Name: Ezgi

Surname: Demir

Section: 1

ID: 22103304

Date: 11.10.2023

EE202-Circuit Theory

Lab1

Time Domain and Frequency Domain Analysis

Introduction:

In the first part of Lab1, the main purpose is to learn how to analyze the time domain(transient) and frequency domain from the RL and Op-amp circuits that we design with our values, use the LTSpice application, and make an explication from the graphs of our the RL and Op-amp circuits. Secondly, in the lab's second part, we will try to actually build our circuit(hardware part) with the components given in the lab environment.

Part 1: Software Implementation

Analysis:

In Part 1 I simply designed a voltage divider circuit with R1= 8Ω and R2= 12Ω and a voltage source that gives a 10V peak-to-peak 12kHz sine wave. I used ".tran 0.0003" to get approximately 3 to 4 periods of the waves. With the input voltage of 10V, it is expected to get 6V with the voltage divider formula (eq. 1). The results from LTSpice(Figure 1) satisfied the result we get from the voltage divider formula.

$$Vout = \ Vin.\left(\frac{R_2}{R_2+R_1}\right) \text{(eq. 1)}, \ \ Vout = 6V = 10.\left(\frac{12\Omega}{20\Omega}\right)$$

After that, I changed my resistor(R1) value to 24Ω and put an inductor with the value of 8.2µH, and created another simulation. I observed a high-pass filter. Because of the reactance that the inductor is very low in lower frequencies (eqn. 2.1), Vout gets higher in higher frequencies from the formula of voltage divider of RL circuit (eqn. 2.2). When we take the limit, as frequency tends to infinity(very high), Vout reaches the input voltage we apply. To test in the simulation we used the values 10kHz, 100kHz, and 500kHz and checked if our results (see Table 1) match with the calculations from our formula (Figures 2.1, 2.2, and 2.3).

$$X = j\omega L \text{ (eqn. 2.1)}$$
$$Vout = Vin.\left(\frac{j\omega L}{\sqrt{R_2^2+(j\omega L)^2}}\right) \text{ (eqn. 2.2)}$$

In part 2 instead, I performed a frequency-domain(AC) analysis of the RL circuit in a logarithmic scale (instead of transient) and observed how the circuit behaves in different frequencies(Figure 3.1). I also observed the cut-off frequency and when I added 50Ω to my circuit in series with my resistance 24Ω(Figure 3.2), which stands for the internal resistance of the signal generator affected our cut-off frequency. (Got higher compared to the circuit that we didn't take into account the internal resistance). Which is really crucial because it changes our calculations. (Later on, I will be comparing my LTspice simulation results, cut-off frequency, saturation(clipping) points, and transient and AC analysis in the RL circuit with the hardware part.)

In part 3, I imported the LM324 OPAMP model into LTSpice, because we created an inverting amplifier and used it in our circuit. In the simulation, I fed my Op-amp in a range (6 to 10) as said the the lab manual, and gave an input voltage, sinusoidal wave with an amplitude of 2V, and frequency of 1 kHz. I set R3 to 2 kΩ and set R1 to 330Ω and R2 to 990Ω. At first feeding in the range (6 to 10) wasn't enough and there was a clipping I increased the value and saw a full sine wave without clipping(saturation)(Figure 4.1). Since we assumed that the opamp is not saturated meaning it is in the linear region, we can apply KVL in the negative and the positive input parts and assume, there is zero current there. Also, we assume "$Vin_{(-)} = Vin_{(+)}$" and solve our KVL, as a result, since we chose our resistor's resistance ratio 3 (330Ω to 990Ω), we expect to get 3 times gain from the eqn. 3.1 After these, in step 4, I changed the input to a square wave (pulse) with the 1V amplitude with 2 ms period and % 50 duty cycle and set the rise and fall times to 10ns. We are expected to see a square wave (Figure 4.2). This time when I increased the ratio of R_2/R_1 by increasing the value of only R_2 at some point I observed that V_{out} doesn't change (see Table 2, after ratio 8.18 saturation observed, Vout didn't change). This indicates a saturation point(Figure 4.3). Finally ,in step 6, I changed R_2 with a capacitor with 3n capacitance as a result, we got a integartor opamp (Figure 4.4). Hence, once we add the capacitor now we had a differntial part in our equation(eqn. 3.2). In the equation's context, the capacitor functions as an integrator, which means that when a square wave is applied as the input, it results in the generation of a ramp-like response in the output.

$$\frac{V_{output}}{V_{input}} = -\frac{R_2}{R_1} \text{ (eqn. 3.1)}$$

$$V_{output} = -\frac{1}{C.R_{1.}} \int_0^t V_{input}\, dt \text{ (eqn. 3.2)}$$

Simulations:

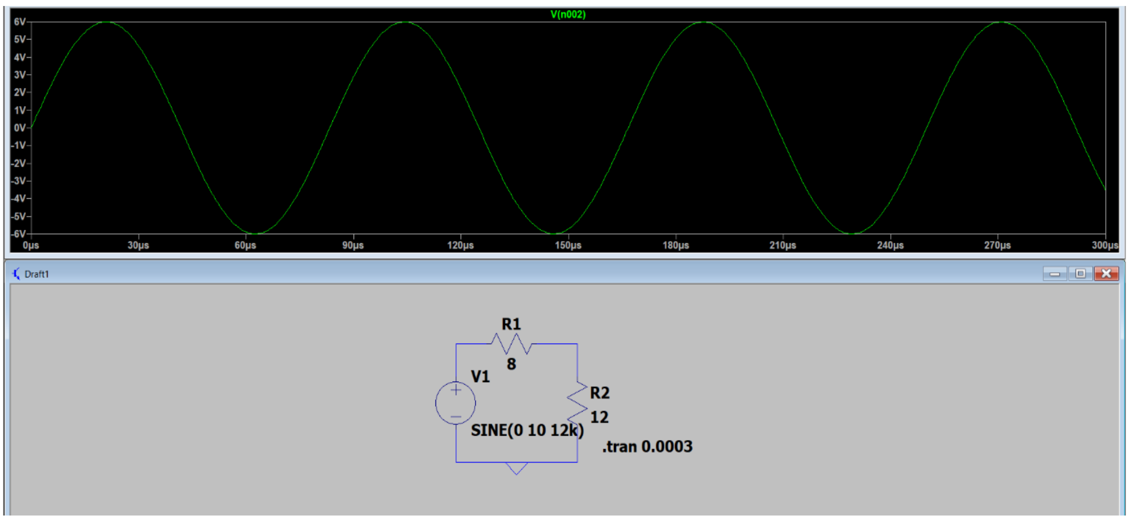


Figure 1: Voltage Divider and simulation of transient analysis

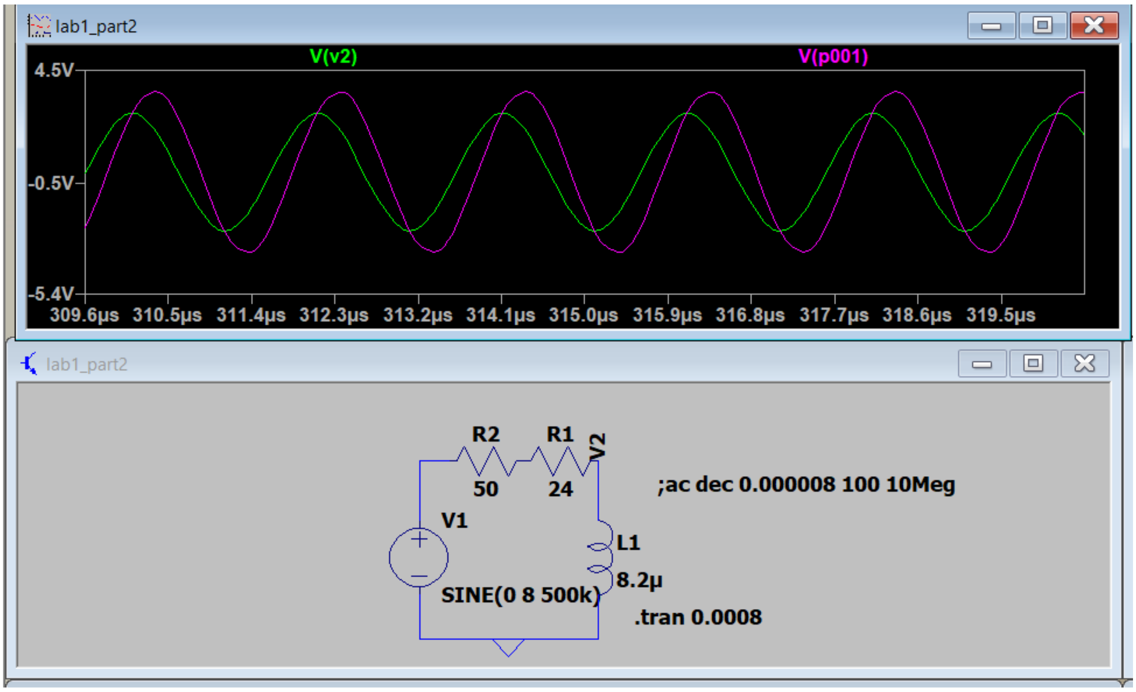


Figure 2.1: Transient analysis of the high-pass filter at 500kHz

