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

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)	$\mathbf{V_P}$ (volts)	$\mathbf{V_{max}}$ (volts)	$\mathbf{V_{min}}$ (volts)	$\mathbf{V_{P\text{-}P}}$ (volts)	$\mathbf{V_{rms}}$ (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. Show your analysis in the workspace provided, and record your results in **Table 2.0**.

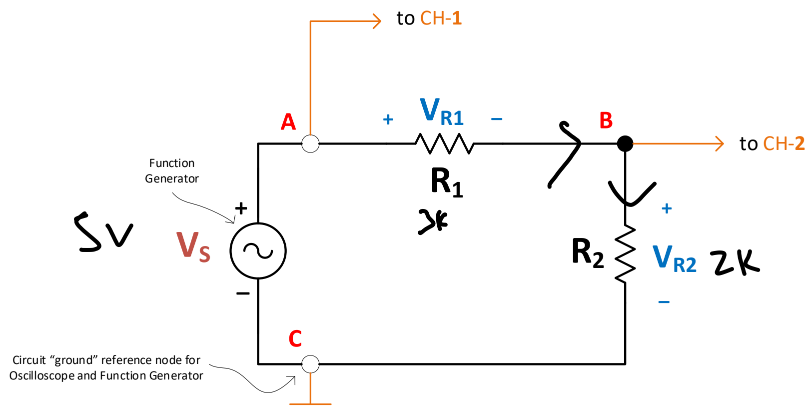


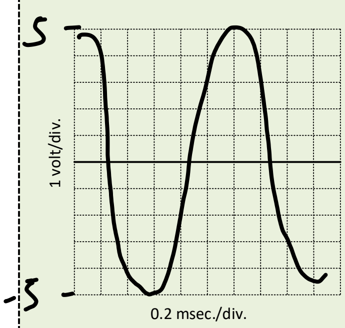
Figure 2.0: Simple AC circuit for *nodal* voltage measurements

Pre-Lab workspace Show your analysis here =>

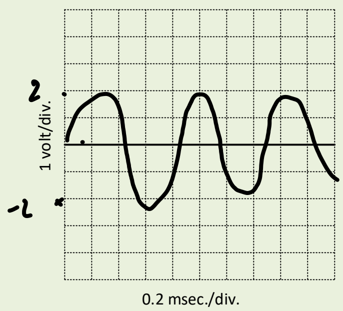
f (Hz)	V_A (volts)	V_B (volts)	V_{R1} (volts)	V_{R2} (volts)
1000	5	2	3	2

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

2π 1000t

N B:

$0 = \frac{V_B - V_A}{3k} + \frac{V_B}{2k}$

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

$\frac{1}{3}V_A = \left(\frac{5}{6k}\right)V_B$

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

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

$2 = V_B$

(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: $\boldsymbol{\theta}^\circ = \boldsymbol{tan}^{-1}(2\pi \mathbf{f}.\mathbf{C}.\mathbf{R})$

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**.

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

0.1×10^{-6} F

C·R

$(0.1 \times 10^{-6})(2000)$

$= 0.0002$

$= \tan^{-1}(2\pi \cdot 300 \cdot (0.1 \times 10^{-6})(2 \times 10^3))$

$= 20.655$

$= \tan^{-1}(2\pi \cdot 1000 (0.1 \times 10^{-6})(2000))$

$= 51.488$

$\tan^{-1}(20000\pi (0.1 \times 10^{-6})(2000$

$= 85.45$

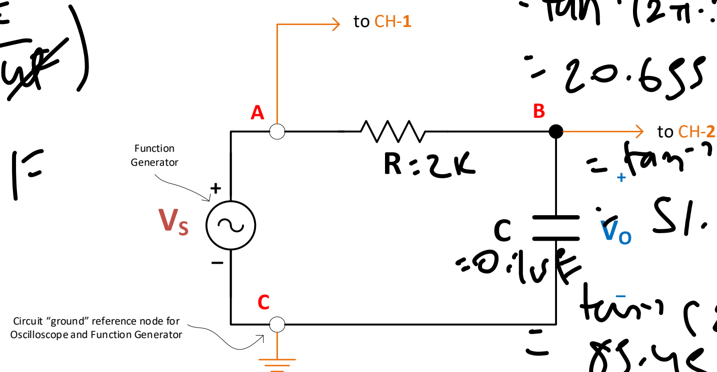


Figure 3.0: Resistance - Capacitance (R-C) Circuit

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 $\mathbf{R_1}$ = 3kΩ and $\mathbf{R_2}$ = 2kΩ. Ensure that *node* **C** of the circuit is used as the *common ground* for all equipment.

