Bilkent University

Electrical and Electronics Department

EE202-01 Lab 1 Report:

"Time-Domain and Frequency-Domain Analyses in LTSpice"

12/02/2024

Fatih Mehmet Çetin - 22201689

Purpose:

The main purpose of the software part of this lab assignment was to perform time-domain and frequency-domain analyses in LTSpice in a series of circuits, including RL&OPAMP circuits. The software lab involved importing the LM324 OPAMP model to LTSpice and using this specific type of OPAMP in the circuits we designed.

The main purpose of the second part of the experiment, was to implement the RL and OPAMP circuits we designed in the software part of the lab onto a breadboard or a PCB via solder and compare the hardware findings with the software ones.

Software Lab

Part 1: Transient (time-domain) Analysis

In the part 1.1 of the software lab, we were assigned to create a voltage divider circuit. Here you can see the circuit I've designed **(Figure 1.1.1),** and the input&output voltage plot **(Figure 1.1.2)**. The input voltage is a sinusoidal wave whose peak value is 8 Volts. The output voltage was measured as 5.50 Volts as expected **(Equation 1)**.

$$\frac{V_{out}}{V_{in}} = \frac{R_2}{R_2+R_1} = \frac{15}{21.8} = 5.50\,V \quad \textbf{(Equation 1)}$$

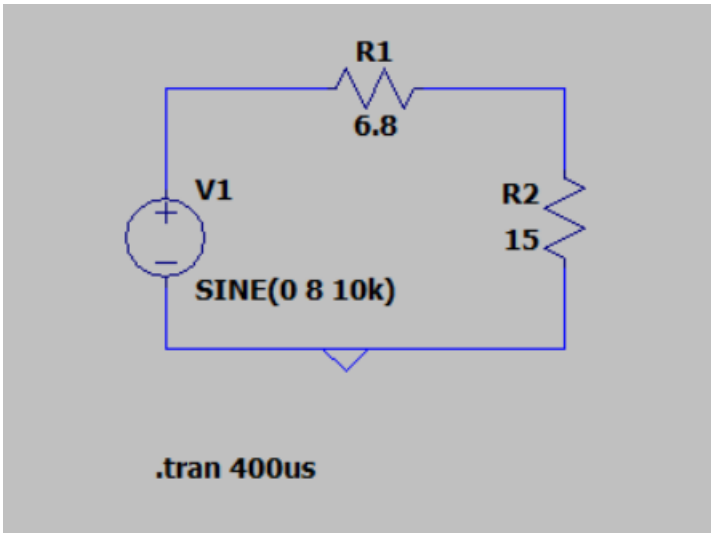

Figure 1.1.1: The Voltage Divider Circuit

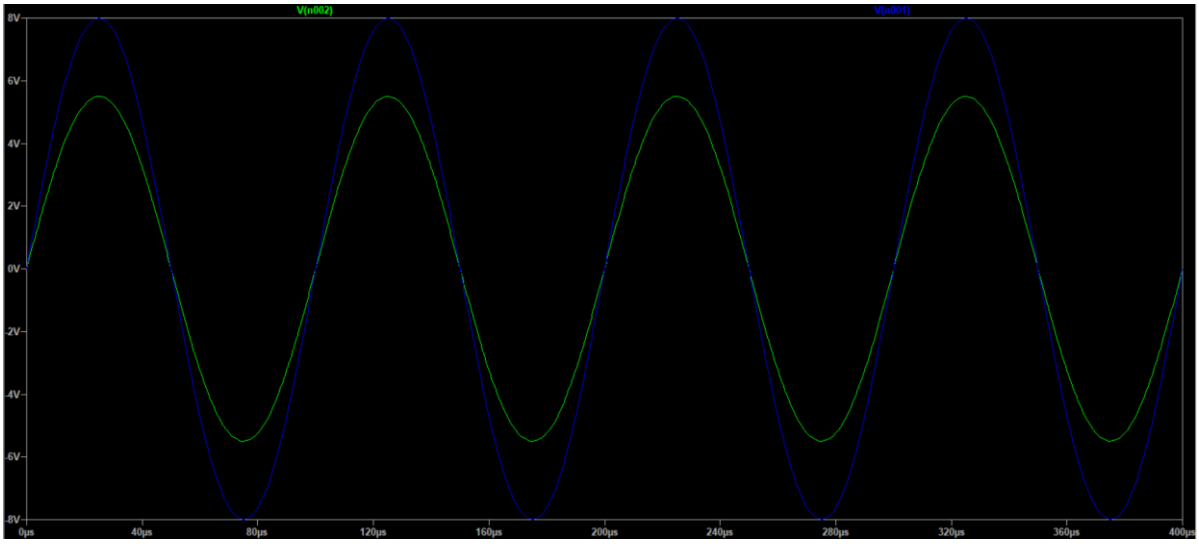

Figure 1.1.2: The Input&Output Voltage Plot of the voltage divider circuit

In the part 1.2 of the software lab, we were assigned to replace the R_2 with an inductor and compare the output voltage in different frequencies. Here are the circuits I've designed **(Figures 1.2.1, 1.2.3 & 1.2.5)** and their output voltage response accordingly **(Figures 1.2.2, 1.2.4 & 1.2.6)**. You can see the voltage divider formula on phasor domain **(Equation 2)**. This circuit gives high output voltage when the input frequency is high, therefore it is a high-pass filter.

