calculated results.
- Repeat the experiment with different resistors.

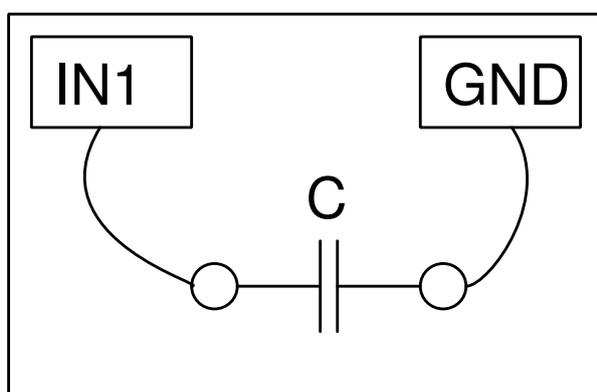
6.5 Measuring Capacitance

Adding electric charge Q to an isolated body results in a potential difference V between that body and its surroundings. The ratio is called its capacitance C , given by the equation $C = Q/V$. A capacitor is generally formed by two conducting surfaces separated by a non-conducting material. The capacitance is the ratio charge in one plate and the potential difference between the plates. The capacitance depends on the area of the plates, distance between them, dielectric constant of the separating material and the geometry of the plates.

For a parallel plate capacitor, $C = \frac{\epsilon_0 \epsilon_r A}{d}$, where A is the area of the plates facing each other, d is the distance between plates, ϵ_r is the dielectric constant of the material between the plates and ϵ_0 is the permittivity of free space whose value is $8.8541878210^{-12} Fm^{-1}$.

Objective

Measuring a capacitance using the IN1 terminal.



Procedure

- Connect the capacitor between IN1 and Ground.
- Click on “Capacitance on IN1”. Should not touch the circuit while measuring.
- Repeat with different values of capacitors, starting from $100 pF$ to several micro Farads.

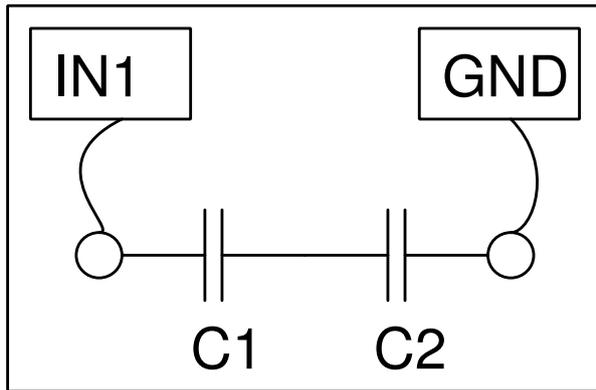
Discussion

You can make the capacitors by pasting thin metal foils on both sides of insulators like paper, polythene or glass. It is possible to find out the dielectric constant of materials by this method.

6.6 Measuring Capacitance in series combination

Objective

Measuring the capacitance of series combination of capacitors.



Procedure for two capacitors

- Connect the capacitors as shown in the figure.
- Click on “Capacitance on IN1”. Should not touch the circuit while measuring

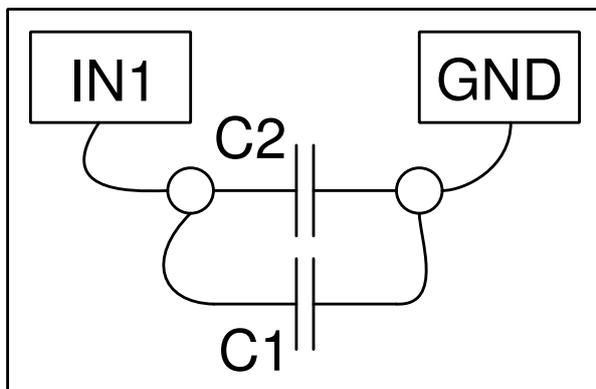
Discussion

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

6.7 Measuring Capacitance of a parallel combination

Objective

Measuring the capacitance of a parallel combination of capacitors.



Procedure for two capacitors

- Connect the capacitors as shown in the figure.
- Click on “Capacitance on IN1”. Should not touch the circuit while measuring

Discussion

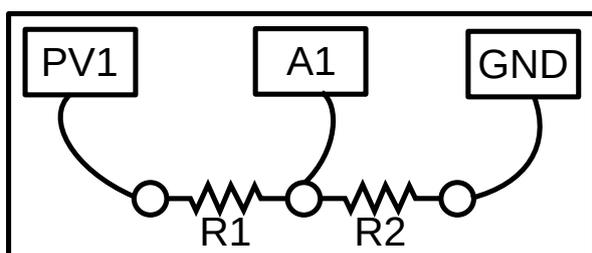
For parallel combination, the effective capacitance is given by $C = C_1 + C_2 + \dots$

6.8 Measure resistance by comparison

According to Ohm's law, current through a conductor and the potential difference between its end are proportional. The constant of proportionality is called the resistance. Mathematically $R = \frac{V}{I}$. When two resistors are connected in series, the current will be same through both. $I = \frac{V_1}{R_1} = \frac{V_2}{R_2}$. if the value of one resistance and the voltage across both are known, the other resistance can be calculated from $R_1 = R_2 \times \frac{V_1}{V_2}$

Objective

Find the value of an unknown resistance by comparing it with a known resistance, using the equations given above. Assume R1 is the unknown resistance and R2 is 1000Ω



Procedure

- Fix the two resistors in series on a bread board.
- Connect the junction to A1
- Connect the other end of R2($1\text{ k}\Omega$) to Ground.
- Connect one end of R1 to PV1
- Set PV1 to 4 volts.
- Enable the Checkbutton on top right, to measure the DC voltage at A1.

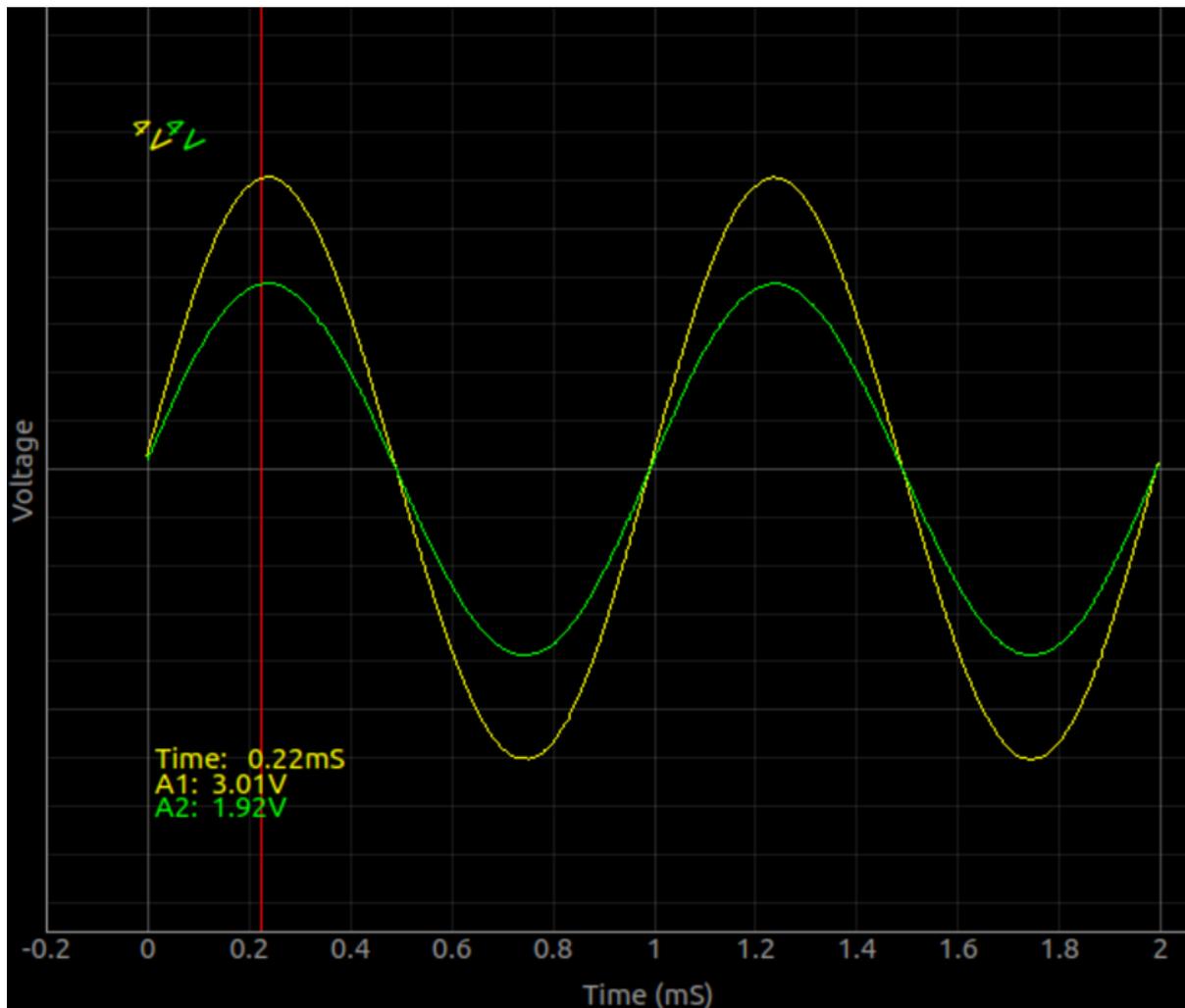
Current $I = \frac{V_{A1}}{R_2}$ and R1 can be calculated using $R_1 = \frac{V_{PV1} - V_{A1}}{I}$.

6.8.1 Ohm's law in AC circuits

It can be easily shown that this measurement can be done using AC also. We will use both A1 and A2 inputs here.

- Fix the two resistors in series on a bread board.
- Connect the junction to A2
- Connect the other end of R2($1\text{ k}\Omega$) to Ground.
- Connect one end of R1 to both WG and A1
- Set WG to 1000Hz
- Enable A1 and A2
- Enable the Cursor Check button to display the voltages at the cursor

Taking voltage reading from the picture below, $I = \frac{1.92}{1000}$ and $R1 = \frac{3.01-1.92}{0.00192} = 576.7$



Discussion In this measurements we have made the assumption that no current flows in to A1 and A2. This is not true, they both have an input impedance of $1M\Omega$. This will matter when we use resistance values of mega Ohms range. To illustrate this connect WG to A1 using a wire and the same signal to A2 through a $1M\Omega$ resistor. Try to explain the results using Ohm's law.

6.9 Direct and Alternating Currents

The magnitude and direction of current from a drycell does not change with time. It is called DC or direct current. The mains supply we get is of different kind. The mains supply in India is 230V at 50 Hz. A frequency of 50 Hz means the change in voltage repeats every 20milliseconds. If we measure the voltage across the phase and neutral terminals of a household power socket, the voltage will increase from zero to $325V (230\sqrt{2})$ in 5 mS and it will be back to zero in the next 5 mS. During the third 5mS it will reach $-325V$ and will again reach zero during the fourth 5mS. These type of current is called alternating current (AC). To study the behavior of AC, an oscilloscope is required.

Objective

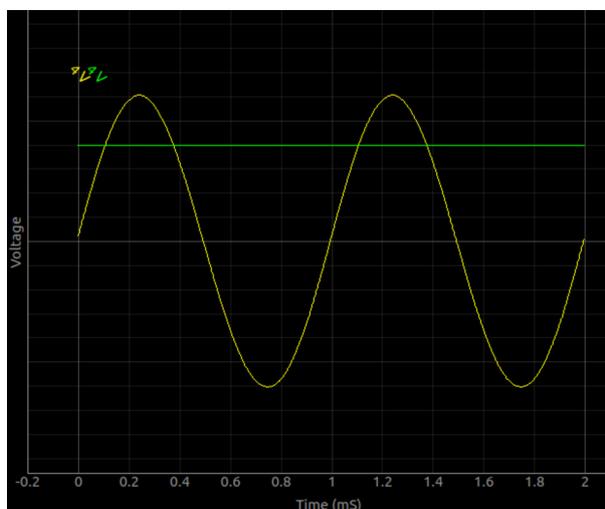
Introduce the concept of time dependent voltages, using a $V(t)$ graph. Compare the graph of DC and AC.



Procedure

- Connect WG to A1 and PV1 to A2, using wires.
- Set PV1 to 2 volts and Set WG to 1000 Hz
- Enable analyse on A1, to measure amplitude and frequency.
- Enable A2

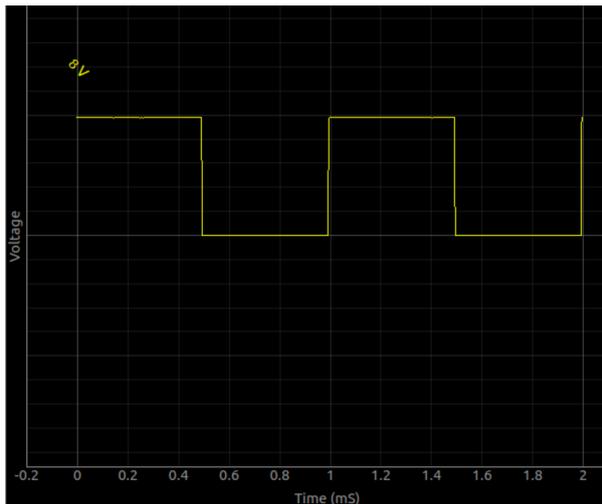
The observed results are shown here.



This should not lead to a conclusion the voltages are either DC or AC. There could be combination of both. For example, take the case of a squarewave that changes between 0 and 5V.

- Connect SQ1 to A1
- Set range of A1 to 8V. Adjust trigger a bit for a stable trace.
- Set PV1 to 2 volts and Set WG to 1000 Hz

Is the graph shown below is AC or DC ? It is a 2.5 DC plus an AC changing from -2.5V to +2.5V. Separating these components will be explained in the coming sections.



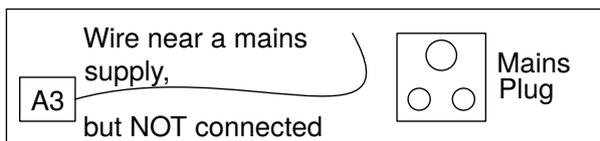
If the voltage is not changing with time, it is pure DC. If it is changing with time, it has an AC component. If the average voltage is zero, it is pure AC.

6.10 AC mains pickup

A changing magnetic field will be present near the wires carrying AC. Voltage will be induced between the ends of a conductor placed in this field. We can explore this by connecting one end of a long wire to the measuring equipment.

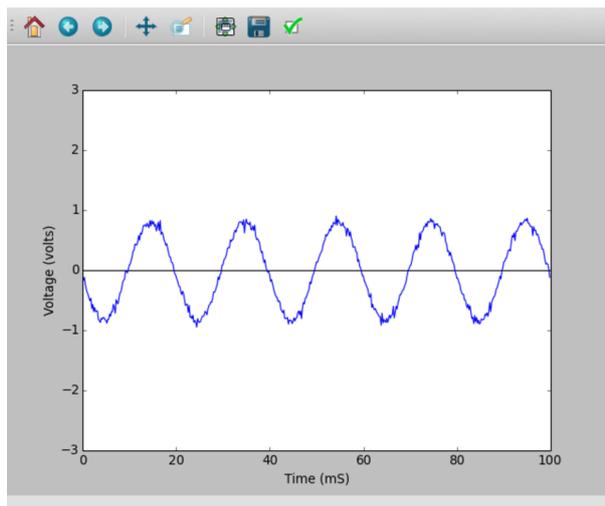
Objective

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



Procedure

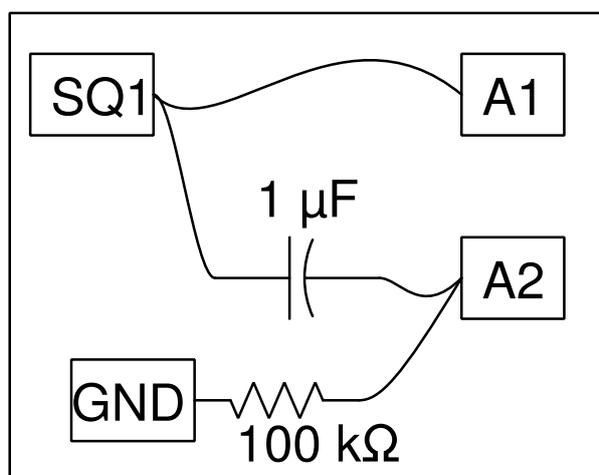
- Connect a long wire to A1
- Take one end of the wire near the AC mains line, without touching any mains supply.
- Enable A1, and it's analysis.



The amplitude of the pickup depends on the wattage of the electrical equipment operating nearby, distance to them and on the length of the wire used.

6.11 Separating DC & AC components

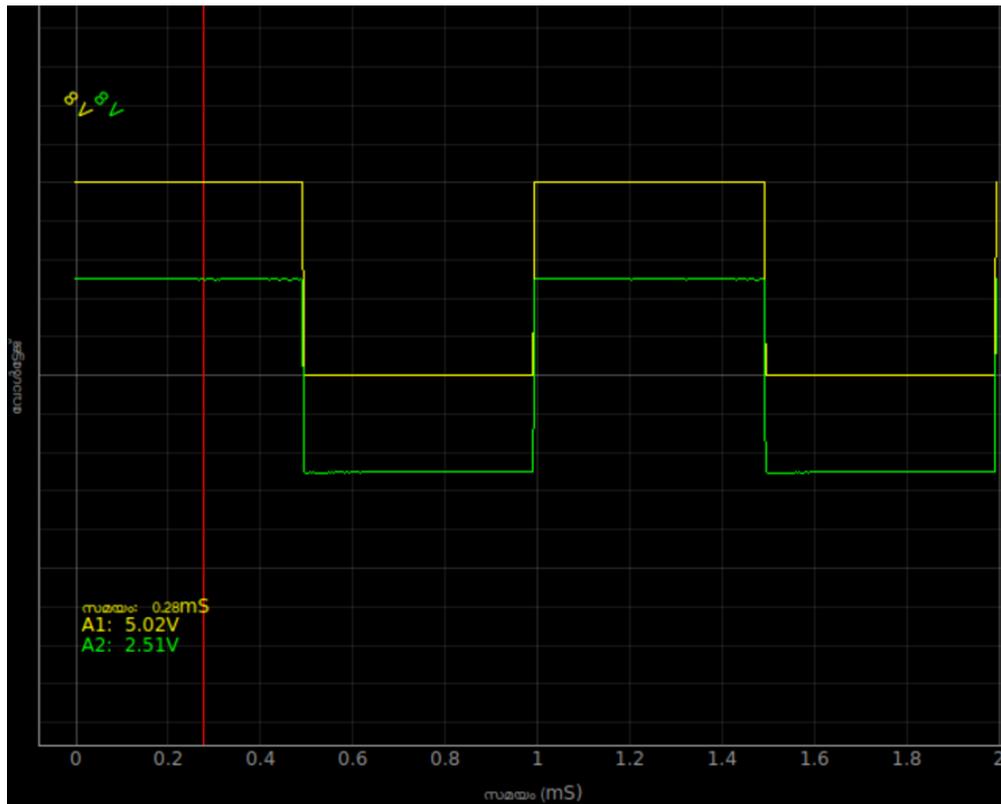
A capacitor does not allow DC to pass through it. This property can be demonstrated using a squarewave swinging from 0 to 5V.



Objective Separate the AC component of a 0 to 5V squarewave.

Procedure

- Set SQ1 to 1000 Hz.
- Connect SQ1 to A1
- Conenct SQ1 to A2 through a $0.1\mu F$ capacitor



Discussion

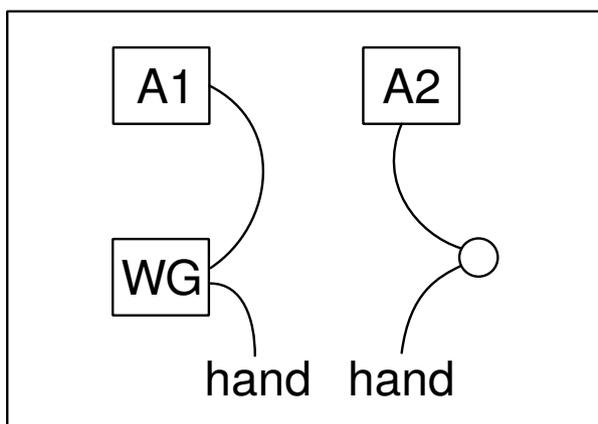
The observed waveforms with and without the series capacitor are shown in figure. The voltage is swinging between 0 and 5 volts. After passing through the capacitor the voltage swings from -2.5 volts to +2.5 volts. What will you get if you subtract a 2.5 from the y-coordinate of every point of the first 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. In case the output on A2 is having a DC component, connect a $100k\Omega$ resistor from A2 to GND.

6.12 Human body as a conductor

It is well known that touching the mains supply is fatal. This is because our body conducts electricity. At the same time you cannot light an LED from a drycell by using your fingers to make the connection. We can explore this further using the low voltage AC and DC sources of ExpEYES.

