Syed Ali Zaidi (zaidis33)

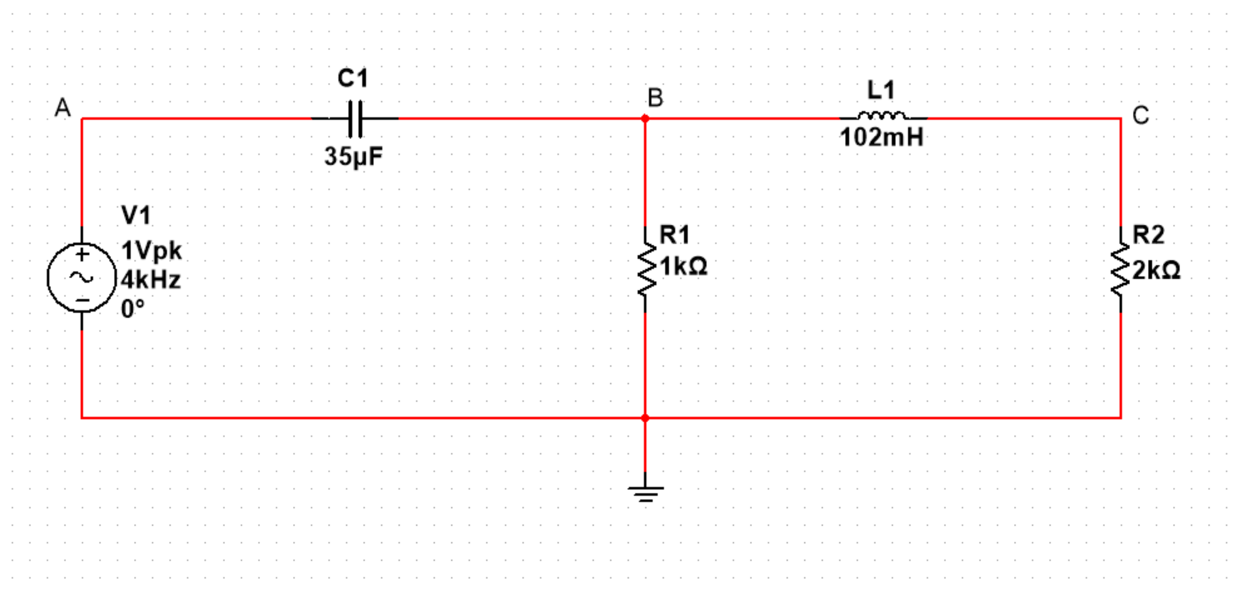
Lab 2

Introduction:

The circuit shown below is a modified circuit of sample lab 2. The solution to this circuit is divided into three stages: analysis, digital and physical solution. For my analysis, I used the voltage divider formula and mesh analysis to provide two set of result. I used Maple to solve the equations of both methods and found their results to agree with each other. My digital part involved me making the circuit in NI Multisim and using single frequency sweep to find the voltage and currents. I also used the Tektronix Oscilloscope to measure voltages at A, B and C along with comparing their phase difference. My physical build involved making the circuit on a breadboard and using a Hantek Oscilloscope to measure values.

Problem:

Consider the following circuit:



Determine the current through the inductor and the voltages at nodes A, B and C…

1. Analytically, by solving your circuit [by working in the phasor domain] in two ways:
   a. By using either Kirchhoff's laws OR using voltage dividers, and
   b. By using either mesh analysis OR nodal analysis

2. Digitally, in Multisim using both:
   a. single frequency AC analysis, and
   b. any one of the oscilloscopes.

3. Physically, by building and measuring your circuit using the oscilloscope.

Analysis:

The first part of my analysis was to assign a reference voltage. This will be Va and it will be used throughout the lab. This is because there are no components between the voltage source and the node. I will now demonstrate Va in its phasor domain and time domain form.

We can represent the voltage though A as:

$v_a(t) = A\cos(\omega t + \varphi)$

We will then use the equation $\omega = 2\pi f$ and plug in the amplitude, frequency, and time to get the time domain:

$v_a(t) = 1 \cdot \cos(2\pi(4000)t)$

$v_a(t) = \cos(8000\pi t)$

Because this is the reference voltage and because there are no components in between the input and node, there is no phase shift thus $\varphi = 0$.

Now we will convert to phasor domain:

$V = 1V \cdot e^{j\varphi}$

$V = e^{j(0)} = 1V$

I will use this method to calculate the phasor and time domain form of the other node voltages through Maple.

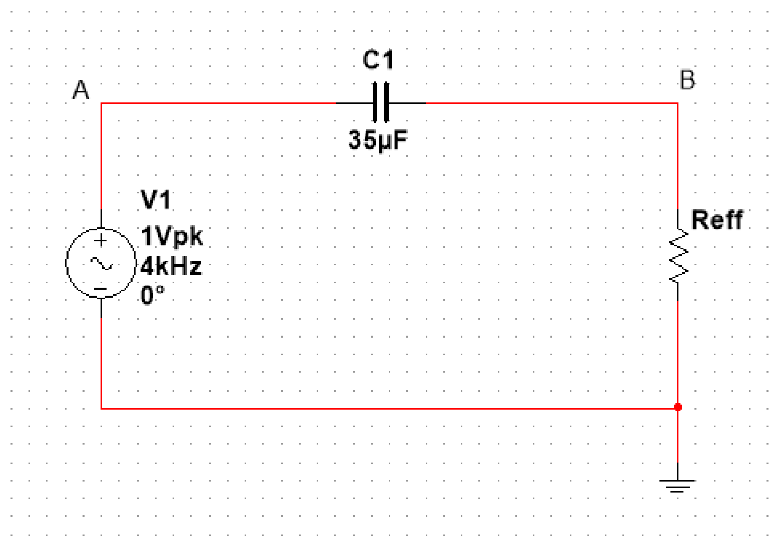
Voltage dividers:

First, I defined the impedance equations of the inductor and capacitor, respectively as:

$$Z_L = j\omega L$$

$$Z_C = \frac{1}{j\omega C}$$

Then the right-hand side mesh was simplified as having one resistor with an impedance of Reff. The simplified circuit looks like this:



This allows the voltage divider formula to be applied efficiently. As there are now two components in series, we can use the ratio of the effective resistance to the total resistance of the circuit to calculate the drop in voltage through C1.

$$v_B(t) = \frac{R_{eff}}{R_{eff} + Z_{C1}} \cdot v_A$$

$$where\ R_{eff} = \left(\frac{1}{R_1} + \frac{1}{Z_{L1} + R_2}\right)^{-1}$$

Now using the voltage at node B as the main voltage source though L1 and R2, we can calculate Vc using the formula below:

$$v_C(t) = \frac{R_2}{R_2 + Z_{L1}} \cdot v_B$$

These voltages at A, B and C will allow us to calculate the voltage across each component (see Appendix):

$$\therefore V_{C1} = v_A - v_B$$

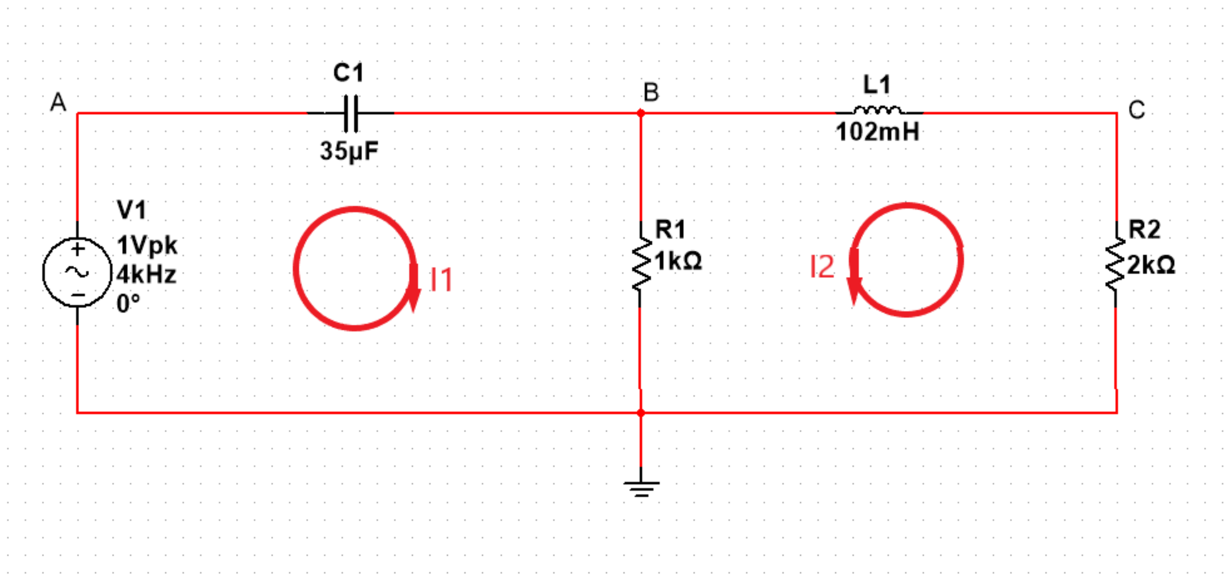
$$V_{R1} = v_B$$

$$V_{L1} = v_B - v_C$$

$$V_{R2} = v_C$$

Nodal Analysis:

For performing nodal analysis, I first set I1 and I2 as the currents through the two meshes as shown in the diagram below. Then I performed calculations on each mesh (using the sum of currents for the common wire in the circuit).



Clockwise through left mesh:

$$V_1 - I_1 Z_{C1} - \left(I_1 + I_2\right) R_1 = 0$$

Clockwise through right mesh:

$$\left(I_1 + I_2\right) R_1 + I_2 Z_{L1} + I_2 R_2 = 0$$

These two equations contain I1 and I2 as the variables and go clockwise in each mesh. Solving the two equations in Maple (see appendix) we get:

Maple Input	Maple Output
`VC1 := -0.0002739145777 - 0.001352632208*I; VC1_A:=abs(VC1); phi_VC1:= argument(VC1);`	VC1 := -0.0002739145777 - 0.001352632208 I VC1_A := 0.001380088072 phi_VC1 := -1.770599229
`VR1 = 1.000273915 + 0.001352632208*I;assign(%); VR1_A:=abs(VR1); phi_VR1:= argument(VR1);`	VR1 = 1.000273915 + 0.001352632208 I VR1_A := 1.000274830 phi_VR1 := 0.001352260979
`VL1=-0.6211468751-0.4859535907*I; assign(%); VL1_A:=abs(VR1); phi_VL1:= argument(VL1);`	VL1 = -0.6211468751 - 0.4859535907 I VL1_A := 1.000274830 phi_VL1 := -2.477707611
`VR2=0.3791270396- 0.4846009587*I;assign(%); VR2_A:=abs(VR2); phi_VR2:= argument(VR2);`	VR2 = 0.3791270396 - 0.4846009587*I VR2_A := 0.6152848132 phi_VR2 := -0.9069112841
`IC1 = 0.001189837411 - 0.0002409482185*I; assign(%); IC1_A:=abs(IC1); phi_IC1:= argument(IC1);`	IC1 = 0.001189837411 - 0.0002409482185 I IC1_A := 0.001213988924 phi_IC1 := -0.1998032056
`IR1 = 0.001000273915 + 1.352632182e-6*I; IR1_A:=abs(IR1); phi_IR1:= argument(IR1);`	IR1 = 0.001000273915 + (1.352632182*10^(-6))*I IR1_A := 0.001000274830 phi_IR1 := 0.001352260953
`IL1 = 0.0001895635198 - 0.0002423004793*I; assign(%); IL1_A:=abs(IL1); phi_IL1:= argument(IL1);`	IL1 = 0.0001895635198 - 0.0002423004793 I IL1_A := 0.0003076424066 phi_IL1 := -0.9069112840
`IR2 = 0.0001895635198 - 0.0002423004794*I; assign(%); IR2_A:=abs(IR2); phi_IR2:= argument(IR2);`	IR2 = 0.0001895635198 - 0.0002423004794 I IR2_A := 0.0003076424066 phi_IR2 := -0.9069112842