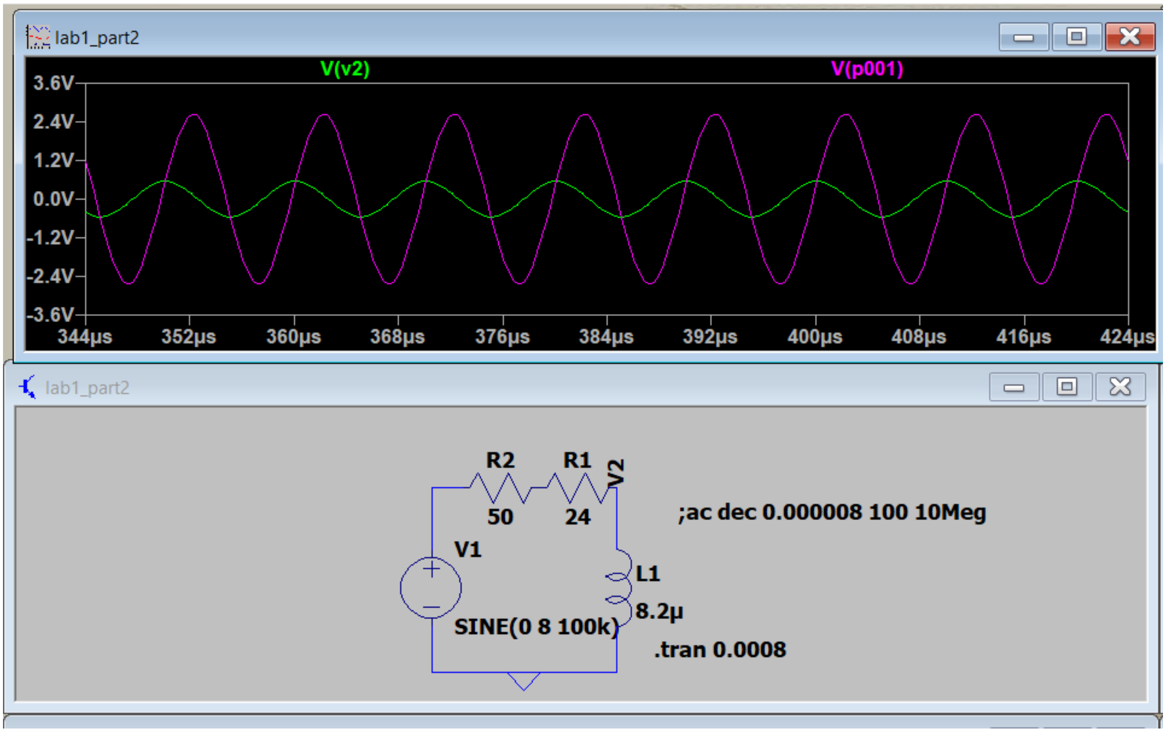


Figure 2.2: Transient analysis of the high-pass filter at 100kHz

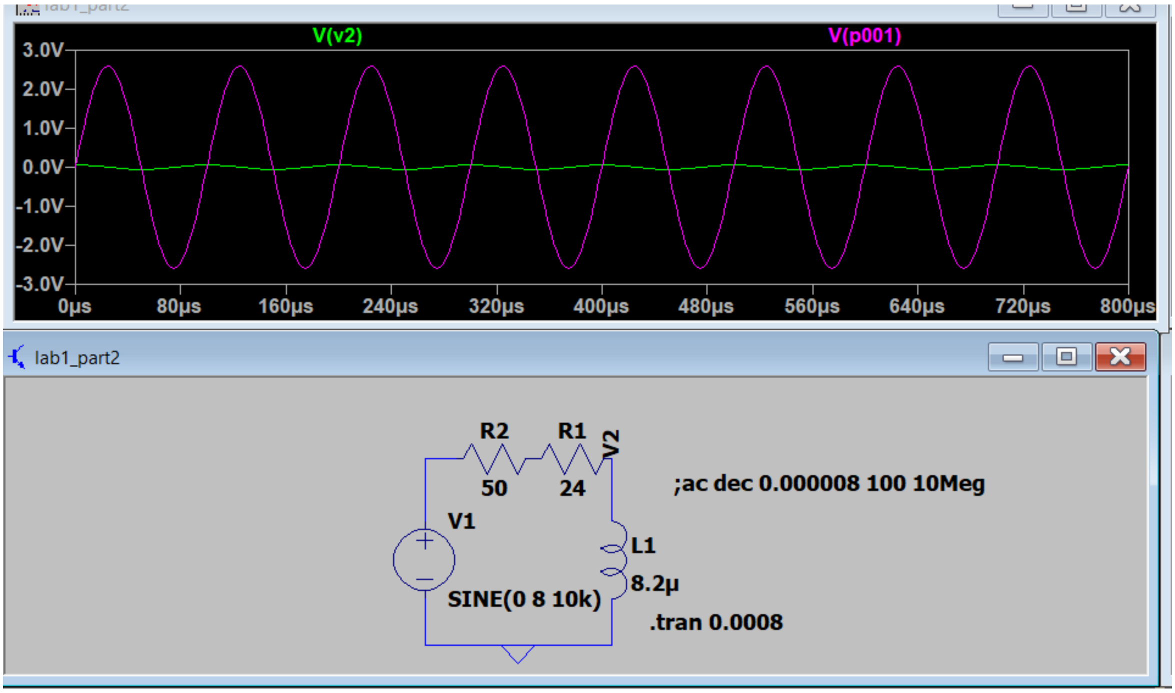


Figure 2.3: Transient analysis of the high-pass filter at 10kHz

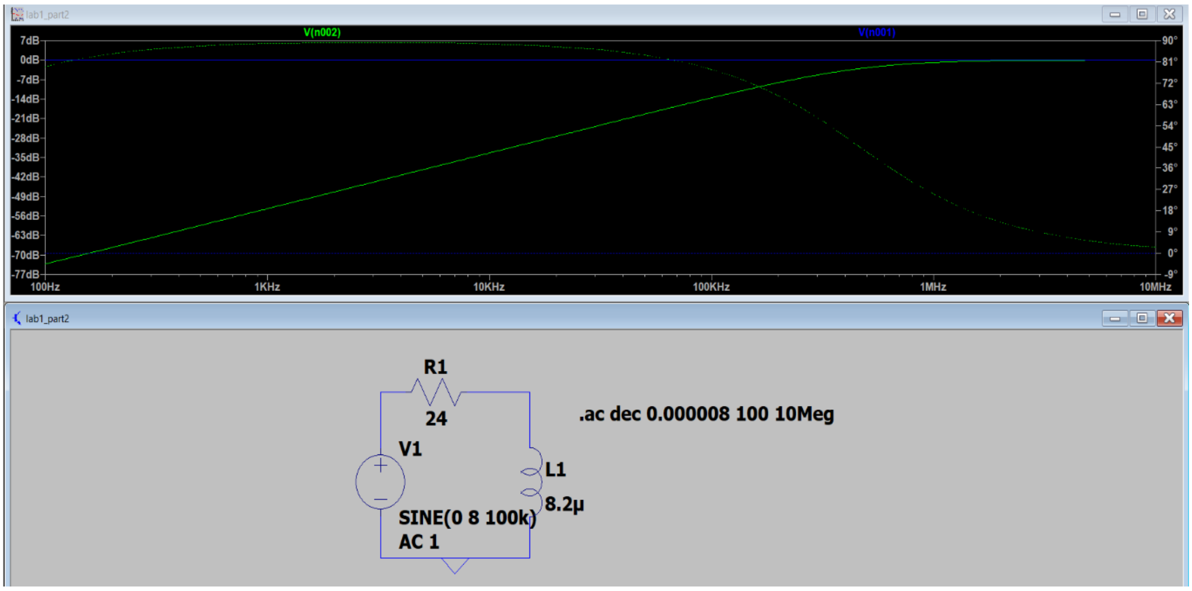


