Experiment 7
Rectifiers, Capacitors and Inductors

Ahmet Akman 2442366
Assistant : Uğur Berkay Saraç

December 28, 2021

Contents

1 Introduction

In this experiment, as students, we are expected to experiment with rectifiers, capacitor and inductor circuits by completing the steps described in the seventh experiment laboratory manual. Throughout these steps, the half full rectifier circuit structures and ripple voltages are expected to be learned. The output versus input characteristics is observed by connecting the signal generator to the oscilloscope and the circuit. Also, the measurement techniques for capacitance of capacitors and inductance of inductors are expected to be expressed and experimented. The results of the steps were recorded and plotted for further comments.

2 Experimental Results

In this section, the results of Experiment 7 are discussed.

2.1 Step 1

In this step, circuit shown in the Figure 1 is constructed.

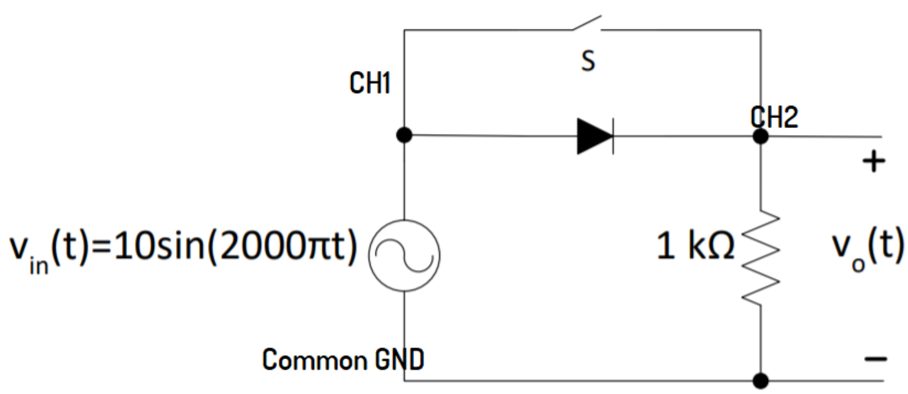

Figure 1: Half wave rectifier circuit

2.1.1 a)

By connecting the channels of the oscilloscope to the CH1 and CH2 nodes, the plot given in Figure 2 is obtained.

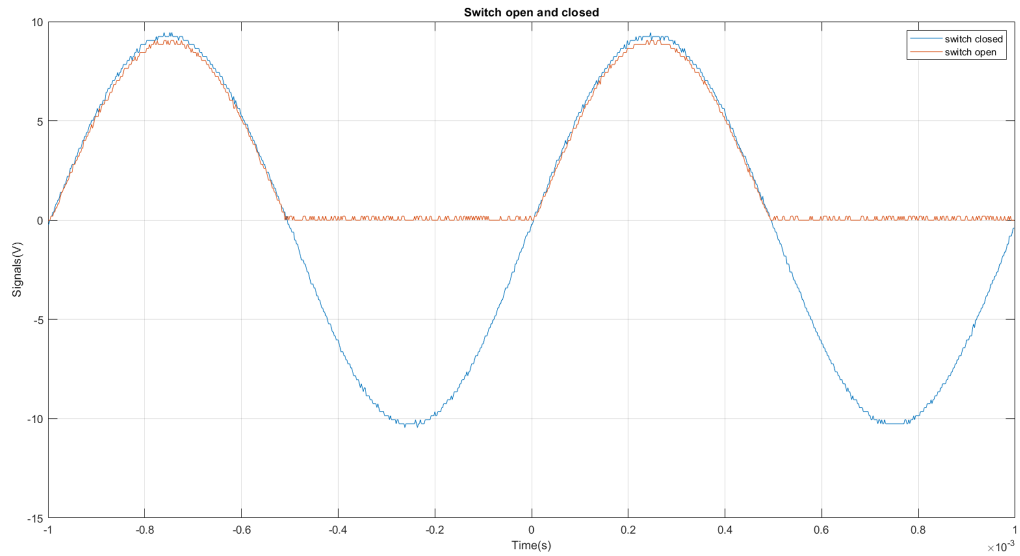

Figure 2: Output waveforms

As a result, it can be said that half wave rectifier is able to allow only positive current to flow.

