Ryerson University
Department of Electrical, Computer, & Biomedical Engineering
Faculty of Engineering & Architectural Science

ELE 202
Laboratory #5

Introduction to Function Generators, Oscilloscopes, & AC Measurements

1.0 INTRODUCTION

The lab exercises up to now focused on Direct Current (DC) signals whose values are constant over time. The DC power supply served to produce the DC input signals, and the DMM multimeter was used to measure the DC voltages and currents. In addition to DC signals, **Alternating Current (AC)** signals are an important part of circuit theory and design. AC voltage signals vary with time and are generally *periodic*, meaning they repeat with a specific time interval. There are many different AC waveforms but the three most common ones are *sine*, *square* and *triangle* waveforms. However, the widely used time-varying waveform is the ***sine-wave*** whose amplitude varies sinusoidally with time, as shown in **Figure 1.0a**.

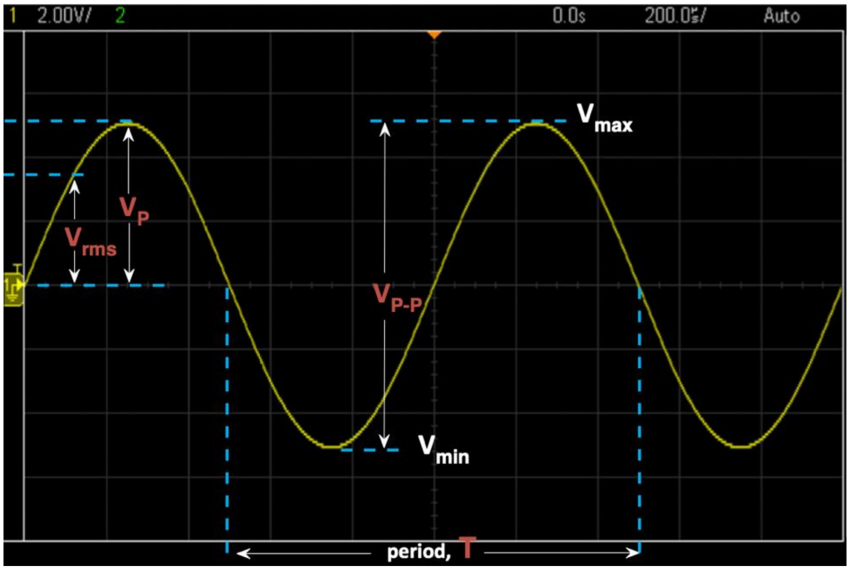


Figure 1.0a: Characterization of a time-varying sinusoidal signal

In order to make AC voltage measurements, some properties of periodic waveforms need to be pre-defined. For a sinusoidal signal that periodically varies with time, it can be characterized by a number of parameters, some of which are shown in **Figure 1.0a**:

- Peak-To-Peak Voltage: $\mathbf{V_{P-P}} = \mathbf{V_{max}} - \mathbf{V_{min}}$
- Amplitude of the sinusoidal signal is defined as its Peak Voltage: $\mathbf{V_P} = \frac{\mathbf{V_{P-P}}}{2}$
- Root-Mean-Squared Voltage: $\mathbf{V_{rms}} = \frac{\mathbf{V_P}}{\sqrt{2}} = 0.707\mathbf{V_P}$
- The period, **T** is defined as the time in which the signal does one full cycle. It can be determined and measured as the length from any point to the next matching point on the signal (e.g., from one peak to the next)
- The frequency, **f** is equal to the number of repetitions per unit of time, and can be calculated from the period, **T**: $\mathbf{f}(\text{Hz}) = \frac{1}{\mathbf{T}(\text{sec})}$ or $\mathbf{T}(\text{sec}) = \frac{1}{\mathbf{f}(\text{Hz})}$.
  When frequency is given in radians/sec then the symbol, **ω** is used, where $\boldsymbol{\omega} = 2\pi\mathbf{f}$

- Phase Shift, $\boldsymbol{\Theta}$ of one sinusoidal signal with respect to another (of the same frequency) occurs when there is time-offset, $\Delta \mathbf{T}$ between them. Phase-Shift => $\boldsymbol{\theta} = \frac{\Delta \mathbf{T}}{\mathbf{T}} . 2\pi \text{ (radians)} = \frac{\Delta \mathbf{T}}{\mathbf{T}} . 360° \text{(degrees)}$
- Sinusoidal AC voltage as a function of time: $v(\mathrm{t}) = \mathbf{V_P} \sin(\boldsymbol{\omega}\mathrm{t} + \boldsymbol{\theta}) = \mathbf{V_P} \sin(2\pi \mathbf{f} \mathrm{t} + \boldsymbol{\theta})$

For this lab exercise, two new pieces of equipment are introduced, namely: (1) the **Function Generator** to produce the AC signals; and (2) an **Oscilloscope** to visualize and measure AC voltage signals in the circuit.

1.1 The Function Generator

Function generator is an instrument that is capable of producing a variety of voltage signals that vary with time. Common functions include the ***sine-wave*** (sinusoid), the ***square-wave*** (alternates between two voltage values) and the ***triangular-wave*** (linearly ramps between two voltage values) over a wide range of frequencies. The frequency generator used in the lab is **Keysight Model EDU33211A** shown in **Figure 1.0b**. *Prior to this lab exercise, students are required to familiarize with the basic setup and operations of the Function Generator by reviewing its User Manual, related FAQs and video tutorials on the course website (D2L).*
The AC signal waveform that is output from the function generator can be used as the input signal to different circuits in a variety of applications. This AC signal can be easily and intuitively configured using some basic functions on the instrument:

- Waveform type: *selections* of *Sine, Square or Triangular (Ramp) are most common.*
- Frequency: *controls the frequency value which can be set in units of Hz (one cycle per second), KHz or MHz.*
- Amplitude: *controls the waveform amplitude, as Peak-Voltage (**V_P**) or Peak-to-Peak Voltage (**V_{P-P}**).*
- DC Offset: *adds a DC offset voltage (with respect to the "ground" reference) to the generated AC signal.*
- Frequency Sweep: *controls the ability to automatically vary (sweep) the frequency of the generated AC signal between a minimum and maximum value.*

Figure 1.0b: Keysight Function Generator

1.2 The Oscilloscope

Oscilloscope is often regarded by engineers as the most useful of the various electronic instruments to measure and display a variety of AC signals as plots of ***input voltage*** versus ***time***. Oscilloscopes vary widely depending on the manufacturer and model but the basic operation remains the same. Most oscilloscopes will have two input channels, normally labelled either as 1-2 or A-B, allowing two input voltage signals for ***simultaneous*** comparison and analysis.