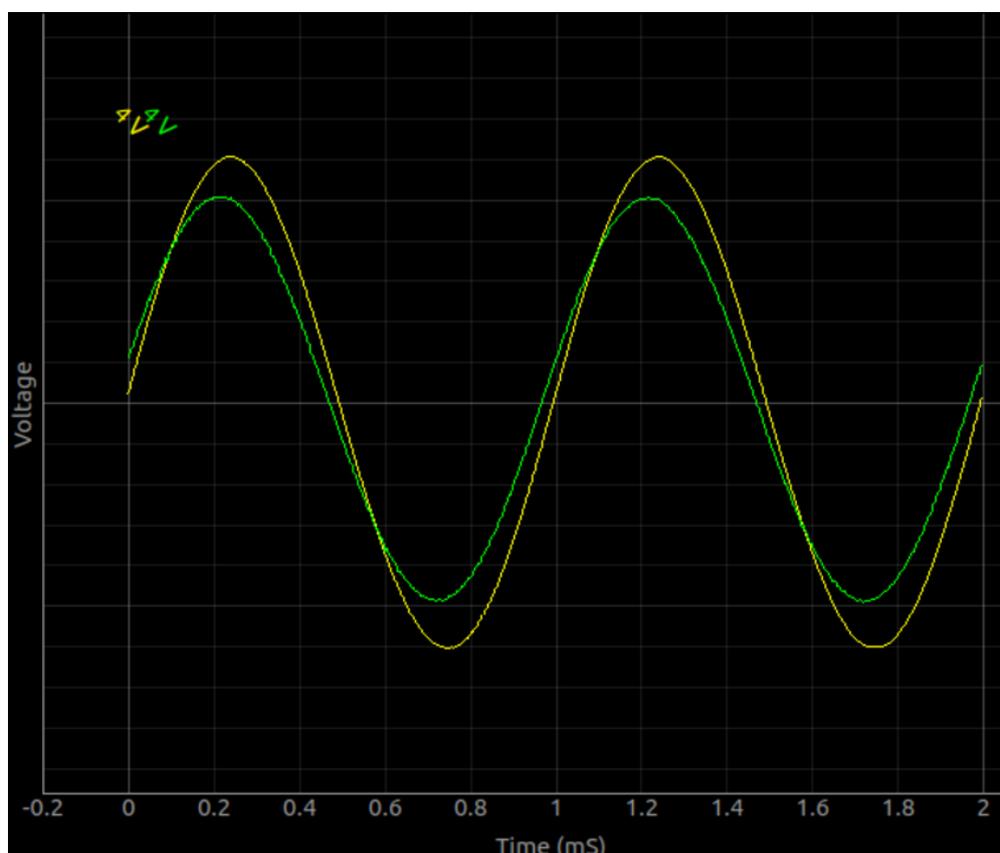
Objective To study the electrical conduction of human body

Procedure



- Connect a wire from WG to A1
- Connect one end of a wire to WG
- Connect one end of another wire to A2
- Enable A1, A2 and their amplitude and frequency display.
- Hold the unconnected ends of both wires by your hands.
- Repeat it using a 3 volt DC signal from PV1.

The observed voltage waveforms are shown below. The voltage on A2 is slightly less than 3volts, due to the resistance of the body.



Discussion Using DC the voltage reaching A2 is smaller, means body conducts AC better than DC The voltage measured at A2 is decided by the ratio of the resistance offered by the body

and the input impedance ($1M\Omega$) of A2. The phase difference between the two waves implies that there is some capacitance present in the circuit.

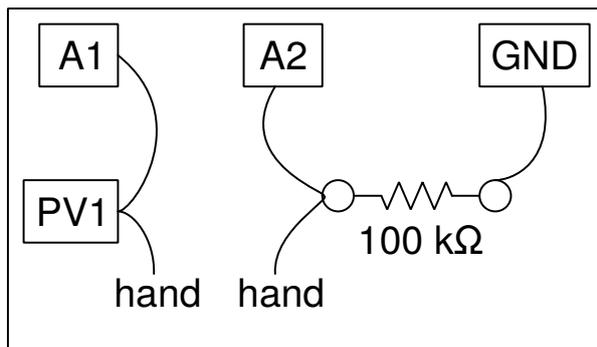
Is the conduction happens through the surface of the body or through the blood stream. How to answer this question. What is the salt content of blood. Try measuring the resistance of salt water. What is the role played by the skin.

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

We have seen that an unknown resistance can be measured by comparing with a known resistance by connecting them in series and measuring the voltages across them. This technique can be used to measure the resistance of human body.

Objective Measure the resistance of human body by comparing it with a known resistor, for DC and AC voltages.



Procedure

- Set PV1 to 3 volts
- Connect a wire from PV1 to A1
- Connect one end of a wire to PV1
- Connect one end of another wire to A2
- Connect a 100Ω resistor from A2 to ground.
- Enable the Checkboxes to display A1 and A2
- Hold the unconnected ends of both wires by your hands.
- Repeat using SINE instead of PV1.
- Enable amplitude and frequency display for A1 and A2.

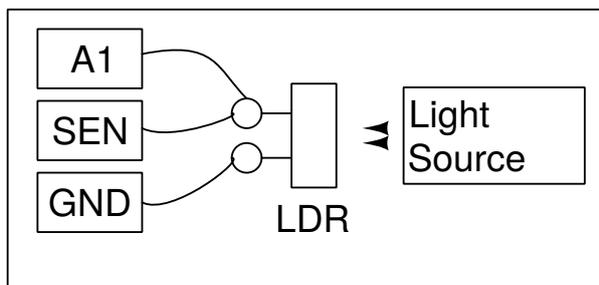
Discussion The AC resistance is less than the DC resistance. The resistance is due to our skin and AC can pass through this, like it passes through the dielectric material of a capacitor. A bit of exploring will reveal that a capacitor is formed between the tip of the wire and blood inside with the skin acting as a dielectric. Explore the effect of a metal plate at the tip of the wire.

6.14 Light dependent resistors

The resistance of an LDR reduces with the intensity of light falling on it. This can be measured using the SEN terminal.

Objective

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



Procedure

- Connect the LDR from SEN to ground
- Connect a wire from SEN to A1
- Measure the LDR's resistance, for different light intensities.
- Illuminate LDR using a fluorescent lamp, A1 should show ripples
- Set the time base to 200mS full scale
- Put A1 in AC mode, using the switch, and measure ripple frequency

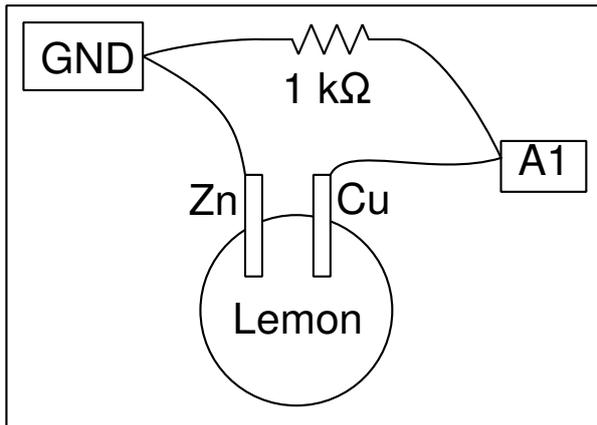
Discussion The resistance vary from $1k\Omega$ to around $100 k\Omega$ depending on the intensity of light falling on it. The voltage is proportional to the resistance. The resistance decreases with intensity of light. If you use a point source of light, the resistance should increase as the square of the distance between the LDR and the light source. The light from a fluorescent lamp operating at 50Hz has 100Hz ripples and these can be measured.

6.15 Voltage of a lemon cell

A lemon cell is formed by inserting Copper and Zinc plates in to a lemon, or any dilute acid. The voltage developed across the electrodes is very small and the cell has a very high internal resistance. Due to this the voltage drops when a load resistor is conected.

Objective

Make a lemoc cell and explore it's internal resistance.



Procedure

- Connect the electrodes to A1 and Ground
- Enable Checkbutton for displaying the voltage at A1
- Connect a 1000Ω resistance across the electrodes.
- Measure the voltage again, to note down the reduction in it.

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 ?

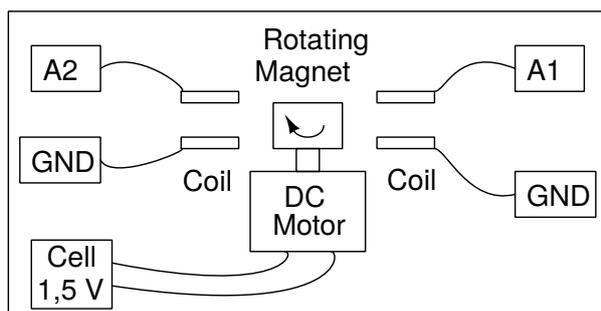
What is the internal resistance of the cell ?

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

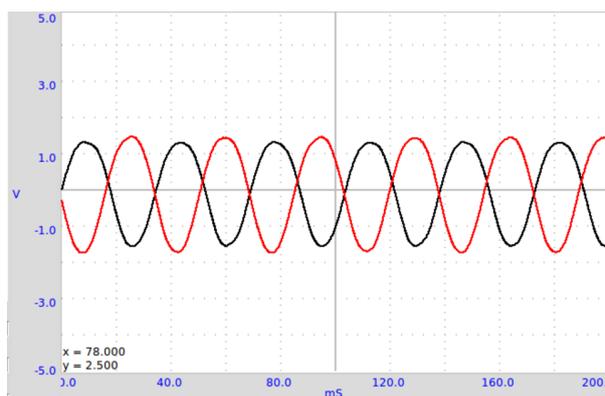
A voltage will be induced across a conductor placed in a changing magnetic field. This can be demonstrated by making a simple AC generator. Use the 10 mm x 10 mm magnet and the 3000T coils that comes with the kit.

Objective Measure the frequency and amplitude of the voltage induced across a solenoid coil kept near a rotating magnet. Use more than one coils to generate multi-phase AC.



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.

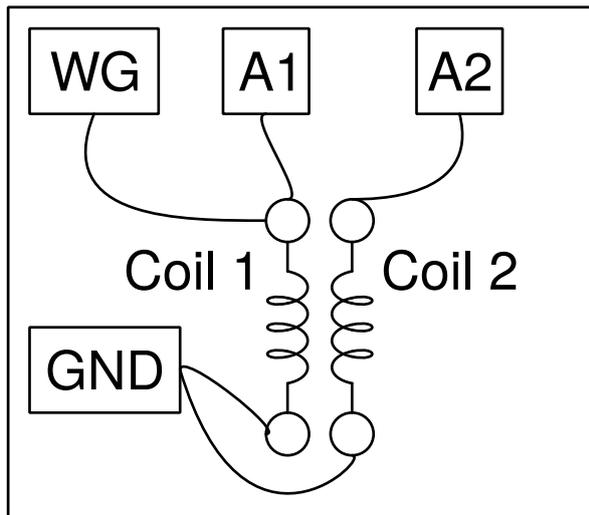
6.17 AC Transformer

There will be a time varying magnetic field around a conductor carrying AC. A voltage will be induced across another conductor placed in this field. This is the working principle of a transformer, that can be demonstrated using the two coils included in the kit.

Objective

Construct a transformer demonstrate mutual induction using two coils. One coil, the primary, is connected between WG and Ground. The axes of the coils are aligned and any ferromagnetic material may be inserted for better coupling.

Procedure



- Make connections as shown in the figure
- Enable A1 and A2
- Set WG to 1000 Hz
- Bring the coils close and watch the voltage on A2.
- Try inserting an iron core, a nail or screwdriver also would do.



Discussion

The applied waveform and the induced waveform are shown in figure. A changing magnetic field 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.

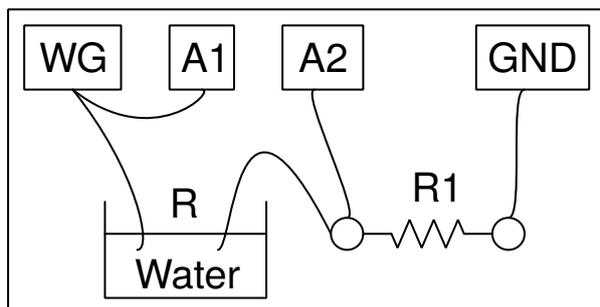
Try doing this experiment using a squarewave. Connect a $1k\Omega$ resistor across secondary coil to reduce ringing.

6.18 Resistance of water, using AC

Resistance of water is an indication of its purity. Water conducts mostly due to the dissolved salts. If you have never tried measuring the resistance of ordinary tap water try doing it with a multimeter. Are you getting a stable reading ?

Objective

Measure the resistance of ionic solutions, using AC voltages, using normal tap water.

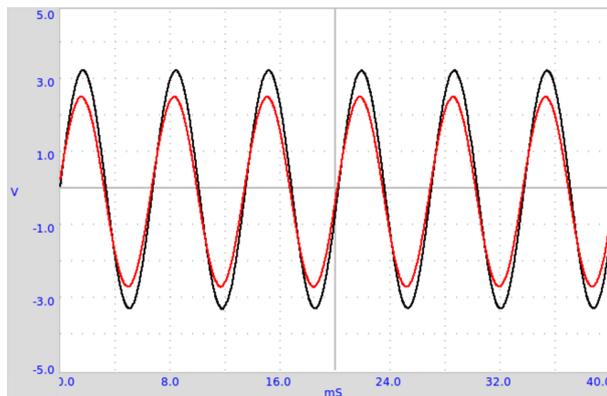


Procedure

- Make connections as shown in the diagram
- Choose the resistor comparable to that of water, start with 10k.
- Enable A1 and A2 with the amplitude and frequency display options.

Calculate the resistance using the method of comparing with a known resistance. Current flowing $I = \frac{V_{A2}}{R_1}$

Resistance offered by water $R_w = \frac{V_{A1}-V_{A2}}{I} = \frac{V_{A1}-V_{A2}}{V_{A2}} \times R_1$.



Discussion

The experiment may be repeated using DC from PV1. Use the DC displays of A1 and A2 while using DC. With DC, the resistance of the liquid changes with time, but AC gives a steady reading. When done using water in a cup with two wire ends dipped in it, the resistance does not depend much on the distance between the electrodes, the area of the electrode is having some effect. To find out the resistivity of water we need to use a long column of water in a tube, with electrode plates at both ends.

The resistance depends on the ion concentration and presence of impurities in the water used. 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 ?

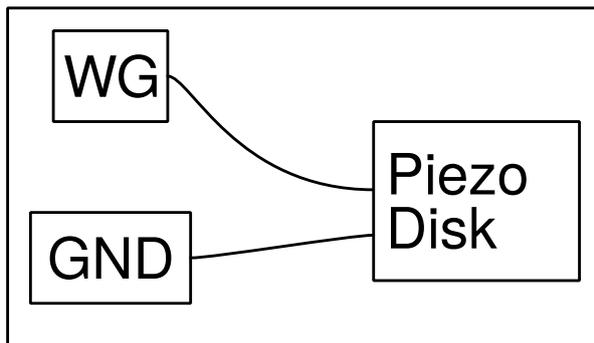
6.19 Generating sound

Electrical signals can be converted into sound using devices like loudspeaker, Piezo buzzer etc. We use a buzzer because available loudspeakers have a low impedance and require more current.

Objective

Generate sound from electrical signals, using a Piezo-electric buzzer.

Procedure



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

Discussion

When you change the frequency of the voltage that excites the Piezo, both the frequency and the intensity 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 the mechanical properties.

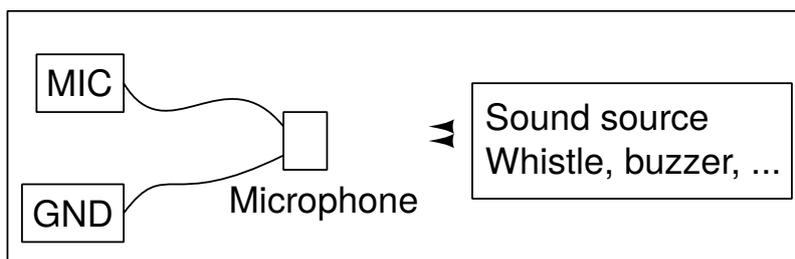
6.20 Digitizing sound

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

Objective

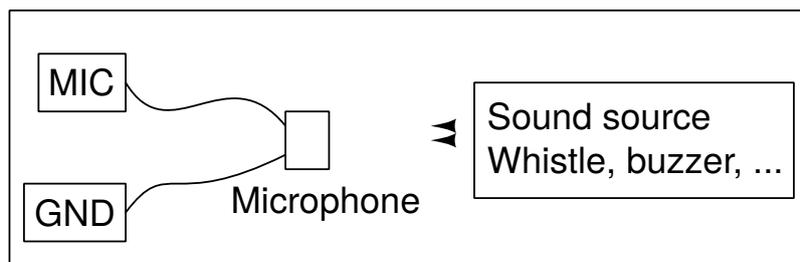
Digitize sound signals from a microphone, and measure its frequency. Use the Piezo buzzer or any other source of sound like a tuning fork.

Procedure



- Enable CheckButton MIC and it's amplitude and frequency display.
- Position the buzzer facing the microphone
- Set WG to 1000Hz, change it and watch the MIC output
- Use a whistle instead of the buzzer
- Press 'Fourier Transform' while singal is steady to analyze it.

The microphone output is shown in figure.



Discussion

The ability to capture sound and measure it's frequency leads to several experiments. Making sound by blowing air in to one end closed tubes can be used for measuring the velocity of sound in air.

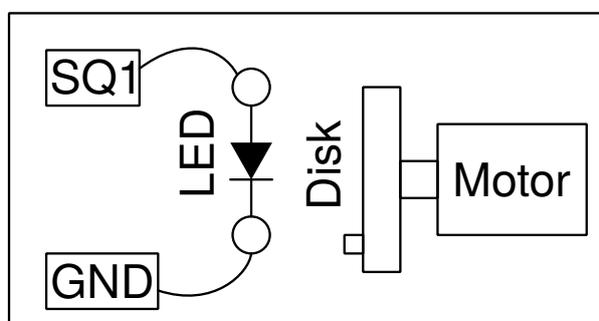
6.21 Stroboscope

A stroboscope is an instrument used to make a cyclically moving object appear to be slow-moving, or stationary. It consists of light source which produces brief repetitive flashes of light. If a body rotating at some frequency is illuminated with a light flashing at the same frequency, it is visible only when it is at a particular position. This gives an impression that it is stationary. If the two frequencies differ slightly, the body will appear to move very slowly.

Objective

Observation of a rotating body with a periodic flashed light, using a disk with a marker.

Procedure



- The disk is rotated by powering the motor by a 1.5 V cell.
- Connect a white LED to SQ1

- Set the dutycycle of SQ1 to 20% or less
- Adjust the frequency of SQ1, to make the disk appear stationary
- You need a dark area, no other light should fall on the body.

Discussion

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

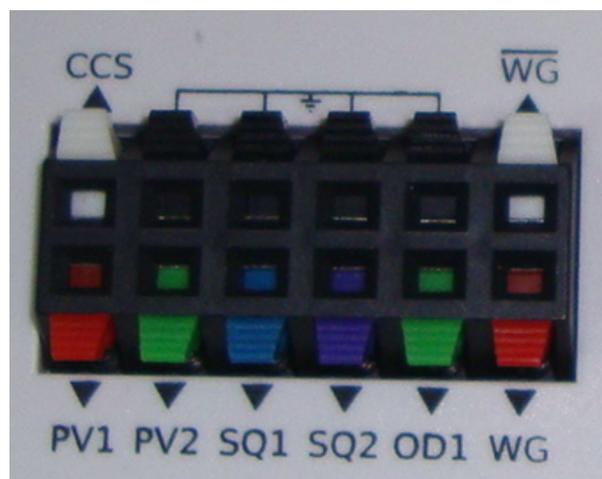
What happens when the frequency of the light is slightly increased, or slightly decreased?

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

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 Oscilloscope and other Equipment

The first thing that appears when you open the ExpEYES GUI program is an Oscilloscope window. The graphs of the four channel scope appears on the left side. On the right side we have the Buttons, Sliders etc. to control the scope and most of the hardware features of ExpEYES, like DC supplies, waveform generators etc. To understand ExpEYES, one should start with the Input/Output terminals and their functions as explained below.

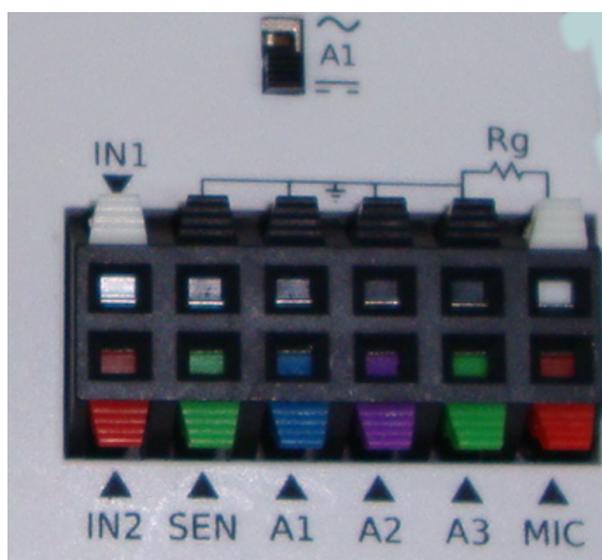


Output Terminals

- **CCS:** 1.1 mA Constant Current Source. If you connect a resistor from this terminal to ground, a current of 1.1mA will flow irrespective of the value of the resistor. The

maximum allowed value of the resistor is $2k\Omega$. Using the Check-Button given on the GUI, you can enable CCS.

- **PV1 and PV2: Programmable Voltages** The output of this terminal can be set anywhere from $-5V$ to $+5V$, using software. The voltage can be verified by connecting a voltmeter between PV1 and ground. PV2 is a similar output but the range is only from $-3.3V$ to $+3.3V$. The voltages can be set from the GUI, using Sliders or Text-Entry widgets.
- **SQ1: Square Wave Generator** This terminal can generate a Square wave that changes continuously from 0 to $5V$. The frequency can be set from the GUI from $1Hz$ to $5kHz$, but it can be programmed to generate $0.1Hz$ to $1MHz$. SQ1 has a 100Ω series resistor so that LEDs can be directly connected.
- **SQ2: Square Wave Generator** SQ2 is available as an option of WG, means only one is available at a time. SQ2 has no series resistor.
- **OD1: Digital Output** The output of this terminal can be set to either 0 or $5V$. Using the Check-Button given on the GUI, you can change the voltage level at OD1.
- **WG: Waveform Generator** This terminal is an arbitrary waveform generator. The frequency can be varied from $5Hz$ to $5kHz$. Amplitude can be set to $3V$, $1V$ or $80mV$. The shape can be set to Sine, Triangular or Square. In SQR mode the output will appear on SQ2, with 0 to $5V$ swing. $\bar{W}G$ is just an inverted output of WG.