2.1.2 b)

There is a little difference of maximum values of the waveforms. This can be stemmed from the fact that the diode component do not behave ideally and have an opening voltage. This opening voltage should be passed in order diode to allow current pass through.

2.2 Step 2

For this step circuit shown in Figure 3 is set with LEDs.

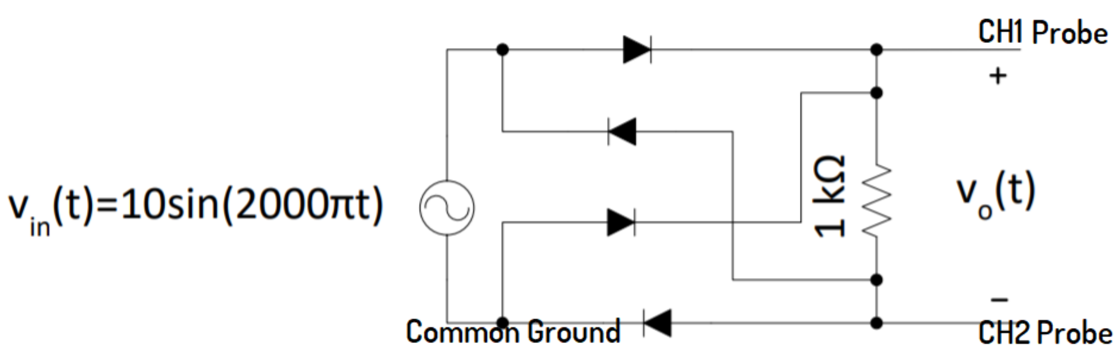

Figure 3: Full wave rectifier circuit

2.2.1 a)

The plot given in Figure 4 is obtained using the math function of the DSO.

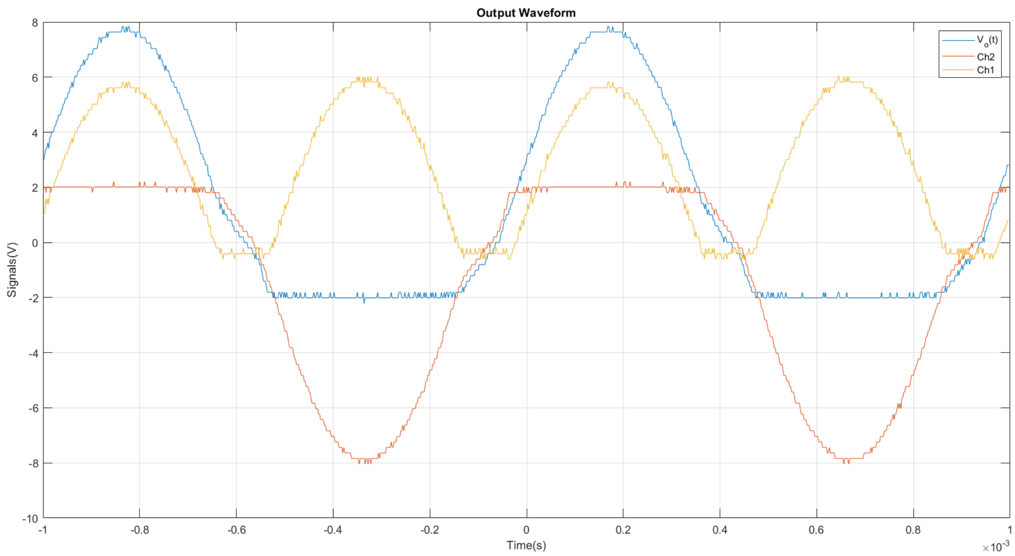

Figure 4: Output waveforms

To solve the grounding problem oscilloscope probes are connected to the nodes expressed in Figure 3. Then the CH2 signal is subtracted from the CH1 signal to get resulting signal. Also even though the input Vp is 10V the resulting signal have the amplitude of 6V. This difference means diodes have 2V opening voltage (2 volts per diode).