A digital storage oscilloscope, [**Keysight Model DSOX1202G**] shown in **Figure 1.0c** is used in the lab. *Prior to this lab exercise, students are required to familiarize with the basic setup and operations of the Oscilloscope by reviewing its User Manual, related FAQs and video tutorials on the course website (D2L).*
Unlike their analog counterparts, digital oscilloscopes work by digitizing the input signal at a very high rate. Because the signal waveform is acquired as series of digital data points over time, a digital oscilloscope can internally process the signal and auto-measure its amplitude, frequency, period, rise time and fall time. In addition, a digital oscilloscope may have a number of built-in mathematical functions and can perform Fast Fourier Transforms (FFT) in addition to capturing and storing the display for direct printing or retrieval using a USB storage device. Notwithstanding, all oscilloscopes share certain ***basic features***, the typical controls for which should be found on the oscilloscope instrument:

- Input Voltage Channels: *there are two (2) input channels, CH1 (Yellow) and CH2 (Green). The same colored and numbered function key turns channel ON or OFF when that specific channel is selected to be controlled. Input voltage signal on a channel can be coupled as: direct – "**DC**" **coupling**" (default setting) or through an internal blocking capacitor – "**AC**" **coupling**" which removes any DC information present in the time-varying input signal.*
- Vertical Sensitivity (volts/div): *the amplification applied for each input channel to control the amount of its signal to be displayed per division, vertically along the y-axis.*
- Horizontal Time Base (time/div): *controls the time it takes to sweep the screen to capture the amount of waveform cycle(s) of the signal to display along the x-axis. The higher the number the more compact the signal will look by capturing more of the waveform cycles across the screen.*
- Trigger Source: *selects the source to trigger the sweep, either an "INT - Internal source" (CH1 or CH2) or the "EXT - external source". The trigger function is an important setup variable as it synchronizes the horizontal sweep to produce a stable waveform on the display. The most common choice is the use of INT source, with the waveform signal on CH1 used as the primary trigger reference. The mode for the trigger point can be set to AUTO or in the "normal" mode on positive slope trigger with the level set to 0 volts, ensuring it will trigger the scope at the start of every cycle.*
- Position Knob: *allows for relative positioning of each CH1 and CH2 input waveforms over the vertical plane of the display.*

In addition to the above basic controls, the Keysight Oscilloscope allows measurement of different values of the signal waveform such as peak or peak-to-peak voltages, rms voltage, phase, period, frequency, etc., from convenient use of ***cursors***. Like in the MultiSIM simulator, there are ***two vertical*** and ***two horizontal*** cursor lines that can be moved and positioned on the displayed waveform to make ***differential*** measurements. For example, if the vertical cursors lines are used, the oscilloscope will display the ***time difference*** between them. On the other hand, if the horizontal cursor lines are used, the oscilloscope will display the ***voltage difference*** between them.

Figure 1.0c: Keysight Oscilloscope

2.0 OBJECTIVES

- To introduce the basic operations of: **(i)** the function generator as an AC signal source, and **(ii)** the oscilloscope as an AC voltage measuring instrument.
- To use the oscilloscope to display, record and measure period, frequency and voltages (amplitude, peak-to-peak, rms, maximum and minimum) of a sinusoidal time varying waveform.
- To use simple AC circuits with sinusoidal input source to verify the KVL circuit law on AC voltages, and to introduce and measure the phase-shift between sinusoidal AC voltage waveforms.
- To visually observe the concept of bi-directional current flow of an AC signal through use of a "blinking" LED circuit. The "DC-Offset" feature of the function generator will be examined for altering the reference level of a square-wave (digital) waveform.

3.0 REQUIRED LAB EQUIPMENT & PARTS

- Function Generator (FG) and Oscilloscope
- ELE202 Lab Kit: various components, breadboard, wires and jumpers.

4.0 PRE-LAB: ASSIGNMENT

(a) *Even though the properties of a sinusoidal signal may not have been covered in the lectures as of yet, the information provided in **Section 1.0** should be more than sufficient to answer the below pre-lab questions. This understanding of the basic sinusoidal waveform properties should help you better understand how to use an Oscilloscope to display and make meaningful measurements of time-varying waveforms.*

Pre-Lab workspace

Referring to the time-varying sinusoidal waveform shown in **Figure 1.0**:

(i) If the period, **T** = 0.01 sec (or 10 msecs), what is its frequency, **f**?

(ii) If the frequency, **f** = 2000 Hz (or 2 kHz), what is its period, **T**?

(iii) If the frequency is given as **ω** =100 rad/sec, what is its frequency, **f** in Hz?

(iv) If you could use two moveable "***horizontal cursors***" on the displayed waveform to make a differential measurement, describe the relative positions of these two cursors to measure: (a) the peak-voltage, V_P? (b) the peak-to-peak voltage, V_{P-P}? and (c) the rms voltage, V_{rms}?

i) $f = \frac{1}{T}$
$= \frac{1}{0.01}$
$f = 100 \frac{1}{s}$

ii) $T = \frac{1}{f}$
$= \frac{1}{2000}$
$= 0.0005 s$

iii) $\omega = 2\pi f$
$f = \frac{\omega}{2\pi}$
$= \frac{100}{2\pi}$
$f = 50\pi$

iv)
a) one cursor at a peak
b) both cursors at peaks
c) one cursor at $\frac{1}{4}\lambda$, one cursor λ away

(v) If you could use two moveable "***vertical cursors***" on the displayed waveform to make a differential measurement, describe a way to position these two cursors to measure the period, **T**.