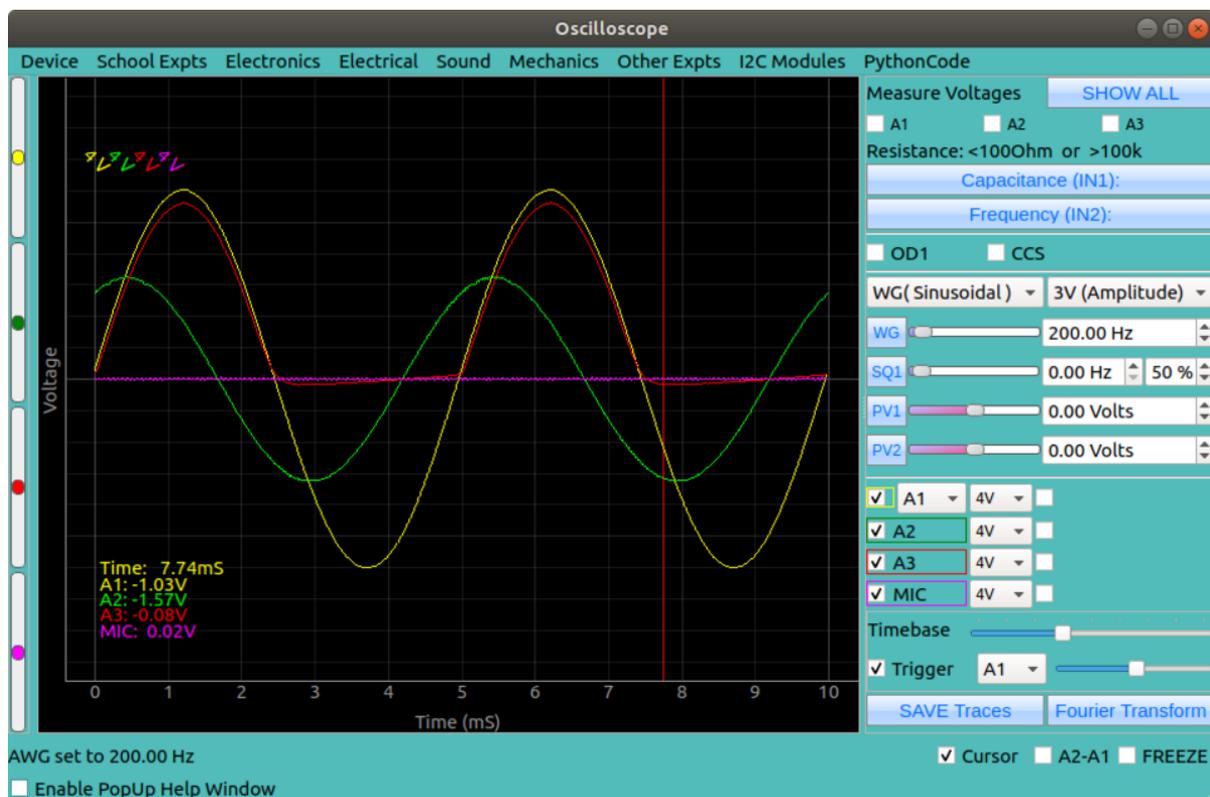


Input Terminals

- **IN1: For measuring Capacitance** This has a built-in capacitance meter. By connecting between IN1 and ground, you can measure capacitance values ranging from few picoFarads to several microFarads. A Push-Button is provided for measurement.
- **IN2: Frequency counter for logic signals** This terminal can measure the frequency and duty cycle of digital signals, swinging between 0 and 3 to $5V$. A Push-Button provided for measurement.

- **SEN: Input for measuring resistance** This is actually a voltage measurement terminal that is internally connected to 3.3 V via a 5.1 kΩ resistor. We calculate the value of the externally connected load by using Ohm's law. This terminal is mostly used for connecting resistive sensors like photo-transistors, for time interval measurements.
- **A1: Voltage measurement terminal** Functions as voltmeter and oscilloscope inputs. Maximum Input range ± 16 V. An AC/DC mode selection slider switch is provided on the box. The maximum input voltage range is selectable from pull down menu. Input A2 is same as A1, but no AC coupled mode.
- **A3: Voltage measurement terminal** This has only a range of ± 3.3 V. Inputs to A3 can be internally amplified by connecting a resistance from Rg to Ground. This helps measuring very small signals.
- **MIC: Microphone input** A Condenser microphone can be directly connected to this. The 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 kΩ resistor gives a gain of 11.
- **I2C Interface** I2C standard circuits can be connected to this. A wide range of sensors to measure temperature, pressure, velocity, acceleration, magnetic field etc. are available in the market. ExpEYES supports many of them from the GUI.
- ± 6V/10mA DC Supply This sockets provide DC power for Op-Amp circuits, with very limited current.

Graphical User Interface



The GUI menubar consists of several pulldown menus for different categories of experiments. The left side of the the screen is the four channel oscilloscope window. On the right side there are Buttons, Sliders and Text Fiields for accessing the hardware features explained earlier.

Oscilloscope Controls

- **Channel Selection** The four channels A1, A2, A3 and MIC can be selected for display using the Check buttons on lower half of the right side.
- **Input Volatge Range** Pulldown menus are provided against each channel for selecting the input voltage range. A1 and A2 are having a maximum range of $\pm 16 V$.
- **Amplitude and Frequency** Check buttons on the right side of the range selection menu enables mathematical fitting of the captured data using a Sine function. The calculated Amplitude and Frequency are displayed.
- **Time base Slider** The time base slider is for selecting the duration of the X-axis. The allowed full range can be set from 1mS to 500mS.
- **Trigger Controls** A digital storage oscilloscope digitizes periodically captures the input for a given duration and displays the result. To get a stable trace, the amplitude and slope of the input and the beginning of each capture operation should be kept same. This starting value is decided by the trigger level setting.
- **Saving Data** The data of the traces can be saved to a file in text format. The time and voltage values of each selcted trace will be written to a file.
- **Cursor** A vertical cursor can be enabled to view the instantaneous values of each channel at the cursor position along with time.
- **A1-A2 Option** This option draws a trace showing the difference of A1 and A2 inputs.
- **Fourier Transform** The Fourier transform converts the data from time domain to frequency domain. The amplitudes of the frequency components will be displayed. The value of the fundamenta frequency will be displayed.

Other Controls and Readbacks

- On the top right side there are Check boxes labelled A1, A2 and A3. Selecting them will display the instantaneous values of the voltages at terminals A1, A2 and A3. These are useful for measuring DC volatges only.
- The value of the resistance connected to **SEN** is displayed below the DC voltage displays.
- Next is a Button for measuring the capacitance connected to IN1.
- A Button is available for measuring the frequency of a digital pulse at IN2. The signals should be swinging from zero to 5 volts.
- Two CheckButtons are provided for selecting OD1 and CCS.
- A pulldown Menu is given for selecting the waveshape of WG. When SQR is selected, the output shifts to SQ2. There is also a menu to select the amplitude of WG output. The allowed values are 3V, 1V and 80mV.
- There are four vertical sliders on the left side of the GUI. They are used for horizontally shifting the traces. This i useful in experiments like the study of logic gates.

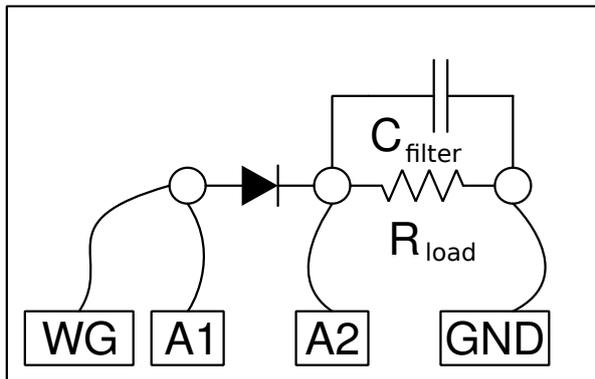
7.2 Half wave rectifier using PN junction

A PN junction conducts only in one direction. Only one half of a pure AC signal can pass through a PN junction. This device is widely used for generating DC from AC.

Objective

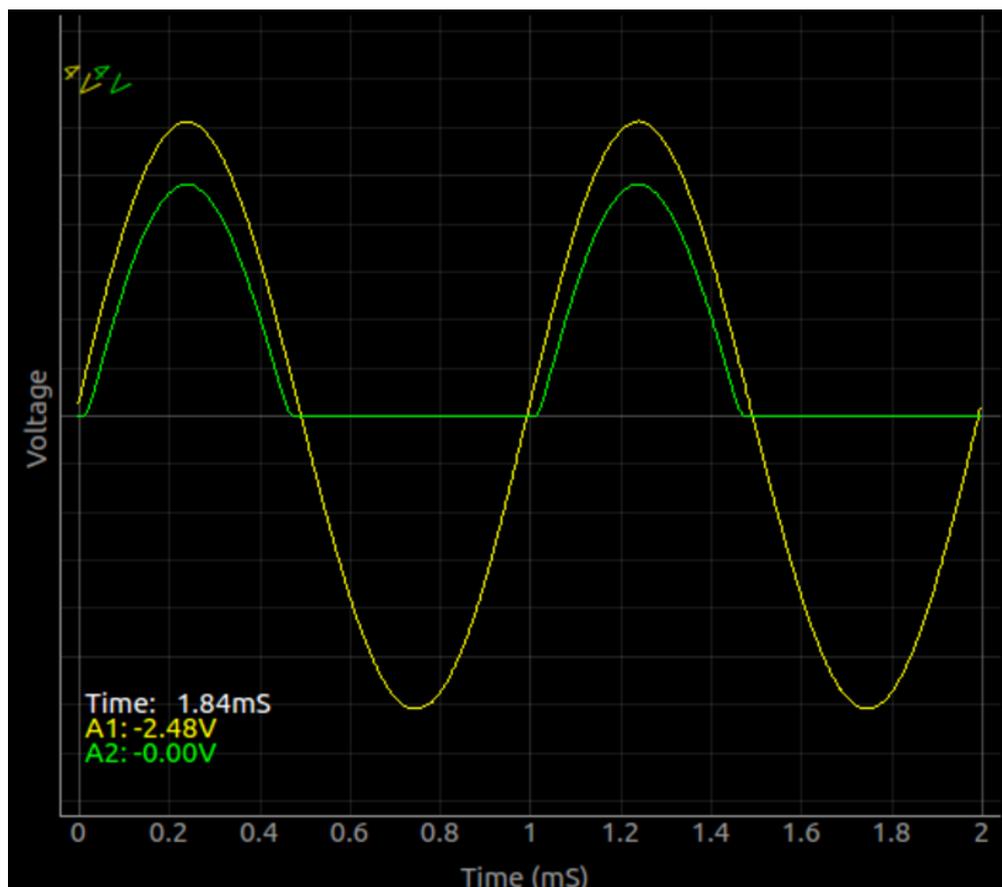
Learn the working of a PN junction diode as a rectifier. Use 1N4148 or 1N4003. RC filtering to reduce the ripple (the AC component).

Procedure



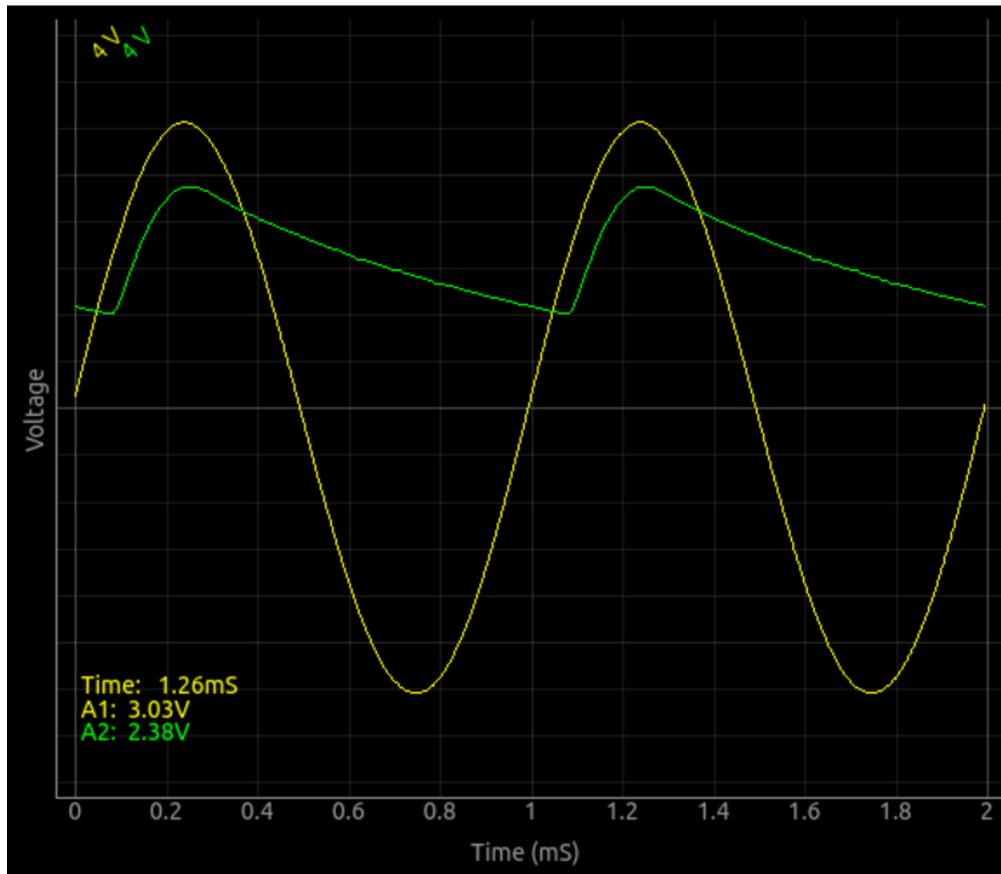
- Fix the diode on a bread board
- Make connections and observe the output
- Connect a $1\text{ k}\Omega$ load resistor, note the difference in amplitude

The output is shown below.



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

The effect of the RC filter is shown below.

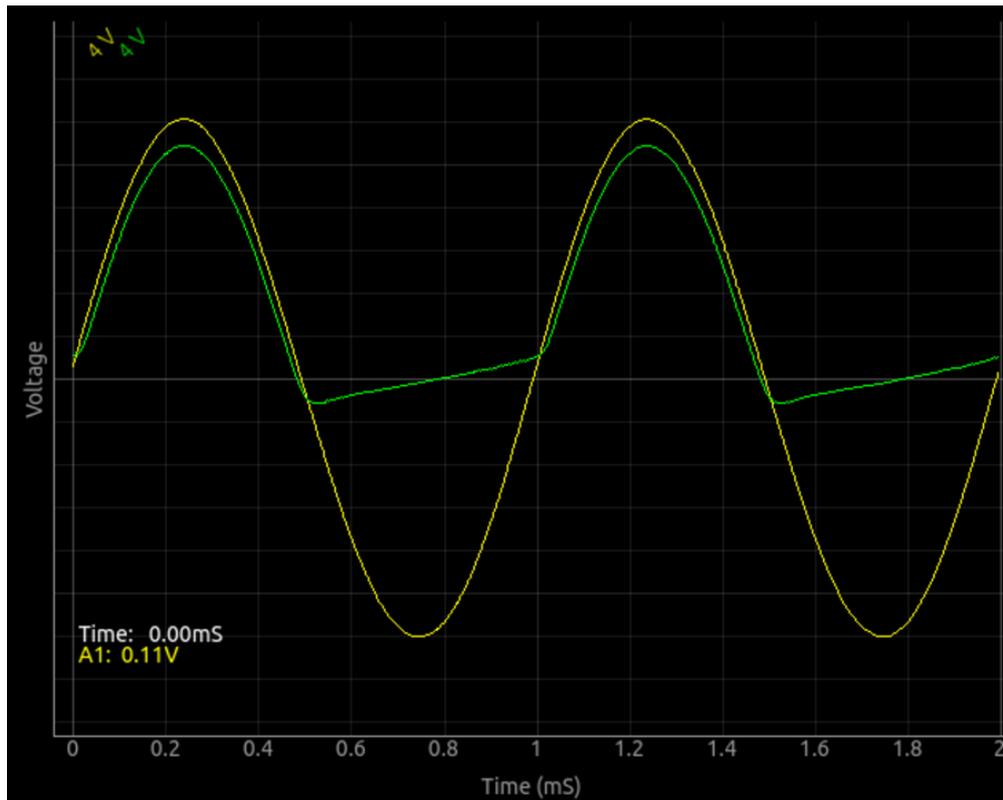


Discussion

The negative half is removed by the diode as shown in figure. The output is a bit noisy and there is some reduction in the voltage. When you connect the load resistor, the output becomes clean and the voltage drop becomes 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 $1k\Omega$ 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 ?

If you use a 1N400x diode without a load resistor, there is some voltage appearing at the output during the negative half also. This will vanish when load resistor is connected. Think of parasitic elements , like the capacitance of the PN junction. Why it is less for 1N4148.

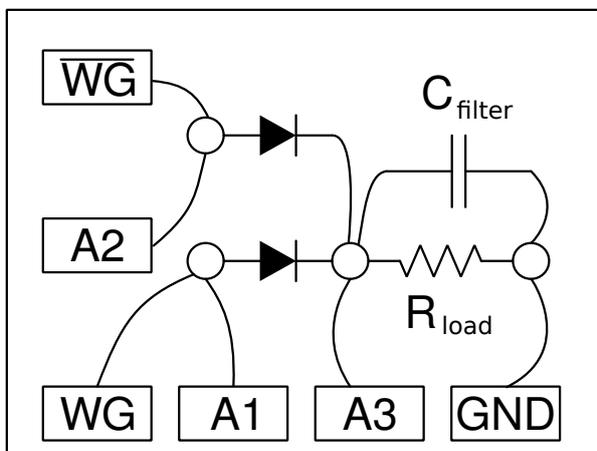


7.3 Fullwave rectifier using PN junctions

A half wave rectifier output depends on the filter capacitor for a long duration to provide DC output. This results in larger ripple and not suitable for higher current. A fullwave rectifier solves this by providing output for both negative and positive halfcycles. However it requires two out of phase AC inputs, generally provided by a transformer with a center tap.

Objective

Make a full wave rectifier, using two diodes. Two AC waveforms, differing by 180 degree in phase, are provided by the WG and $\bar{W}G$ outputs.



Procedure

- Make connections on a breadboard
- Enable A1, A2 and A3
- Set WG to 1000Hz 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

A PN junction diode conducts when the anode voltage exceeds the cathode voltage by an amount greater than the forward voltage of the diode. We can feed an AC voltage to the anode through a resistor and clip a part of it by applying a DC voltage at the cathode.

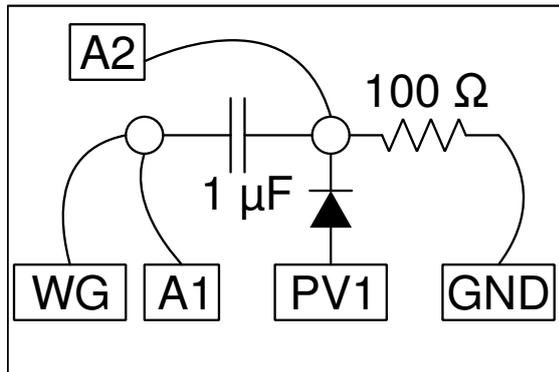
Objective

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

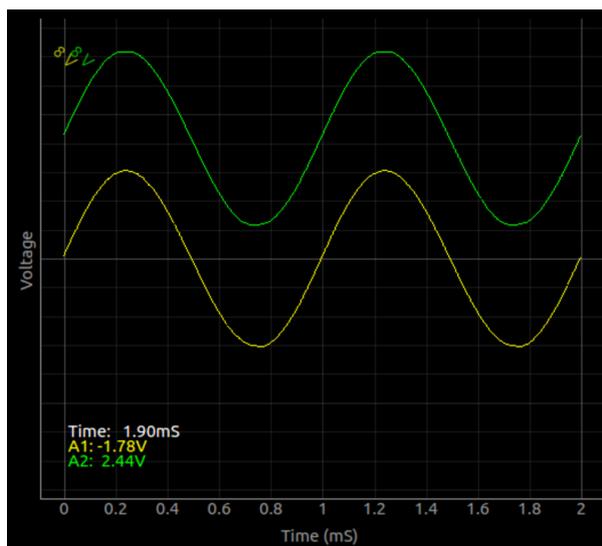
Procedure

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

Procedure



- Make connections as shown in the figure.
- Set WG to 1000 Hz
- Load resistor may be omitted, if it distorts 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.

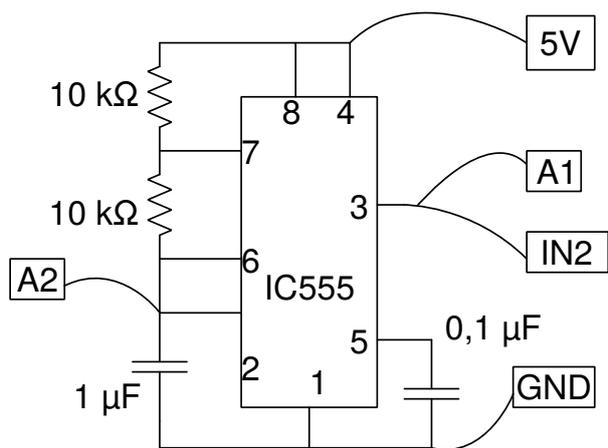
7.6 IC555 Oscillator

NE555 is a very popular time circuit IC. It can be configured as an astable or monostable multivibrator. The values of frequency and duty cycle are decided by two resistors and a capacitor.