Translating these values in time and phasor domain gives us:

	Time Domain	Phasor Domain
Voltage of Capacitor (VC1)	1.38 cos($8000\pi t - 1.77$)	1.38$e^{-1.77j}$
Voltage of Resistor (VR1)	1000 cos($8000\pi t$-0.00135)	1000$e^{-0.00135j}$
Voltage of Inductor (VL1)	1000 cos($8000\pi t$-2.48)	1000$e^{-2.48j}$
Voltage of Resistor (VR2)	615.3 cos($8000\pi t$-0.907)	615.3$e^{-0.907j}$
Current of Capacitor (IC1)	1.21 cos($8000\pi t$-0.200)	1.21$e^{-0.2j}$
Current of Resistor (IR1)	1.00 cos($8000\pi t$ +0.00135)	1.00$e^{0.00135j}$
Current of Inductor (IL1)	0.308 cos($8000\pi t$-0.907)	0.308$e^{-0.907j}$
Current of Resistor (IR2)	0.308 cos($8000\pi t$-0.907)	0.308$e^{-0.907j}$

Digital Solution:

My digital solution includes two parts. First, I did a single-frequency AC sweep across the whole circuit. This allowed me to get the individual voltages and currents across each component.

Design2
Single Frequency AC Analysis @ 4000 Hz

	Variable	Real	Imaginary
1	V(a)	1.00000	0.00000e+00
2	V(b)	1.00027	1.35263 m
3	V(c)	379.12704 m	-484.60096 m
4	I(C1)	1.18984 m	-240.94785 u
5	I(R1)	1.00027 m	1.35263 u
6	I(R2)	-189.56352 u	242.30048 u
7	I(L1)	189.56352 u	-242.30048 u
8	I(V1)	-1.18984 m	240.94785 u

Design2
Single Frequency AC Analysis @ 4000 Hz

	Variable	Magnitude	Phase (deg)
1	V(a)	1.00000	0.00000e+00
2	V(b)	1.00027	77.47885 m
3	V(c)	615.28481 m	-51.96219
4	I(C1)	1.21399 m	-11.44786
5	I(R1)	1.00027 m	77.47885 m
6	I(R2)	307.64241 u	128.03781
7	I(L1)	307.64241 u	-51.96219
8	I(V1)	1.21399 m	168.55214

Maple input	Maple output

va:= 1: convert(%,polar);
vb:= 1.00027+1.35263e-3*I: convert(%,polar);
vc:= 379.12704e-3 - 0.48460096*I: convert(%,polar);
IC1:= 1.18984e-3 - 240.94785e-6*I: convert(%,polar);
IR1:= 1.00027e-3 + 1.35263e-6*I: convert(%,polar);
IR2:= -189.56352e-6 + 242.30048e-6*I: convert(%,polar);
IL1:= 189.56352e-6 - 242.30048e-6*I: convert(%,polar);
	polar(1, 0)
	
	polar(1.000270915, 0.001352264064)
	
	polar(0.6152848144, -0.9069112849)
	
	polar(0.001213991389, -0.1998024848)
	
	polar(0.001000270915, 0.001352264064)
	
	polar(0.0003076424072, 2.234681369)
	
	polar(0.0003076424072, -0.9069112849)

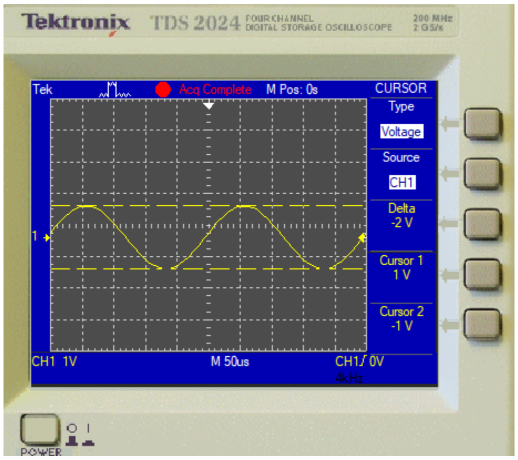
	Phasor Form	Time Domain
Voltage at node A	1	1cos(8000πt)
Voltage at node B	$1.00e^{0.00135j}$	1 cos(8000πt + 0.00135)
Voltage at node C	$0.615e^{-0.907j}$	0.615 cos(8000πt – 0.907)
Current through C1	$0.00121e^{-0.200j}$	0.00121 cos(8000πt - 0.2)
Current through R1	$0.00100e^{0.00135j}$	0.001 cos(8000πt + 0.00135)
Current through R2	$0.000308e^{2.23j}$	0.000308 cos(8000πt + 2.23)
Current through L1	$0.000308e^{-0.907j}$	0.000308 cos(8000πt – 0.907)

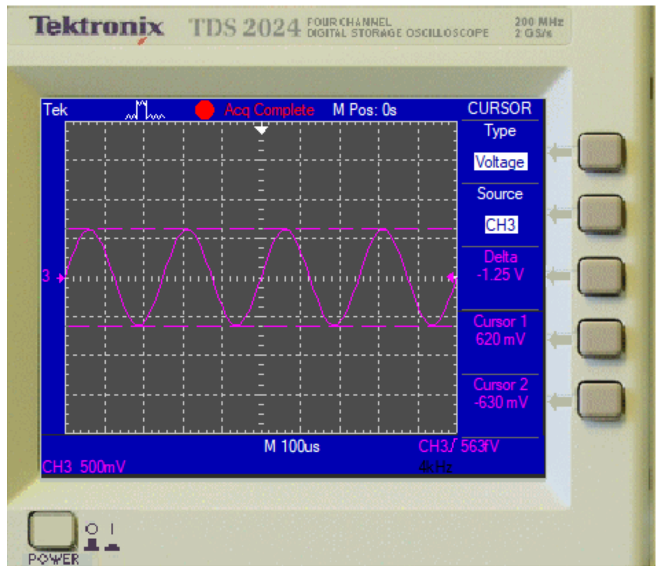
The table shows the time domain and phasor form for voltages at nodes and the currents passing through each component. They have the same values as that of the analytical solution. Thus, these values agree with the analysis calculations above.