Put a cursor at a peak, another at the peak π away, then find Δx as period.

(vi) The equation of a sinusoid waveform is given as: $v(t) = 5\sin(6283.2\mathbf{t} + 0)$ volts. Determine the values of the sine wave properties listed in **Table 1.0**. Show your analysis.

$\omega = 6283.2 = 2\pi f$
$\therefore f = 1000$
$\phi = 0$

$V_{rms} = 0.707 V_p$
$= 3.535$

T (msec.)	**f** (Hz)	V_P (volts)	V_{max} (volts)	V_{min} (volts)	V_{P-P} (volts)	V_{rms} (volts)
0.001	1000	5	5	5	10V	3.5

Table 1.0: Theoretical values of the sinewave properties

(b) For the simple AC circuit shown in **Figure 2.0**, the resistor values chosen for $\mathbf{R_1} = \mathbf{3}$ kΩ and $\mathbf{R_2} = \mathbf{2}$ kΩ. The input AC voltage source, $\mathbf{V_S}$ is a sinusoidal signal with peak voltage amplitude, $\mathbf{V_P} = \mathbf{5}$V and frequency, $\mathbf{f} = \mathbf{1000}$Hz. That is, $V_S(t) = \mathbf{5}\sin(2\pi\mathbf{1000}t + 0)$ volts.
Use ***nodal analysis*** to determine the respective peak-voltage of the voltage signals, $\mathbf{V_A}$ (at *node* **A**) and $\mathbf{V_B}$ (at *node* **B**) with respect to the circuit ground (at *node* **C**); and for the voltage signals, $\mathbf{V_{R1}}$ and $\mathbf{V_{R2}}$ as shown. <u>Show your analysis in the workspace provided</u>, and record your results in **Table 2.0**.

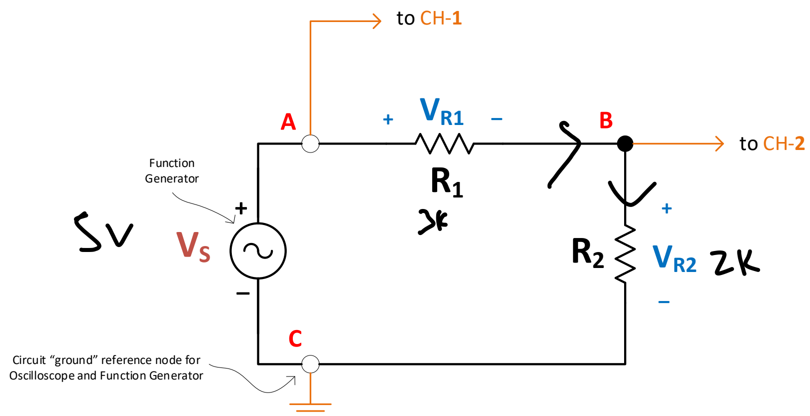


Figure 2.0: Simple AC circuit for *nodal* voltage measurements

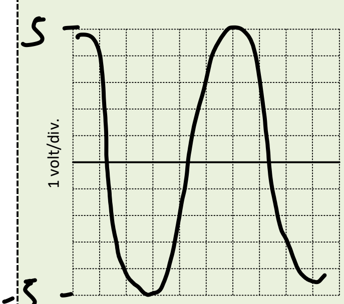
$W = 2\pi f$

Pre-Lab workspace Show your analysis here =>

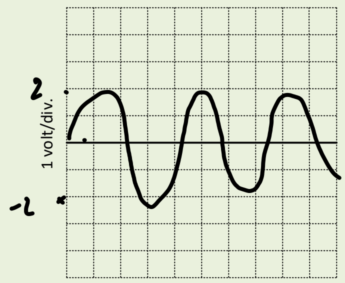
f (Hz)	V_A (volts)	V_B (volts)	V_{R1} (volts)	V_{R2} (volts)
1000	5	2	3	2

Table 2.0: Theoretical voltage values of Figure 2.0 circuit

From the above results, sketch and *clearly* label the waveforms expected at voltage nodes **A** and **B**:



V_A



V_B

$NA = 5A$

$V_{R1} = 3$

$V_{R2} = 2$

$2\pi\, 1000t$

$NB:$

$0 = \frac{VB - VA}{3k} + \frac{VB}{2k}$

$= \left(\frac{1}{3k} + \frac{1}{2k}\right)VB - \frac{1}{3k}VA$

$\frac{1}{3}VA = \left(\frac{5}{6k}\right)VB$

$\frac{5}{3k} = \frac{5}{6}VB$

$\frac{5}{3}\left(\frac{6}{5}\right) = VB$

$2 = VB$

(c) The R-C circuit in **Figure 3.0** introduces a ***phase-shift*** (**Θ**) between the output signal, V_O and the input signal source, V_I, the amount of phase-shift depends on the frequency, **f** of the input signal. This specific phase relationship can be expressed as: $\theta^\circ = \boldsymbol{tan}^{-1}(2\pi \mathbf{f}.\,\mathbf{C}.\,\mathbf{R})$

If the resistor, **R** = **2** kΩ and the capacitor, **C** = **0.1** µF (note: 1 µF = 10^{-6} F), use the above equation to determine the phase-shift, **Θ°** at each input-signal frequency listed in **Table 2.1**.