Objective

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

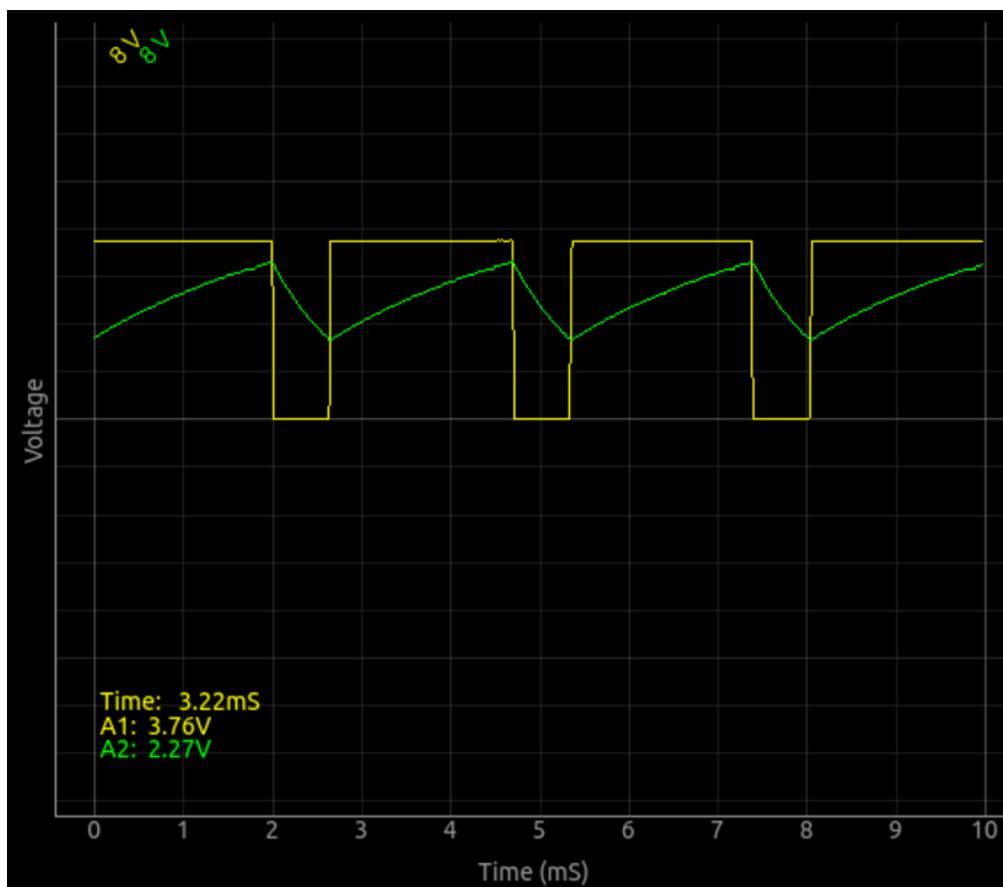
Procedure



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

The output waveform is shown below. The frequency and dutycycle can be calculated using the equations given below and compared with the measurements.

$$T1 = 0.693(R1 + R2)C \text{ and } T2 = 0.693R2C. \text{ Frequency } f = \frac{1}{(T1+T2)} \text{ and duty cycle} = \frac{T1}{(T1+T2)}$$



Discussion

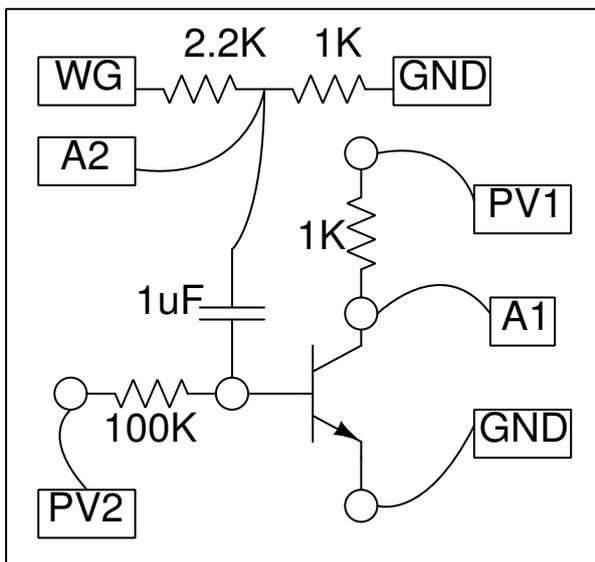
The output waveform is shown in figure. Change the value of resistors or the capacitor, and compare the frequency and duty cycle with the calculated values. Variable resistors may be used to change the frequency and duty cycle.

7.7 Transistor Amplifier CE

The experiment to draw the transistor characteristics can be modified to convert it into a common emitter amplifier. All we need to do is to make a potential divider network to reduce the amplitude of WG to a value less than 80mV. This reduced output is connected to the base through a 1uF capacitor.

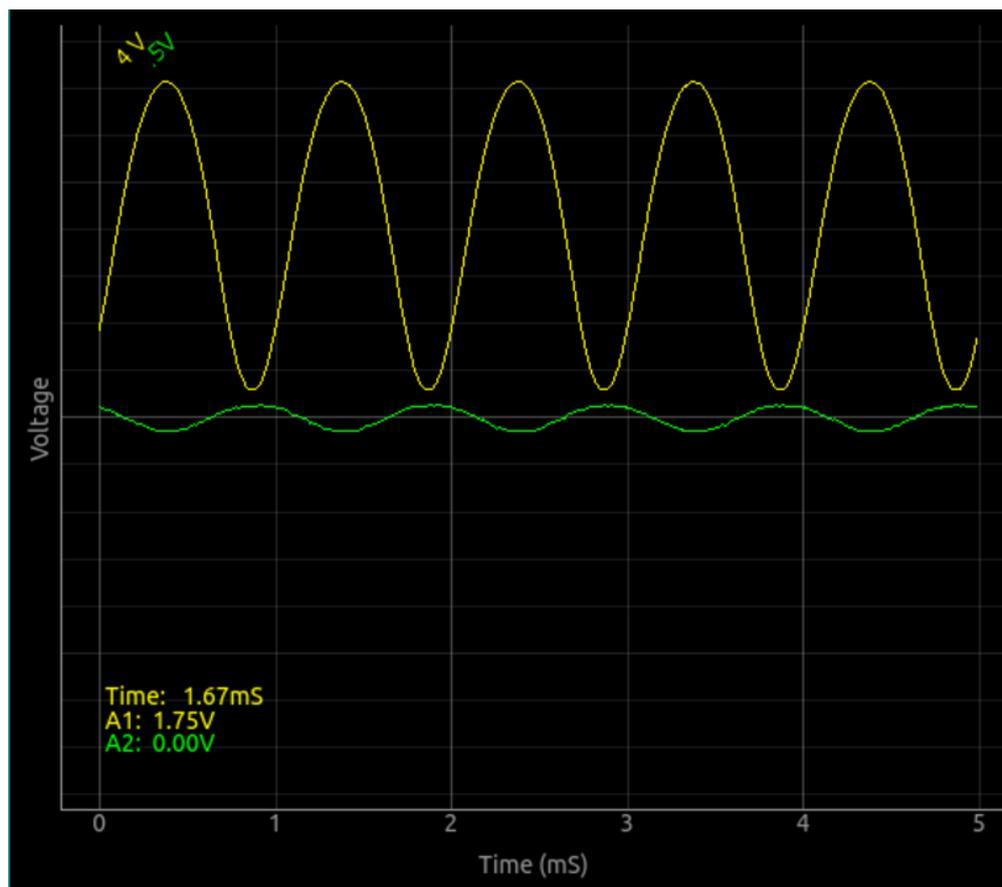
Objective Demonstrates the amplifier action of an NPN transistor in the common emitter configuration.

Procedure



- Fix the transistor on bread board and make connection as shown in the figure.
- Set base voltage to the 1 volt and START.
- Set WG amplitude to 80mV.
- Enable A1 and A2
- Use the slider to change base voltage to observe the shift in operating point.

The output is shown below. It can be seen that the input and output signals are 180 degree out of phase.



Discussion

The base current is set the PV2 voltage applied via the $100\text{ k}\Omega$ resistor. This sets DC operating point of the circuit. Choose this to get minimum distortion in the output.

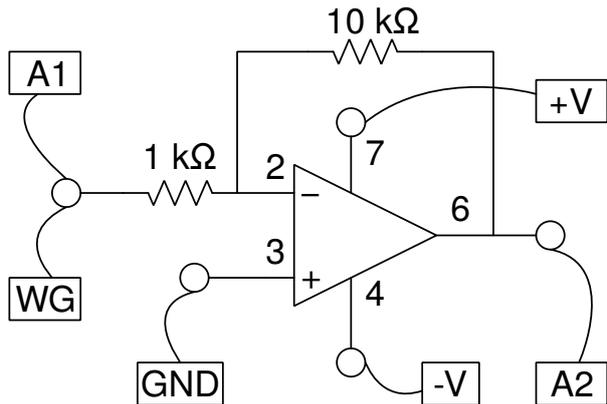
7.8 Inverting Amplifier

It is very easy to make an amplifier using an Operational Amplifier IC. The amplification factor is decided the ratio of two resistors.

Objective

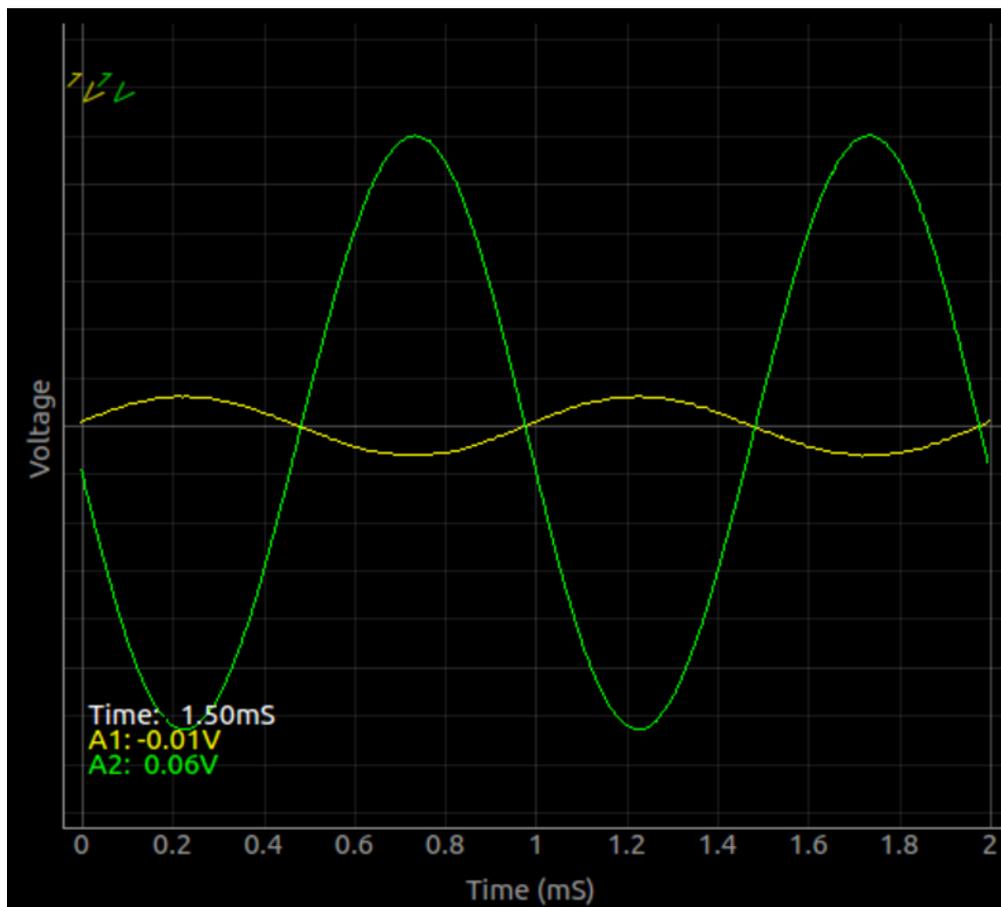
Wire an Inverting amplifier using an Op-Amp, like uA741 or OP07, and test it.

Procedure



- Make the connections on bread board as shown in the figure.
- Set WG amplitude to 80 mV and frequency to 1000 Hz
- Enable A1 and A2, and set ranges to 1V
- Enable the amplitude and frequency displays of A1 and A2.
- Make connections and calculate the voltage gain from the output
- Change gain by changing the resistor values.

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



Discussion

Using expeyes, it is not possible to study the high frequency response.

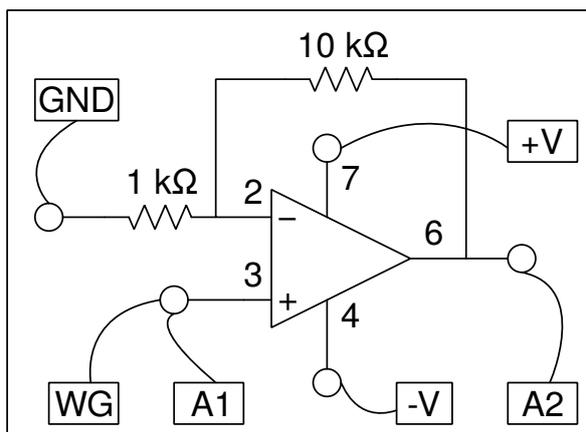
7.9 Non-inverting Amplifier

This experiment is very similar to the previous one, the inverting amplifier.

Objective

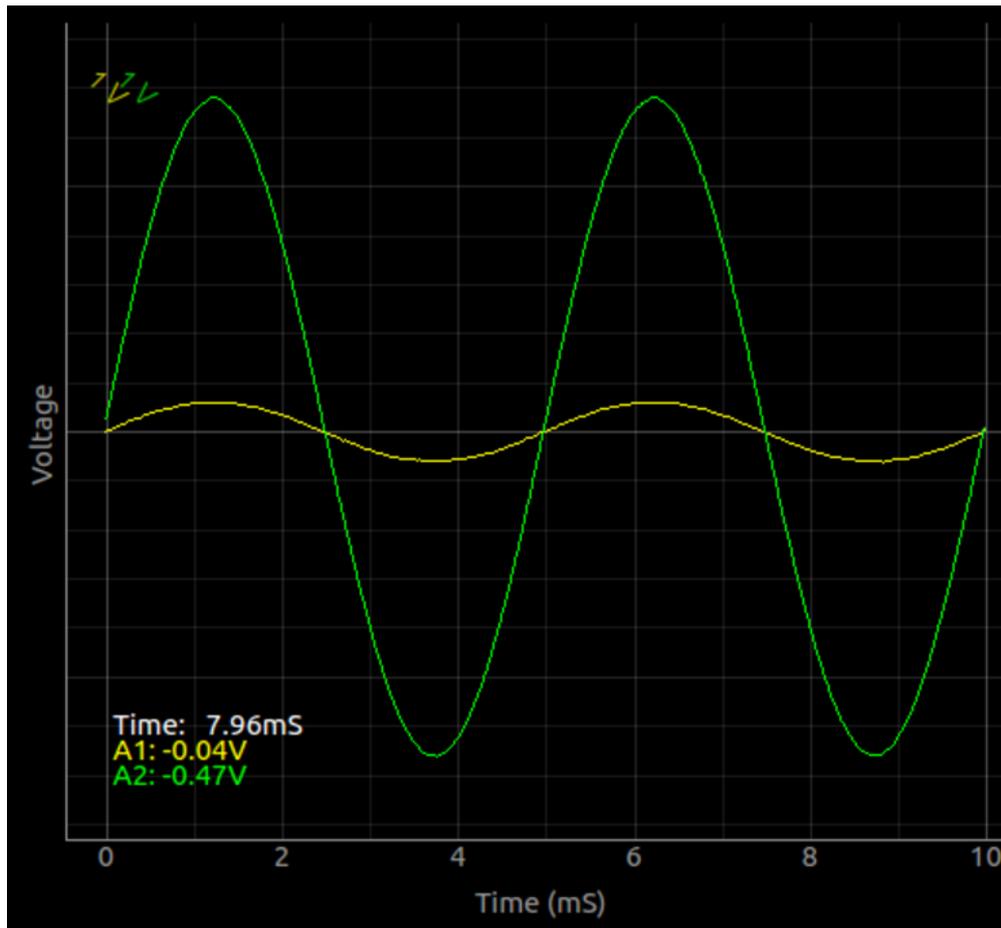
Wire an non-inverting amplifier using an Op-Amp, like uA741 or OP07, and test it.

Procedure



- Make the connections on bread board as shown in the figure.
- Set WG amplitude to 80 mV and frequency to 1000 Hz
- Enable A1 and A2, and set ranges to 1V
- Enable the amplitude and frequency displays of A1 and A2.
- Make connections and calculate the voltage gain from the output
- Change gain by changing the resistor values.

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



Discussion

Using expeyes, it is not possible to study the high frequency response.

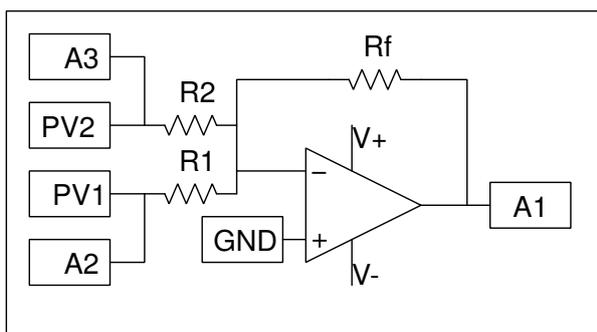
7.10 Summing Amplifier

Operational amplifiers can be used for simulating mathematical operations like addition, integration, differentiation etc.

Objective

Make a summing amplifier using an Op-Amp and test it.

Procedure



- Make the circuit on bread board, as shown. Use $1k\Omega$ for all resistors
- Set PV1 and PV2 to 1V each
- Enable A1, A2 and A3
- Observe the outputs. Verify that the output is the sum of inputs

The output is given by $V_o = \frac{R_1}{R_f}V_1 + \frac{R_2}{R_f}V_2 + \dots$

Discussion The summation can be done using AC waveforms also. Our supply voltage is only 6V. So the output should be kept less than 4V.

- Replace R1 with $10k\Omega$ and feed SQ1 instead of PV1

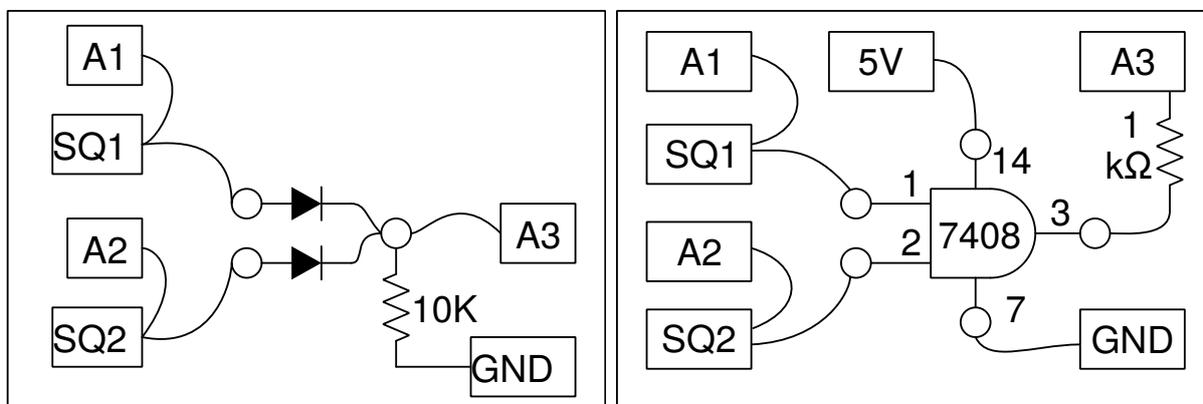
7.11 Logic gates

Logic gate circuits can perform operations like AND, OR etc. They can be implemented using PN junction diodes for demonstration purpose. A wide range of logic gate ICs are also available.

Objective

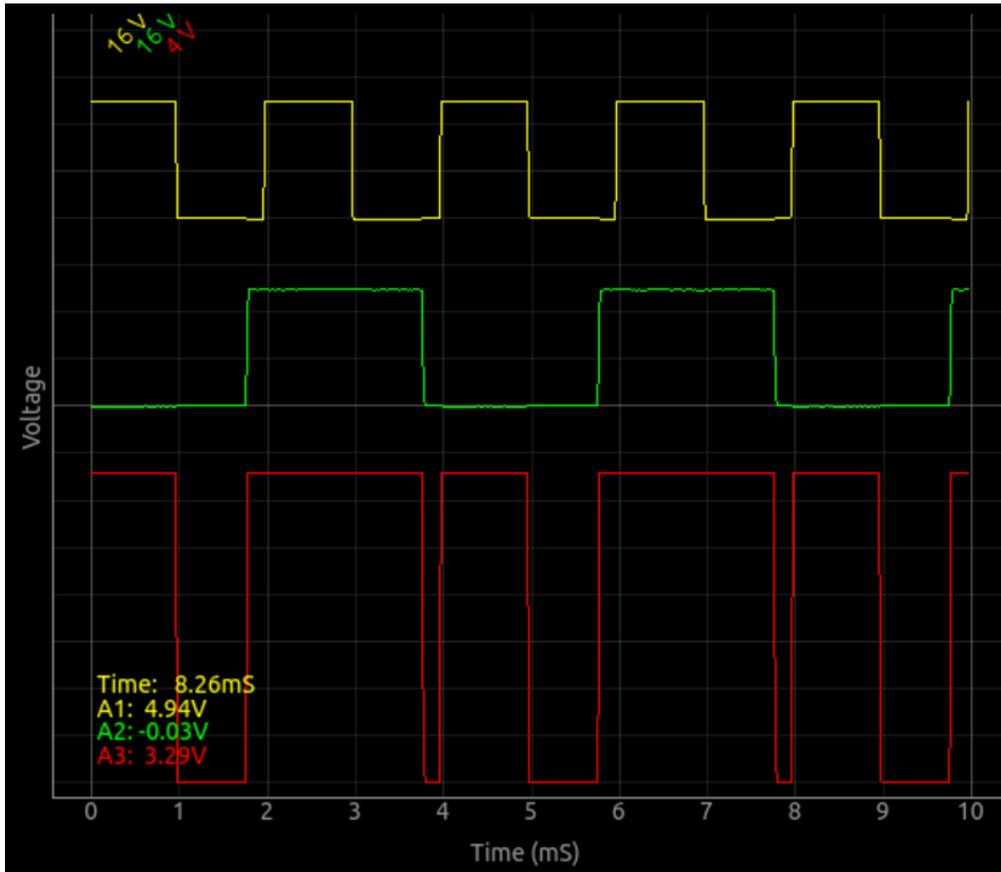
Study of logic gates using SQ1 and PV1 as inputs, using TTL logic ICs 7408 and 7432.