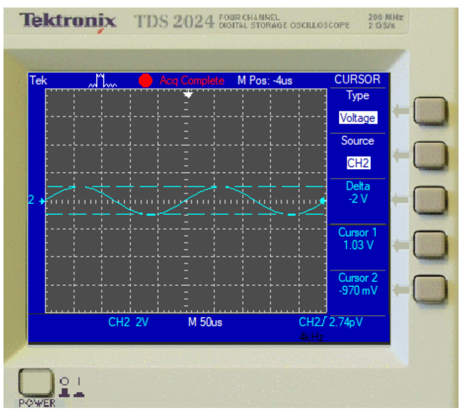
I also used the Tektronix Oscilloscope to be able to visualize the frequency generation. First, I measured the voltages at each node.

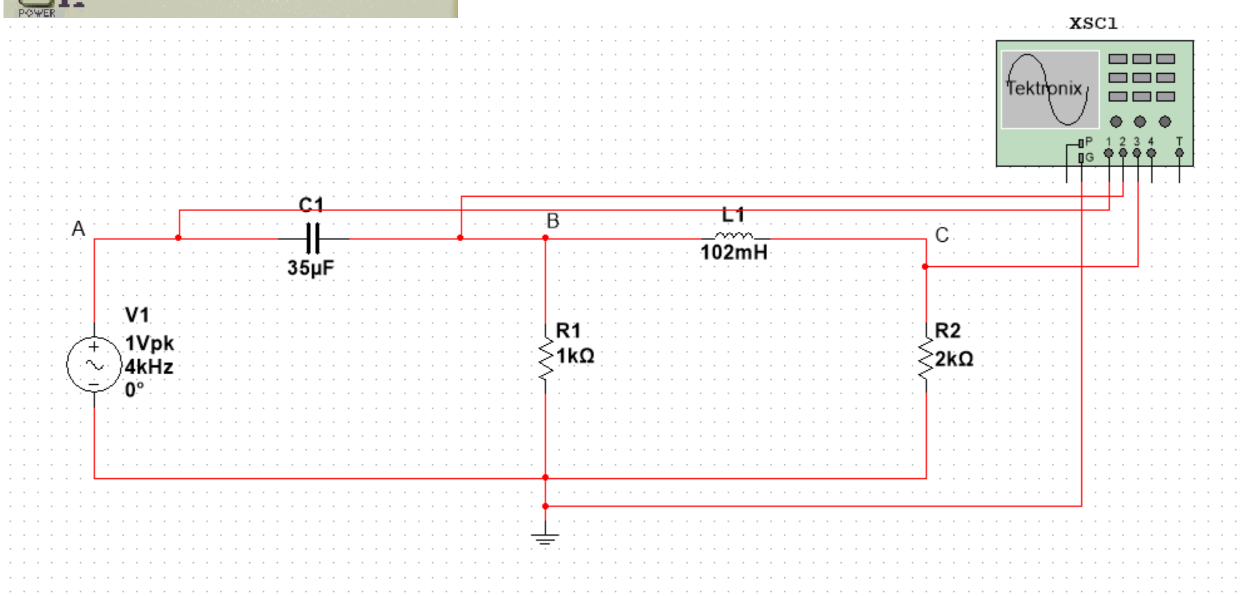
The peak voltages measured were 2 V, 2 V and 1.25 V respectively.

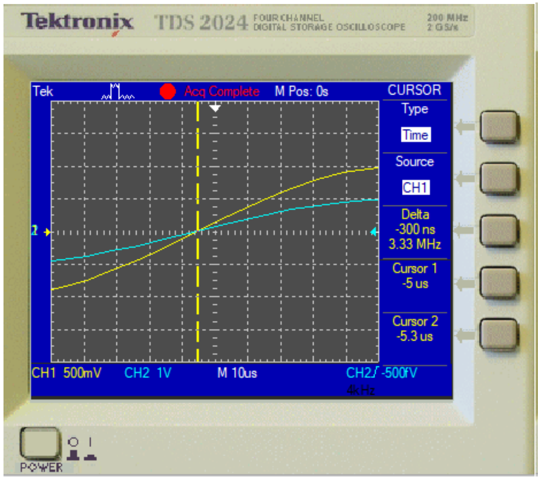
Next, I used the node A as the reference node (channel one in the multisim circuit) and compared it to node B and C.