$$\frac{V_{out}}{V_{in}} = \frac{jwL}{jwL+R_1} \quad \textbf{(Equation 2)}$$

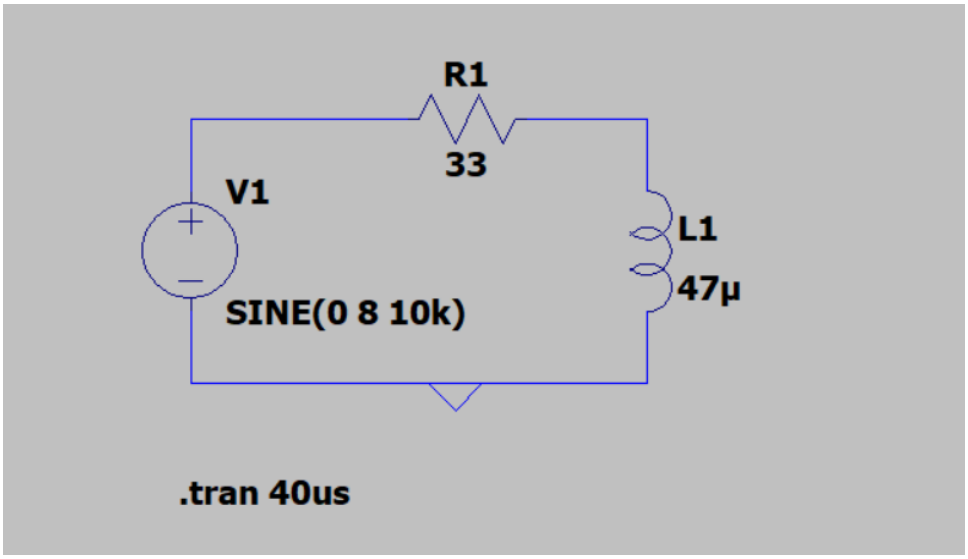

Figure 1.2.1: The circuit with input voltage frequency 10k Hz

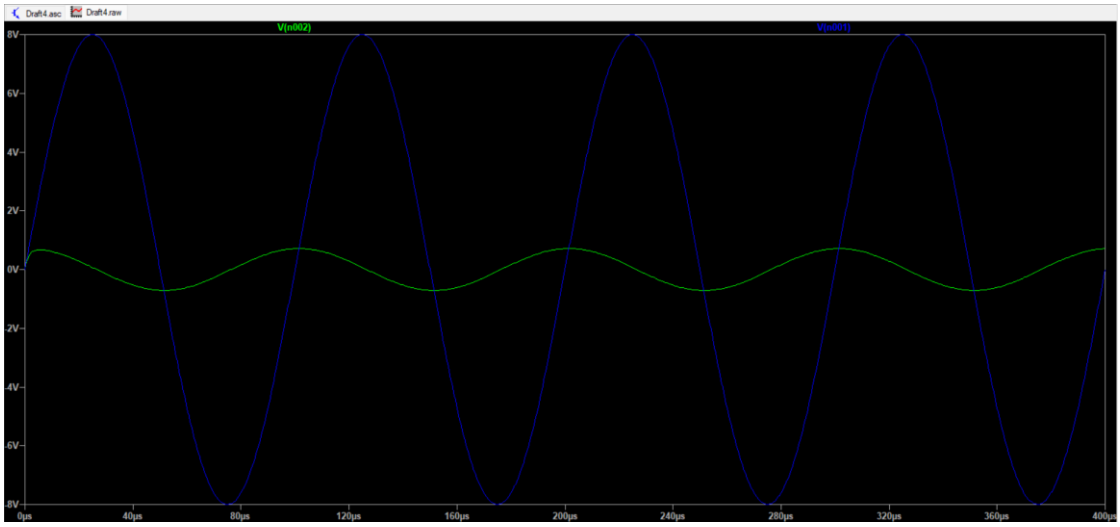

Figure 1.2.2: The input&output voltage plot with input voltage frequency 10k Hz

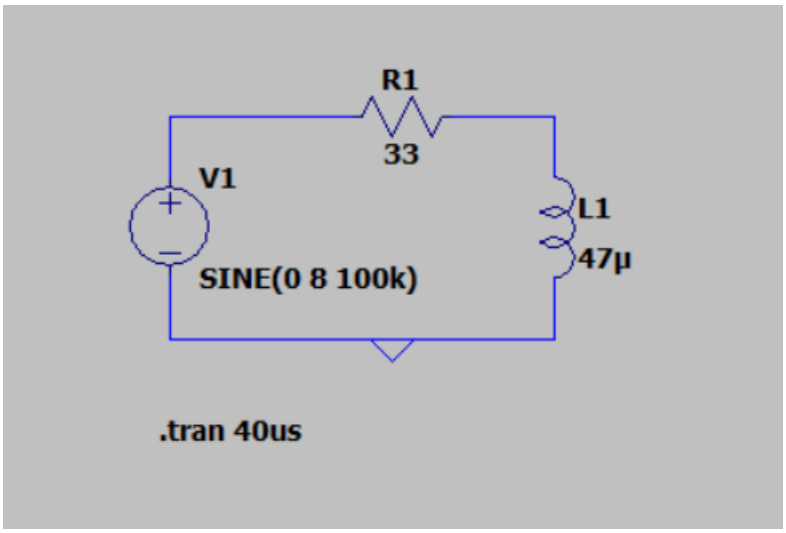

Figure 1.2.3: The circuit with input voltage frequency 100k Hz

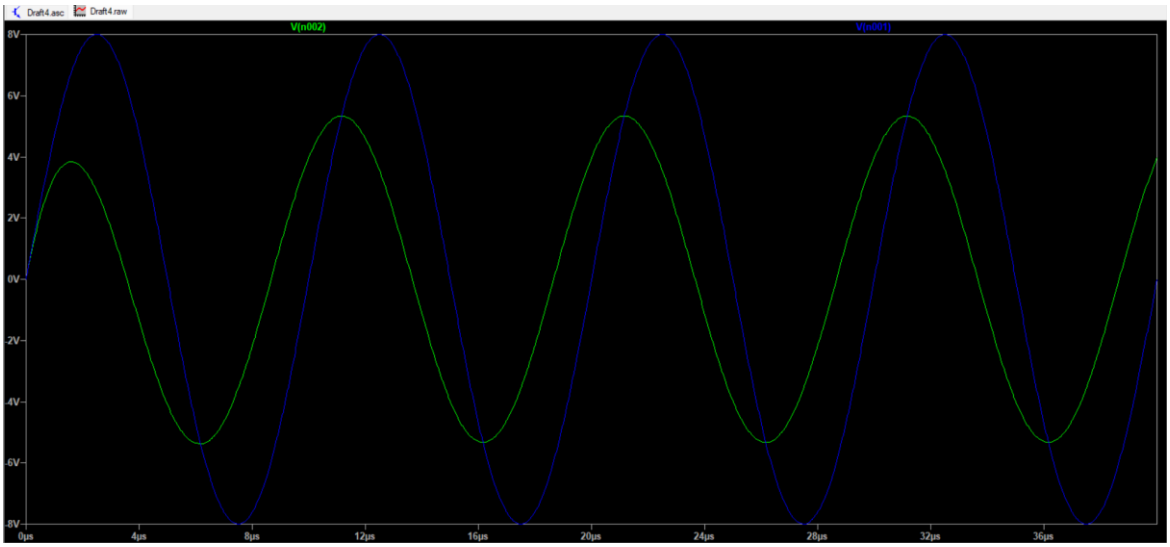

Figure 1.2.4: The input&output voltage plot with input voltage frequency 100k Hz

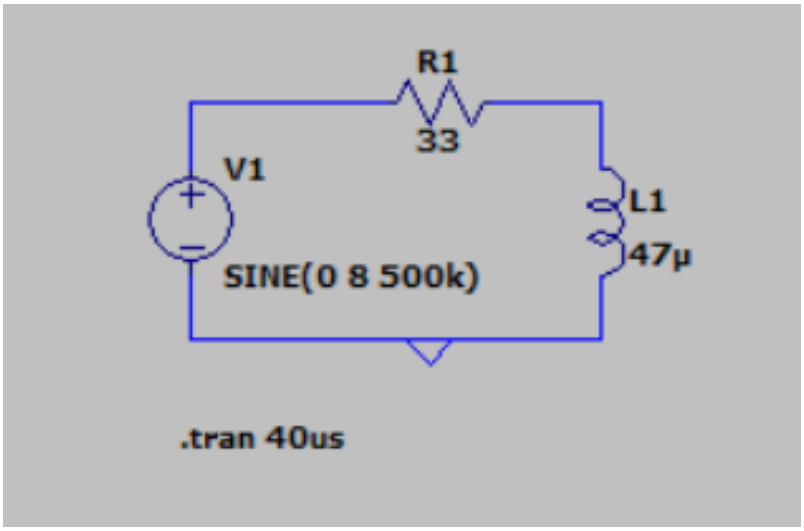

Figure 1.2.5: The circuit with input voltage frequency 500k Hz

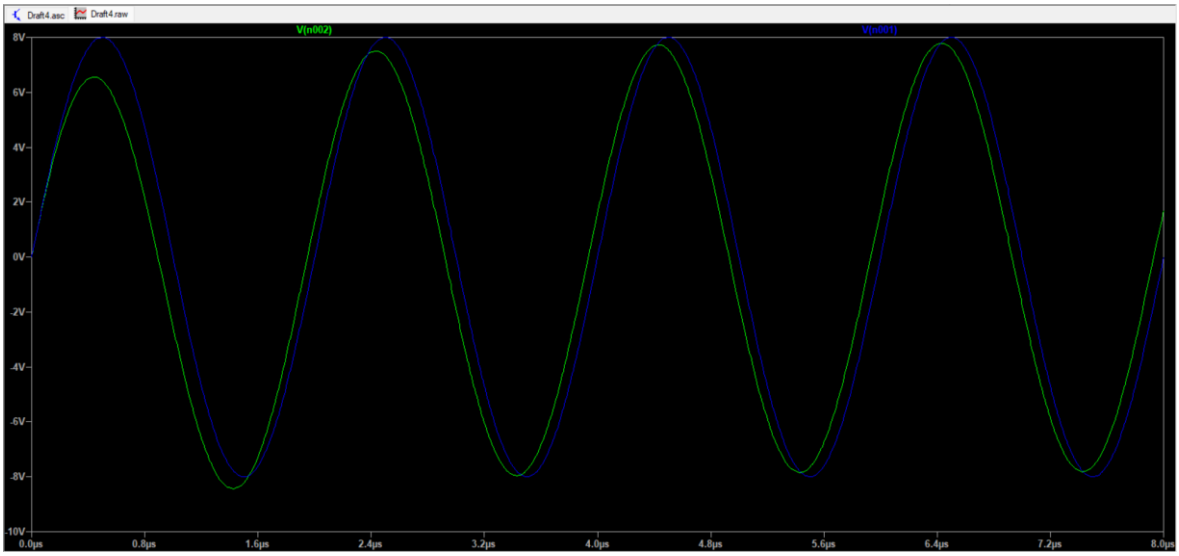

Figure 1.2.6: The input&output voltage plot with input voltage frequency 500k Hz

Part 2: AC (frequency-domain) Analysis

In the part 2 of the software lab, we were first assigned to do a frequency-domain analysis of the RL circuit we've constructed, by using AC analysis method in LTSpice. Here is the circuit **(Figure 2.1.1)** and the change of the magnitude and phase of the output with respect to frequency plotted on a logarithmic axis **(Figure 2.1.2)**.