Figure 3.1: AC analysis of RL circuit

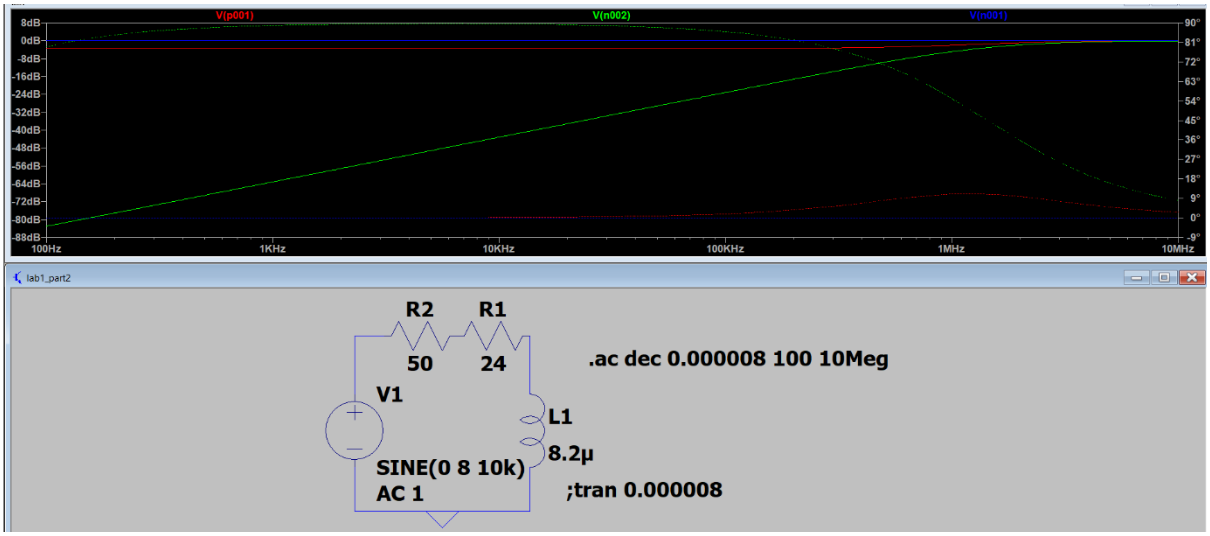


Figure 3.2: AC analysis of RL circuit with 50Ω (realistic circuit) (blue line: input, red line: realistic one, green line: output)

