ElecEng 2CF3

ASSIGNMENT 3

Resonant Circuits and Filters

Nooshin V. Shahmirzadi and Natalia K. Nikolova

Instructions and objective

- Perform all exercises.
- For each exercise, there are instructions on what to include in a short report about your observations. The report also must contain the answers to the posed questions.
- For each exercise, you are required to submit: (i) your final LTspice (*.asc) file and (ii) your short report (*.pdf) file. Submit to the respective Dropbox on A2L by the respective deadline. Do not include intermediate *.asc files. Failure to submit either in the correct file format will be penalized.
- Each exercise report (the PDF) must include (at the top of the first page) a suitable short title, your full name as it appears on Avenue to Learn, your student ID number, and your MacID login name.
- If you need help in using LTspice, please refer to LTspice *QuickStart Guide* PDF file on A2L. Help is also available from the TAs from 3:30 pm to 4:30 pm every day from Monday to Friday.
- There is a requirement to include SPICE netlists in the report. Just copy-paste the netlist text from LTspice. Do not export or submit any netlist files.

The objective of this assignment is to practice the analysis and design of basic filter networks.

EXERCISE #1: *RLC* SERIES NETWORK AT RESONANCE

A series *RLC* circuit (see Fig. 1) resonates at ω_0=1000 rad/s. The voltage-source phasor is given by $\mathbf{V}_s = 1\angle 0°$. The capacitance is C = 20 μF and the network impedance at the terminals of the voltage source at resonance is $\mathbf{Z}(\omega_0)$=5 Ω.

In preparation for the simulations, calculate: (a) the inductance L, (b) the quality factor Q, (c) the resonant frequency f_0 in Hz, (d) the magnitude and phase of the inductor's voltage $\mathbf{V}_L(f_0)$ at resonance, and (e) the magnitude and phase of the capacitor's voltage $\mathbf{V}_C(f_0)$ at resonance. Save your calculations and answers for inclusion in your report.

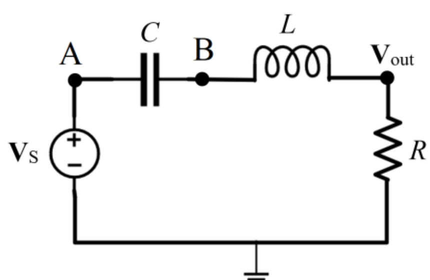

Fig.1. *RLC* series circuit.

Build the series *RLC* circuit in LTspice using the value of L that you calculated. Define $\mathbf{V}_s$ as a sinusoidal voltage source with amplitude of 1 V and frequency f_0 (the resonant frequency that you calculated). Choose transient analysis for your simulation. Suitable simulation parameters are:

```
. tran 0 5.1 5 1e-2
```

Simulate your circuit and plot the voltage waveforms as a function of time over the capacitor, $v_C(t)$, and over the inductor, $v_L(t)$.

Hint: For plotting $v_L(t)$, you need the potential difference between **B** and $\mathbf{V}_{\text{out}}$ (see Fig. 1). To obtain this plot, move the cursor to point **B**, and put by a left click and holding the left click drag the mouse to the point $\mathbf{V}_{\text{out}}$ and place another there by releasing the left click. An alternative way is to define $\mathbf{V}_{\text{out}} - \mathbf{V}_{\text{B}}$ in "`Expression(s) to add`" in the `Add Traces to Plot` window.

For the Report

1. Include the complete schematic (screenshot or image export).
2. Include the complete netlist (`View→SPICE Netlist`).
3. Include the LTspice (*.asc) file, named properly, e.g., `Exercise1.asc`.
4. Include your calculations and final answers for L, Q, f_0, and the phasors $\mathbf{V}_L$ and $\mathbf{V}_C$ (magnitude and phase).
5. Include the images of the $v_C(t)$ and $v_L(t)$ waveforms resulting from the simulation.
6. Do the simulation results for $v_C(t)$ and $v_L(t)$ at the resonant frequency confirm the magnitude and phase calculation for the phasors $\mathbf{V}_L$ and $\mathbf{V}_C$ in part 4? Justify your answer.

