EEE 202 CIRCUIT THEORY
LAB 1

Time-Domain and Frequency-Domain Analyses in LTSpice

In this lab assignment, you will perform time-domain and frequency-domain analyses in LTSpice in a series of circuits in the first two parts. In the last part, you will make OPAMP circuit analysis. Please make sure that you have downloaded LTSpice to your computer and imported the LM324 OPAMP model provided on Moodle before coming to the lab.

Do not forget to save the output plots in LTSpice and make a note of your voltage values after every simulation! Make a table of your results after each part.

Software Lab

Part 1: Transient (time-domain) Analysis

1. Open a new schematic and construct the following simple voltage divider circuit (you can use the component library for all of the required circuit elements or use the shortcuts for some of them):
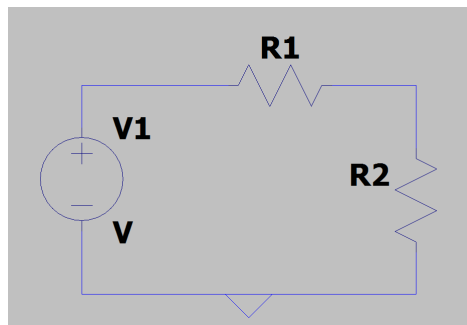

Figure 1: A simple voltage divider

Now, you are supposed to simulate this circuit. You should first assign the component values.

- Set R1 between 1-5 Ω, R2 between 10-20 Ω and give a sinusoidal voltage signal with an amplitude between 5-10 V and frequency between 1-10 kHz.

- Then, go to edit Simulation Cmd and set the stop time according to your frequency (e.g., 3-4 periods of the chosen frequency)

- Run the circuit and observe the input and output voltages on the same plot. Here, consider the voltage on R2 as the output.

- Make sure that you see 3-4 periods of the signal. Check that the output value in the plot is correct. Save your results.

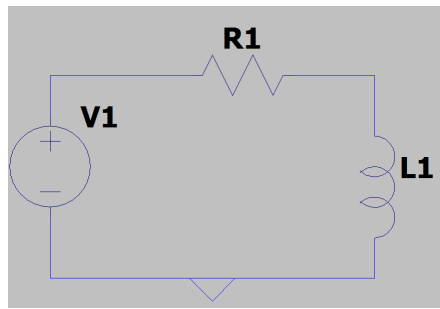**2**. Now, replace R2 in Figure 1 with an inductor and construct the following RL circuit:

Figure 2: RL circuit

- Set voltage source to a sinusoidal wave with an amplitude of 5V, frequency of 100 kHz.

- Set R1 between 10-50 Ω, choose the value for L1 as one of the following: 8.2 μH, 10 μH, 47 μH, 100 μH

- Set the stop time to see at least 3-4 periods of the chosen frequency or at least 10 time constants of the RL circuit based on which one is greater.

- Run the circuit and observe the input and output voltages on the same plot. Here, consider the voltage on L1 as the output. Save your results.

- Repeat the previous item with frequency 10 kHz and 500 kHz. Comment on your results. What kind of filter is this?

Part 2: AC (frequency-domain) Analysis

In this part, you are supposed to perform frequency-domain analysis of the RL circuit in Figure 2. You have observed that the behaviour of the circuit changes with frequency. However, as it is hard to simulate the circuit at each frequency, you can see its behaviour in a frequency range in logarithmic plots (i.e., Bode plots) by AC analysis.

1. Edit simulation parameters to do AC analysis instead of transient analysis. Set type of Sweep as decade and set the start and stop frequencies to make an analysis from 100 Hz to 10 MHz. You also should adjust the voltage source for small signal AC analysis with an AC amplitude of 1.

2. Run the circuit and check the output signal. You should see the change of the magnitude and phase of the output with respect to frequency, plotted on a logarithmic axis. Here, consider the voltage on L1 as the output.