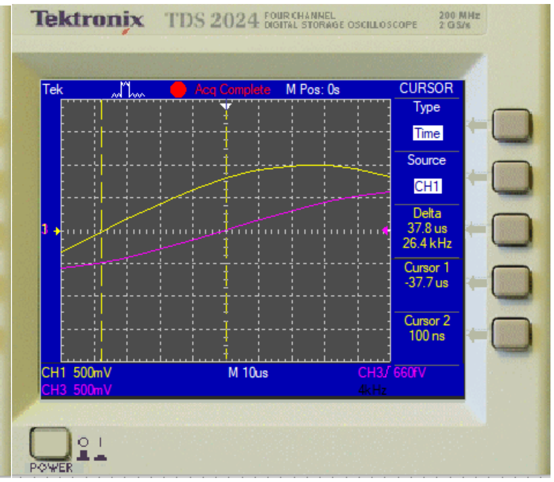








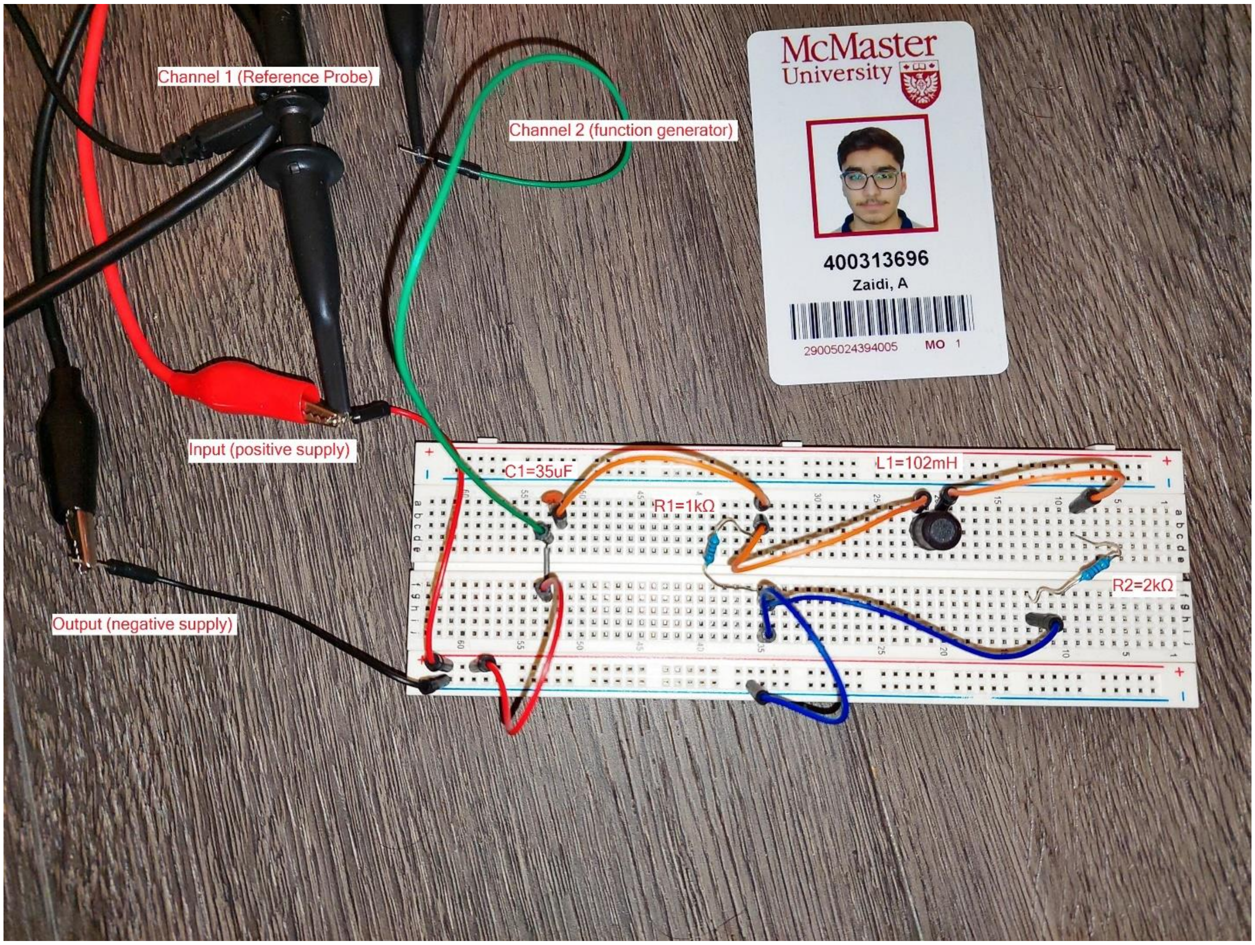




The time difference for B was close to 0 s but the oscilloscope measured 300 ns which was a result of a low sensitivity of using the cursor in multisim. Moreover, for C the time difference was 37.8 μs.

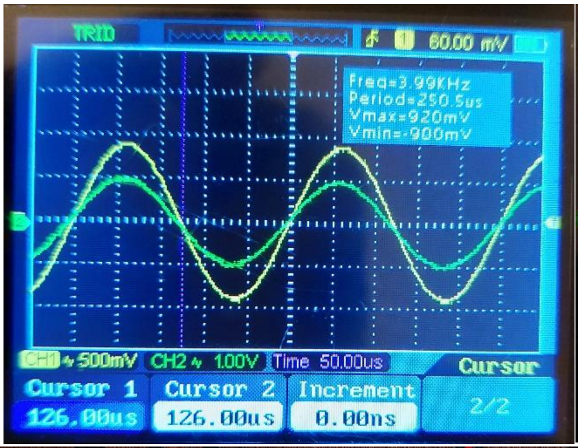
Physical Solution:

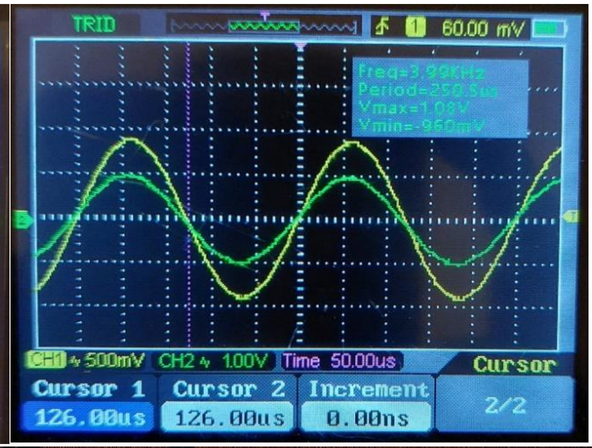
This is my physical solution which I made on a breadboard.