EXERCISE #2: *RLC* SERIES NETWORK AS A BANDPASS FILTER

In this Exercise, you will use again the *RLC* series circuit in Fig. 1 with the same values for R, L, and C. Calculate the 3-dB bandwidth (BW) and the cut-off frequencies f_{LO} and f_{HI} (all in Hz) for the voltage-gain transfer function $\mathbf{G}_v(j\omega) = \mathbf{V}_{\text{out}}(j\omega)/\mathbf{V}_{\text{s}}$.

Modify the simulation of the series *RLC* circuit (Fig. 1) in LTspice so that its frequency-domain analysis can be carried out. The setup for frequency-domain analysis is explained on page 10 of the LTspice *QuickStart Guide* PDF file on A2L. Suitable simulation parameters are:

```
.ac dec 500 10 1e3
```

Simulate the circuit and observe the voltage-gain magnitude and phase versus frequency. To achieve that, use "`Expression(s) to add`" in the `Add Traces to Plot` window and define `V(vout)/V(a)`.

For the Report

1. Include the complete schematic (screenshot or image export).
2. Include the complete netlist (`View→SPICE Netlist`).
3. Include the LTspice (*.asc) file, named properly, e.g., `Exercise2.asc`.
4. Include the calculation of the 3-dB BW, f_{LO}, and f_{HI}.
5. Include the magnitude-dB and phase plots of $\mathbf{G}_v$ versus frequency from the simulation. They can be plotted on a common graph. With the simulation settings suggested above, the frequency axis should be automatically set to a logarithmic scale.
6. Does the magnitude-dB plot of $\mathbf{G}_v$ versus frequency confirm your calculations for BW, f_{LO}, and f_{HI}? To answer this question, you need to readout the values of f_{LO} and f_{HI} from the plot.
   Hint: To read a value from a graph, place your cursor over the graph and look at the bottom left of the LTspice window at the status bar. It displays the values at the cursor as you move it over the graph. Use zoom in or zoom out options on the toolbar to observe more accurate values.
7. What are the values of the phase of $\mathbf{G}_v$ at the resonant frequency f_0 as well as the cut-off frequencies, f_{LO} and f_{HI}, according to the simulation plot? What are these values according to theory? Is there an agreement?

EXERCISE #3: LOW-PASS FILTER

In this exercise, you will design an *RC* circuit to filter out high-frequency noise. You will set the values of the capacitor *C* and the resistor *R* to achieve half-power (3-dB) cut-off frequency of 25 kHz. For this purpose, set *R* to be equal to the sum of the last two digits of your student number in kΩ and then calculate *C* from the cut-off frequency and *R*.

Build the *RC* circuit in LTspice and simulate it. Suitable frequency-domain simulation parameters are:

```
.ac dec 500 50 5e6
```

For the Report

1. Include the complete schematic (screenshot or image export).
2. Include the complete netlist (`View→SPICE Netlist`).
3. Include the LTspice (*.asc) file, named properly, e.g., `Exercise3.asc`.
4. Include the values of *R* and *C*. Show the calculation for *C*.
5. Include the magnitude-dB plot of the transfer function obtained from the simulation.
6. From the magnitude-dB plot of the transfer function, determine the cut-off frequency. Is it in agreement with the required value of 25 kHz?

EXERCISE #4: HIGH-PASS FILTER

Change the schematic of **Exercise #3** so that it now represents a **high-pass filter** with cut-off frequency of 100 kHz. Again, the resistor value *R* must be equal to the sum of the last two digits of your student number in kΩ. You can then calculate *C* from the cut-off frequency and *R*.

Build the circuit in LTspice and simulate it in order to obtain the magnitude-dB plot of the transfer function as a function of frequency. Suitable simulation parameters are:

```
.ac dec 500 50 500e3
```

For the Report

1. Include the complete schematic (screenshot or image export).
2. Include the complete netlist (`View→SPICE Netlist`).
3. Include the LTspice (*.asc) file, named properly, e.g., `Exercise4.asc`.
4. Include the values of *R* and *C*. Show the calculation for *C*.
5. Include the magnitude-dB plot of the transfer function versus frequency.
6. From the magnitude-dB plot of the transfer function, determine the cut-off frequency. Is it in agreement with the required value of 100 kHz?