(i) For the input AC voltage source, $\mathbf{V_S}$ set the Function Generator to a sinusoidal signal with peak-voltage amplitude, $\mathbf{V_P}$ = **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 $\mathbf{V_A}$ (at *node* **A**) and $\mathbf{V_B}$ (at *node* **B**) waveforms. Record your results in **Table 3.1**. From these measurements, determine the peak-voltage values of $\mathbf{V_{R1}}$ and $\mathbf{V_{R2}}$, 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)	$\mathbf{V_A}$ (volts)	$\mathbf{V_B}$ (volts)	$\mathbf{V_{R1} = V_A - V_B}$ (volts)	$\mathbf{V_{R2} = V_B}$ (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 discrepencies could've come from calution differences.

(ii) Is the peak voltage of $\mathbf{V_A}$ signal same as that of the input source, $\mathbf{V_S}$? Why?

Peak V_A is close to V_S. Makes sense because V_S directly connects to V_A

(iii) Do you expect the frequency of the sinusoidal voltage signal across $\mathbf{R_1}$ and $\mathbf{R_2}$ 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, $\mathbf{V_S} - \mathbf{V_{R1}} - \mathbf{V_{R2}} = 0$ 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**.
(**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, 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 repeat the above steps **(ii)** and **(iii)**. For each frequency setting, record your result in **Table 3.2**.

Pre-Lab workspace *Show you calculations here =>* y(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

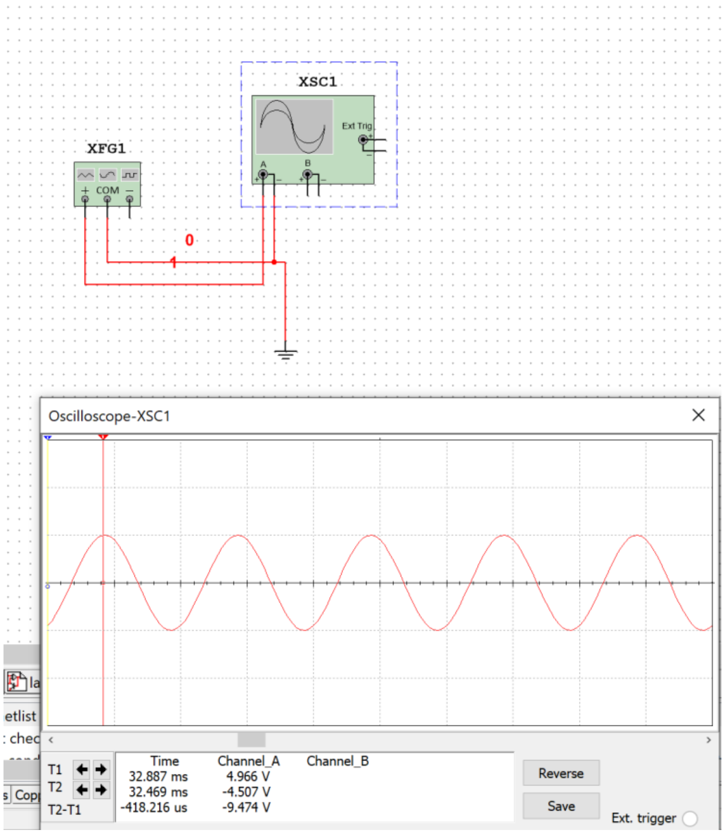
(i) Compare your simulation results in **Table 3.1** with the respective theoretical ones from **Table 2.1**, and explain the causes of any discrepancies.

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

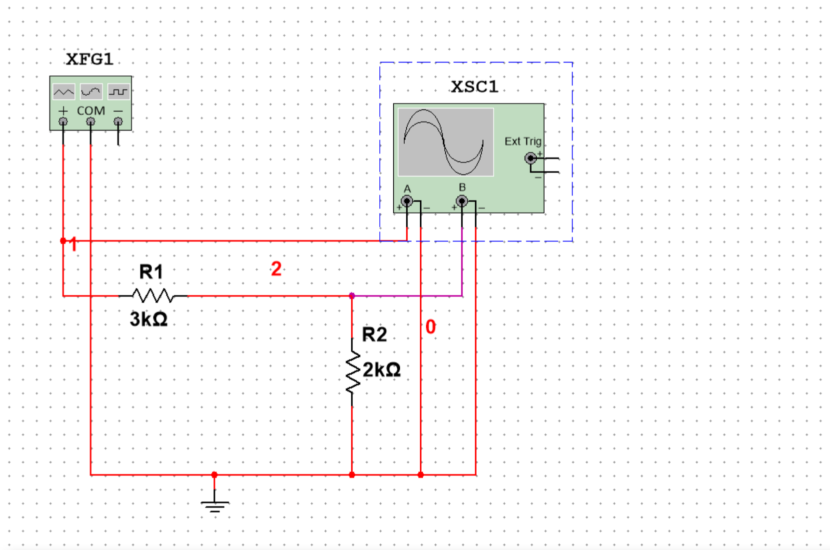
(ii) Was the waveform, V_O ***leading*** or ***lagging*** the input waveform, V_S? Explain how you would make such a determination from the displayed waveforms.