$0.1\,\cancel{\mu}F\left(\frac{10^{-6}F}{1\,\cancel{\mu F}}\right)$

$0.1 \times 10^{-6}\,F$

$C.R$

$(0.1\times10^{-6})(2000)$

$= 0.0002$

$= \tan^{-1}(2\pi \cdot 300 \cdot (0.1\times10^{-6})(2\times10^3))$

$= 20.655$

$= \tan^{-1}(2\pi \cdot 1000(0.1\times10^{-6})(2000))$

$= 51.488$

$\tan^{-1}(20000\pi(0.1\times10^{-6})(2000$

$= 85.45$

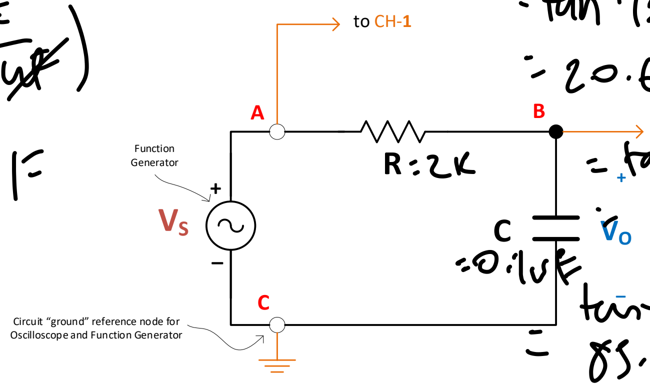


Figure 3.0: Resistance - Capacitance (R-C) Circuit

Pre-Lab workspace

f =>	300 (Hz)	1000 (Hz)	10000 (Hz)
Θ°	20.65	51.49	85.45

Table 2.1: Theoretical phase-shift values of Figure 3.0 circuit

(d) On **MultiSIM**, select the Function Generator and the Oscilloscope as the instruments.

(i) Set the Function Generator to produce a **sine** waveform at a frequency, **f** of **1000** Hz (or **1** kHz) with a peak-voltage amplitude (**V_P**) of **5** volts (or **10** volts peak-to-peak).

(ii) Connect the Function Generator output directly to the CH-1 (or CH-A) input of the Oscilloscope, with a common reference connection between them.

(iii) Adjust the CH-1 horizontal/vertical scale settings on the Oscilloscope to display at least *two complete waveform cycles* over most of the display area.

(iv) Use the ***horizontal/vertical*** cursors to take measurements of the sine wave parameters listed in **Table 3.0**, and to record your results. (**Note:** *Refer to the MultiSIM **FAQ** on D2L on use of **horizontal and vertical cursors** in MultiSIM to take reliable measurements of the waveform properties*.)

- Copy and paste a screenshot showing your MultiSIM measurement readings on the circuit. Include the MultiSIM circuit file (**.ms14**) in your Pre-Lab submission.
- All screenshots should show your name printed on the center-top of the MultiSIM screen and the timestamp at the bottom-lower corner.

Pre-Lab workspace

T (msec.)	**f** = 1/**T** (Hz)	**V_P** (volts)	**V_{max}** (volts)	**V_{min}** (volts)	**V_{P-P}** (volts)	**V_{rms}** (volts)
2	1/2	4.992	4.992	-4.992	9.984	3.53

Table 3.0: MultiSIM measured values of the sinewave properties

(i) Compare the above MultiSIM measured values in **Table 3.0** with your theoretical values in **Table 1.0**. Explain any discrepancies.

The values are very close in magnitude, Slight discrepancies could be result of calculation rounding.

(ii) To determine the peak voltage, **V_P**, often measuring the peak-to-peak voltage, **V_{P-P}** (and dividing it by 2) may be more accurate than directly measuring the peak voltage on the waveform. Why is that the case? (Hint: Since measurement of **V_P** is relative to the sine-wave's zero baseline reference, can this baseline be reliably established for a displayed waveform? Why not?)

Since the |amplitude| can vary without effecting Vp overall, the Vp at one place might not be numerically close to the other Vp. ∴ It's better to find an average

(e) Use **MultiSIM** to construct and simulate the circuit in **Figure 2.0**, with resistors R_1 = 3kΩ and R_2 = 2kΩ. Ensure that *node* **C** of the circuit is used as the *common ground* for all equipment.

(i) For the input AC voltage source, V_S set the Function Generator to a sinusoidal signal with peak-voltage amplitude, V_P = **5**V and frequency, **f** = **1000**Hz. That is, $V_S(t) = \mathbf{5}\sin(2\pi\mathbf{1000}t + 0)$ volts.

(ii) Connect *node* **A** of the circuit to CH-1 (or CH-A) of the Oscilloscope, and *node* **B** to its CH-2 (or CH-B). Adjust the horizontal/vertical scale settings on the Oscilloscope to display *at least two complete cycles* of the waveforms across the screen, and with adequate display coverage.
(**Note:** *The vertical positioning knob can be used to position CH-1 waveform* *above* *CH-2 waveform, however this may not be necessary because the overlapped waveforms are easily identifiable from their respective colors.*)

(iii) Use the two "***horizontal cursors***" on the displayed waveforms to reliably measure the respective peak voltages of V_A (at *node* **A**) and V_B (at *node* **B**) waveforms. Record your results in **Table 3.1**. From these measurements, determine the peak-voltage values of V_{R1} and V_{R2}, and record the values in **Table 3.1**.
(**Note:** *Unlike the DMM multimeter, the voltage* V_{R1} *across resistor* R_1 *cannot be measured directly because the outer conductor of the oscilloscope input channel (and cable) is always connected internally to "ground", which in turn gets referenced to the "common ground" of a circuit, like the* *node* ***C****. If you attempt to directly measure the differential voltage across* R_1 *with the oscilloscope, then* *node* ***B*** *will get shorted to "ground", thereby invalidating the circuit's operation. Therefore, in such situations, you* ***must use*** *node voltages (*V_A *and* V_B*) to determine the voltage across* R_1*. For the circuit in* ***Figure 2.0****, you should verify that* $V_{R1} = V_A - V_B$ *and* $V_{R2} = V_B - 0$)

- Copy and paste a screenshot showing your MultiSIM readings of the circuit. Include the MultiSIM circuit file (**.ms14**) of each circuit in your Pre-Lab submission.
- All screenshots should show your name printed on the center-top of the MultiSIM screen and the timestamp at the bottom-lower corner.