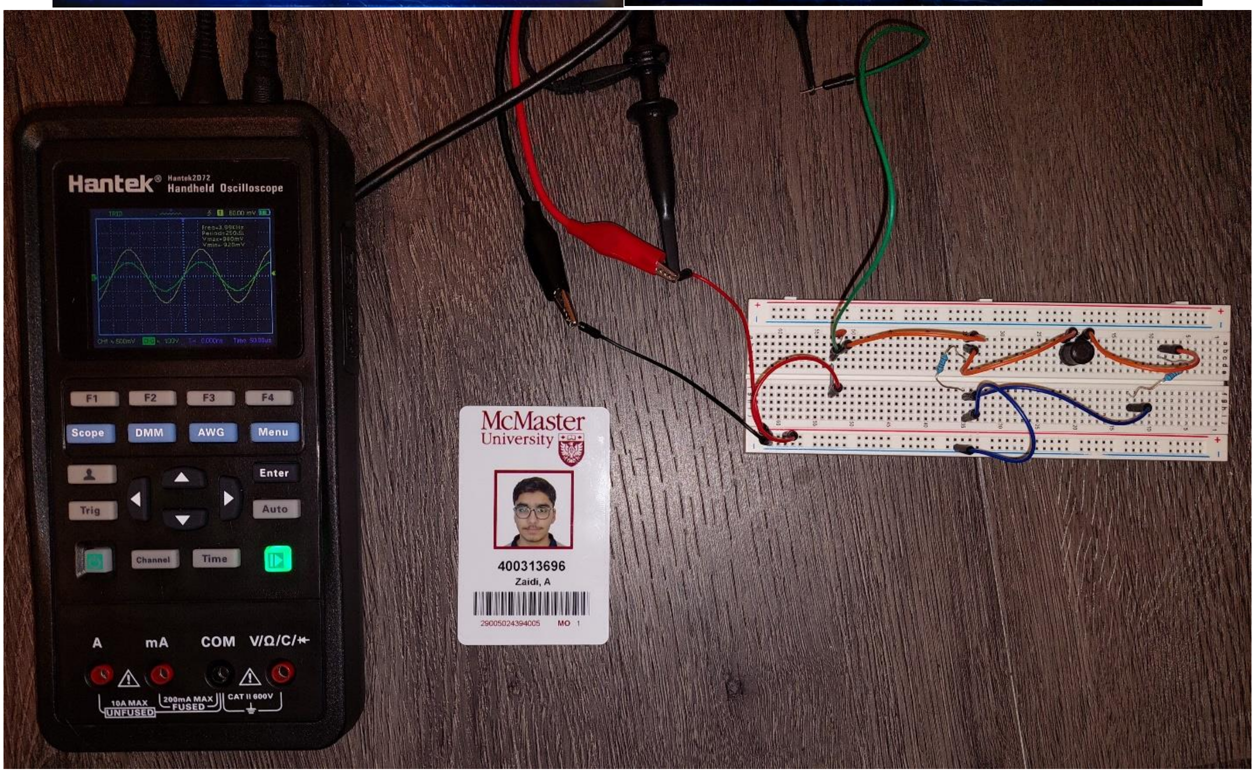


I measured the voltage at node A, B and C using the channel 2 (green wire) of the Hantek Oscilloscope. I connected the channel 1 to the input voltage to be a reference voltage for the measurements.

Node A:

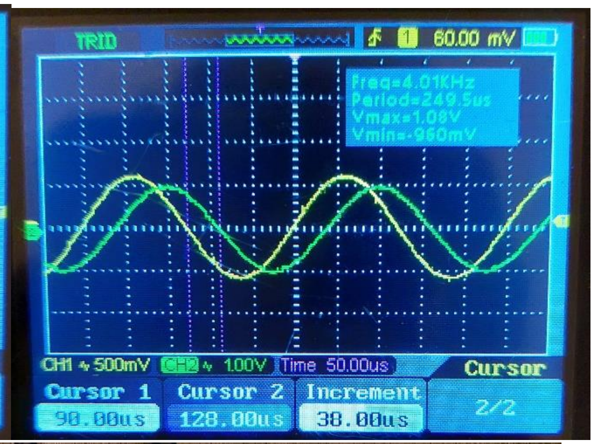

60.00 mV
Freq=3.99KHz
Period=250.5us
Vmax=920mV
Vmin=-900mV
CH1 500mV
CH2 1.00V
Time 50.00us
Cursor
Cursor 1
126.00us
Cursor 2
126.00us
Increment
0.00ns
2/2