Procedure

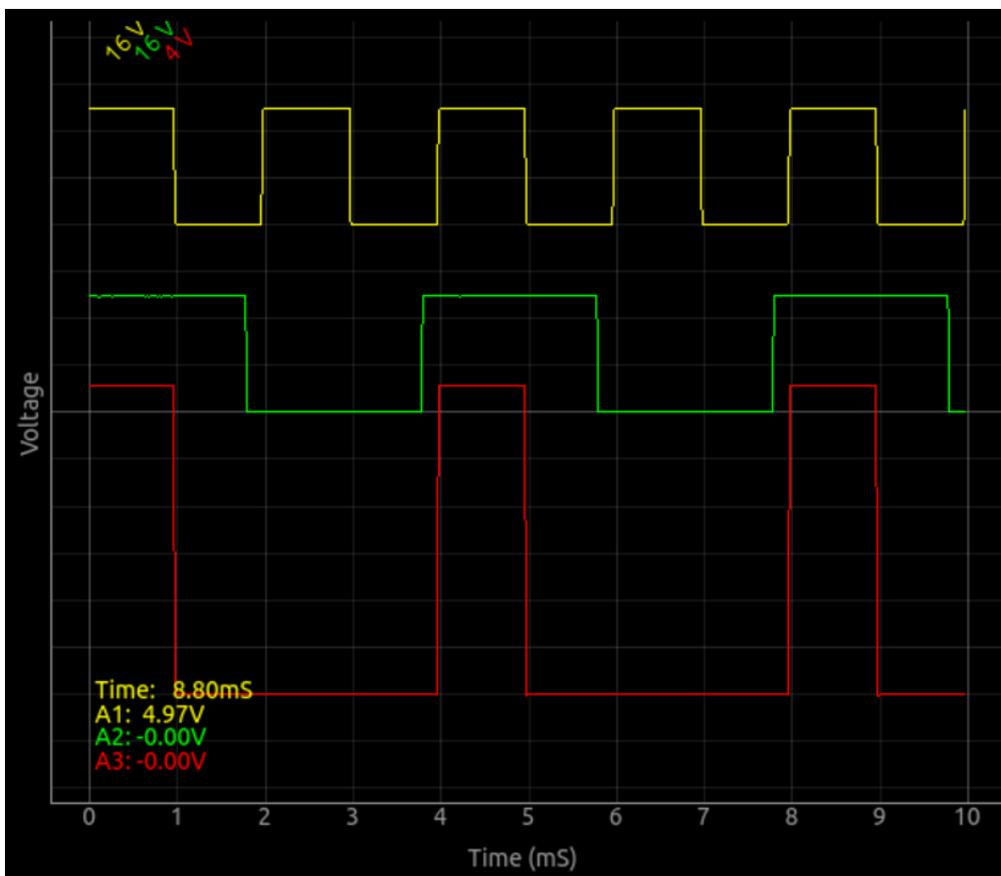


- On a bread board, wire one of the circuits shown above.
- Enable A1, A2 and A3. Set input range on A1 and A2 to 16V
- Set SQ1 to 500 Hz and SQ2 to 1000 Hz
- Select SQ2 from the WG wave shape
- Shift the traces using the vertical sliders on the left, for better view.

The results for the OR gate made using the diodes is shown below.



The results for the AND gate made using the IC 7408 is shown below.



Discussion

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

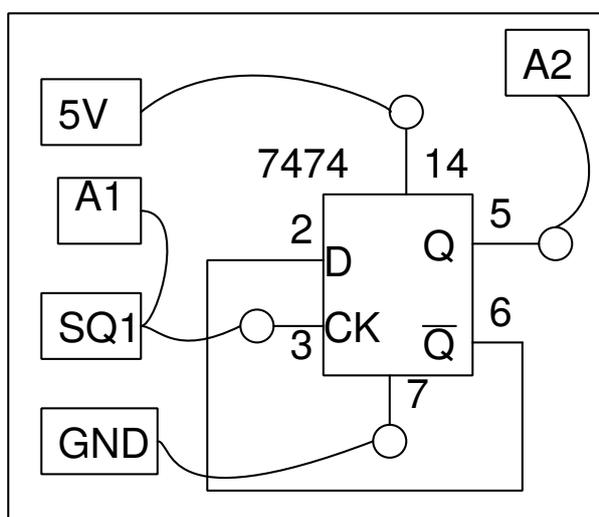
7.12 Clock Divider

A D-type flip-flop can be configured as a clock divider.

Objective

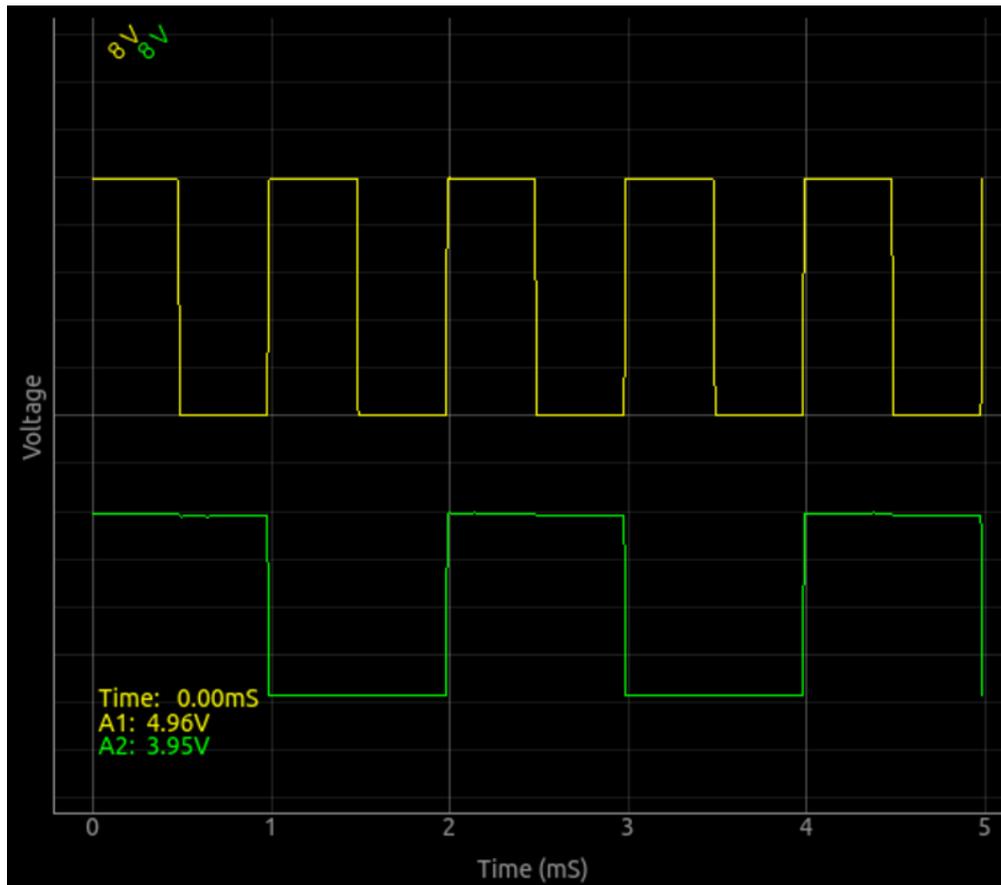
Study of a clock divider, using a D flip-flop (TTL family, 7474).

Procedure



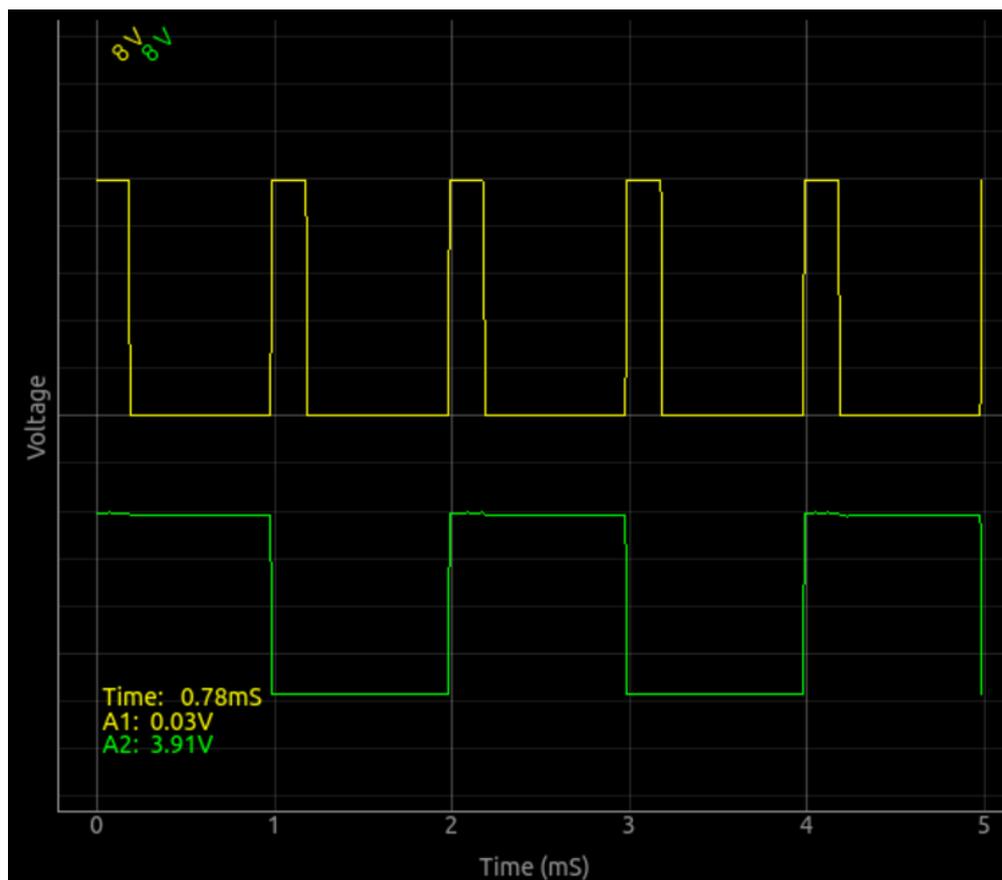
- Enable A1 and A2, set range to 8 volts fullscale
- Set SQ1 to 1000 Hz

For a symmetric input, the input and output waveforms are shown below.



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, as shown below. Every rising edge of the input results in a level change at the output.



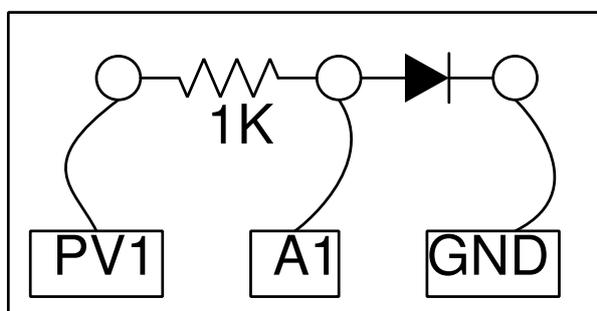
7.13 Diode I-V characteristics

The current through a PN junction varies in a non-linear fashion with the voltage applied across it. The current is very small until the applied voltage exceeds the forward voltage of the diode. This can be visualized by plotting the voltage against current.

Objective

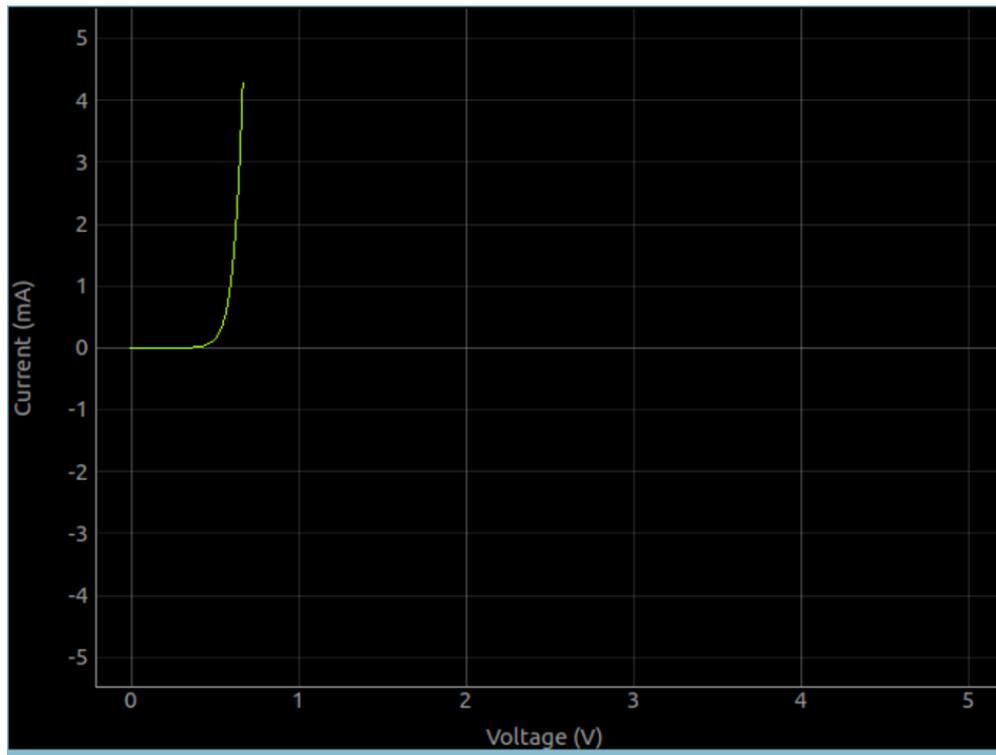
Draw the I-V Characteristic of diode and compare the result with the theory.

Procedure



- Make the connections as shown in the figure. Use 1000Ω resistor
- Click on START to draw the characteristic curve.
- Analyse the data

- Plot the IV of LEDs



It is possible to do this by manually taking the reading, from the oscilloscope GUI. The steps involved are:

- Set PV1 to 100mV
- Read the voltage at A1, voltage across the diode
- Calculate current from $I = \frac{V_{PV1} - V_{A1}}{1000}$
- Increment PV1 by 100mV , up to 5V, and repeat the process
- plot the readings of A1 against I.

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 1N4148 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.

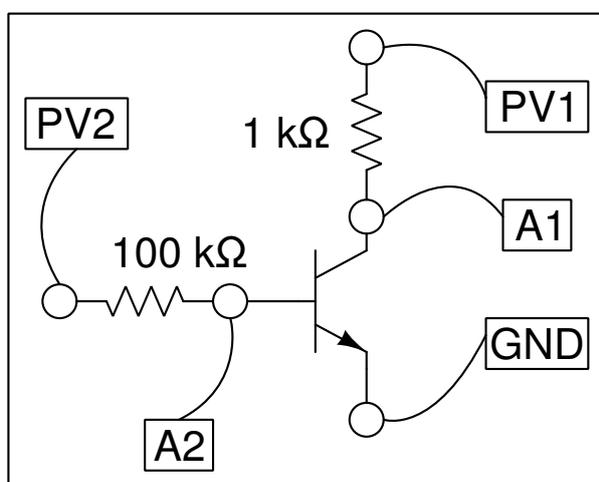
7.14 NPN Transistor Output characteristics (CE)

The basic working principle of a transistor is controlling a bigger current in circuit using a small current in another circuit. Being a three terminal device, one terminal has to be common for the two circuits. The common emitter configuration is very common in most of the applications. We can explore the working of the transistor by studying the relationships between various voltages at different terminals and currents through them.

Objective

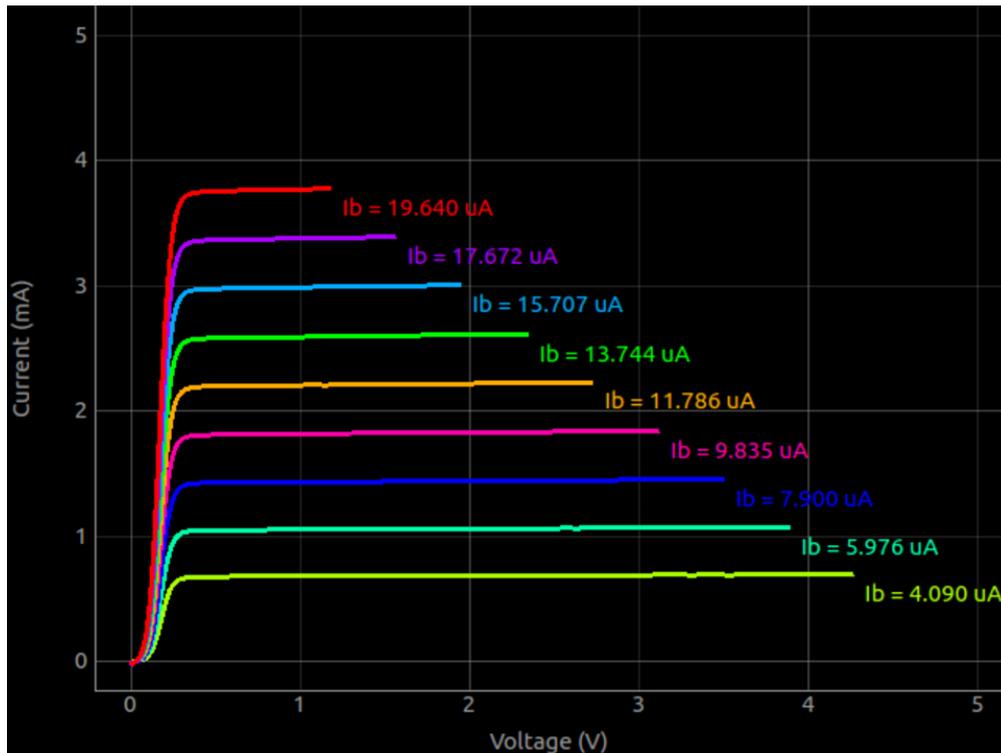
Using an NPN transistor, plot the Collector voltage against the collector current in a common emitter configuration. Repeat it for different base currents. Collector current is calculated from the voltage across the a 1K resistor, in the collector circuit.

Procedure



- Fix the transistor on a bread board and make the connections.
- Set base voltage to the 1 volt and press the START button.
- Repeat for different base currents, increment PV2 by 0.3 volt steps.

The resulting graphs are shown below. This experiments also can be done manually from the oscilloscope GUI, by noting down the readings.



Discussion

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

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

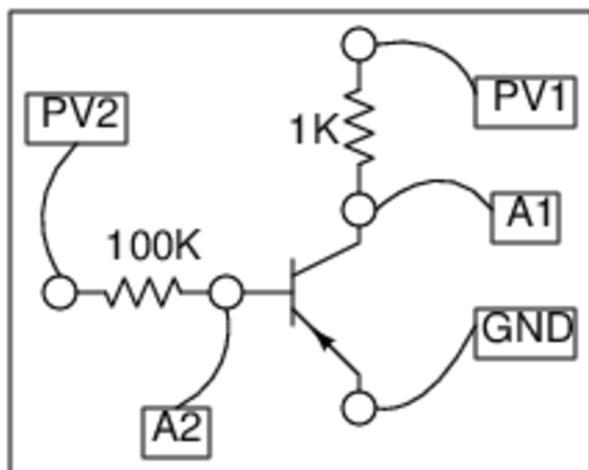
7.15 PNP Transistor Output characteristics (CE)

The basic working principle of a transistor is controlling a bigger current in circuit using a small current in another circuit. Being a three terminal device, one terminal has to be common for the two circuits. The common emitter configuration is very common in most of the applications. We can explore the working of the transistor by studying the relationships between various voltages at different terminals and currents through them.

Objective

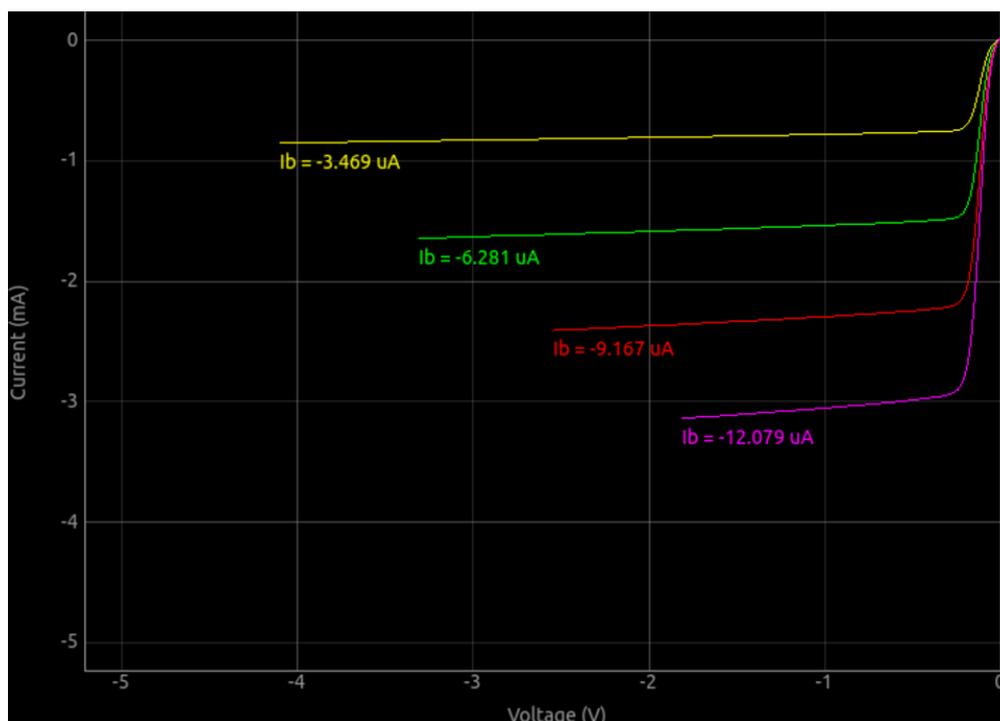
Using a PNP transistor, plot the Collector voltage against the collector current in a common emitter configuration. Repeat it for different base currents. Collector current is calculated from the voltage across the a 1K resistor, in the collector circuit.

Procedure



- Fix the transistor on a bread board and make the connections.
- Set base voltage to the 1 volt and press the START button.
- Repeat for different base currents, increment PV2 by 0.3 volt steps.

The resulting graphs are shown below. This experiments also can be done manually from the oscilloscope GUI, by noting down the readings.



Discussion

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

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

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.

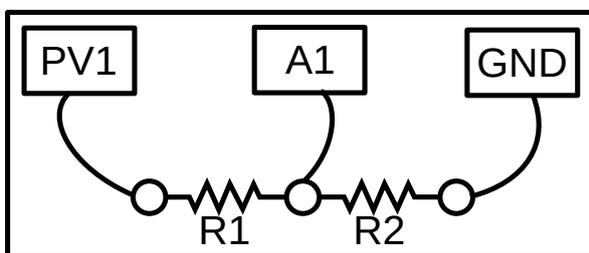
8.1 Plot I-V Curve

Here we plot voltage across an element against the current flowing though it. The current is calculated from the voltage across a series resistance. The graph of resstors is a straiight line, but not for non-linear elemets like the diodes.

Objective

Plot the I-V curve of several circuit elements.