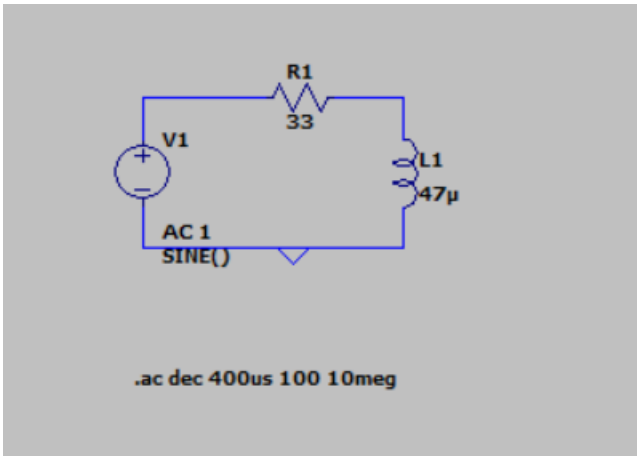

Figure 2.1.1: The circuit used for small-signal AC Analysis

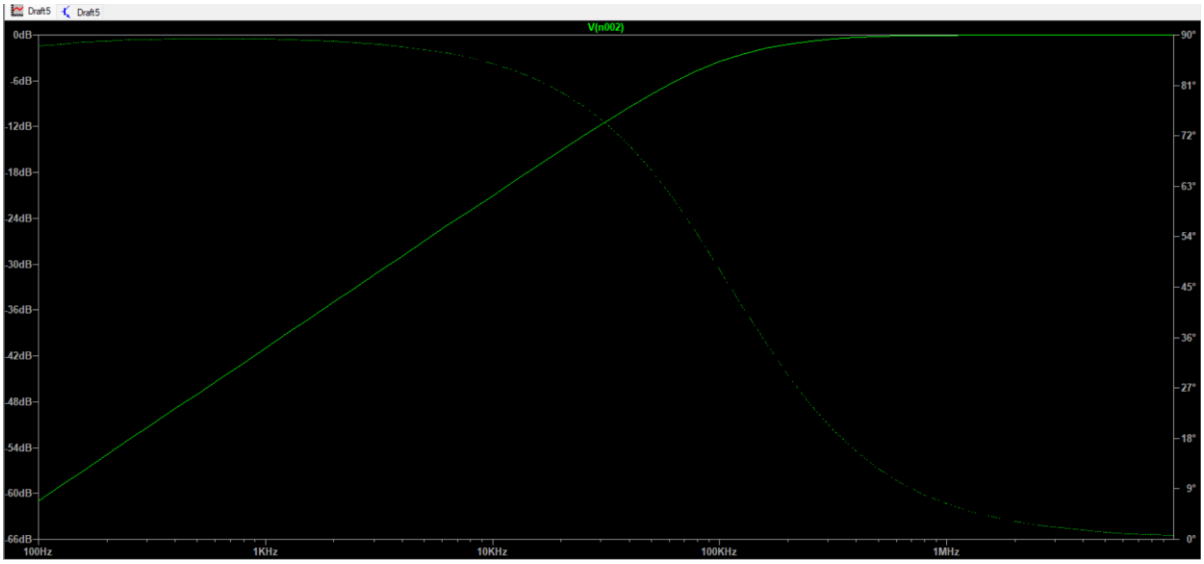

Figure 2.1.2: The change of the magnitude and phase of the output with respect to frequency

Next task we were assigned in part 2 was to add the serial output resistance of 50 Ω of the signal generators present in the hardware labs to the circuit. Here is the modified circuit **(Figure 2.2.1)** and the output signal plot **(Figure 2.2.2)**. We can observe that the gain in the modified circuit is smaller than the original circuit without the serial output resistance. This is due to the additional resistance we added to the circuit.

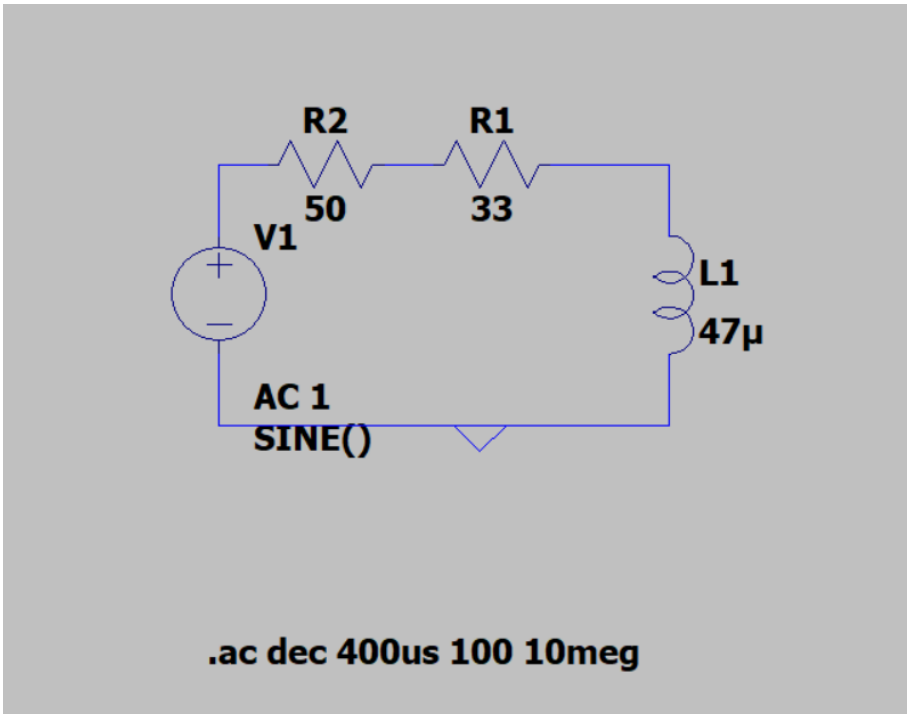

Figure 2.2.1: The modified circuit which has a serial output resistance of 50 Ω

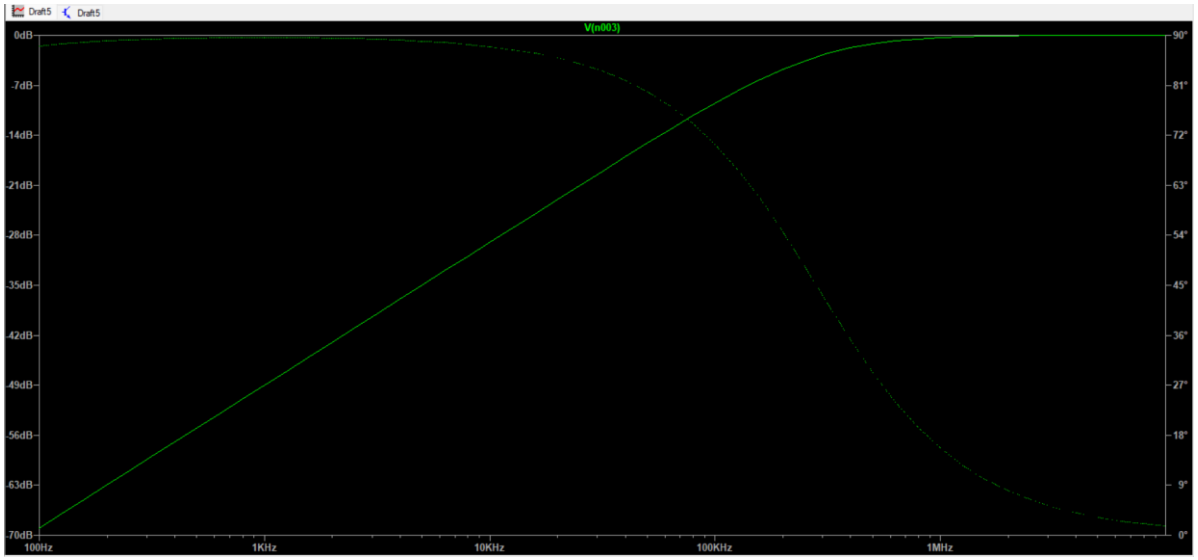

Figure 2.2.2: The change of the magnitude and phase of the output with respect to frequency in the modified circuit

The final task we were assigned in part 2 of the software lab was to observe the output voltage with respect to the voltage at the output of the realistic signal generator model. Here is the ratio plot between the nodes **(Figure 2.3.1)**. I've used the circuit in Figure 2.2.1 making this analysis.