2.2.2 b)

The frequency of the signal generator is adjusted to 1Hz. As a result the LEDs have started to blink visibly. Two of the LEDs blinked at the same time, which means when the voltage is positive two of the diodes are active and when the voltage is negative other two of the diodes are active. When the frequency increases , the blink becomes less noticable.

2.2.3 c)

The frequency is set to 1kHz and a 0.47μF capacitor is connected parallel to the resistor. The plot given in Figure 5 is obtained.

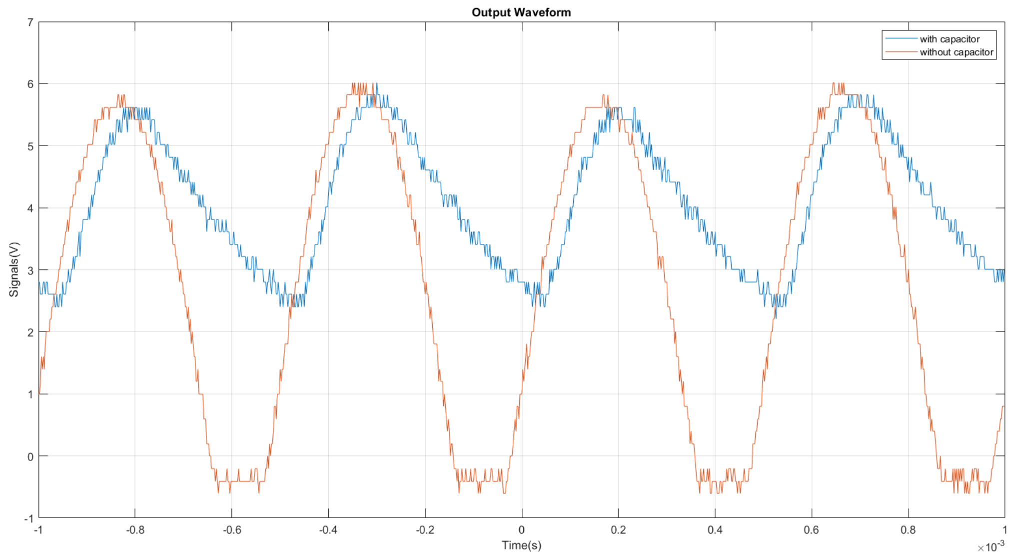

Figure 5: Output waveforms

It can be stated that, when a capacitor is connected the ripple voltage decreases ,and the output look more alike DC. The capacitor discharges when voltage decreasing and helps the node remain its voltage.

2.3 Step 3

The circuit given in Figure 6 is built for the Step 3.

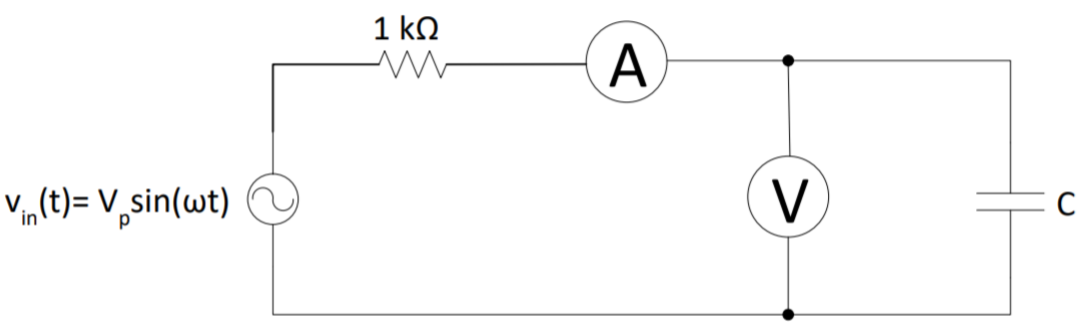

Figure 6: Capacitor measurement circuit

Then the RMS voltage measurements are made using digital multimeter. The measurements are given in Table 1.