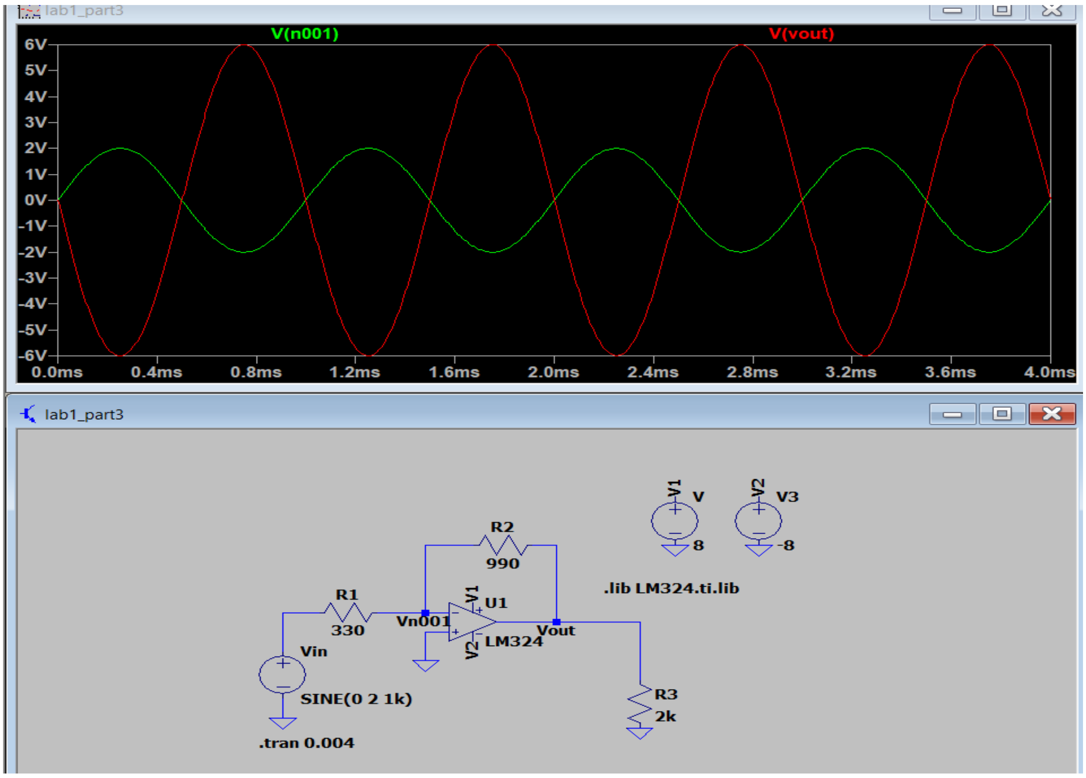


Figure 4.1: Inverting amplifier circuit and the simulation in the lineer region (not saturated), with $R_1 = 330\Omega$ *and* $R_2 = 990\Omega$

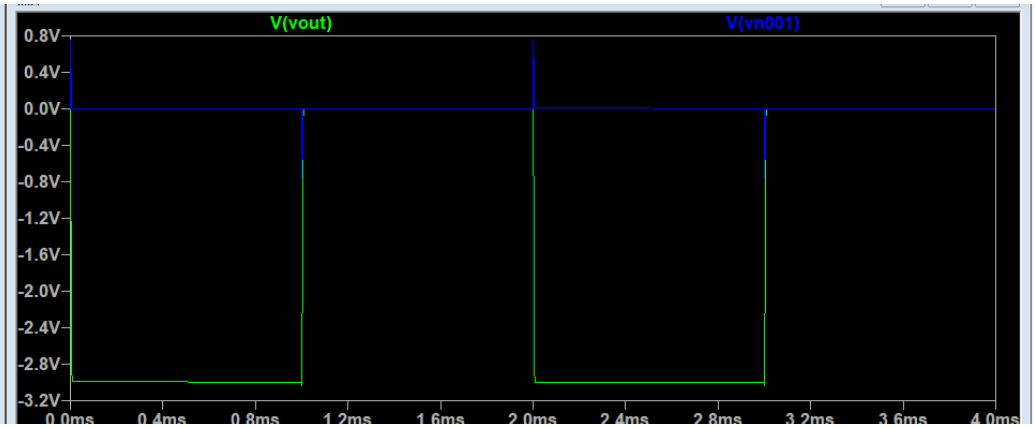


Figure 4.2: Inverting amplifier simulation with $R_1 = 330\Omega$ *and* $R_2 = 990\Omega$ and square wave input (not saturated)

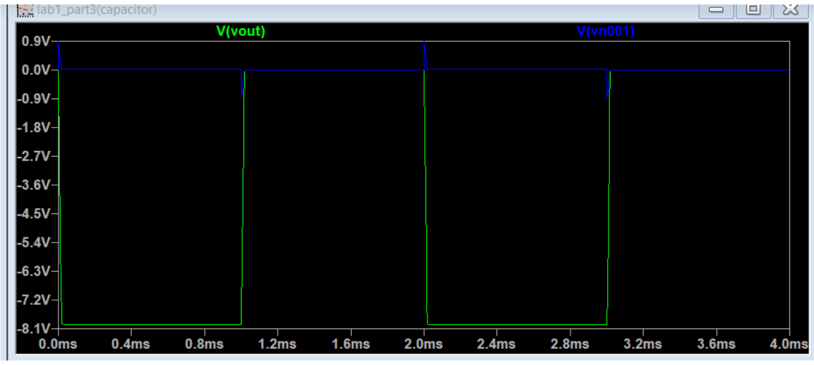


Figure 4.3: Inverting amplifier with $R_1 = 330\Omega$ *and* $R_2 = 2.7K\Omega$ and square wave input (saturated)