Pre-Lab workspace

f (Hz)	V_A (volts)	V_B (volts)	$V_{R1} = V_A - V_B$ (volts)	$V_{R2} = V_B$ (volts)
1000	4.992	1.997	2.995	1.997

Table 3.1: MultiSIM measured values of Figure 2.0 circuit

(i) Compare your simulation results in **Table 3.1** with the respective theoretical ones from **Table 2.0**, and explain the causes of any discrepancies.

They are extremly close. Slight discrepancies could've come from calulation differences.

(ii) Is the peak voltage of V_A signal same as that of the input source, V_S? Why?

Peak Va is close to Vs. Makes sense because Vs directly connects to VA

(iii) Do you expect the frequency of the sinusoidal voltage signal across R_1 and R_2 to be the same as the input source frequency? Why?

No, I believe the voltage will change since there are

(iv) Was the KVL relationship, $V_S - V_{R1} - V_{R2} = 0$ verified using the peak-voltage values of the respective AC signals? Explain.

Yes. $V_S \simeq 5$, $V_{R1} \simeq 3$, $V_{R2} \simeq 2$.

(f) Use **MultiSIM** to construct and simulate the circuit in **Figure 3.0**, with resistors **R** = 2kΩ and **C** = **0.1**μF. Ensure that *node* **C** of the circuit is used as the *common ground* for all equipment.

(i) For the input AC voltage source, V_S set the Function Generator to a sinusoidal signal with peak-voltage amplitude, V_P = **5**V and frequency, **f** = **300**Hz.

(ii) Connect *node* **A** of the circuit to CH-1 (or CH-A) of the Oscilloscope, and *node* **B** to its CH-2 (or CH-B). Adjust the horizontal/vertical scale settings on the Oscilloscope to display *at least two complete cycles* of the waveforms across the screen, and with adequate display coverage.

(iii) Use the two "***vertical cursors***" on the displayed waveforms to reliably measure the time-shift, Δ**t** between the two waveforms in order to determine the relative phase-shift of the output signal, V_O (on CH-2) with respect to the input source signal, V_S (on CH-1). Record your result in **Table 3.2**.
(<u>**Note:**</u> *A way to measure the phase-difference between **CH-1** and **CH-2** waveforms is to position the **vertical cursors** (or traces) between the two adjacent peaks of the respective sinusoidal waveforms to determine the relative time-shift, Δ**t**. The equation* $\Theta = 2\pi f \times \Delta t$ *(radians) or* $\Theta = 2\pi(\Delta t/T)$ *(radians) can be used to calculate the phase-difference in radians. You will need to convert Θ (radians) to degrees, noting that* π *radians = 180º. Refer to **Section 1.0***)

- Copy and paste a screenshot showing your MultiSIM readings of the circuit. Include the MultiSIM circuit file (**.ms14**) of each circuit in your Pre-Lab submission.
- All screenshots should show your name printed on the center-top of the MultiSIM screen and the timestamp at the bottom-lower corner.

(iv) For **each** remaining frequency, **f** listed in **Table 3.2**, set the function generator accordingly, and then <u>repeat</u> the above steps **(ii)** and **(iii)**. For each frequency setting, record your result in **Table 3.2**.

Pre-Lab workspace *Show you calculations here =>* v(t) =

$\Theta = \tan^{-1}\left(\frac{2\pi CR}{T}\right)$

$= \tan^{-1}\left(\frac{2\pi \cdot 0.1\mu F \cdot 2k}{T}\right)$

$= \tan^{-1}($

f	**T** calculated from freq., **f**	**ΔT** measured on MultiSIM	**Θº** determined from **ΔT**
300 (Hz)	0.0033	5.0 ms	2.51
1000 (Hz)	0.001	1.506 ms	6.34
10000 (Hz)	0.0001	150 μs	8.377

Table 3.2: MultiSIM measured phase-shift values of Figure 3.0 circuit

(i) Compare your simulation results in **Table 3.1** with the respective theoretical ones from **Table 2.1**, and explain the causes of any discrepancies.

My results are extremely different, Discrepancies probably stemmed from calculation errors.

(ii) Was the waveform, V_O ***leading*** or ***lagging*** the input waveform, V_S? Explain how you would make such a determination from the displayed waveforms.

Vo should be lagging the Vs, since there is some resistance before the Vo.

5.0 IN-LAB Experiment: IMPEMENTATION & MEASUREMENTS

(a) AC Measurements of Sinewave (Sinusoid) Signal Properties

1. Turn ON the Function Generator. Set it up to produce a **sine** waveform at a frequency, **f** of **1000** Hz (or **1** kHz) with a peak-voltage amplitude (**V_P**) of **5** volts (or **10** volts peak-to-peak).

2. Connect the Function Generator output directly to the CH-1 input of the Oscilloscope, with a common reference connection between them.

3. Adjust the CH-1 ***horizontal/vertical scale*** settings on the Oscilloscope to display at least *two complete waveform cycles* over most of the display area. The ***trigger*** setting for CH-1 should be set to Auto mode.

4. Use the ***horizontal*** and the ***vertical*** cursors to take respective measurements of the sinewave parameters listed in **Table 4.0**, and record your results. (**Note:** *Refer to the Oscilloscope **FAQ** on D2L on use of **horizontal and vertical cursors** to take reliable measurements of the waveform properties*.)

5. For **any one** of the above measurements, *capture* the Oscilloscope screen and record the saved image. Include the captured image(s) in your In-Lab report. (**Note:** *You can save the captured screen image in your USB key, otherwise you must take picture of the captured screen using your phone.*)

T (msec.)	**f** = 1/**T** (Hz)	**V_P** (volts)	**V_{max}** (volts)	**V_{min}** (volts)	**V_{P-P}** (volts)	**V_{rms}** (volts)
0.001	1000	5.0	5.0	5.0	10.0	3.535

Table 4.0: In-Lab measurement values of the sinewave properties.