Table 1: Measurements for the capacitor circuit

	Capacitor	Resistor
Voltage Reading	1.1518 V	1.69 V

By dividing the voltage of the resistor to its resistance value, the current value is obtained as " 0.00169A". Then using the equation,

$$|z_c| = \frac{1}{\omega C} = \frac{V_{RMS}}{i_{RMS}}$$

so,

$$C = \frac{i_{RMS}}{V_{RMS}\omega}$$

The capacitor value is obtained and the LC meter measurement are given in Table 2ç

Table 2: Measurements for the Capacitor

Datasheet	Calculation	LC meter Measurement
0.47μF	0.4670461 μF	0.48 μF

It can be said that the calculations are consistent with the datasheet and LC meter measurement. The deviation might be stemmed from the negleteced resistances of cables and capacitor. Also, the resistor value might not be exactly 1kΩ.

2.4 Step 4

For Step 4 the circuit given in Figure 7 is constructed.

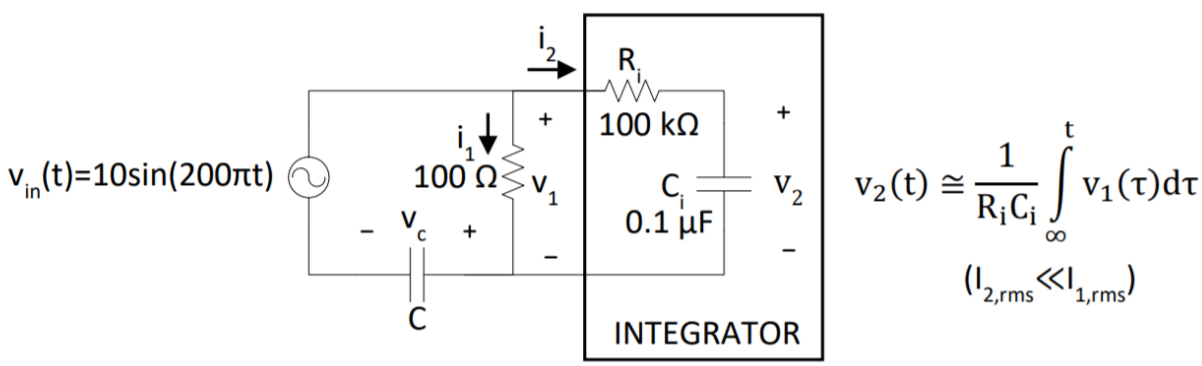

Figure 7: Circuit for the capacitance finding method

The circuit is reconstructed in LTSPice simulation environment which is shown in Figure 8.