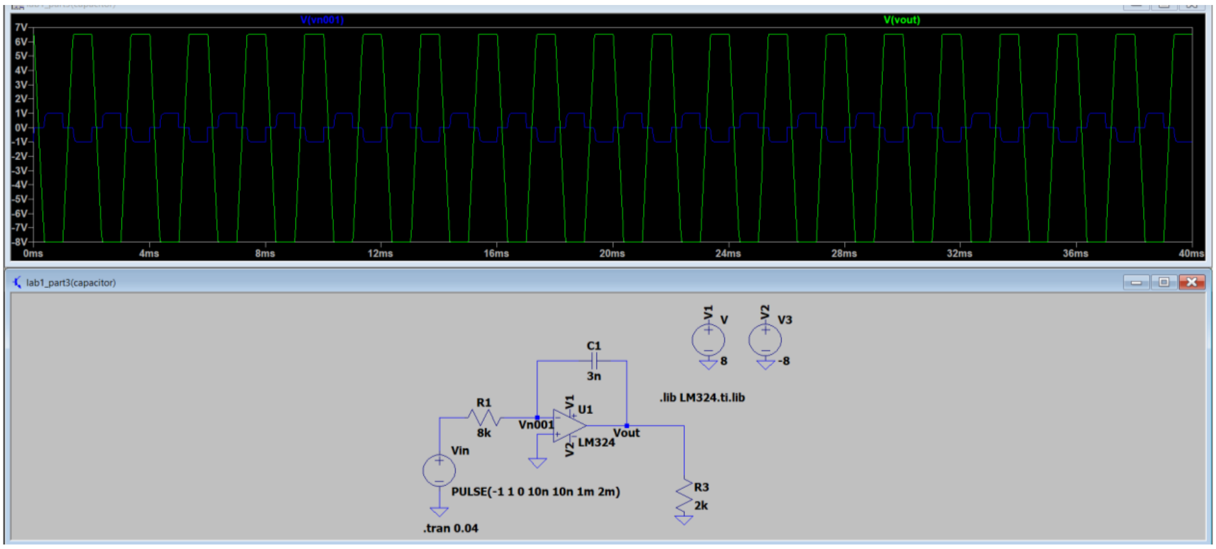


Figure 4.4: Inverting integrator circuit with $R_1 = 8K\Omega$ *and* C_1*= 3nF* and square wave input

Frequency(kHz)	Vinput	Voutput
10	8 Vpp	1.1 Vpp
100	8 Vpp	1.11 Vpp
500	8 Vpp	4.2 Vpp

Table 1: High-pass filter results with different frequencies

R2/R1	Vinput	Voutput
3	2 Vpp	6 Vpp
6	2 Vpp	12 Vpp
7	2 Vpp	14 Vpp
8.18 (it is the saturation ratio)	2 Vpp	16.36 Vpp

Table 2: Inverting Amplifier Results with different resistance ratios

Part 2: Hardware Implementation

In the hardware part, responses of the high-pass filter fits with the simulation, however the error of 10kHz was a little higher. Also from the "Graph 1" it is clear that cut-off frequency close to 1.4 (gain is very close to -3dB at this point) which meets again with the software part. In the Figures 9 and 11 also, the yellow wave represents output and blue wave represents the input. I observed 10.2 Vpp in the integrator opamp compare to the simulation the error is 15% and observed 13.7 Vpp in the inverting integrator opamp which had an error 5.5% (Figure 9 and Figure 11).