(b) AC Measurements on the Voltage-Divider circuit (Figure 2.0)

1. Construct the circuit of **Figure 2.0** on your breadboard using resistors **R_1** = **3**kΩ and **R_2** = **2**kΩ from your Kit components. Ensure that *node* **C** of the circuit is used as the *common ground* for all equipment connections.

2. For the input AC voltage source **V_S** of the circuit, set the Function Generator to a sinusoidal signal with peak-voltage amplitude, **V_P** = **5**V and frequency, **f** = **1000**Hz.

3. From your circuit, connect *node* **A** to CH-1 input of the Oscilloscope, and *node* **B** to its CH-2 input. Adjust the horizontal/vertical scale settings on the Oscilloscope to display *at least two complete cycles* of the waveforms across the screen, and with adequate display coverage. Set the ***trigger*** mode on CH-1 for signal reference. (**Note:** *The vertical positioning knob can be used to position CH-1 waveform above CH-2 waveform, however this may not be necessary because the overlapped waveforms are easily identifiable from their respective colors.*)

4. From one of the displayed waveforms on the Oscilloscope, use the ***vertical cursors*** to measure the period, **T**. Record your result in **Table 4.1**. *Capture the screen image* showing your cursor placements, and save the image. Include the captured image(s) in your In-Lab report.

5. Use the ***horizontal cursors*** to measure the peak-voltage, **V_P** of the signal, **V_A** (node **A**) on CH-1, and the signal, **V_B** (node **B**) on CH-2. Record yours result in **Table 4.1**. *Capture the screen images* showing your cursor placements on each measurement, and save the images. Include the captured image(s) in your In-Lab report.

6. Turn OFF the Function Generator.

5.25
− 2.44
2.81

f (Hz)	**V_A** (volts)	**V_B** (volts)	**$V_{R1} = V_A - V_B$** (volts)	**$V_{R2} = V_B$** (volts)
1000	5.25	2.44	2.81	2.44

Table 4.1: In-Lab measurement values of Figure 2.0 circuit

(c) AC Phase Measurement on the Simple R-C circuit (Figure 3.0)

1. Construct the circuit of **Figure 3.0** on your breadboard using the resistor value of **R** = **2**kΩ and the capacitor value of **C** = **0.1**μF from your Kit components. Ensure that *node* **C** of the circuit is used as the *common ground* for all equipment connections.

2. Turn ON the Function Generator. For the input AC voltage source **V_S** of the circuit, set the Function Generator to a sinusoidal signal with peak-voltage amplitude, **V_P** = **5**V and frequency, **f** = **300**Hz.

3. From your circuit, connect *node* **A** to CH-1 input of the Oscilloscope, and *node* **B** to its CH-2 input. Adjust the horizontal/vertical scale settings on the Oscilloscope to display *at least two complete cycles* of the waveforms across the screen, and with adequate display coverage. Set the ***trigger*** mode on CH-1 for signal reference. (**Note:** *The vertical positioning knob can be used to position CH-1 waveform above CH-2 waveform, however this may not be necessary because the overlapped waveforms are easily identifiable from their respective colors.*)

4. Use the "***vertical cursors***" on the displayed waveforms to reliably measure the time-shift, Δ**t** between the two waveforms in order to determine the relative phase-shift of the output signal, **V_O** (on CH-2) with respect to the input source signal, **V_S** (on CH-1). Record your result in **Table 4.2**.
(**Note:** *A way to measure the phase-difference between* ***CH-1*** *and* ***CH-2*** *waveforms is to position the* ***vertical cursors*** *(or traces) between the two adjacent peaks of the respective sinusoidal waveforms to determine the relative time-shift,* $\Delta\mathbf{t}$. *The equation* $\boldsymbol{\Theta} = 2\pi \mathbf{f} \times \Delta\mathbf{t}$ *(radians) or* $\boldsymbol{\Theta} = 2\pi(\Delta\mathbf{t}/\mathbf{T})$ *(radians) can be used to calculate the phase-difference in radians. You will need to convert* Θ *(radians) to degrees, noting that* $\boldsymbol{\pi}$ *radians = 180º. Refer to* ***Section 1.0*** *and related FAQ on phase-measurement on D2L*).

   - *Capture the screen image* showing your cursor placements and readings, and save the image for the record. Include the captured image in your In-Lab report.

5. For **each** remaining frequency, **f** listed in **Table 4.2**, set the Function Generator accordingly using the same sinusoidal peak-voltage amplitude, **V_P** = **5**V; and then **repeat** the above steps **(3)** and **(4)**.

6. Turn OFF the Function Generator.

f	**T** calculated from freq., **f**	**ΔT** In-Lab measurements	**Θº** determined from **ΔT**
300 (Hz)	0.003	113 μs	2.366
1000 (Hz)	0.001	82 ms	515.22
10000 (Hz)	0.0001	1.6 μs	100.53

Table 4.2: In-Lab measurement results of Figure 3.0 circuit

(d) "Blinking LED" Circuit – use of Square-Wave Signal with DC Offset

Background: Basically, a **L**ight **E**mitting **D**iode (LED) is a semiconductor device (a **p-n** junction diode) that emits light due to electro-luminescence effect. When the LED is forward-biased, it acts as a light source device releasing light when current passes through it. A physical structure image of a red LED and its electrical symbol (similar to the **p-n** junction diode) are shown in **Figure 4.0**. The polarity on the symbol is an indication that the LED, like its **p-n** junction diode counterpart, is not symmetric as it allows current to flow only in *one direction*, from the **anode** to **cathode**, and blocks the current flow in the reverse direction.