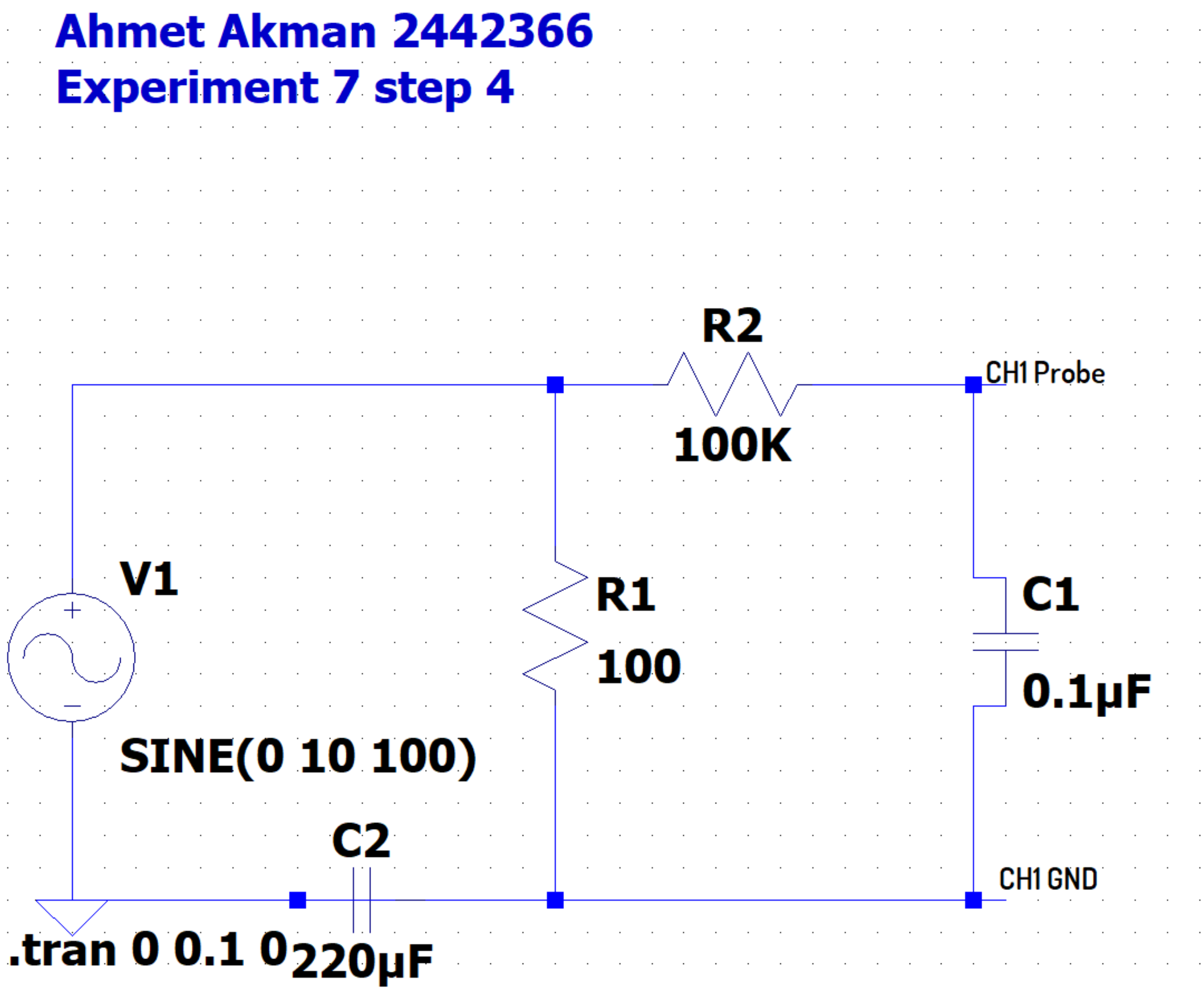

Figure 8: Simulation circuit for the capacitance finding method

Then the q-v characteristics of the capacitor C is plotted as given in Figure 9.