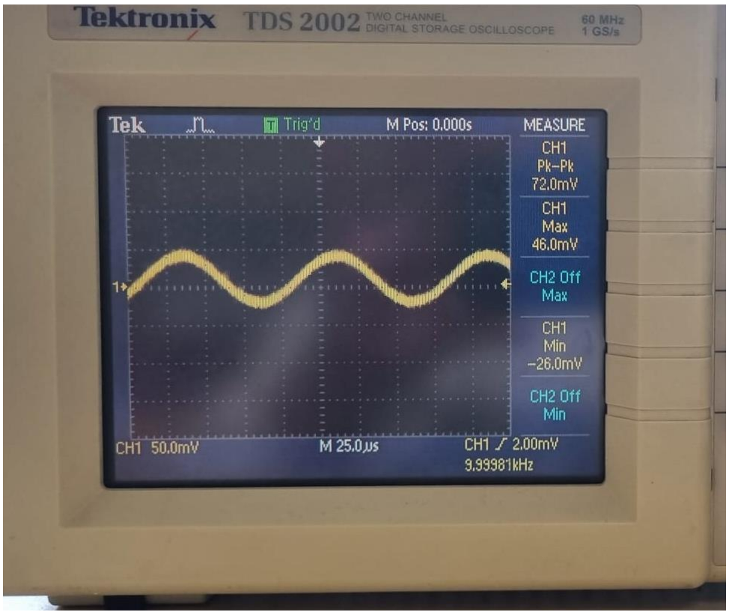


Figure 5: High-pass filter in 10kHz

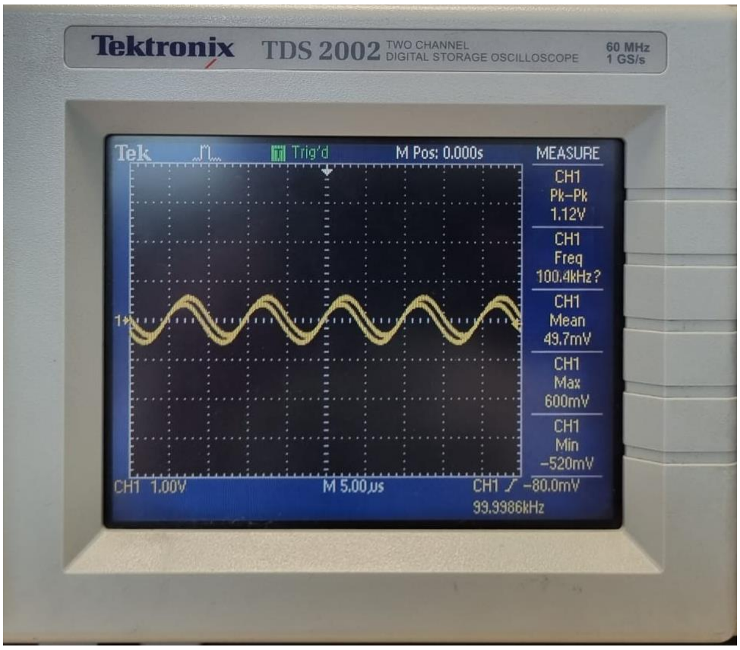


Figure 6: High-pass filter in 100kHz

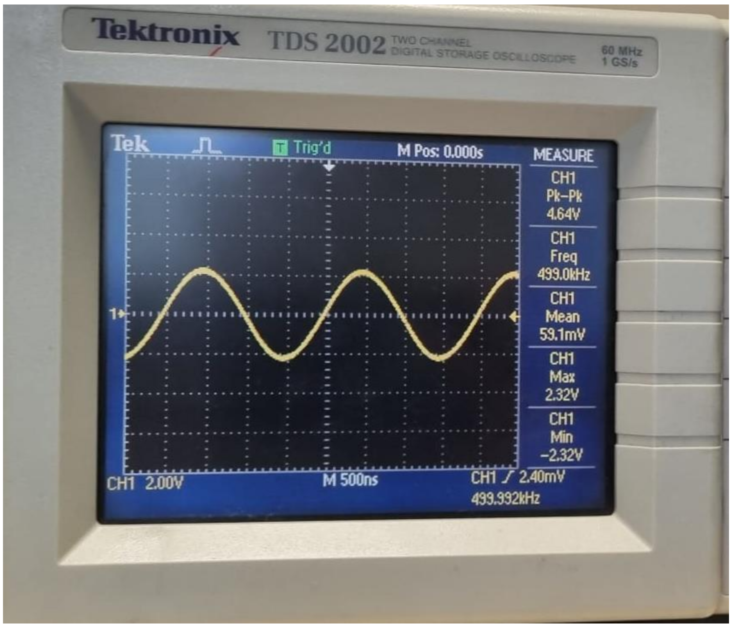
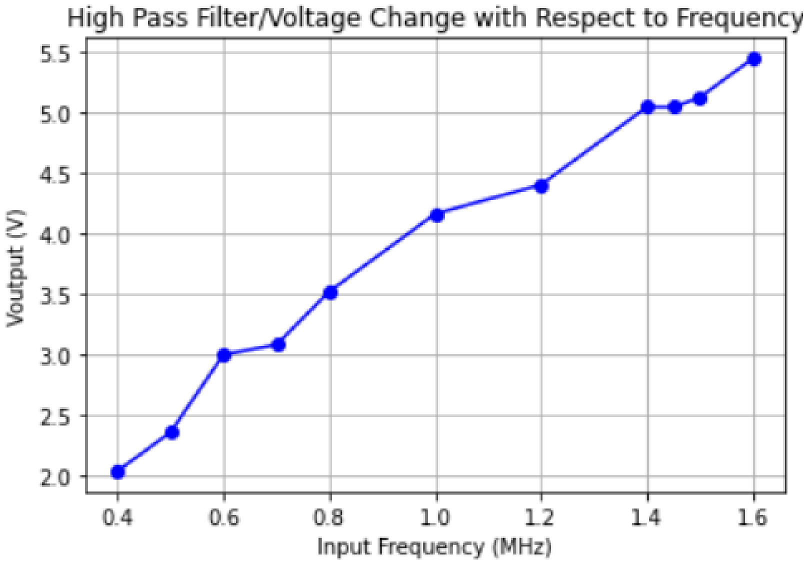


Figure 7: High-pass filter in 500kHz

Frequency(kHz)	Vinput(V)	Voutput(Vpp)	%error(compare to the simulation)
10	8 Vpp	72 mVpp	34
100	8 Vpp	1.12 Vpp	0.89
500	8 Vpp	4.64 Vpp	10.77