Procedure



- 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 in steps. Voltage across R1 with current is plotted

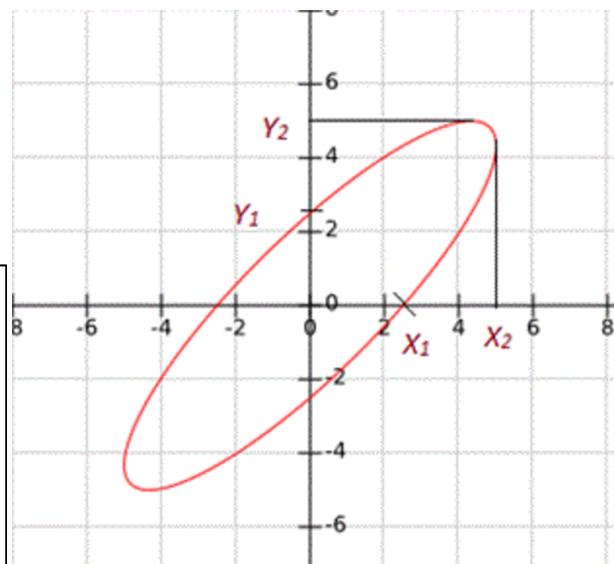
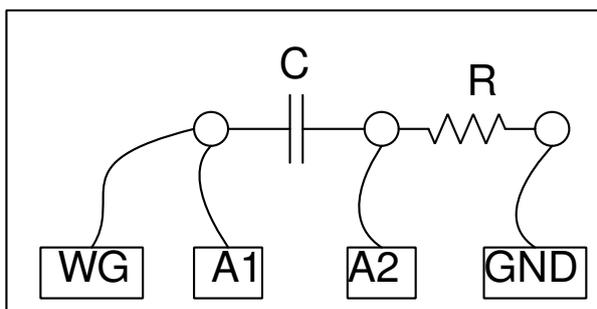


8.2 XY plotting

Oscilloscopes generally plots time X-axis. They also provide and X-Y mode, where one input is plotted on the X-axis. This feature is used for showing lissajou’s figures and also for measuring the phase difference of two input waveforms.

Objective

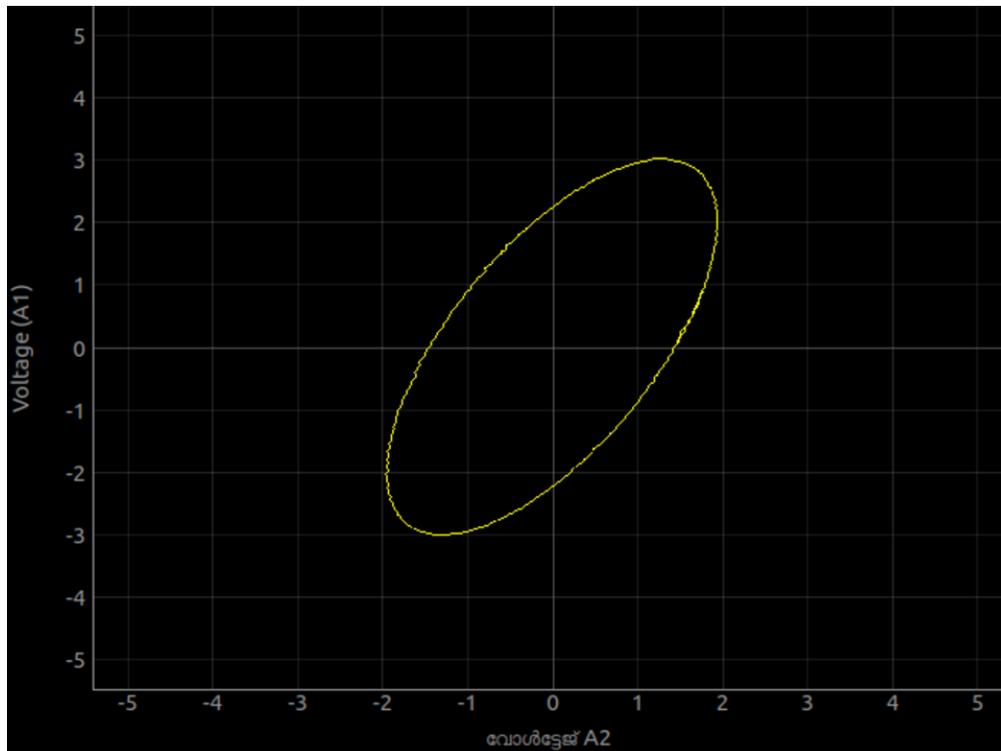
Plot two signals in the X-Y mode and find out their phase difference. The phase difference is be generated by CR circuit.



Procedure

- Make the connections as shown in the figure. Use 1K and 1uF
- Note down the maximum value and the y-intercept

The phase difference is given by the equation $\theta = \sin^{-1}\left(\frac{y_{int}}{y_{max}}\right)$



Discussion There is an option to plot (A1-A2) against A2. In the above circuit, this plots the voltage across the capacitor against the voltage across the resistor.

- Select C = 1uF, R = 1 kOhm and plot (A1-A2) vs A2. A
- Adjust the frequency to make them equal.

8.3 RLC circuits, steady state response

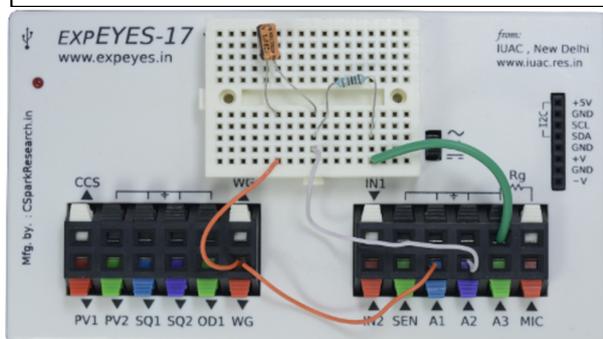
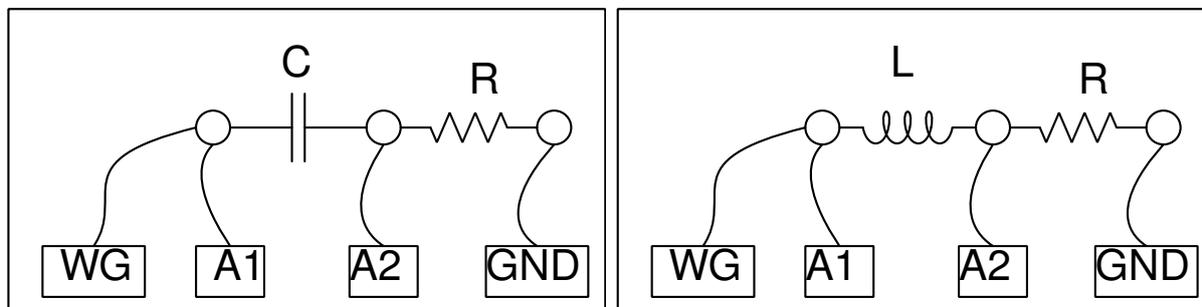
Under a sinusoidal signal, the three basic passive components Resistance, Inductance, and Capacitance shows different phase relationships between the voltage and current. For a resistor the voltage waveforms are “in-phase” with the current. In a pure inductance the voltage waveform “leads” the current by 90 degree. In a pure capacitance the voltage waveform “lags” the current by 90 degree. Using ExpEYES we can study these things experimentally. We will also explore the phase relationships between voltages at various points in the circuit. Three different cases RC, RL and RLC will be studied. The voltage and phase values at series resonance condition will be explored. An Impedance calculator is provided on the right side of the GUI, to compare the measure values with calculations.

Objective

Measure the amplitude and phase values in an RC circuit. The phase difference of voltages at the two ends of the capacitor is given by

$$\theta = \tan^{-1} \left(\frac{X_c}{R} \right) \text{ where the capacitive reactance } X_c = \frac{1}{2\pi fC}$$

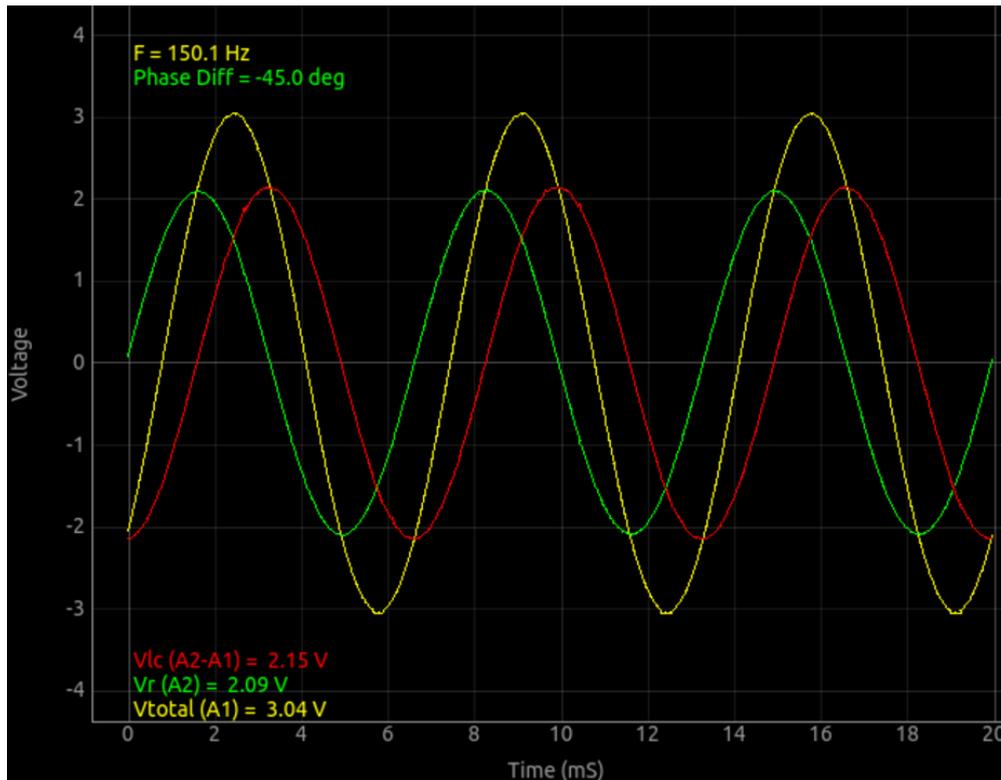
Procedure



- Use a bread board as shown in the figure.
- Fix a 1uF capacitor and 1kΩ resistor.
- Make the connections as shown in the first schematic.
- Repeat the measurements, at different frequencies.

The resulting waveforms for the RC circuit are shown below. The green trace (voltage at A2) is the voltage waveform across the resistance. This can be considered as the current waveform since voltage and current are in phase across a resistance. The red trace is the voltage across the capacitance, and it can be seen that the current waveform is leading by 90 deg.

The phase difference across the capacitor is displayed at the top corner. This can be compared with the calculated values.



The next step is to study the RL circuit. Here you will observe the current waveform lagging behind the voltage across the inductor.

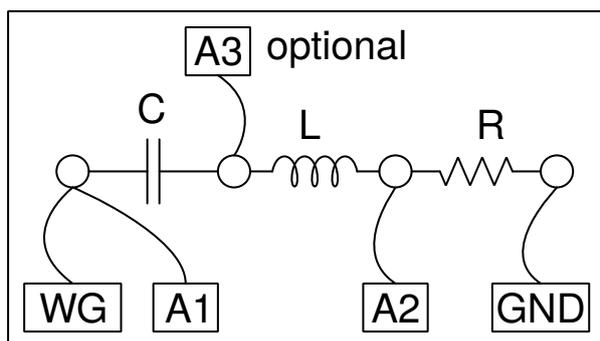
- Replace the capacitor with a 10mH inductor.
- Set frequency to 4000 Hz to get a larger phase shift, because the inductance used is very small.

8.4 LCR and Series Resonance

When both inductor and capacitor are present the phase shift across LC is given by

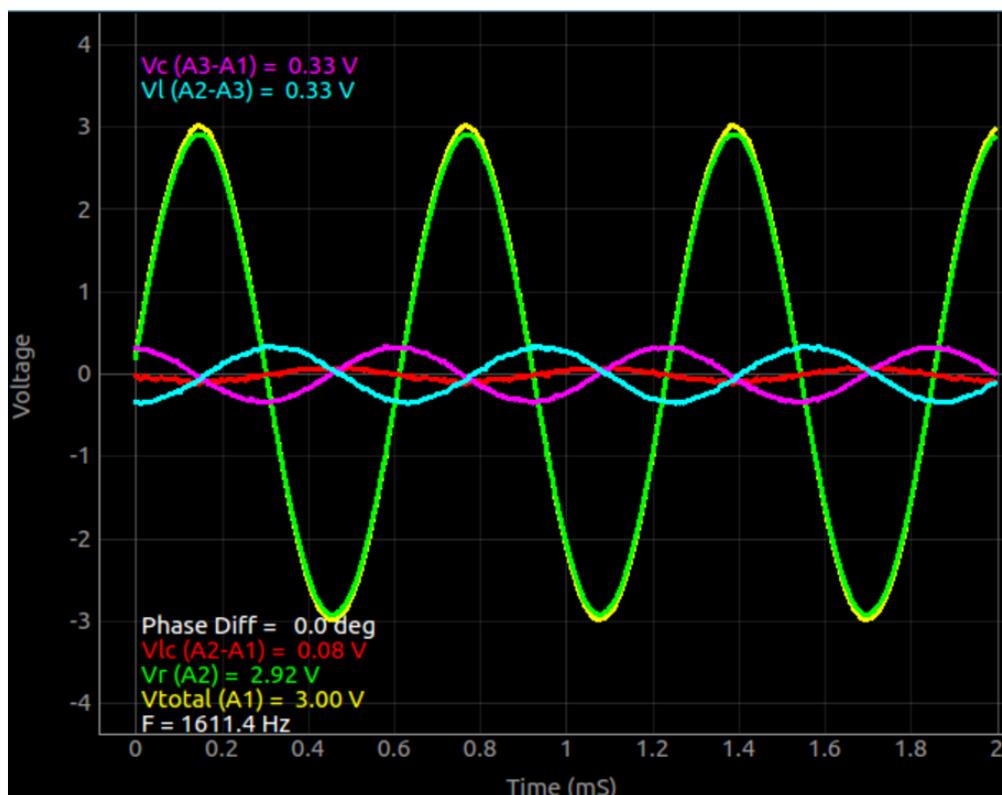
$$\theta = \tan^{-1} \left(\frac{X_c - X_L}{R} \right) \text{ where } X_c = \frac{1}{2\pi f C} \text{ and } X_L = 2\pi f L$$

The inductive reactance increases with frequency while the capacitive reactance decreases with it. At some frequency they will become equal and the phase shift across LC will become zero. The total voltage across LC also will become zero. This condition is called series resonance.



- Make the connections using L, C and R
- Connect A1, A2 and A3 as shown in the figure.
- Note down the amplitude and phase measurements, in each case
- For RLC series circuit, the junction of L and C is monitored by A3
- For resonance select $C = 1 \mu F$, $L = 10 mH$
- Set frequency to $f = 1600 Hz$, adjust it to make phase shift zero

The resonance frequency for the given L and C is 1591.5 hz. We set it nearby to start with. The total voltage across L and C together goes almost to zero. It is not exactly zero because of the ohmic resistance of the coil. The input A3 is connected between L and C, so that the individual voltage drop across L and C can be displayed. It can be see that they are equal and out of phase, as shown below.



Discussion

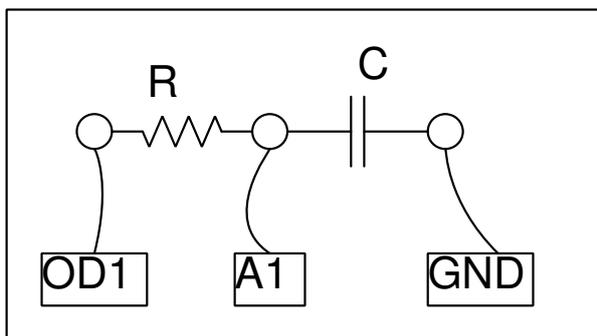
This experiment can be used for measuring the values of unknown capacitors or inductors. Make an RL or RC circuit with a known resistance and measure the phase shift at different frequencies. The L or C values can be calculated using $C = \frac{1}{2\pi f R \tan \theta}$ and $L = \frac{R \tan \theta}{2\pi f}$

8.5 Transient Response of RC circuits

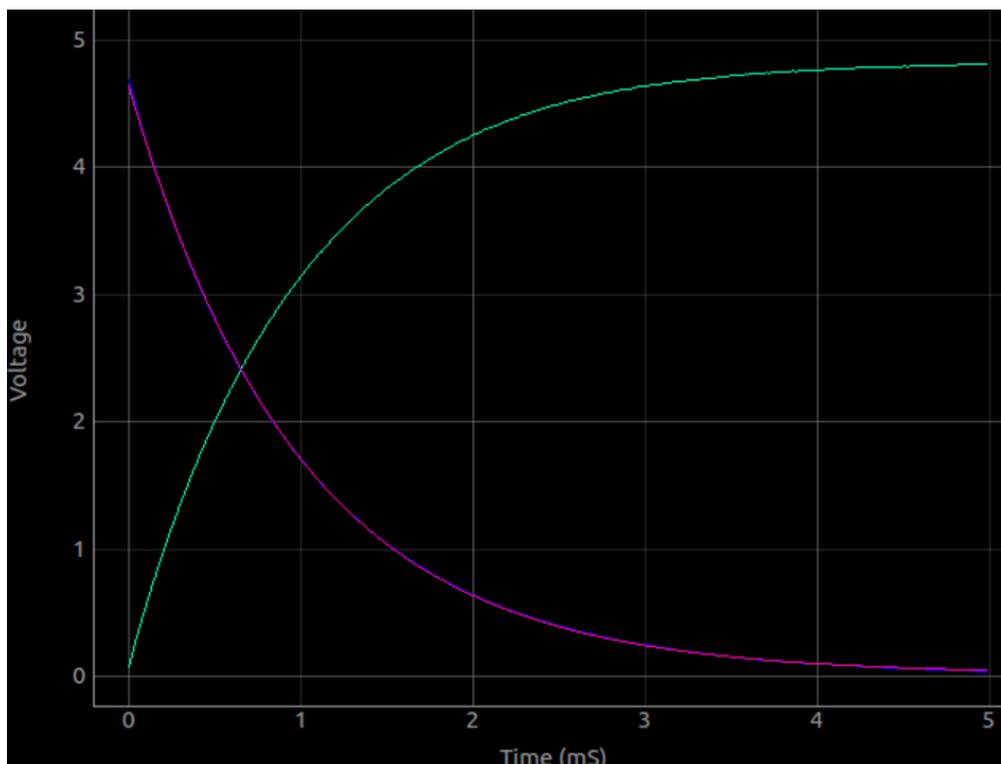
Objective

Plot the voltage across a capacitor, when it is charged by applying a voltage step through a resistor. Extract the value of RC from the graph by curve fitting.

Procedure



- Make connections as shown in the figure
- 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 $V(t) = V_0 \times e^{-t/RC}$, 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 through 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 $V = (I/C) \times t$.

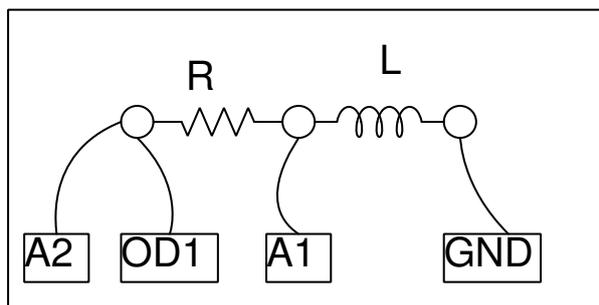
For detailed description of the theory of RC circuits visit [KhanAcademy](#)

8.6 Transient Response of RL circuits

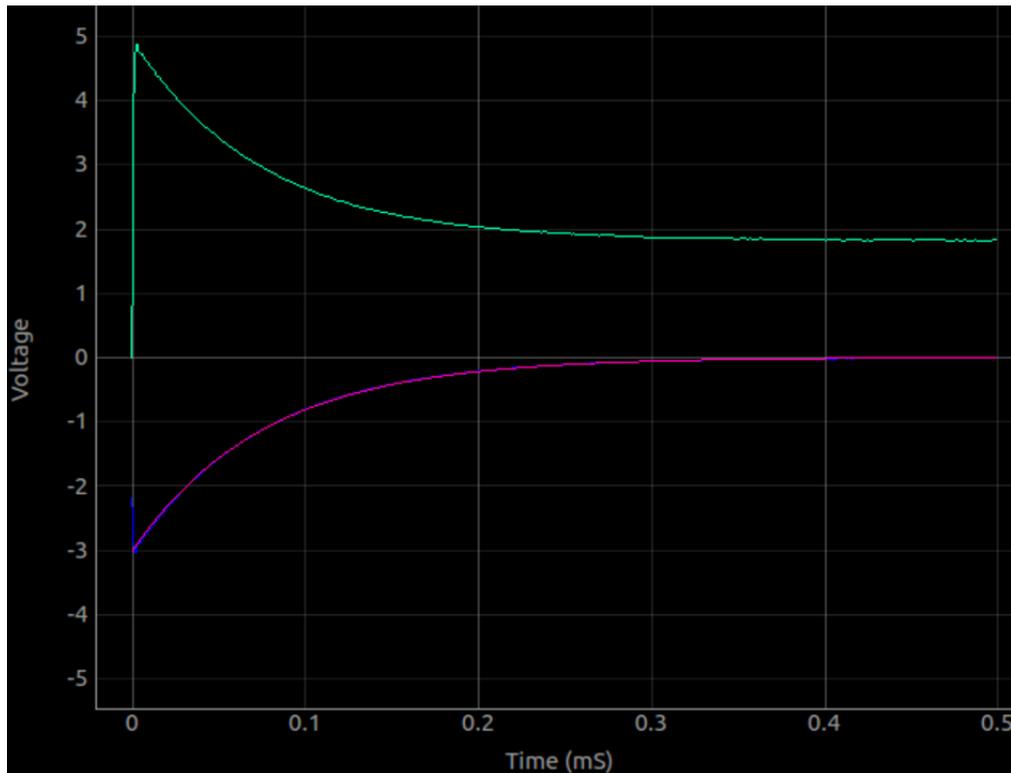
In an RL circuit $V = RI + L(dI/dt)$ and solving this will give $I = I_0 \times e^{-(R/L)t}$. The coefficient of the exponential term R/L can be extracted from the graph of voltage across the inductor. The resistance of the inductor coil should be included in the calculations, $R = R_{ext} + R^*_L$.