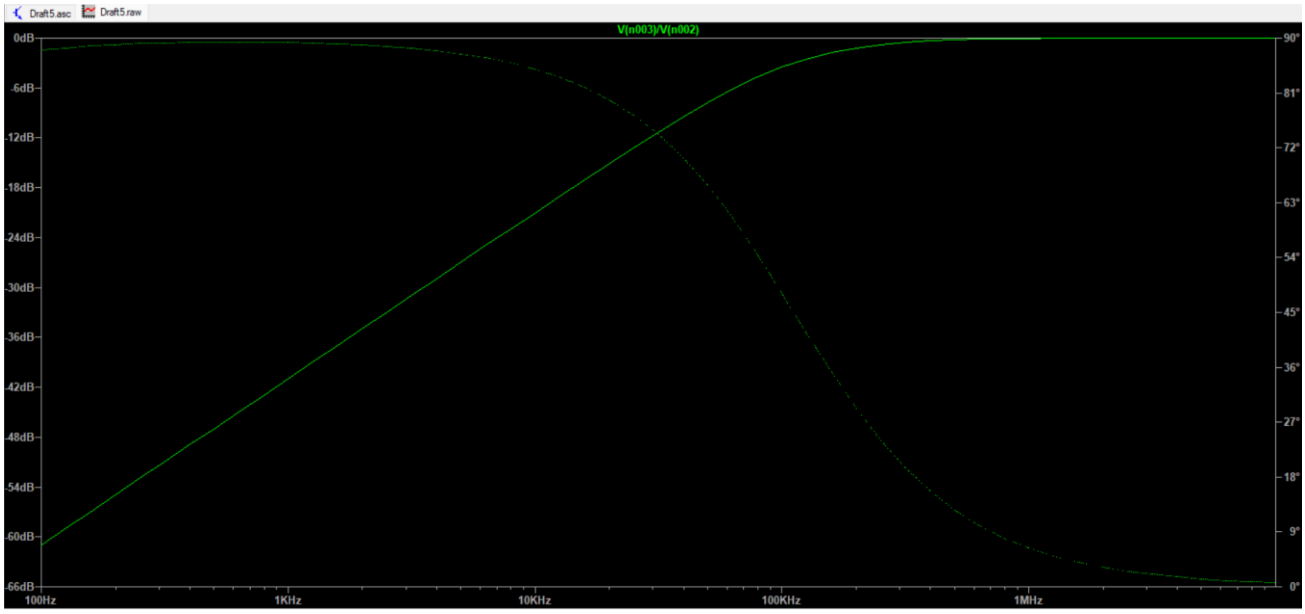

Figure 2.3.1: The ratio of the magnitude and phase at the output with respect to the voltage at the output of the realistic signal generator model plotted with respect to frequency

We can observe that Figure 2.1.2 and Figure 2.3.1 are the same in terms of both magnitude and phase. This means that the ratio of the output voltage with respect to the voltage at the output of the realistic signal generator model in the modified circuit; is the same as the ratio of the output and the input of the original circuit –the circuit with no serial 50 Ω resistance. We previously observed that adding a 50 Ω serial resistance would decrease the gain of the output vs the input. If we consider the input node as the output of the realistic signal generator model; then we would have no such decrease in gain.

Part 3: OPAMP Circuits

In the part 3 of the software lab, we were assigned to first import an OPAMP model -LM324- into LTSpice and use that specific model for the part 3. We were then assigned to create a circuit with different type of inputs. Here are the circuits **(Figures 3.1.1, 3.1.3 & 3.1.5)** and the input&output voltage plots **(Figures 3.1.2, 3.1.4 & 3.1.6)** accordingly. This circuit is an inverting amplifier circuit, OPAMP inverts the input voltage and amplifies it according to the values of R_1 and R_2 **(Equation 3)**.

$$\frac{V_{out}}{V_{in}} = -\frac{R_2}{R_1} \quad \textbf{(Equation 3)}$$

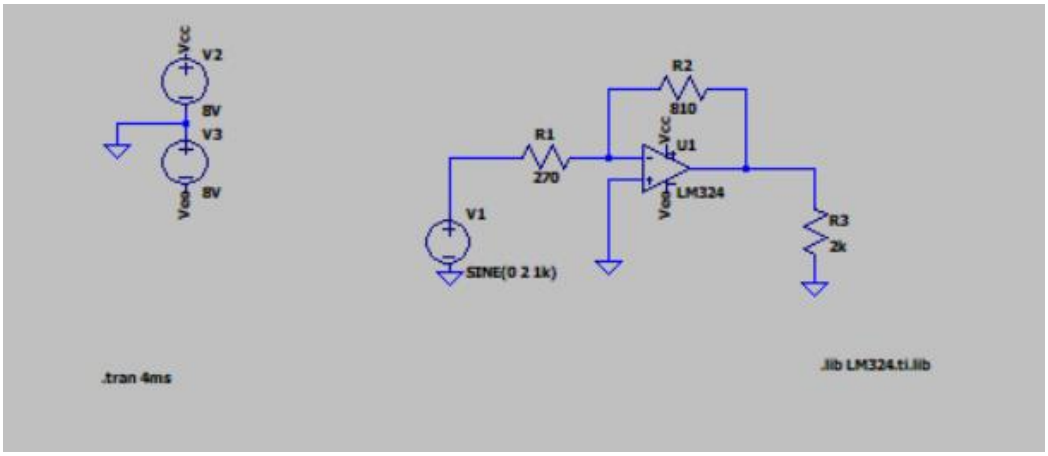

Figure 3.1.1: The inverting OPAMP circuit with a sinusoidal input of 1kHz and 2 Volts

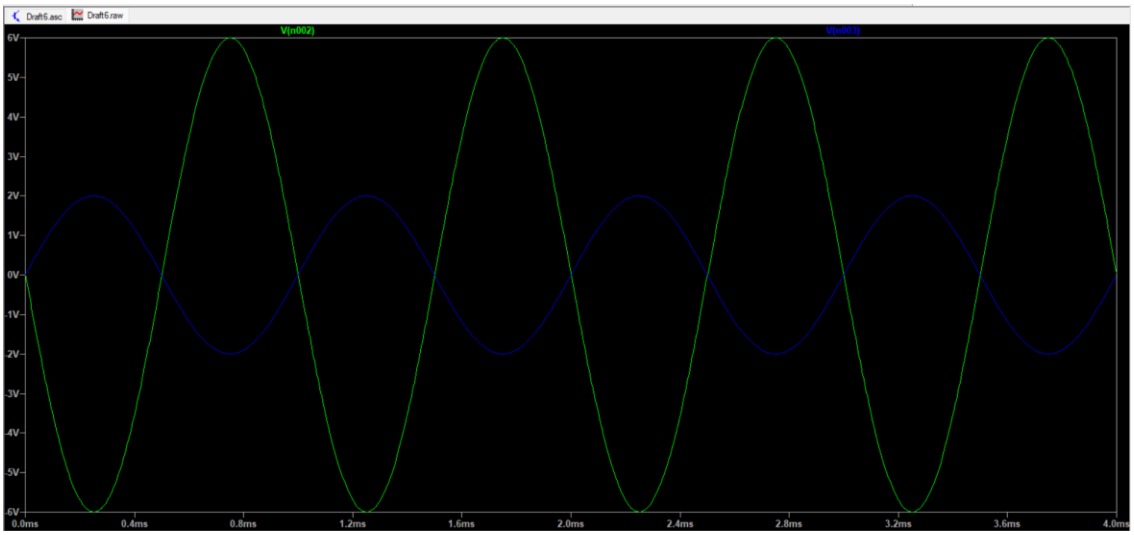

Figure 3.1.2: The input&output plot of the inverting OPAMP circuit with a sinusoidal input of 1kHz and 2 Volts

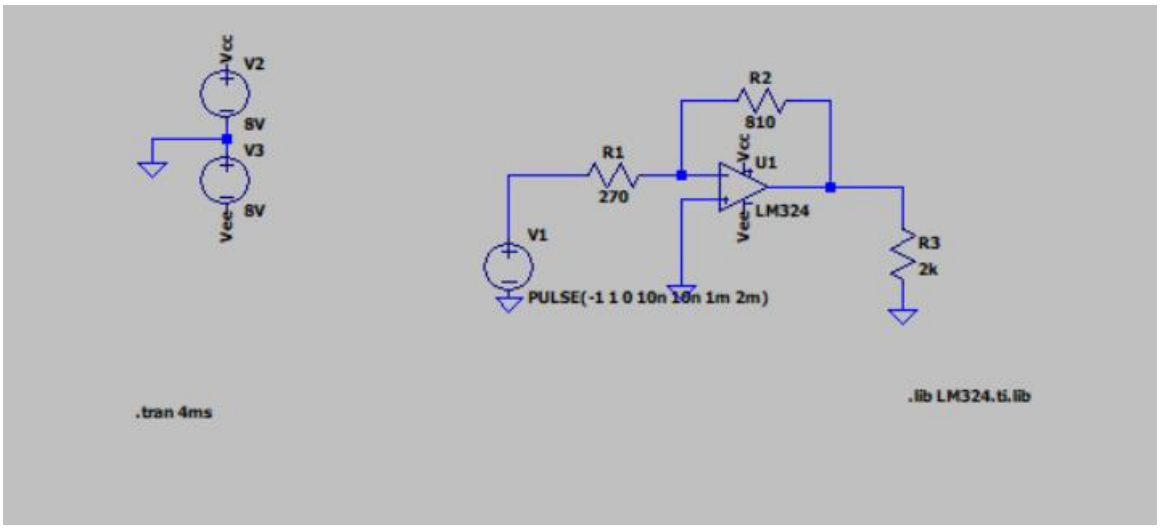

Figure 3.1.3: The inverting OPAMP circuit with a square-wave input of 1 Volts and 2ms period