Table 3: Hardware results compare to software results



Graph 1: Cut-off Frequency of the High- pass Filter

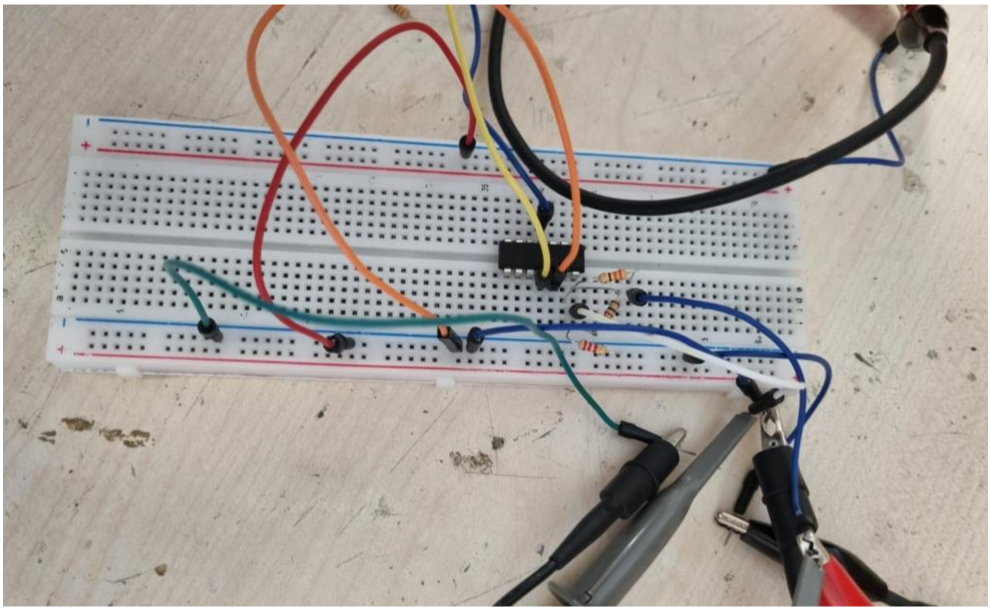


Figure 8: Inverting amplifer circuit implemented on the breadboard

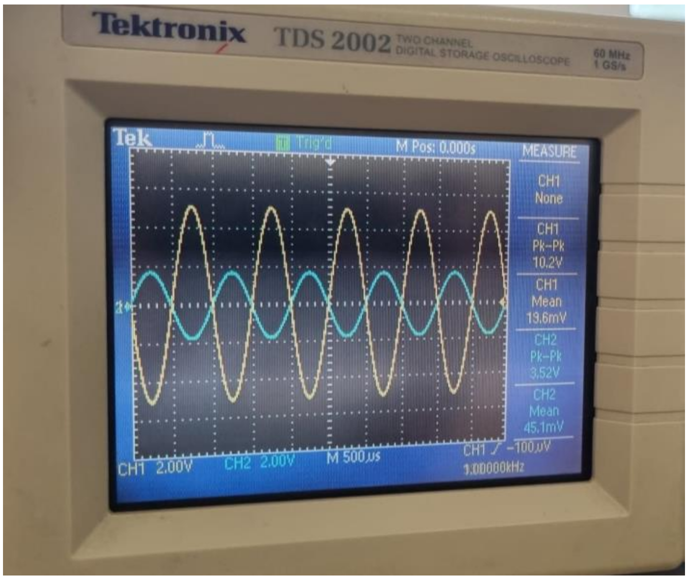


Figure 9: Inverting amplifier input and output

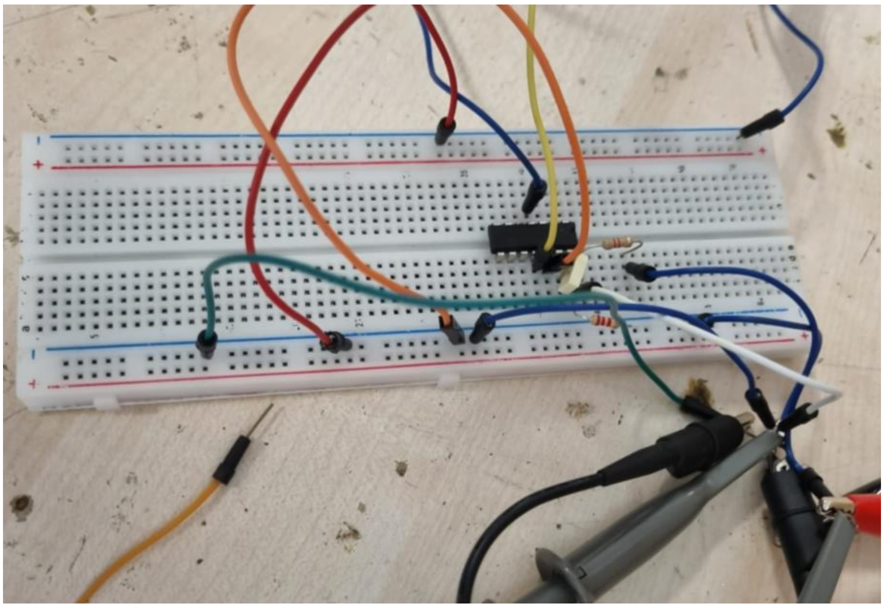


Figure 10: Inverting integrator amplifier circuit implemented on the breadboard

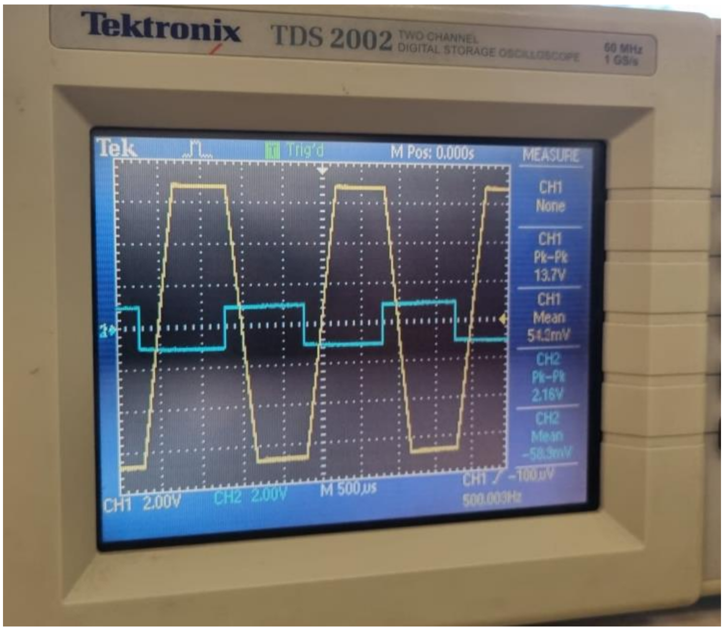


Figure 11: Inverting integrator amplifier input and output

Conclusion:

This experiment allowed us to assess circuit behavior, measure values (cut-off frequency, output voltage, phase), and compare LTSpice results with real-world implementation. Of course errors occured because we are not in the ideal circumstances and also I have used for exapmle 2.2kΩ resistance in the integrator amplifier step because we don't have exactly 2kΩ in the lab. Furthermore, errors could happen because of the internal resistances of the components.

REFERENCES

Equations 1, 2, 3 from:

Book-Electric-Circuits-9th-ed-J.-Nilsson-S.-Riedel-Prentice-Hall-2011.pdf