Objective Capture and analyse the voltage across an inductor when a voltage step is applied to it via a series resistor. By measuring the voltage across the inductor as a function of time, we can extract the R/L ratio.

Procedure



- Use the 3000 turn coil as inductor
- 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. From this the program calculates the value of inductance and displays it.

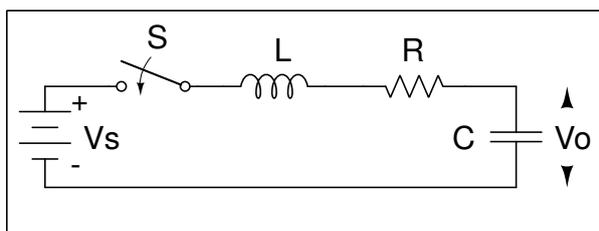
The applied voltages are above zero, but the graph went to negative voltages. Why ?

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.

For detailed description of the theory of RC circuits visit [KhanAcademy](#)

8.7 Transient response of LCR circuits



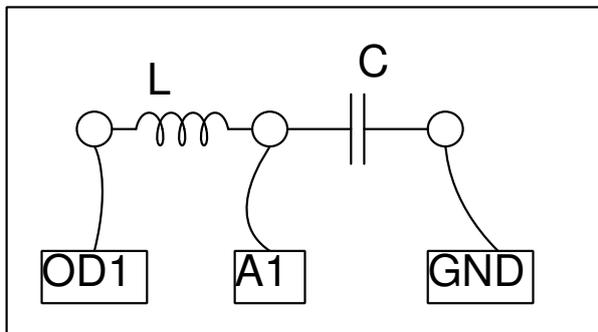
Closing the switch in the circuit shown above will change the voltage across the capacitor. Depending on the relative values of L, C and R , the voltage may show oscillations or it may

change in an exponential manner.

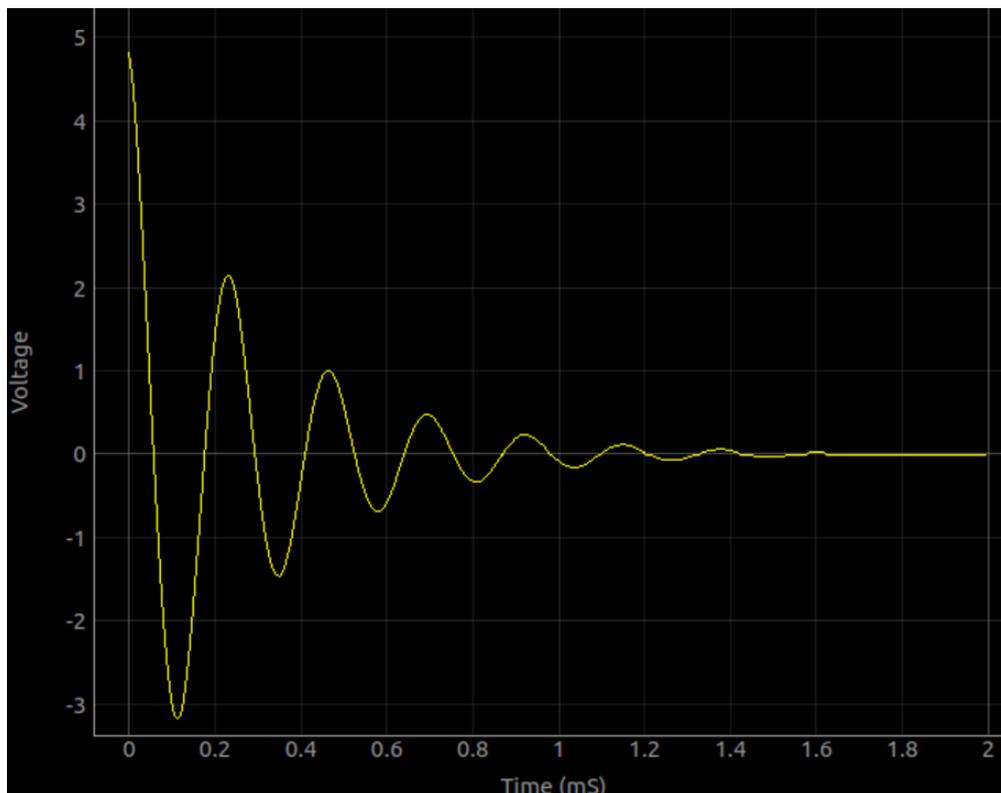
Objective

Apply a step voltage to an LCR circuit. Capture and analyse the resulting voltage across the capacitor.

Procedure



- Use the 3000 turn coil and $0.1 \mu F$ capacitor
- Click on 5->0V STEP. Adjust x-axis and repeat if required.
- FIT the graph to find the resonant frequency & Damping factor.
- Repeat with increased capacitance and resistance values.



The values chosen are for an underdamped response. The coil has an inductance more than 100mH and a resistance of around 500Ω . To increase damping, you may add a series resistor, or use a larger value capacitor.

Discussion

Explore the oscillatory nature of L and C in series. Resonant frequency of series LC circuit is given by $f_0 = 1/(2\pi\sqrt{LC})$. The damping factor is $R/(2\sqrt{LC})$, and it is equal to 1 for critical damping. Depending upon the value of C/L and R, the response could be under-damped, critically-damped or over-damped.

We have used the 3000 turn coil and a 0,1 μF capacitor. The voltage across the capacitor after a 5 to 0V step is shown in figure. The measured resonant frequency tallies with $f = 1/(2\pi\sqrt{LC})$, within the component tolerance values.

http://en.wikiversity.org/wiki/RLC_circuit.

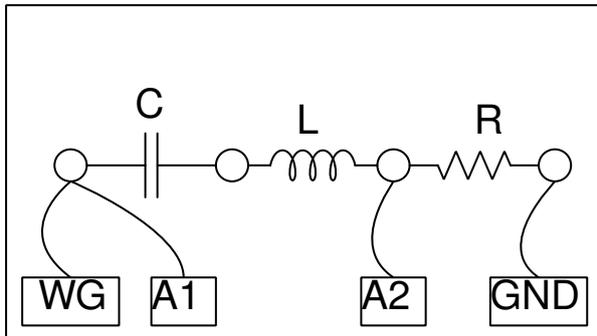
8.8 Frequency response of filter circuit

Transmission of electrical signals through a filter circuit depends on the frequency. The ratio of the output voltage to the input is called gain. Plotting gain as a function of frequency gives the frequency response curve of a filter circuit.

Objective

Draw the frequency response curve of a filter circuit, with $L = 10\mu H, C = 1\mu F, R = 100$. Expected resonant frequency of the bandpass filter $f_0 = \frac{1}{2\pi\sqrt{LC}}$

Procedure



- Assemble the LCR filter circuit on a bread board
- Connect WG and A1 to the filter circuit input
- A2 to the output

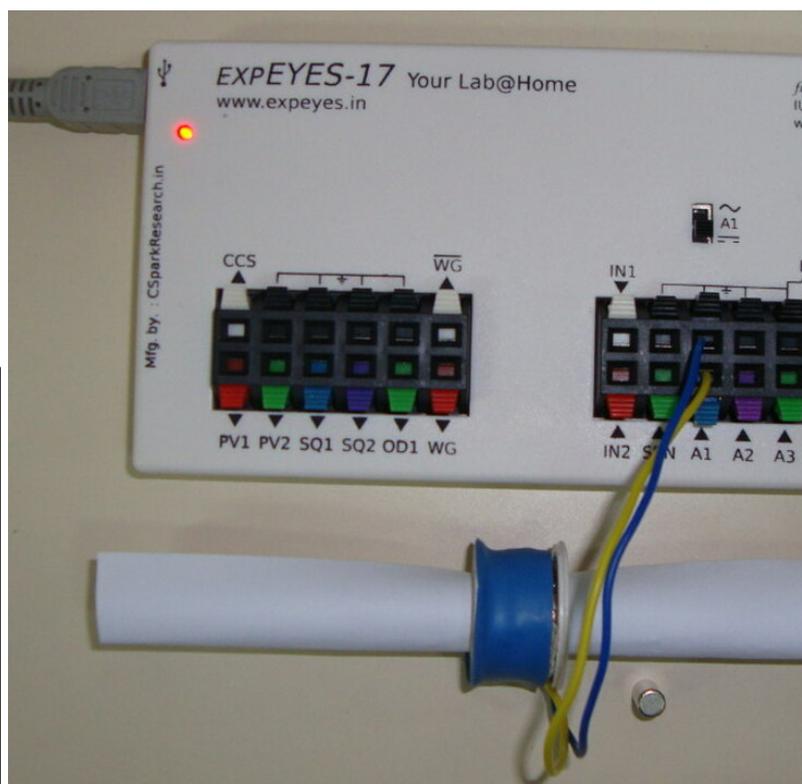
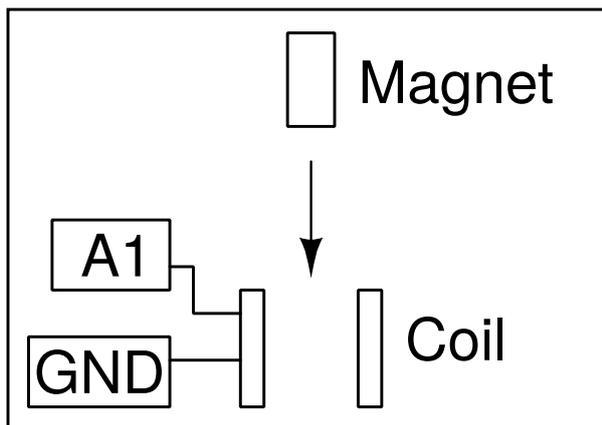
8.9 Electromagnetic induction

A voltage is induced across a conductor kept in a changing magnetic field. This can be demonstrated using a coil and a moving magnet.

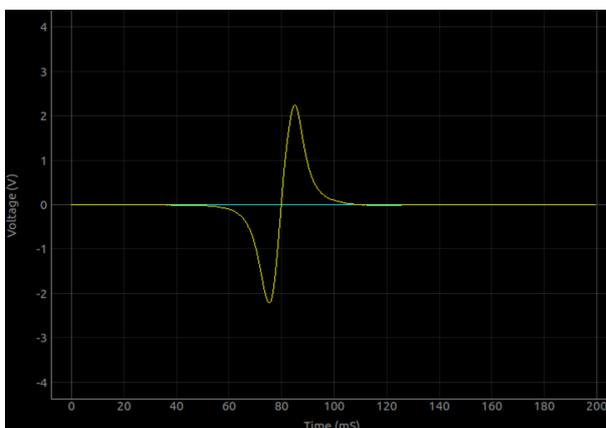
Objective

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.

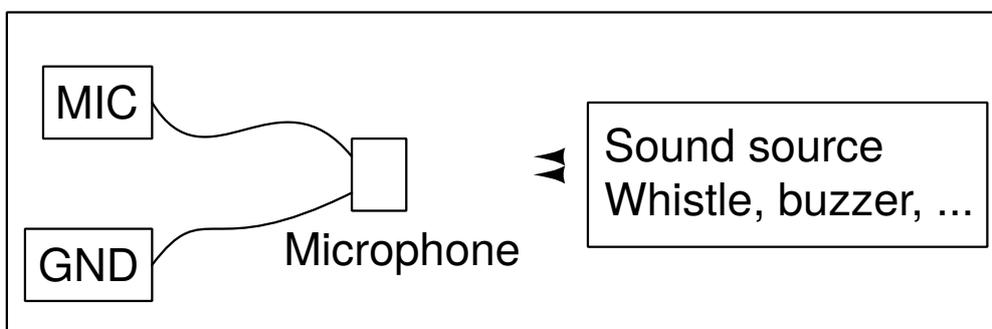
The second peak is bigger than the first peak. Why ? Where will be the magnet at the zero crossing of the induced voltage? Drop the magnet from different heights and plot the voltage vs square root of the height.

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

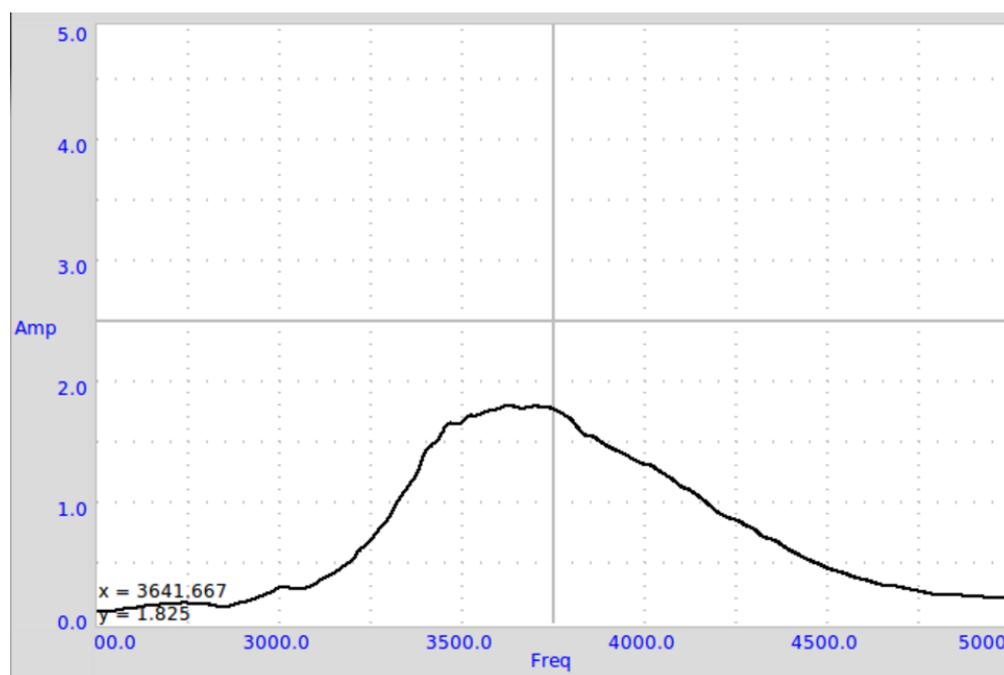
Objective

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

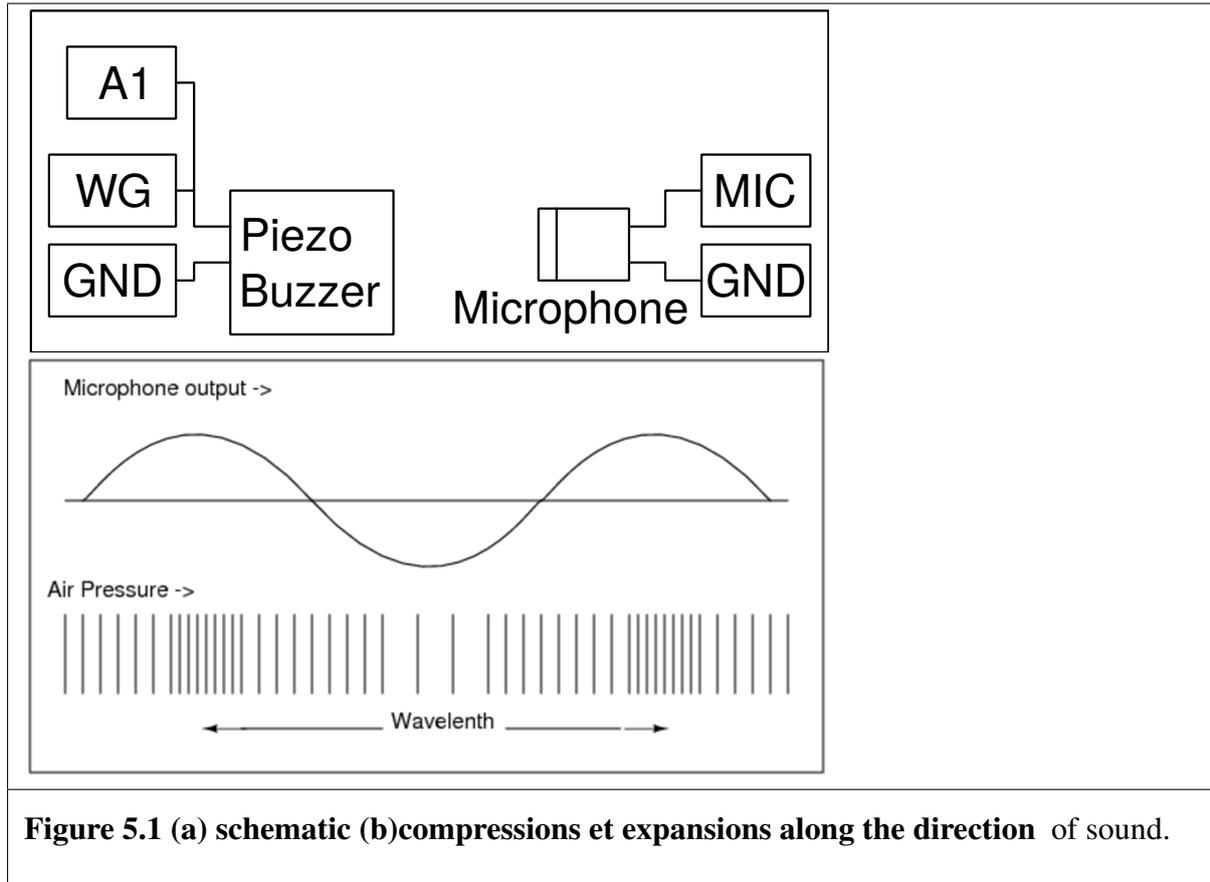
The Frequency Vs Amplitude plot is shown in figure. The amplitude is maximum around 3500 Hz.

9.2 Velocity of sound

Objective

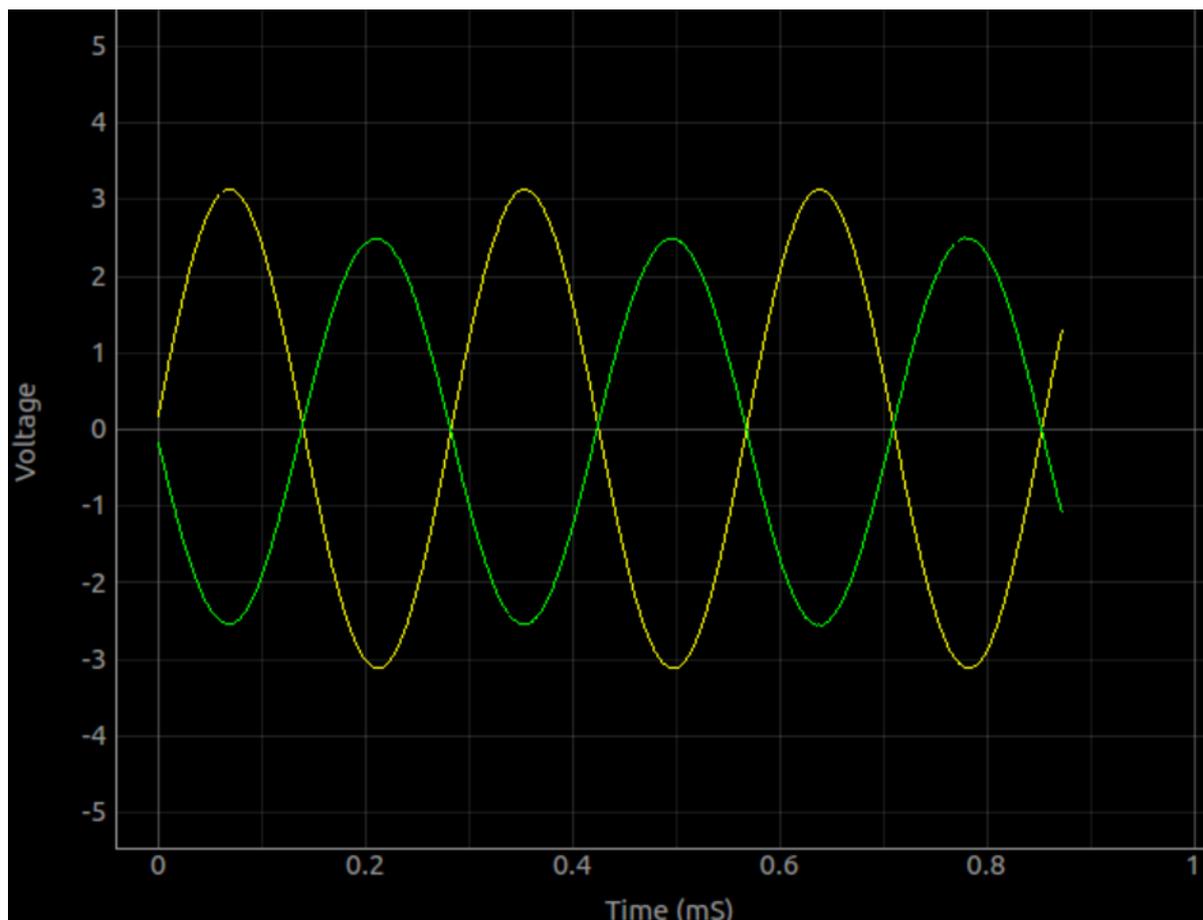
Calculate the velocity of sound by measuring the pressure variation with distance. Sound travels as a series of compressions and rarefactions. Figure (a) shows the High and Low pressure regions along the direction of travel, along with output of a pressure sensor at corresponding positions.

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.



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

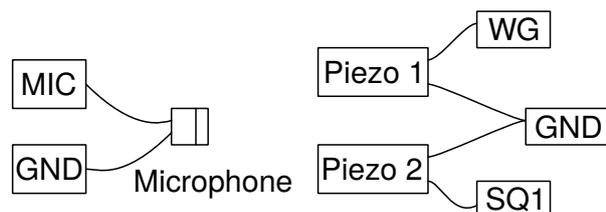
At 3500 Hz, for a 2 cm change in distance the phase changed from 176 to 102. Using the equation, $v = f \times (360 \times \Delta D)/X$, $v = 3500 \times (360 \times 2)/(176 - 102) = 34054 \text{ cm} \cdot \text{s}^{-1}$. It is important to keep the mic and the Piezo disc on the same axis, for accurate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

9.3 Sound beats

Beats are produced when two sinusoidal sound waves of equal amplitude and very nearly equal frequencies mix. The amplitude or intensity of the combined signal would rise and fall at a rate equal to the difference between the two original frequencies. $f_b = f_1 - f_2$.