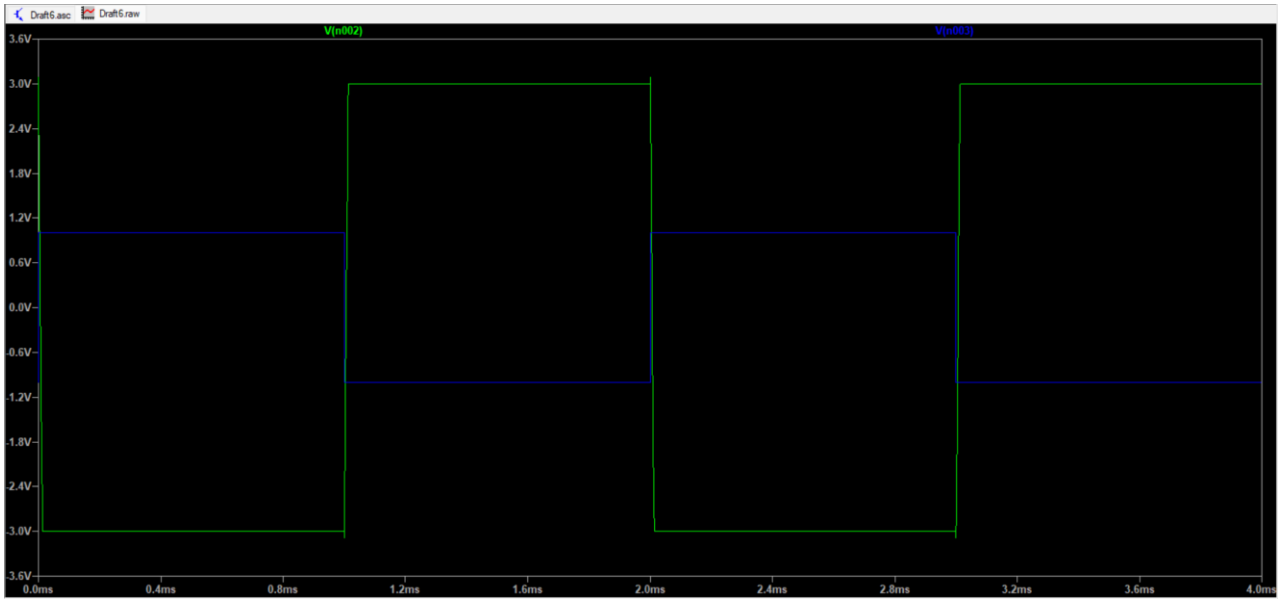

Figure 3.1.4: The input&output plot of the inverting OPAMP circuit with a square-wave input of 1 Volts and 2ms period

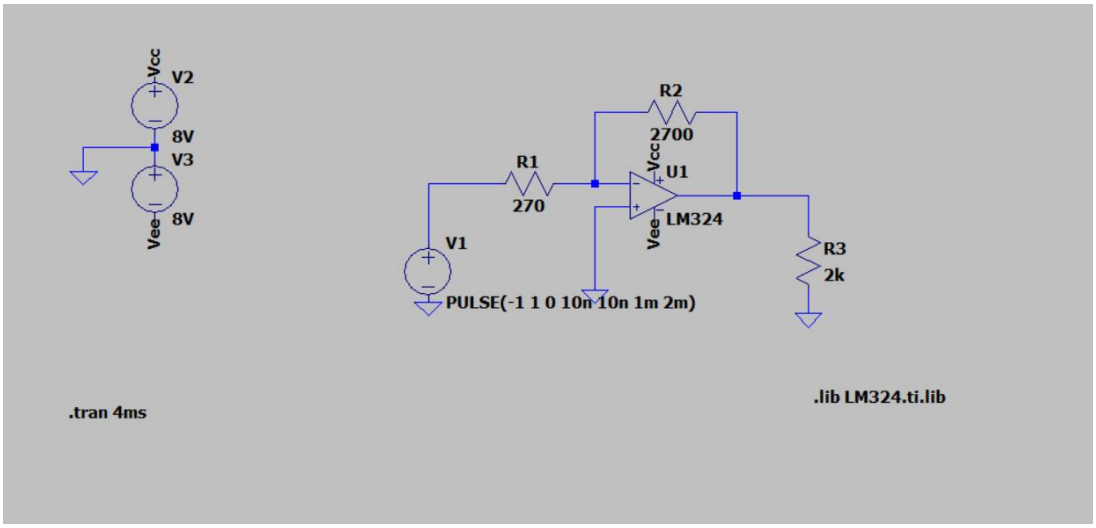

Figure 3.1.5: The saturated inverting OPAMP circuit with a square-wave input of 1 Volts and 2ms period

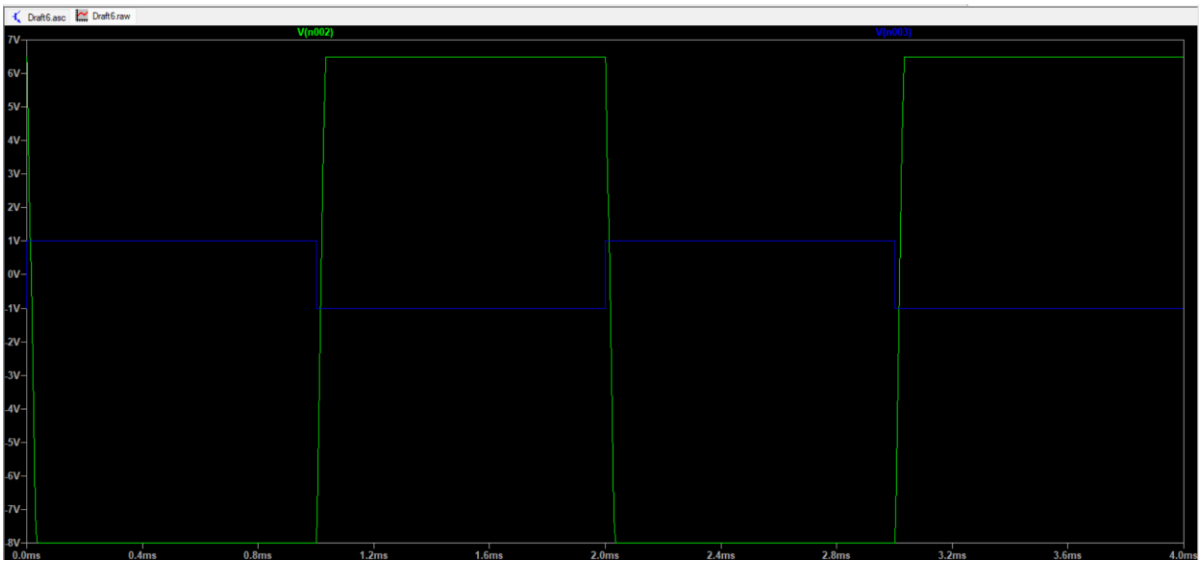

Figure 3.1.6: The input&output plot of the saturated inverting OPAMP circuit with a square-wave input of 1 Volts and 2ms period

The final assignment of the part 3 of the software lab was to change R_2 with a capacitor. Here is the modified circuit **(Figure 3.2.1)** and the input&output plot **(Figure 3.2.2)** of the circuit. This circuit is an integrating circuit, OPAMP integrates the input voltage and gives the output voltage accordingly **(Equation 4)**. This is due to the capacitor we placed instead of R_2.

$$V_{out} = -\frac{1}{R_1 C_1}\int_0^t V_{in}\, dt \quad \textbf{(Equation 4)}$$

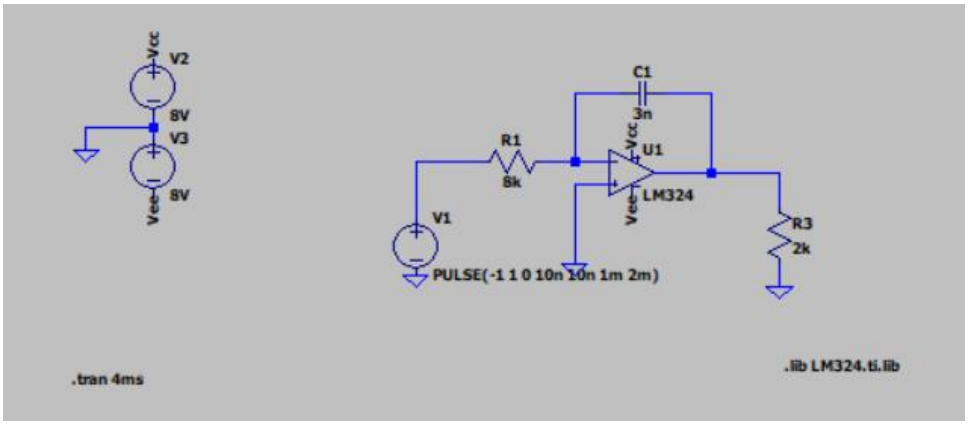

Figure 3.2.1: The integrator OPAMP circuit with a square-wave input of 1 Volts and 2ms period