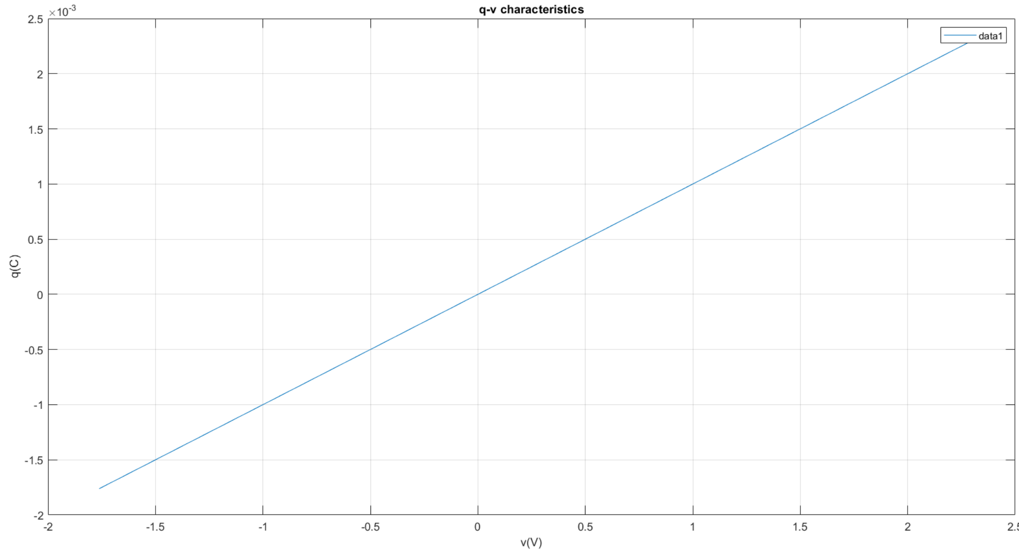

Figure 9: q-v characteristics

The plot is obtained using only V_2 measurement and calculations as follows,

$$i_2(t) = C_1\frac{dV_2}{dt} = \frac{V_1(t)}{R_1}$$

$$V_2(t) = \frac{1}{R_iC_i} * \int_{-\infty}^{t} V_1(\tau)\,d\tau$$

$$V_2(t) = \frac{R}{R_iC_i} * \int_{-\infty}^{t} i_1(\tau)\,d\tau$$

So,

$$q(t) = CV_c(t) = \int_{-\infty}^{t} i_1(\tau)\,\tau = \frac{R_iC_iV_2(t)}{R}$$

This mathematical expression helped us to obtain q-v characteristics of the capacitor C on the circuit given in Figure 8.

2.5 Step 5

For this step the circuit shown in Figure 6 used ,but the only difference is there used an inductor instead of a capacitor. Firstly, the measurements are made using wooden inductor. The measurements are given in Table 3.

Table 3: Measurements for the wooden inductor circuit

	Inductor	Resistor
Voltage Reading	0.68321 V	1.862 V

Then measurements are conducted for the compact type inductor which is given in Table 4.

Table 4: Measurements for the compact inductor circuit

	Inductor	Resistor
Voltage Reading	0.59929 V	1.912 V

Also the inductance and resistance values are measured for both wooden and the compact inductor. For inductance LC meter is used. Resistance is measured via digital multimeter. The results are given in Table 5.

Table 5: Measurements for the inductors

	Wooden	Compact
LC Meter Reading	0.15 H	0.08 H
Resistance Reading	29.13 Ω	3.950 Ω

Therefore the inductance values are calculated using the measurements in Table 7 and the following equation,

$$|Z_c| = \omega L = \frac{V_{RMS}}{i_{RMS}}$$

so,

$$C = \frac{V_{RMS}}{i_{RMS}\omega}$$

The results are given in Table 6.

Table 6: Calculation result for the inductors

	Wooden	Compact
Calculation result	0.1168 H	00.0998 H

The results shows that, our approximations are quite consistent with the real world value. The deviation is stemmed from the neglected resistance of the inductors which are given in Table 5. By adding them to the equation , the calculation can be resulted more accurately.

2.6 Step 6

Circuit given in Figure 10 is set for this step. The variables set as V_{in} = 10sin(2*π ft) V , f = 5kHz and L_1 = H.

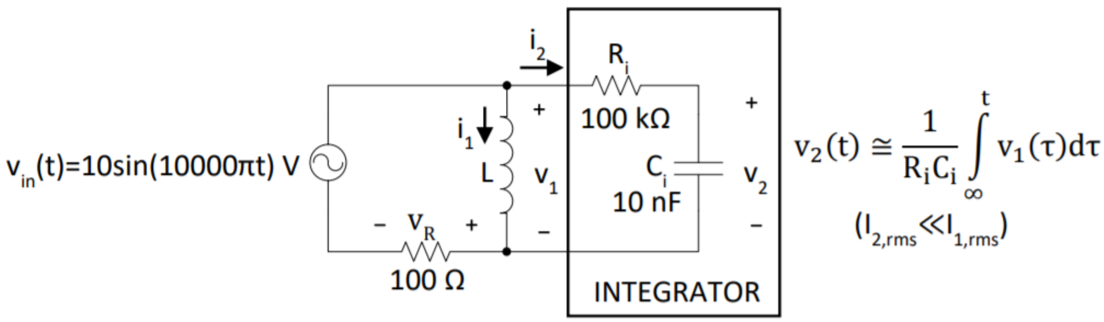

Figure 10: Circuit for the inductance finding method

2.6.1 a)

The circuit schematic given in Figure 11 is constructed in LTSPice simulation environment.