3. In our hardware labs, the signal generator has a serial output resistance of 50 Ω. Therefore, to get consistent results between your design and its hardware implementation, you need to consider this resistance in LTSpice. Hence, modify the circuit in Figure 2 as follows:
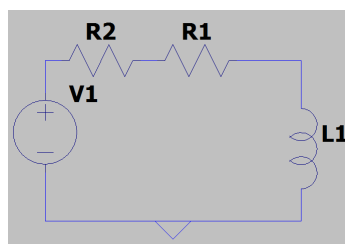

Figure 3: RL circuit with input resistance

- Set R2 to 50 Ω and repeat Step 2 with the same parameters to get the logarithmic plot. Compare the magnitudes of the output voltage with that from Step 2.

4. Run the circuit in Step 3 again and this time, check the output voltage with respect to the voltage at the output of the realistic signal generator model (i.e., the voltage at the node between R2 and R1) instead of the input voltage V1, which is default of LTSpice. To do this, plot the ratio of output voltage and the voltage at the output of the realistic signal generator.

- Compare the result with the results at Step 2 and Step 3. Comment on your results.

Part 3: OPAMP Circuits

1. In this part, you will simulate basic OPAMP circuits. First, construct the following circuit with the basic **opamp2** model from the component library:
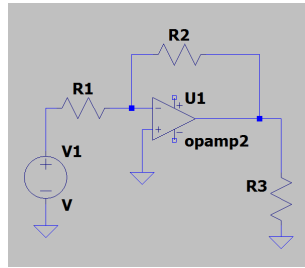

Figure 4: Basic OPAMP circuit

Then, you need to import the **LM324** OPAMP model into LTSpice. Assuming that you have already downloaded it into the computer please go to **.op** shortcut in LTSpice and use the SPICE directive ".lib LM324.ti.lib "to import it in LTSpice. You should also change the value of **opamp2** to **LM324**.

2. Connect separate DC voltage sources (about 6-10 V) to the DC supplies of the OPAMP. You can use labeling through **Label Net** shortcut for this.

3. Now you can simulate the circuit:

- Set the input voltage to a sinusoidal wave with an amplitude of 1V, and frequency of 1 kHz.

- Set R3 to 1 kΩ, and R1 and R2 such that R2/R1 = 4 , where R1 is between 100-500 Ω.

- Run the circuit with transient analysis and observe the amplified output sinusoidal. What type of OPAMP circuit is this?

4. Change the input to a square wave (pulse) with the 1V amplitude with 1ms period and % 50 duty cycle. Set the rise and fall times to 10ns and repeat Step 3.

5. Increase R2/R1 to a value such that you will observe the saturation of the OPAMP and simulate the circuit again.

6. Change R1 to 8 kΩ and replace R2 with a 3 nF capacitor. Then, simulate the circuit with the square wave input from Step 4. Observe the output. What type of OPAMP circuit is this?

Hardware Lab

1. **RL circuit**

- Implement the RL circuit that you constructed in Step 2 of Part 1 in the SW lab. You can use an axial inductor available in the lab.

- Measure the output voltage at 3 different frequencies around 10 kHz, 100 kHz and 500 kHz. Select the frequency values to observe the change in the output voltage magnitude (i.e, to observe the filtering effect). You can use your logarithmic plot from Part 2 of SW Lab to determine the suitable frequencies. Take a note of your output voltage amplitudes.

- Make a table of your results.

- Draw a rough frequency response plot with the measurement data and compare it with the one you have from Part 2 of SW lab.

2. **OPAMP circuit**

- Implement the OPAMP circuit from Step 3 of Part 3 in the SW lab using LM324 OPAMP. Make a table of your input and output voltage amplitudes.

- Implement the OPAMP circuit from Step 6 of Part 3.

Checks

1. SW Lab

- Show your results after Part 2, together with your comments and observations.

- Show your results after Part 3, together with your comments and observations.

2. HW Lab

- Show and explain your results for the RL circuit.

- Show and explain your results for both OPAMP circuits.

Available materials in the lab

Axial inductors with standard values. Toroidal cores to design inductors or transformers: T25-10, T37-7, T38-8, T50-7 from Micrometals. Capacitors with standard values. Resistors with standard values. 10cm x 10cm PCB board pieces to solder your components.