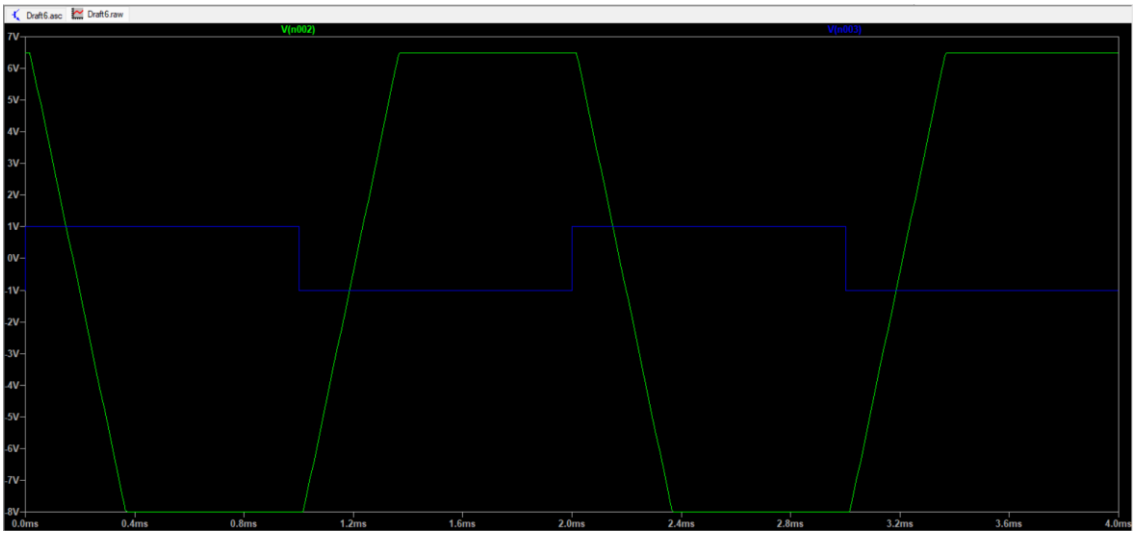

Figure 3.2.2: The input&output plot of the integrator OPAMP circuit with a square-wave input of 1 Volts and 2ms period

Hardware Lab

Part 1: RL circuit

In the first part of the hardware lab, we were assigned to implement the RL circuit we designed in part 1.2 of the software lab. Here you can see the implemented circuit **(Figure 4.1.1)**, the frequency response of the output voltage **(Figures 4.1.2, 4.1.3 & 4.1.4)**, and the table&graph of recorded data **(Figures 4.1.5, 4.1.6 & 4.1.7)**. The findings nearly matched the LTSpice simulation with a small margin of error. We can conclude that our practical data points are consistent with the simulation data **(Please see Figures 2.2.2 and 4.1.7 for this conclusion I've made)**. Also here is the gain equation **(Equation 5)**.

$$Gain\,(dB) = -\,20\,\log\left(\frac{V_{out}}{V_{in}}\right)$$ **(Equation 5)**

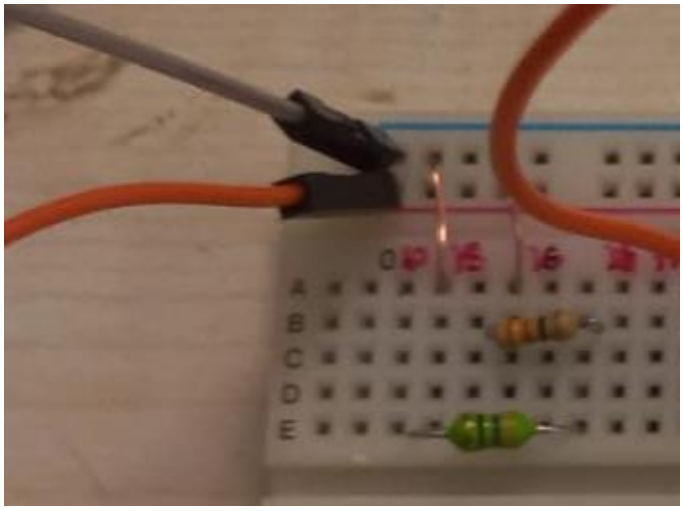

Figure 4.1.1: The RL circuit implemented on a breadboard

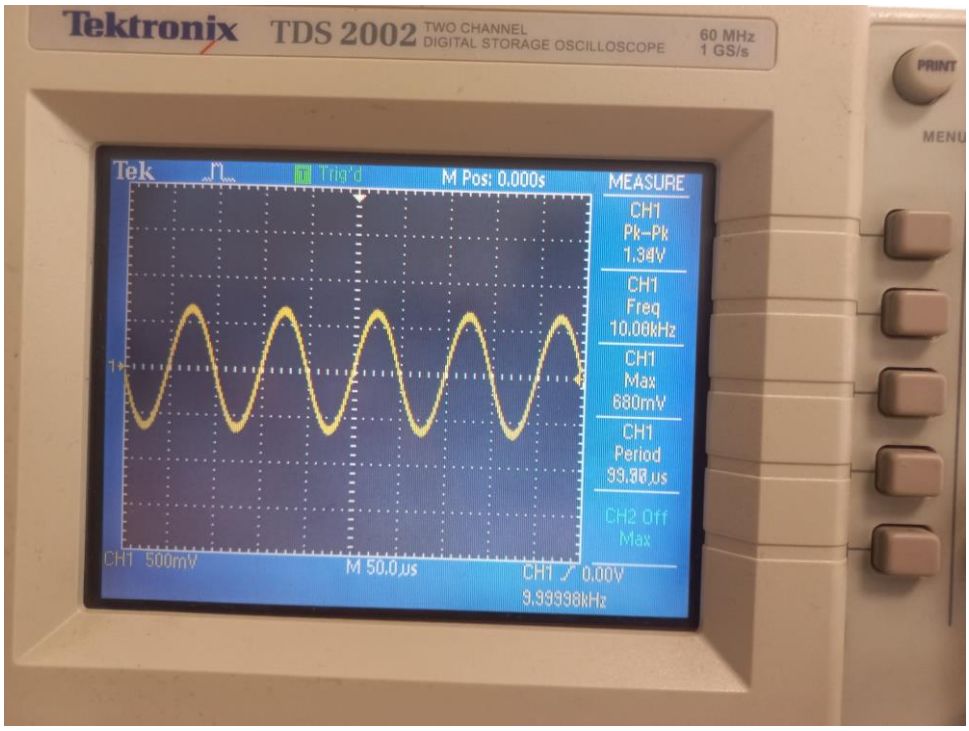

Figure 4.1.2: The output voltage of the RL circuit when input frequency is 10kHz; V_{max} = 0.68V

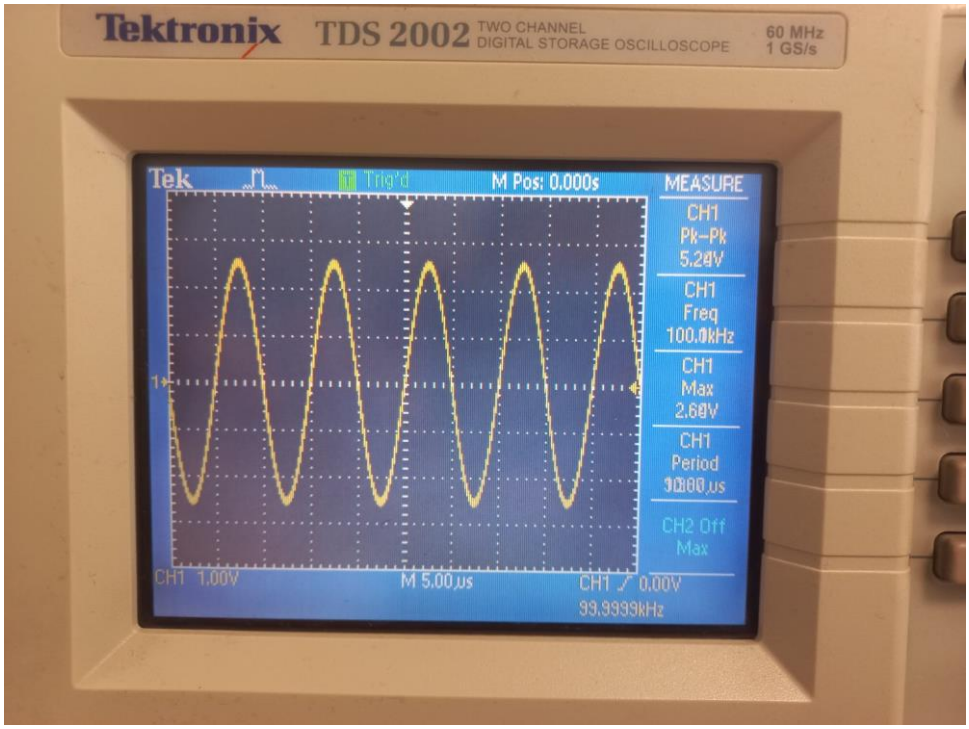

Figure 4.1.3: The output voltage of the RL circuit when input frequency is 100kHz; V_{max} = 2.64V