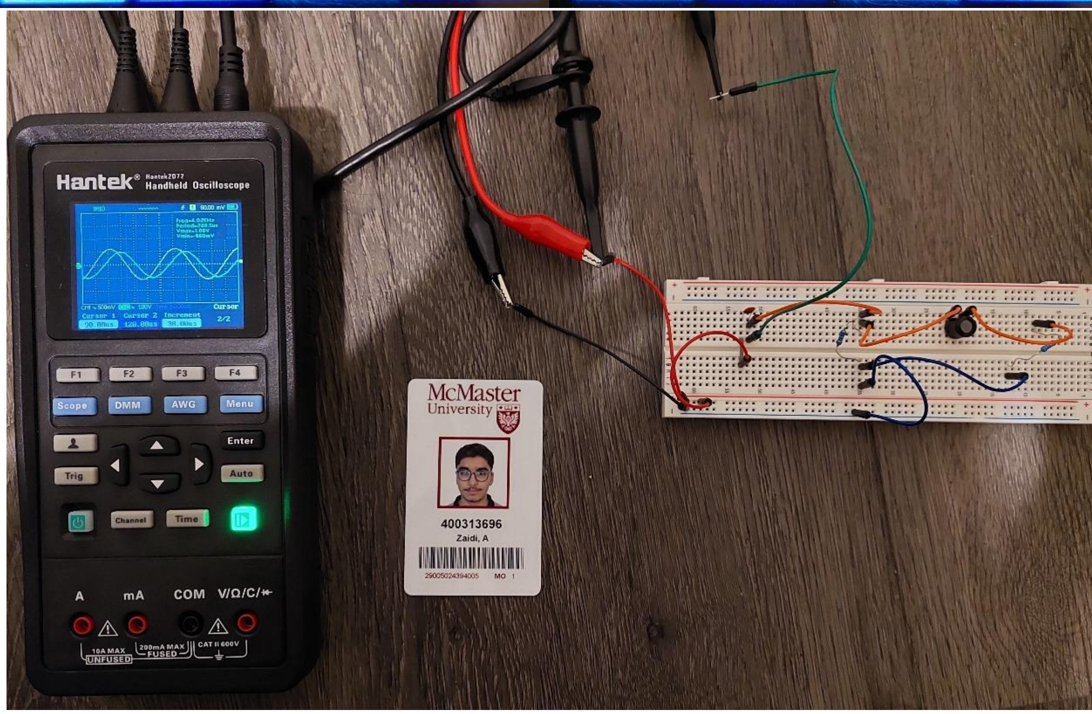

60.00 mV
Freq=3.99KHz
Period=250.5us
Vmax=1.08V
Vmin=-960mV
CH1 500mV
CH2 1.00V
Time 50.00us
Cursor
Cursor 1
126.00us
Cursor 2
126.00us
Increment
0.00ns
2/2


Hantek
Handheld Oscilloscope
F1
F2
F3
F4
Scope
DMM
AWG
Menu
Enter
Trig
Auto
Channel
Time
A
mA
COM
McMaster
University
400313696
Zaidi, A

Node B:

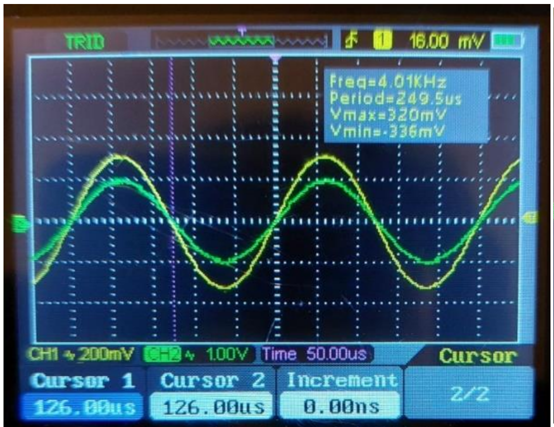

TRIG
60.00 mV
Freq=4.00KHz
Period=250us
Vmax=600mV
Vmin=-600mV
CH1 500mV
CH2 1.00V
Time 50.00us
Cursor
Cursor 1
90.00us
Cursor 2
128.00us
Increment
38.00us
2/2

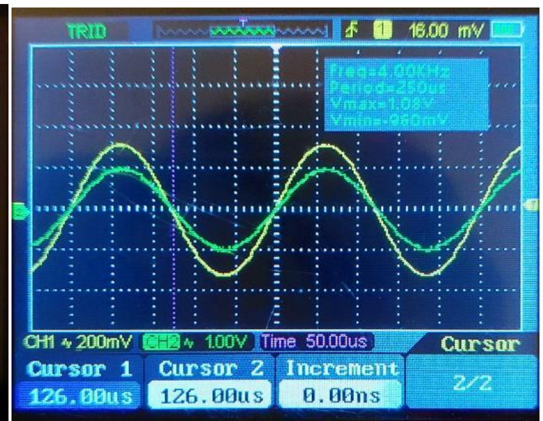

TRIG
60.00 mV
Freq=4.01KHz
Period=249.5us
Vmax=1.08V
Vmin=-960mV
CH1 500mV
CH2 1.00V
Time 50.00us
Cursor
Cursor 1
90.00us
Cursor 2
128.00us
Increment
38.00us
2/2

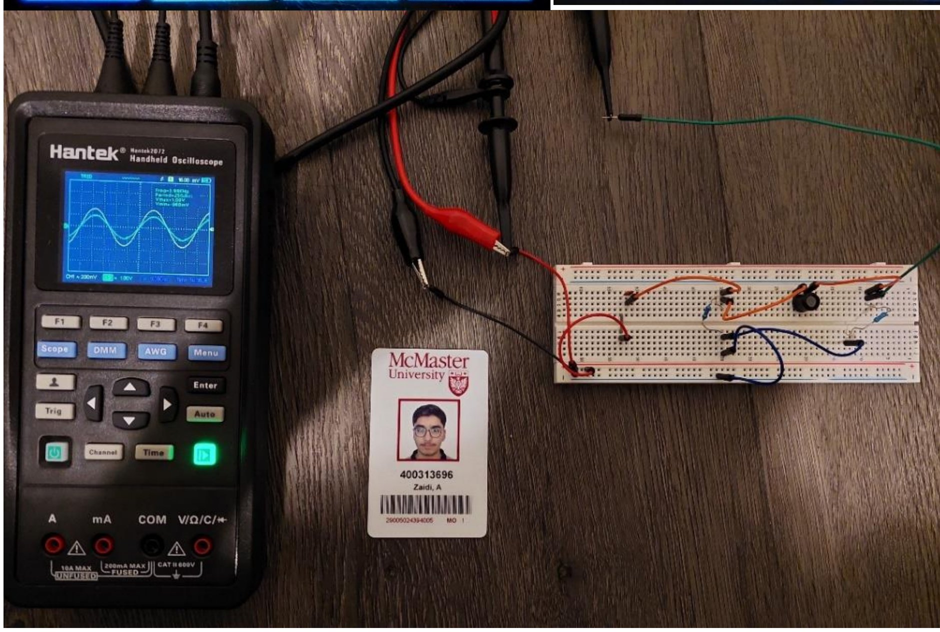

Hantek
Handheld Oscilloscope
F1
F2
F3
F4
Scope
DMM
AWG
Menu
Enter
Trig
Auto
Channel
Time
A
mA
COM
V/Ω/C/
McMaster
University
400313696
Zaidi, A