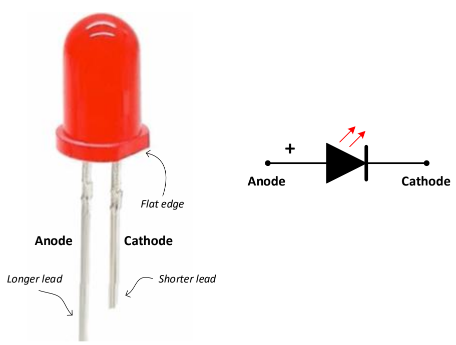


Figure 4.0: LED device and its electrical symbol

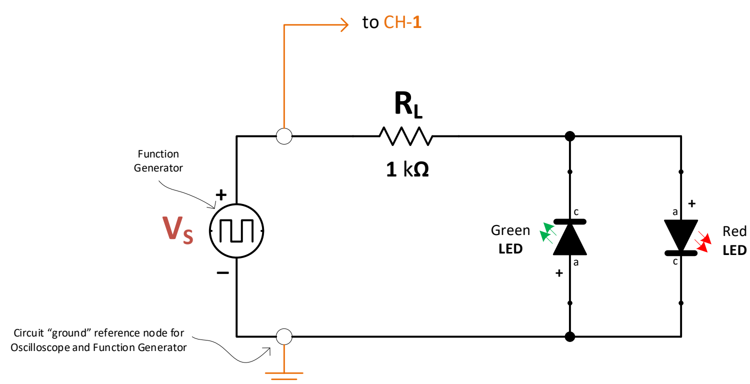


Figure 5.0: Dual "blinking LED" circuit

The purpose of this experiment is to visually observe the concept of ***bi-directional*** current flow of an AC signal through use of a "blinking" LED circuit, shown in **Figure 5.0**. The effects of "DC-Offset" feature of the Function Generator on a square-wave (digital) waveform will be demonstrated.

1. Select a **1** kΩ resistor, the Red and Green LEDs from your Kit to construct the circuit in **Figure 5.0** on your breadboard. **Note:** *In order to protect the LED from excessive current that can damage it, the current-limiting resistor, R_L of **1** kΩ is used in series with the LEDs.*

2. Turn ON the Function Generator. For the input AC voltage source V_S of the circuit, set the Function Generator to a **Square-Wave** signal with the peak-voltage amplitude, V_P = **5**V (that is, $\pm$ **5**V around **0**V reference) as shown in **Figure 6.0 (i)**. Set the frequency, **f** = **1** Hz. (Suggest start with V_P = 3 V)

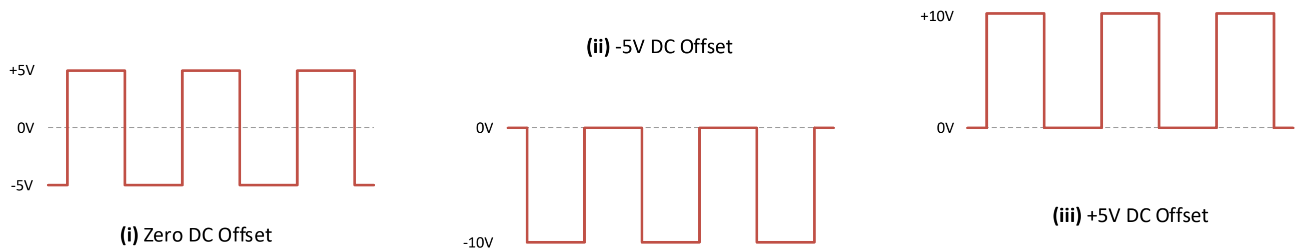


Figure 6.0: Square-wave waveforms showing effect of adding DC-Offset

3. Connect the Function Generator to the circuit, and also to CH-1 of the Oscilloscope as shown in **Figure 5.0**. Monitor the Oscilloscope to confirm the waveform voltage setting of $\pm$ **5**V and the waveform frequency of **1** Hz.

4. Observe the blinking of the LED(s) **on** and **off** once per second (**1**Hz). Record your observations in **Table 5.0** of the status and relative brightness of each LED.

5. Use the DC-OFFSET control on the Function Generator to apply a "**-5**V" DC-Offset to the waveform so that it oscillates between **0**V and **-10**V as shown in **Figure 6.0 (ii)**. Observe the blinking of the LED(s) **on** and **off** once per second (**1**Hz). Record your observations in **Table 5.0** of the status and relative brightness of each LED.

6. Use the DC-OFFSET control on the Function Generator to apply a "+**5**V" DC-Offset to the waveform so that it oscillates between **0**V and **+10**V as shown in **Figure 6.0 (iii)**. Observe the blinking of the LED(s) **on** and **off** once per second (**1**Hz). Record your observations in **Table 5.0** of the status and relative brightness of each LED.

7. Keeping the same setting as in (6), **gradually increase the frequency** on the Function Generator to find the frequency at which the LED appears to stop blinking. The human eye will eventually be unable to perceive the flickering. Record this frequency in **Table 5.0**.

8. Turn OFF all the equipment, and tidy up your work bench space.

1 Hz Square-wave waveform with:	Circle the **color** of blinking LED		Circle the perceived **brightness** level on a scale of 1 to 5				
Zero DC Offset	R	G	1	2	3	4	5
-5V DC Offset	R	G	1	2	3	4	5
+5V DC Offset	R	G	1	2	3	4	5
Frequency at which the LED appears to stop blinking? =>					57 hz		

Table 5.0: Observations of the "Blinking LEDs" of Figure 5.0 circuit.

6.0 POST-LAB: OBSERVATIONS AND ANALYSIS OF RESULTS

(a) AC Measurements of Sinewave (Sinusoid) Signal Properties

Workspace

(i) Compare your experimental results in **Table 4.0** with your theoretical values in **Table 1.0**. Explain the possible causes of any discrepancies.

All values in table 4.0 matched with table 1.0,

(ii) To determine the peak voltage, **V_P**, often measuring the peak-to-peak voltage, **V_{P-P}** (and dividing by 2) may be more accurate than directly measuring the peak voltage on the waveform. Based on your results in **Table 4.0**, does **V_{P-P}**/2 give a better representation of **V_P**? Explain.

Based on my results, it ultimately doesn't matter since they yield the same answer

(iii) Based on your observations of the displayed waveform on the Oscilloscope, did the Function Generator accurately generate the **sine** waveform you programmed for the frequency, **f** of **1000** Hz with the peak-voltage amplitude (**V_P**) of **5** volts?