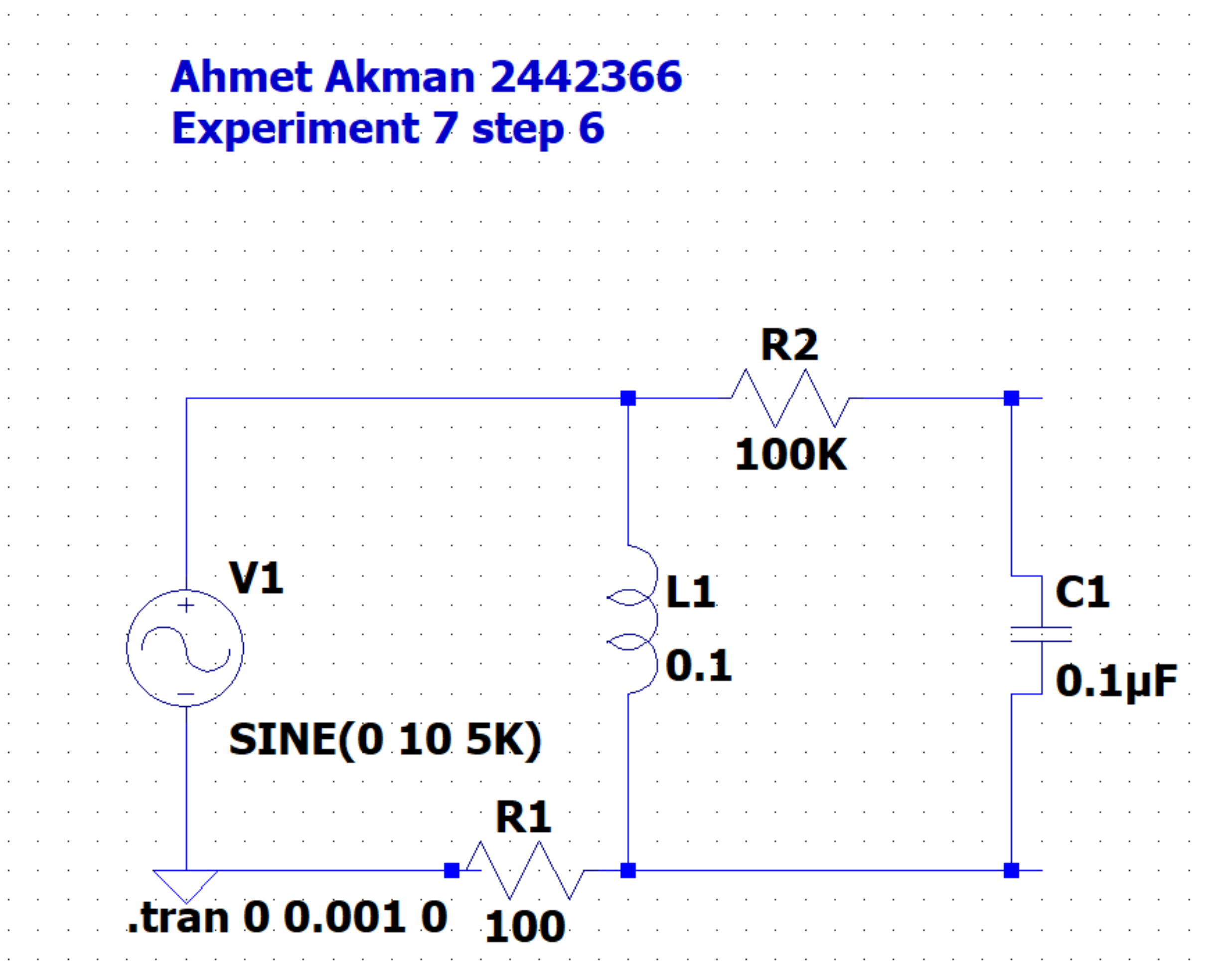

Figure 11: Simulation circuit for the inductance finding method

Then using the following relation obtained in the Preliminary work the plot shown in