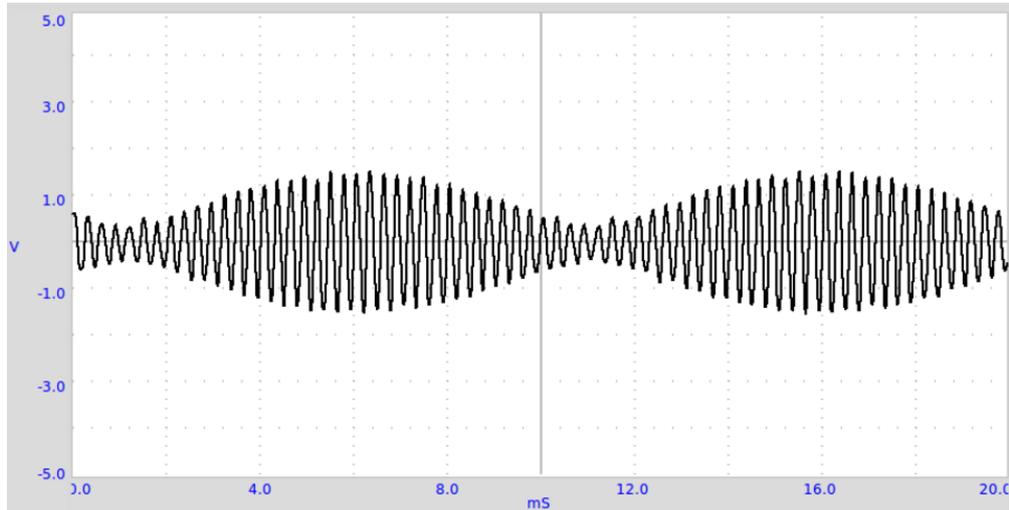
Objective

Study the beats produced by two Piezo buzzers are excited by two nearby frequencies. The sound is captured by a 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 shows the two different frequency components.

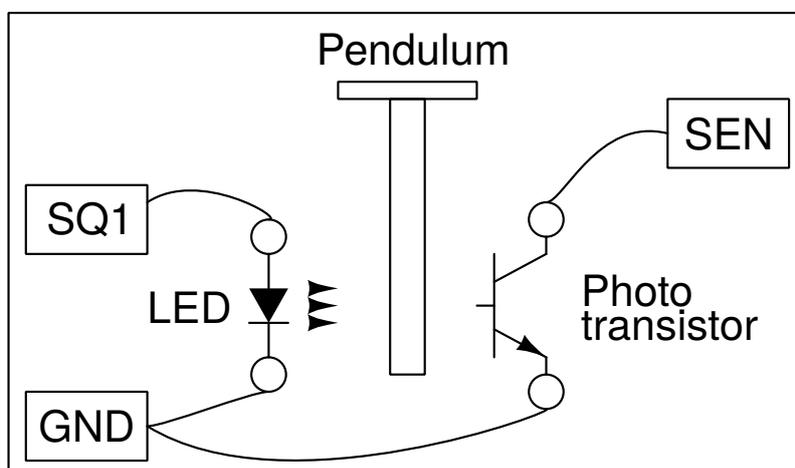
Resonance phenomena is studied using a driven pendulum. Value of acceleration due to gravity is measured using a pendulum.

10.1 Acceleration due to gravity using Rod pendulum

Period of oscillations of a pendulum depends on its length and the value of 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.

Objective

Measure the period of oscillations of a rod pendulum using a photo-transistor and LED arrangement. 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.

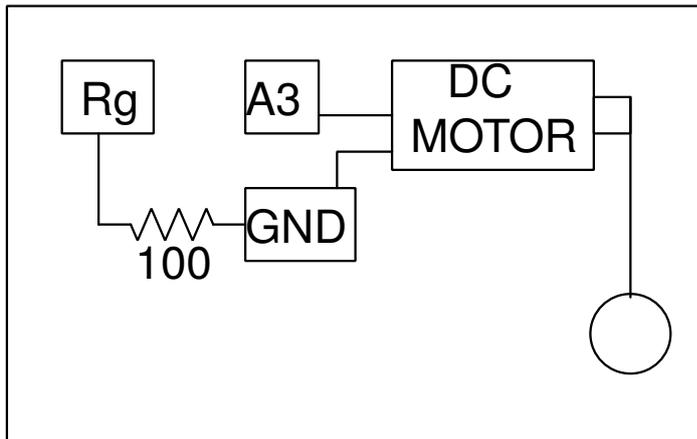
Discussion

The period of oscillation can be measured with an accuracy of 100 microseconds. However Mainly there are two sources of error. 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. Also the calculations assume that the pendulum must be exactly vertical in the resting position, which has to be ensured.

10.2 Digitizing Pendulum Oscillations

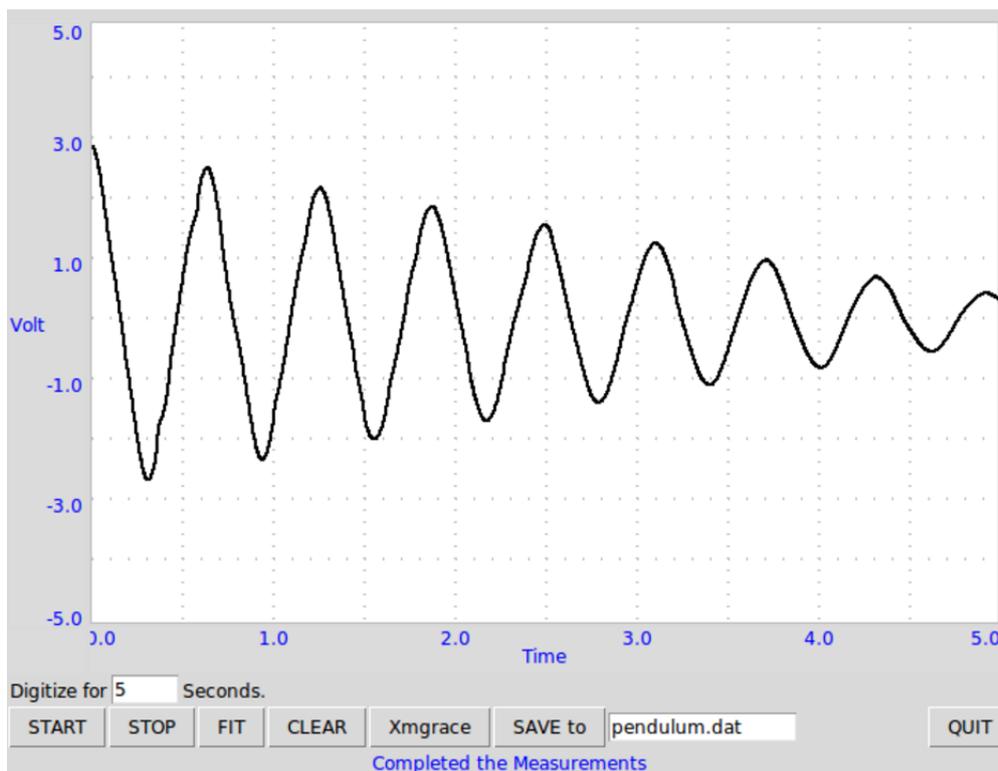
The nature of oscillations of a pendulum can be studied by digitizing the angular displacement as a function of time. By fitting this data with a sinusoid, the time period can be extracted. However, it is much cheaper to digitize the velocity of the pendulum, using a simple DVD motor. Attaching the pendulum to the axis of the motor and oscillating it induces voltage, proportional to the angular velocity of the pendulum.

Objective Digitize the angular velocity of a rod pendulum and calculate the value of ‘g’ from it.



Procedure

- Attach some sort of rigid pendulum to the axis of the motor.
- Connect the motor between A3 and GND
- Connect 100 Ω 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

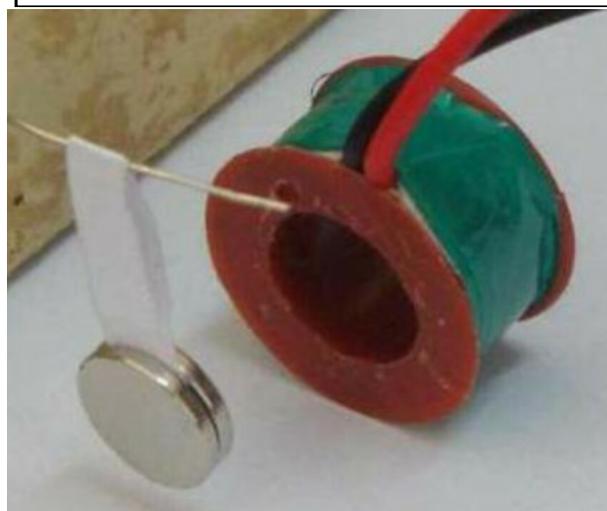
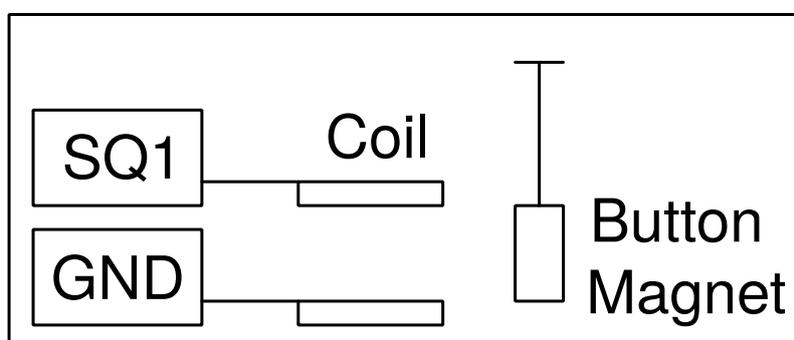
The amplitude of an oscillating body becomes maximum when the frequency of the driving force is equal to its natural resonant frequency. A driven pendulum is a very simple example of such a system.

Objective

Demonstrate the resonance of a driven pendulum. The pendulum is made by two button shaped magnets suspended on a piece of paper. A coil powered from SQ1 provides the driving force.

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
- Adjust 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.

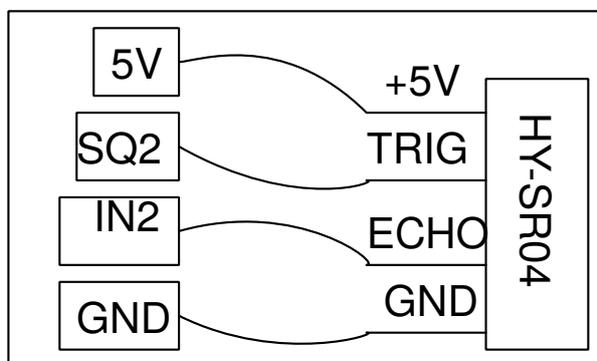
10.4 Distance Measurement, Echo module

Distance can be measured by sending a burst of sound to the target and measuring the time interval to receive the echo. There are low cost electronics modules available for such applications. Hy-SR04 is such a module, that uses a 40kHz Piezo transmitter and receiver combination.

Objective

Measure distance by measuring the time taken by 40 kHz pulse train to echo from a hard surface, using HY-SR04

Procedure



- Make the connections as shown in the schematic
- Keep a flat surface around 10 cm from the SR04 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 measured as function of time, enabling to calculate velocity, acceleration etc.

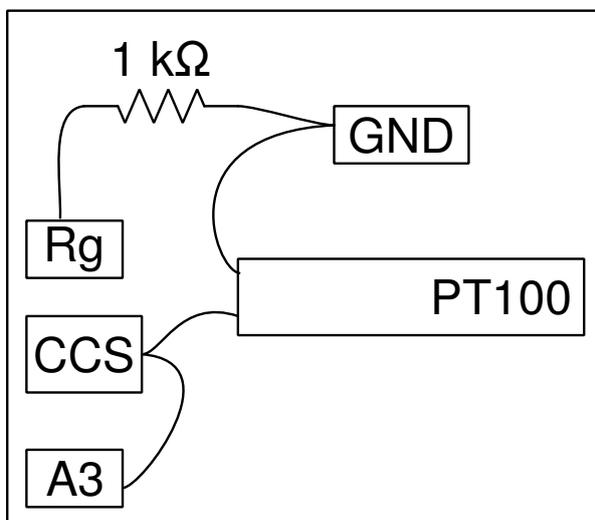
11.1 Temperature measurement using PT100

The Resistance Temperature Detector (RTD) is a temperature sensing device whose resistance changes with temperature, typically built from platinum. To measure the resistance across an RTD, apply a constant current, measure the resulting voltage, and determine the RTD resistance. The Pt100 is a popular RTD with a resistance of 100 ohms at 0°C. 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 $B = -5,775 \times 10^{-7}$.

Objective

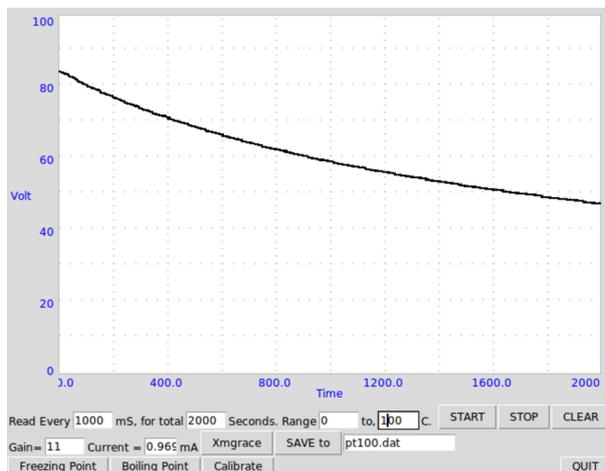
Record the temperature of a liquid by using a Pt100.

Procedure



- Make the connections as shown in the figure.
- Enter the Gain, Offset error and the Current from CCS
- Select the temperature range and time intervals
- Select the required parameters and press START

Cooling curve of water is shown in the figure below.



Discussion

The accuracy of the measurements can be increased by the following steps.

Measure the actual current from CCS and enter it in the GUI, it could be slightly different from 1.1mA. The input to A3 is amplified 11 times by connecting 1 k Ω resistor from Rg to Ground. The gain and offset should be measured and entered in to the program.

11.2 Data Logger

Data logger is a program that allows the user to record the voltage inputs as a function of time.

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

11.3 Advanced Data Logger

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

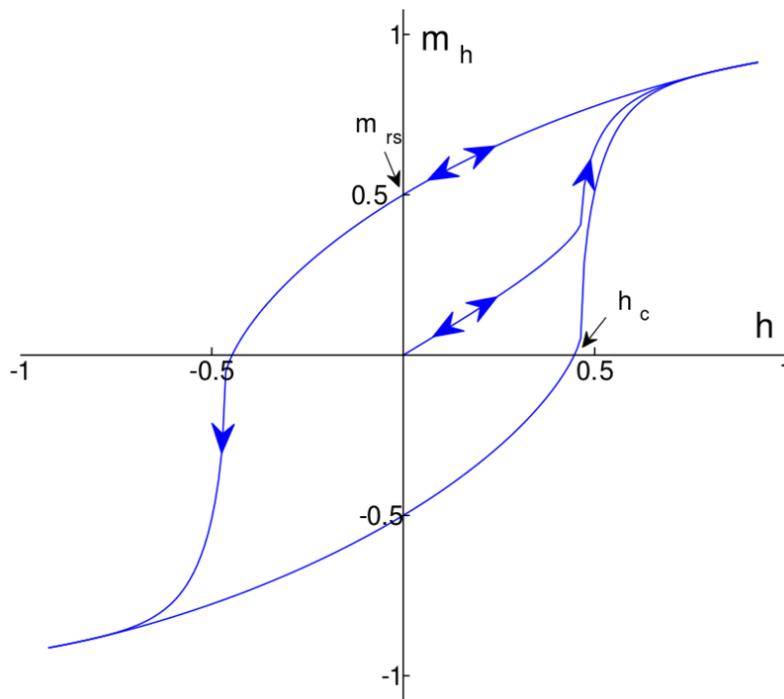
- Voltmeter: A1,A2,A3,IN1,SEN,AN8,CCS
- Capacitance
- Resistance
- Oscilloscope
 - * Extracted frequency,phase,amplitude or offset using a sine fit
 - * Difference in phase between A1(Any analog input) and A2. Also, ratio of amplitudes.
- Frequency on IN2
- Any connected I2C sensor (Magnetometer, accelerometer,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

12.1 B-H Curve

The magnetic field density (H) around a current carrying coil depends on the coil parameters and the current. The magnetic flux density (B) depends on H and the magnetic permeability of the medium. B and H They are related by the equation $B = \mu H$ where μ is the permeability.

Permeability is not a constant for ferromagnetic materials, like iron. If H is increased B increases and saturated at some point. When we reduce H, B does not follow the same path, as shown in the figure.



Objective

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

Procedure

- 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 hysteresis for various different materials.

12.2 Light Sensor Logger

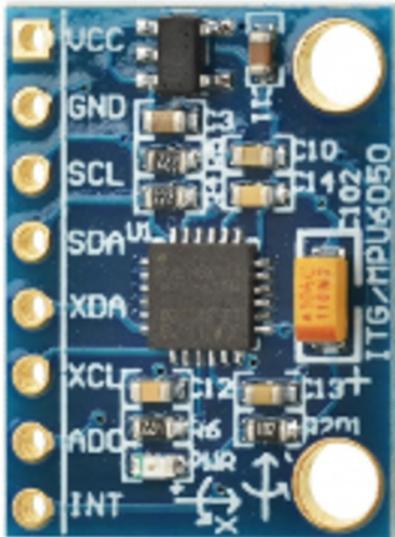
12.2.1 Schematic

- 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

Scan for the I2C sensors connected. The detected sensors can be accessed.

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.

13.1 Programming ExpEYES using 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.

13.1.1 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.

13.1.2 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.

13.1.3 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.

13.1.4 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')
```

13.1.5 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()
```

13.1.6 get_capacitance()

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

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

13.1.7 get_version()

Returns the version number of the Firmware

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

13.1.8 get_temperature()

Returns the temperature of the processor inside eyes17

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

13.1.9 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')
```

13.1.10 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

13.1.11 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.

13.1.12 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

13.1.13 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)
```

13.1.14 set_sqr2(frequency)

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

13.1.15 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
```

13.1.16 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')
```

13.1.17 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')
```

13.1.18 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.

13.1.19 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

13.1.20 select_range(channel, range)

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

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

13.1.21 select_range(channel, range)

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

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

13.1.22 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).

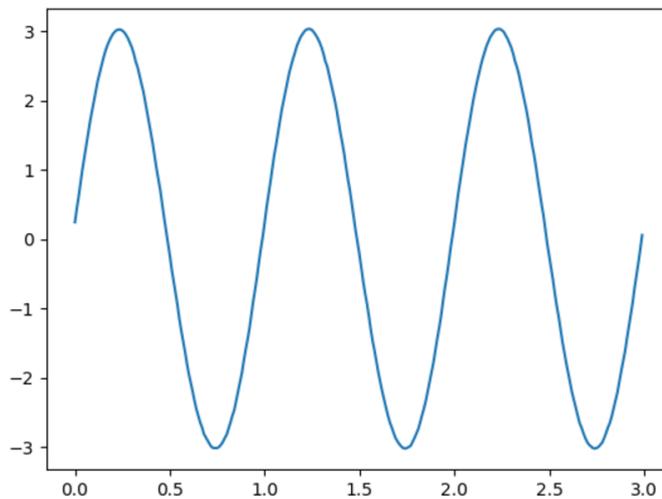
```
print p.capture1('A1', 5, 5)
```

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.



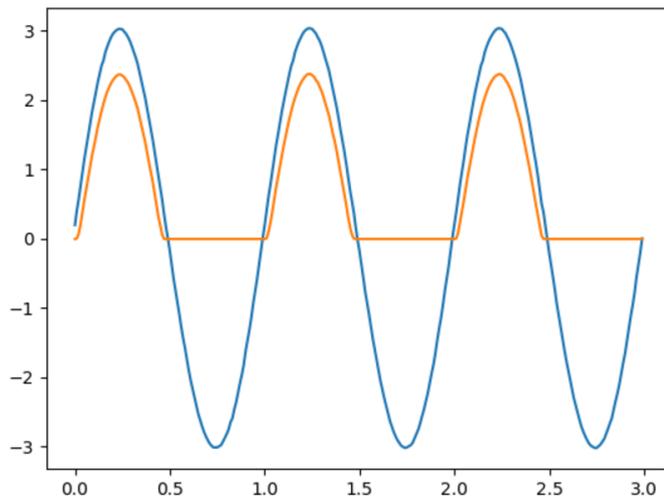
13.1.23 capture2(Number of samples, time interval)

Digitizes the inputs A1 and A2 together. The number of samples could be upto 10000. The time gap between two consecutive samples is 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.

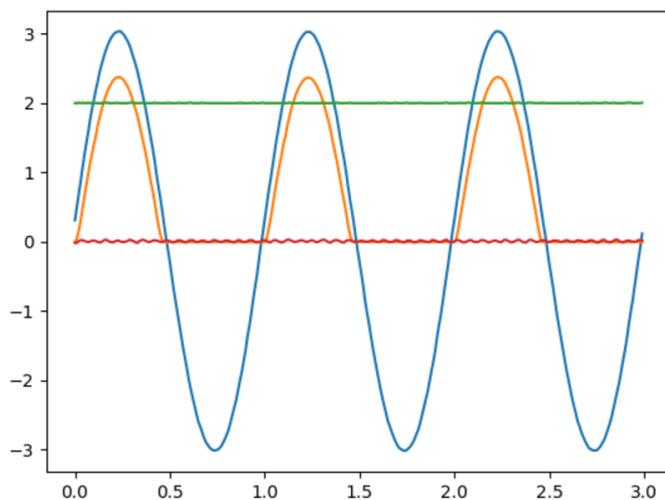


13.1.24 capture4(Number of samples, time interval)

Digitizes the inputs A1,A2,A3 and MIC together. The number of samples could be upto 10000. The time gap between two consecutive samples is 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)
plot(res[4],res[5])      # A3
plot(res[6],res[7])     # MIC
show()
```



13.1.25 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 triangular wave table and generates_
→100Hz
x,y = p.capture1('A1', 500,50)
plot(x,y)
show()
```

13.1.26 load_equation(function, span)

Makes the wave table using the equation. 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)
```

(continues on next page)

(continued from previous page)

```
x,y = p.capture1('A1', 500,10)
plot(x,y)
show()
```

13.1.27 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 up to 255, tracing a triangular 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()
```