Syed Ali Zaidi (zaidis33)

Node C:







The results of the physical solution are summarized in the table below. Note that the amplitude of the voltage functions are $\frac{V_{max} - V_{min}}{2}$ and the phase difference is the time lag multiplied by omega.

	f (kHz)	Time lag (μ s)	Max V (mV)	Min V (mV)	Time Domain (mV)	Phasor Domain (mV)
Node A Voltage	3.99 kHz	0	920	900	10cos(7980πt)	10
Node B Voltage	4.01 kHz	38.00	960	249.5	355.25cos(8020πt+0.957)	355.25$e^{0.957j}$
Node C Voltage	4.00 kHz	0	1080	960	60cos(8000πt)	60

Now we shall find out the current through each resistor by dividing by its impedance. We do this by first calculating the uncertainty of the voltages. This is done by multiplying the voltage by 1.2% and adding a 5 to the smallest digit. Then we calculate the uncertainty in resistances. This is done by multiplying by 1% and adding 3 to the smallest digit.

	Voltage through components [mV]	Resistance of components [kΩ]	Voltage Relative Uncertainty	Resistance Relative Uncertainty	Current Relative Uncertainty	Current through components [μA]
R1	355 ± 9	0.997 ± 0.04	2.53%	4.01%	6.54%	356 ± 20
R2	60 ± 6	2.002 ± 0.05	10.0%	2.50%	12.5%	30 ± 4

By getting these currents we can get the remaining currents (through the inductor and capacitor). The current through the inductor will equal to the current through R2 (both components in series). The current through the capacitor will be the sum of the currents of R1 and R2 (i.e.: 386 ± 24). This is deduced from Kirchhoff's first law (KCL).

Analysis:

In this lab, the values I got from the analysis and digital solutions were very close. The physical, however, had some deviation. This arose because of many errors from the equipment and experimental inaccuracies. The analytical and digital solutions can deal with numbers in high precision, but the Hantek could not thus a rounding error could be a source of error.

The positioning of the cursor by me in the Hantek was also very inaccurate as I could not make out the exact point where the curve would cross the x-axis thus a systemic error was seen in the phase.

Moreover, we could not measure the inductance of the inductor so by taking an average of random inductor values (found in the lab), I found the inductance to be 102mH. This is a very unreliable result as my inductor could have a different value causing another systemic error in the physical solution.

The relationship I found out with this lab is that the voltage leads the current in an inductor and the current leads the voltage in a capacitor. This relationship was deduced after the results of my digital and physical solutions by looking at the phase differences between the sine waves. This is because when a voltage is applied to an inductor, it responds by resisting to that change thus current builds up slowly than voltage. This causes current to lag the voltage. On the other hand, the capacitor accepts current by the equation:

$$V = \frac{Q}{C} = \frac{I \Delta t}{C}$$

This shows that the voltage is directly proportional to the current causing it to lead so that the capacitor can charge up and raise its voltage.

Conclusion:

This lab was a very new area for me to explore. Being exposed to A.C circuits for the first time was very challenging but I loved the approach of working on the practical aspect of it with little theory knowledge and deduce what happens to the current and voltage of components through doing it practically. I am now comfortable in converting a voltage to its phasor and time domain form. The most important thing I learnt in this lab included using the oscilloscope both digitally and physically. Being able to position cursors and reading off the voltage was a very useful skill. I had a few troubles setting up the circuit and had some confusion with the channel 1 and channel 2 probes but with some in-person help in the lab, I figured out how to effectively set up an A.C circuit. I am now more confident in using an A.C supply as the voltage source which will enable me to perform well in future labs.

Appendix:

```
> va:=1: r1:=1e3: r2:=2e3:
c:=35e-6: l:=102e-3: f:=4e3: omega:=2*Pi*f:
j:=I: zc:=1/(j*omega*c): zl:=j*omega*l:
reff:= 1/((1/r1)+(1/(zl+r2))):
solve([
vb=(reff/(reff+zc))*va, vc=(r2/(r2+zl))*vb,
VC1=va-vb,
VR1=vb,
VL1=vb-vc,
VR2=vc,
IC1=VC1/zc,
IR1=VR1/r1,
IL1=VL1/zl,
IR2=VR2/r2])
```

$\{IC1 = 0.001189837411 - 0.0002409482185\,I, IL1 = 0.0001895635198 - 0.0002423004793\,I, IR1 = 0.001000273915 + 1.352632182 \times 10^{-6}\,I, IR2 = 0.0001895635198 - 0.0002423004794\,I, VC1 = -0.0002739150000 - 0.001352632182\,I, VL1 = 0.6211468754 + 0.4859535909\,I, VR1 = 1.000273915 + 0.001352632182\,I, VR2 = 0.3791270396 - 0.4846009587\,I, vb = 1.000273915 + 0.001352632182\,I, vc = 0.3791270396 - 0.4846009587\,I\}$

```
> va:=1: r1:=1e3: r2:=2e3:
c:=35e-6: l:=102e-3: f:=4e3: omega:=2*Pi*f:
j:=I: zc:=1/(j*omega*c): zl:=j*omega*l:
solve([
va-i1*zc-(i1+i2)*r1=0,
(i1+i2)*r1+i2*zl+i2*r2=0,
VC1=i1*zc,
VR1=(i1+i2)*r1,
VL1=i2*zl,
VR2=i2*r2])
```

$\{VC1 = -0.0002739145777 - 0.001352632208\,I, VL1 = -0.6211468751 - 0.4859535907\,I, VR1 = 1.000273915 + 0.001352632208\,I, VR2 = -0.3791270395 + 0.4846009585\,I, i1 = 0.001189837434 - 0.0002409478470\,I, i2 = -0.0001895635197 + 0.0002423004792\,I\}$