Figure 12 is obtained.

$$i_1 * 100 = V_R$$

$$\phi(t) = Li(t) = \frac{LV_R}{100}$$

by the equation given in Figure 10,

$$V_2 R_i C_i = Li(t)$$

$$V_2 R_i C_i = \frac{LV_R}{100}$$

$$L = \frac{100 V_2 R_i C_i}{V_R}$$

also,

$$\phi(t) = V_2 R_i C_i$$

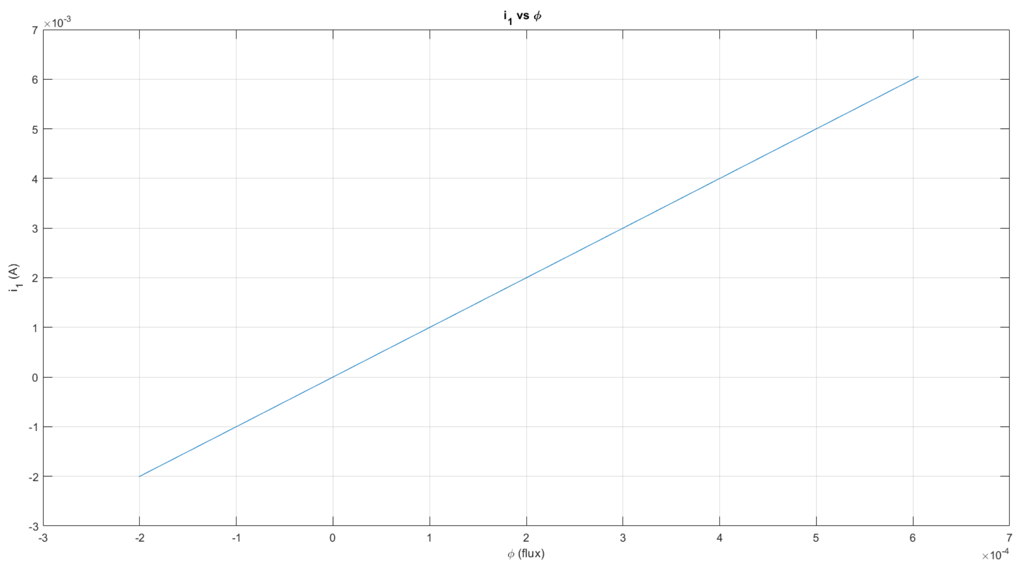

Figure 12: $\phi vs i$ characteristics plot

It can be seen that there is a linear relation between flux and current on an inductor.

2.6.2 b)

3 Conclusion

The non-ideal behavior of the components is compared with the ideal simulation plots.

In conclusion, in experiment 6, "Operational Amplifiers-II," as students, we have learned how various functional circuit setups of Op-Amps can be constructed. Preliminary laboratory work is done via simulations of the Op-Amp circuits in an LTSpice environment and by hand calculations. As students, we have observed how the amplifying job can be done in two-stage and its advantages. We have seen the effect of non-linear feedback on the Op-Amp setup. The characteristics of the negative resistance converter are observed. Also, implementing darkness and lightness sensors has experienced a real-life design approach. Lastly, a difference amplifier circuit is set, and the characteristics are observed with measurements. To sum up, in this experiment, as students, we have experimented with how different kinds of operational amplifier circuits operate.

Appendix I

Total time spent on/during:

- Pre-lab preparation: 6.5 hours (including the preliminary work and simulations)
- Experimental work: 2 hours (hours spent in lab)
- Report writing: 6 hours

Appendix II

The outputs of the simulations are fetched from LTSpice and plotted in MATLAB.