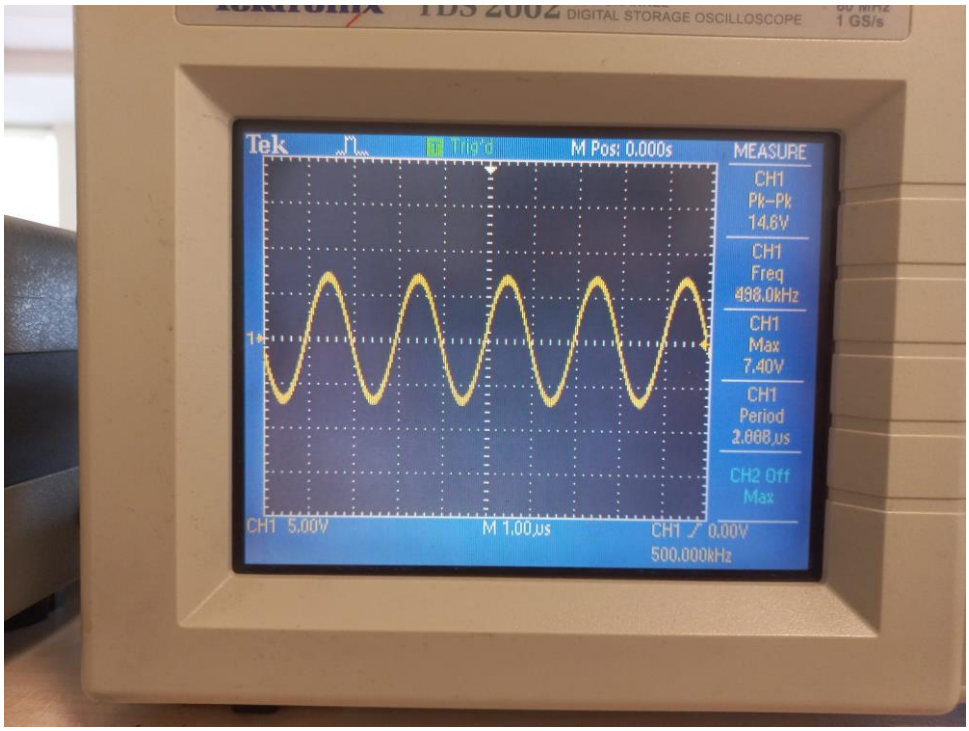

Figure 4.1.4: The output voltage of the RL circuit when input frequency is 500kHz; V_{max} = 7.40V

Frequency	Output Voltage (V)	Gain (dB)
10kHz	0.68V	-21.41dB

30kHz	1.20V	-16.47dB
50kHz	1.72V	-13.35dB
70kHz	2.16V	-11.37dB
100kHz	2.64V	-9.63dB
200kHz	4.40V	-5.19dB
300kHz	5.52V	-3.22dB
400kHz	6.92V	-1.26dB
500kHz	7.40V	-0.67dB

Figure 4.1.5: The output voltage and gain table according to input frequency

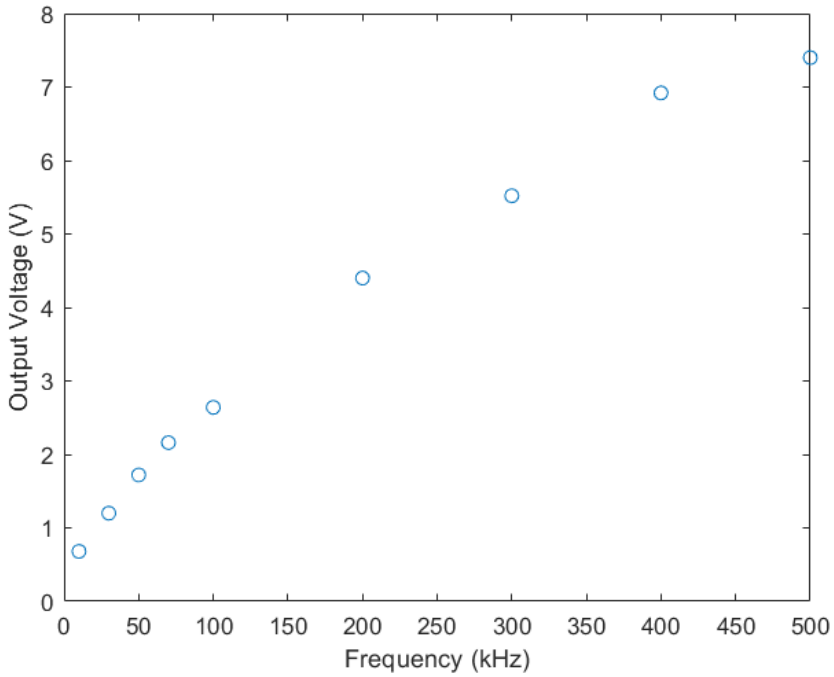

Figure 4.1.6: The output voltage graphed according to input frequency

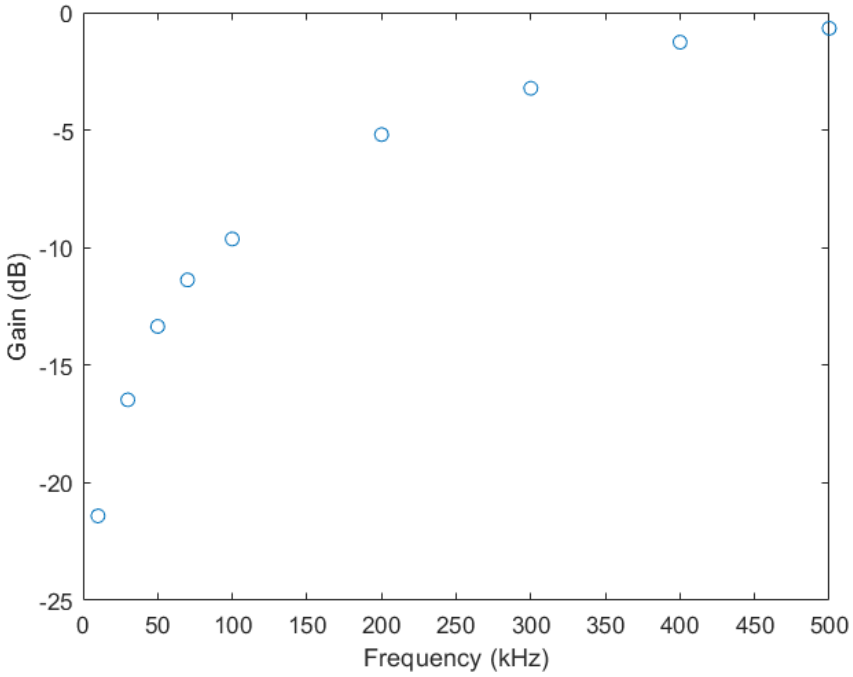

Figure 4.1.7: The gain in dB graphed according to input frequency

Part 2: OPAMP circuits

In part 2 of the hardware lab, we were assigned to create the two different OPAMP circuits we've created in the software lab: inverting amplifier circuit and integrating amplifier circuit. Here are the implemented circuits for both types of amplifier circuits **(Figure 4.2.1)** and the output voltage data **(Figures 4.2.2, 4.2.3, 4.2.4 & 4.2.5)**. We observe that our experimental findings match the simulation data in both type of circuits. The average experimental inverting coefficient is –3.065 which was –3 in the simulation data **(Equation 6)**.

$$Inverting\ Coefficient\ =\ \frac{V_{out}}{V_{in}}$$ **(Equation 6)**

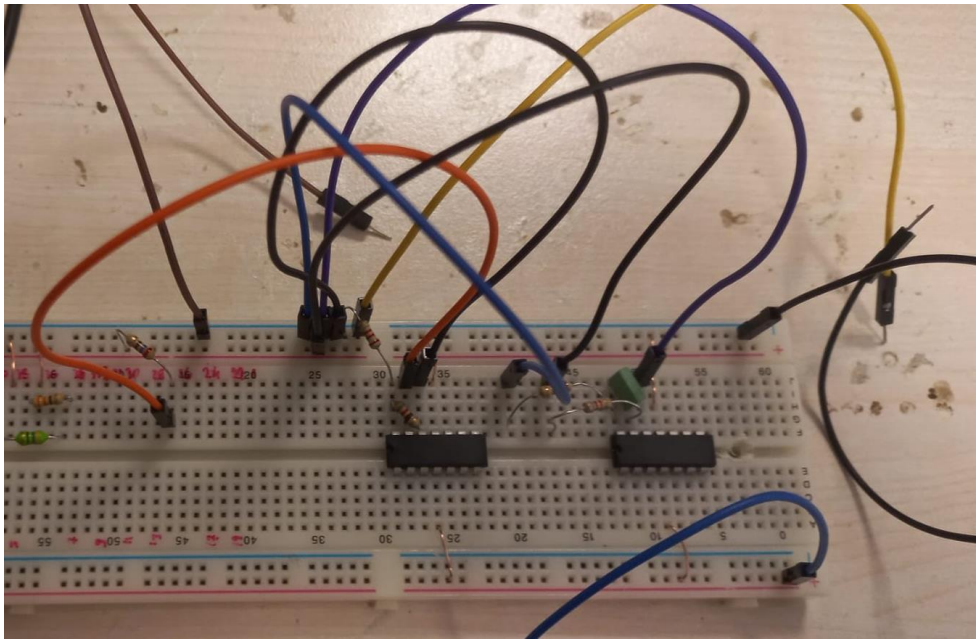

Figure 4.2.1: The two OPAMP circuits implemented on the breadboard

(On the left we have the inverting amplifier; on the right we have the integrating amplifier circuit)