Yes, it accurately generated a sin graph with a fz of 1000Hz & Vp of 5V.

(b) AC Measurements on the Voltage-Divider circuit

Workspace

In regard to the "Simple AC Circuit" in **Figure 2.0**:

(i) Comment on how your measurement results in **Table 4.1** compare to the corresponding theoretical values in **Table 2.0** and MultiSIM simulation values in **Table 3.1**? Explain possible causes of any discrepancies.

there is a difference of approx. 0.25 between experimental and theoretical value tables, which likely stemmed from inaccurately measuring voltage using the vertical lines.

(ii) From your AC measurements in **Table 4.1** of the peak-voltage (**V_P**) values of respective AC signals, was the KVL relationship, **V_S** - **V_{R1}** - **V_{R2}** = 0 verified? Would the same hold true if the peak-to-peak voltage (**V_{P-P}**) values were as used instead? Why? Show your analysis below.

$V_S - V_R - V_{RL} = 0$
$5.25 - 2.81 - 2.44 = 0$
$5 - 3 - 2 = 0$

For the AC measurements in table 4.1, the KVL relationship is verified.
If V_{p-p} was used instead, the KVL should still be followed.

(c) AC Phase Measurement on the Simple R-C circuit

Workspace

In regard to the "Simple RC Circuit" in **Figure 3.0**:

(i) Compare your AC phase measurement results in **Table 4.2** with the theoretical values in **Table 2.2** and MultiSIM simulation values in **Table 3.2**? Explain possible causes of any discrepancies.

Table 2.2 matches my measurement results however the Multisim values in table 3.2 are off, probably due to the fact that table 3.2 wasn't filled correctly.

(ii) Did you find the use of the ***vertical cursors*** to measure the time-difference, **ΔT** (between the two displayed waveforms) a convenient way to determine phase-shift? Would you consider this method a reliable one? Explain.

vertical cursors were used to find ϕ, I would consider this reliable

(iii) Did you observe waveform, V_O ***lagging*** the input waveform, V_S? If so, would it be correct to conclude that V_S must be ***leading*** V_O? How do these ***lag*** or ***lead*** time-shift present itself on the display of the concurrent waveforms?

V_o appears to lag V_s. I believe it would be correct to assume V_s leads V_o. The time-shift is visible at a fixed x ref point (origin), where the point that the graphs cross x-axis can be easily compared

(d) "Blinking LED" Circuit

From your observation of the measurement results in **Table 5.0**:

(i) Explain why both **R** and **G** LEDs were "blinking" when the ± 5V (with zero DC-Offset) waveform was used?

Both LEDs were blinking because both LEDs got an equal amount of voltage.

(ii) Which color LED was blinking when -**5**V DC-Offset was added to the waveform? Likewise, which color LED was blinking when +**5**V DC-Offset was added to the waveform? For each case, what might be the reason?

At -5V, the green led was blinking. At +5V, red blinked. The offset probably

(iii) When the waveform signal was used with either -**5**V or + **5**V DC-Offset, was the resultant brightness level of the blinking LED higher than with use of no DC-Offset? Why? (**Hint:** *Brightness level of the LED is proportional to the amount of current flowing through it*).

It was higher than no offset, likely due to the higher amount of V and ∴ I flowing through a single LED.

(iv) Above a certain frequency, the human eye is unable to perceive the flickering. This frequency limit is typically in range of 30Hz to 50Hz, but can be as high as 65Hz. Did your result confirm this non-flickering phenomenon?

Yes. At 57 Hz the flickering was un-perceivable.

7.0 LAB REPORT REQUIREMENTS & GUIDELINES

Lab reporting is to be completed and submitted separately as **Part I** and **Part II**, noted below:

Part I (**Pre-Lab Work**) => represents **40%** of the pre-assigned Lab weight.

Pre-Lab Work (assignment) of **Section 4.0** that includes handwritten calculations, MultiSIM results, and analysis is to be completed and submitted **prior to the start** of your scheduled lab. *The grading is commensurate with completeness and accuracy of your handwritten calculations, analysis and MultiSIM simulation circuits/plots.*

Note the following requirements for the document submission for Part I:

- A completed and signed "COVER PAGE – **Part I**" has to be included with your submission, a copy of which is available on D2L. The report will not be graded if the signed cover page is not included.
- Your completed handwritten pages of **Section 4.0** should be scanned (via a scanner or phone images), together with the required MultiSIM images. **Note:** *MultiSIM results must be generated using the Department's licensed version of MultiSIM, and the captured screenshots should show your name (at the center-top) and the timestamp (at the bottom-right corner of your screen).*
- Collate and create a .pdf or .docx file of the above, and upload it via D2L **any time prior to the start of your scheduled lab**. Upload instructions are provided on D2L.

Zero marks will be assigned for the entire lab if this Part I is not submitted prior to your scheduled lab.

Part II (**In-Lab Work** and **Post-Lab Work**) => represents **60%** of the pre-assigned Lab weight.

In-Lab Work (**Section 5.0**) and **Post-Lab Work** (**Section 6.0**) that include in-lab results, handwritten analysis and observations are to be completed and submitted **by 11.59 p.m.** of the same day as your scheduled lab. *The grading is commensurate with: - completeness, correctness and collection of all experimental results (data and waveforms); merits of observation of the correlations between the experimental and pre-lab assignment results; and reasonableness of the answers to questions posed.*

Note the following requirements for the document submission for Part II:

- A completed and signed "COVER PAGE – **Part II**" has to be included with your submission, a copy of which is available on D2L. The report will not be graded if the signed cover page is not included.
- Scan your completed pages of **Section 5.0** and **Section 6.0** (via a scanner or phone images), together with any required In-Lab Oscilloscope screen-shot images.
- Collate and create a .pdf or .docx file of the above, and upload it via D2L **by 11.59 p.m. on the same day** your lab is scheduled. Late submissions will not be graded.