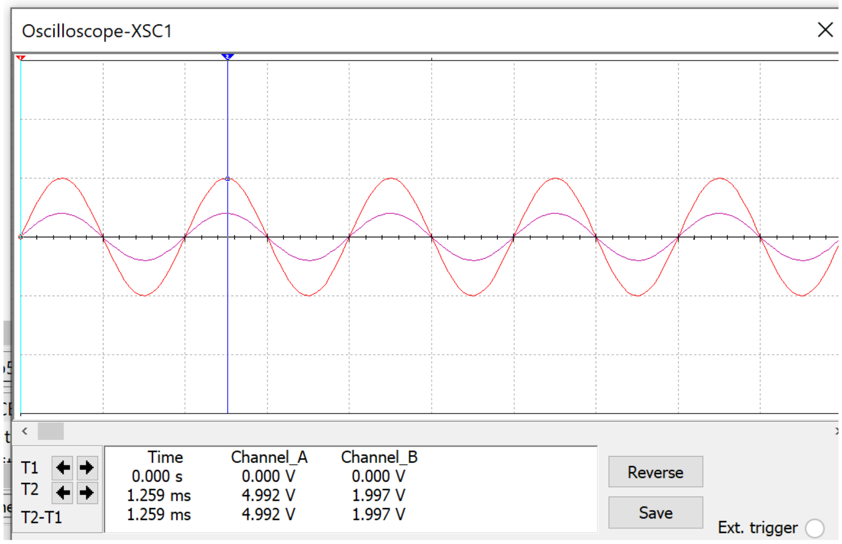
Vo should be lagging the Vs, since there is some resistence before the Vo.

1

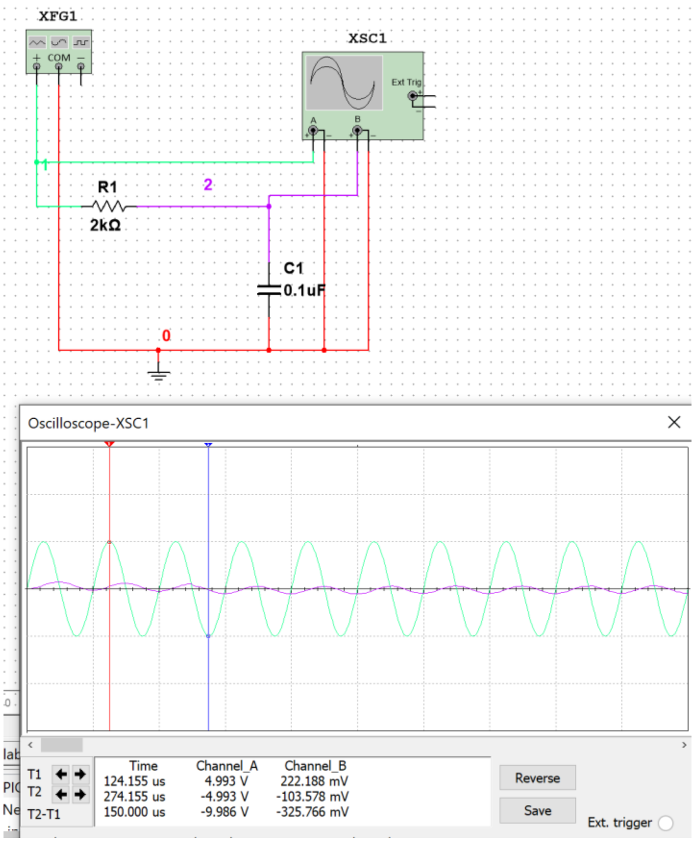

XSC1
Ext Trig
XFG1
COM
0
1
Oscilloscope-XSC1
Time
Channel_A
Channel_B
T1
T2
T2-T1
32.887 ms
32.469 ms
-418.216 us
4.966 V
-4.507 V
-9.474 V
Reverse
Save
Ext. trigger

2


XFG1
COM
XSC1
Ext Trig
A
B
1
R1
3kΩ
2
R2
2kΩ
0


Oscilloscope-XSC1
Time Channel_A Channel_B
T1 0.000 s 0.000 V 0.000 V
T2 1.259 ms 4.992 V 1.997 V
T2-T1 1.259 ms 4.992 V 1.997 V
Reverse
Save
Ext. trigger

3


XFG1
COM
XSC1
Ext Trig
A
B
1
R1
2kΩ
2
C1
0.1uF
0
Oscilloscope-XSC1
Time Channel_A Channel_B
T1 124.155 us 4.993 V 222.188 mV
T2 274.155 us -4.993 V -103.578 mV
T2-T1 150.000 us -9.986 V -325.766 mV
Reverse
Save
Ext. trigger