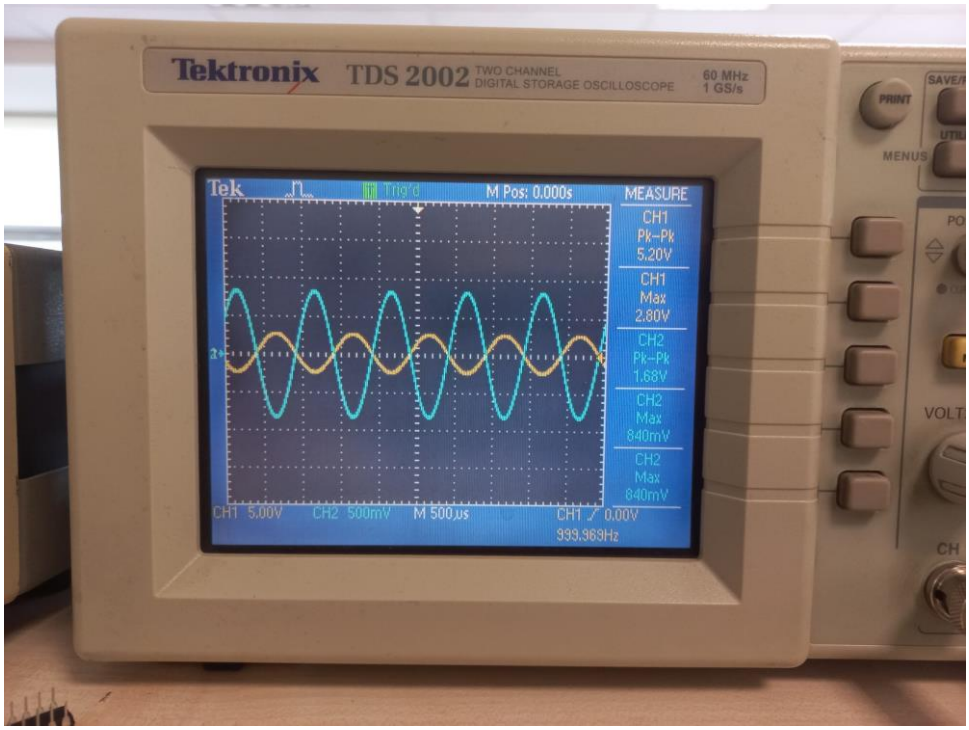

Figure 4.2.2: The inverting amplifier circuit whose inverting coefficient is –3; and with a sinusoidal input whose peak voltage is 0.84V; V_{out} = 2.80V

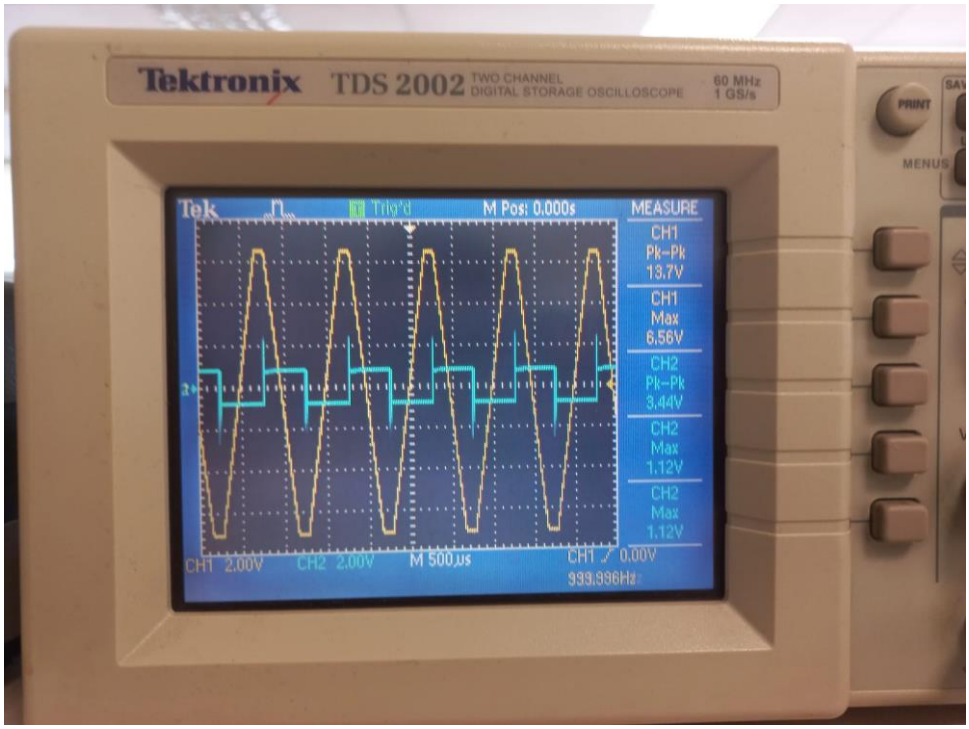

Figure 4.2.3: The integrating amplifier circuit output voltage

Input Voltage (V)	Output Voltage (V)	Inverting Coefficient (V_{out} / V_{in})
0.4	1.24	-3.10
0.6	1.92	-3.20
0.84	2.8	-3.33
1.0	3.04	-3.04
1.2	3.60	-3.00
1.4	4.08	-2.91
1.6	4.60	-2.88

Figure 4.2.4: Input&Output Voltage Relation Table in the inverting amplifier circuit

Average Inverting Coefficient is –3.065

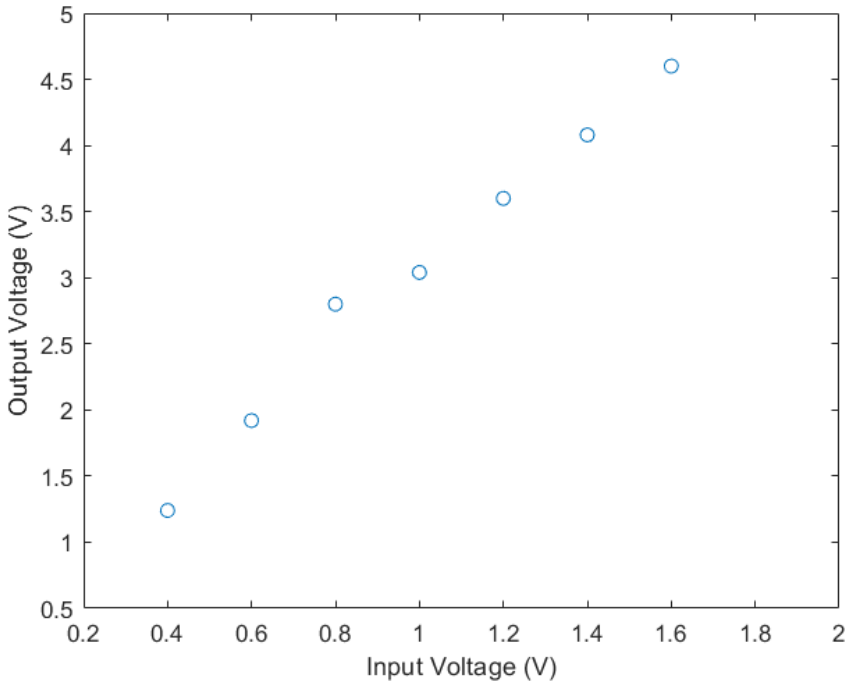

Figure 4.2.5: Input&Output Voltage Relation Graph in the inverting amplifier circuit

Conclusion:

The main purpose of the software part of this lab assignment was to perform time-domain and frequency-domain analyses in LTSpice. We designed RL circuits and OPAMP circuits. On the hardware part, we implemented the RL&OPAMP circuits which was initially designed in the software part.

Both software and hardware part of this experiment resulted as expected ,with some insignificant small error margins present in the hardware lab. These errors might be caused from analog equipment errors such as internal wire resistance, oscilloscope inaccuracy or even LM324 itself.

Throughout the lab, we learned how to use LTSpice more effectively and faster. This will definitely help us not only in our academic student lives, but also in our career in general since we will be using LTSpice software all the time in the future. I find this fundamental lab pretty helpful and important for us.