LTSpice assignments - Week 2.1 (Tuesday) in Tellegen Hall

In this lab in week 2.1, you will analyze and simulate three assignments using LTSpice as follows:

- A second-order circuit (a series RLC circuit)
- AC-DC rectifier using one diode
- Simulation of the nodal analysis circuit (which you had in the Python assignment)

Preparation:

- Make sure that you have the LTSpice installed on your laptop (**note**: do not forget to bring your laptop to EWI)
    - Installation URL link: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

- There are a number of LTSpice tutorial available on internet. (As an example, you can optionally use[1] http://denethor.wlu.ca/ltspice/)
- Always provide your simulation results with a non-black preferably white background waveform (see https://spiceman.net/ltspice-initial-setting/, in Tools > Control Panel > "Waveforms" in Color Scheme - Color Palette Editor. Go to "selected item", click on Background > then set R & G and B at 255)

Assignment 1: Second-order circuit (a series RLC circuit)

You became familiar with the Second-order circuit (a series RLC circuit) in the EE1C11 course in Q1.

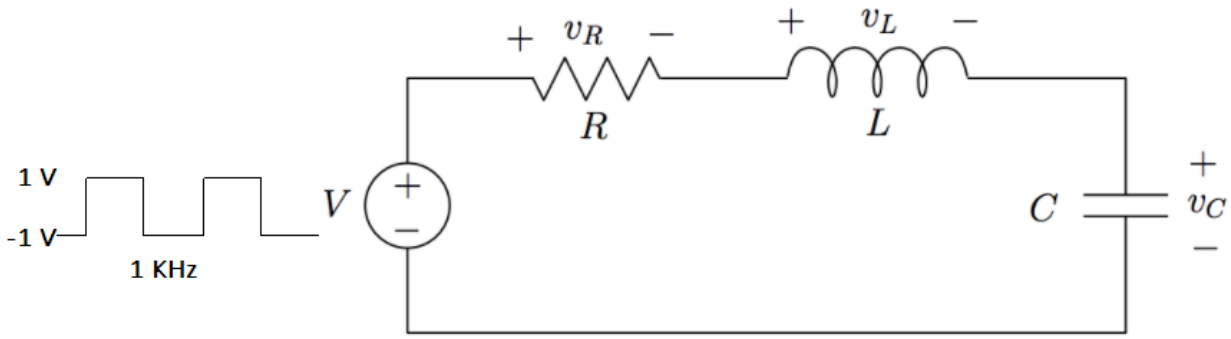

Fig 1. Series RLC circuit with a square wave voltage source.

Consider the RLC circuit shown in Figure 1 (C=100 nF, L=1 mH).

- Calculate the value of the resonant frequency ω_0 (rad/s) and f_0 (Hz).
- Calculate the value for the resistance R where the damping factor (α) is equal to ω_0 (rad/s).
- Calculate the value for the resistance R where the damping factor (α) is equal to 0.25*ω_0 (rad/s).
- Calculate the value for the resistance R where the damping factor (α) is equal to 5*ω_0 (rad/s).

Start to create and simulate the RLC circuit in LTSpice. The input voltage is a square wave (Pulse) with a peak-to-peak voltage of 2 V (-1 to +1 V) with the frequency of 1 KHz. Do not forget to connect the ground and use `.tran` for simulations.

Attention: Make sure you show the simulation results after five or six cycles.

[1] **Disclaimer**: The TUD EE programme does not take the responsibility for the content provided in the (optional) LTSpice links.

- Obtain the simulation result for the voltage across the capacitor using the value of R where the damping factor (α) is equal to ω_0 (rad/s).
- Obtain the simulation result for the voltage across the capacitor using the value of R where the damping factor (α) is equal to 0.25*ω_0 (rad/s).
- Obtain the simulation result for the voltage across the capacitor using the value of R where the damping factor (α) is equal to 5*ω_0 (rad/s).
- Compare your simulation results for the previous three cases with three different values for R (overdamped, underdamped and critically damped).
- For the underdamped case, obtain the value of the damped natural frequency ω_d. Then, give the damped natural frequency from your simulation results. Does the damped natural frequency from your simulation results match the calculated value?

Assignment 2: AC-DC rectifier using one diode

In this assignment, you become familiar with an AC-DC rectifier. The rectifier converts the AC input signal to an output DC signal.

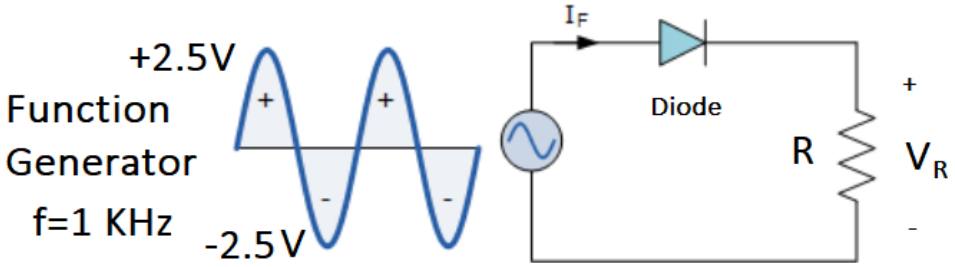

Fig 2. AC-DC rectifier using one diode and a resistor

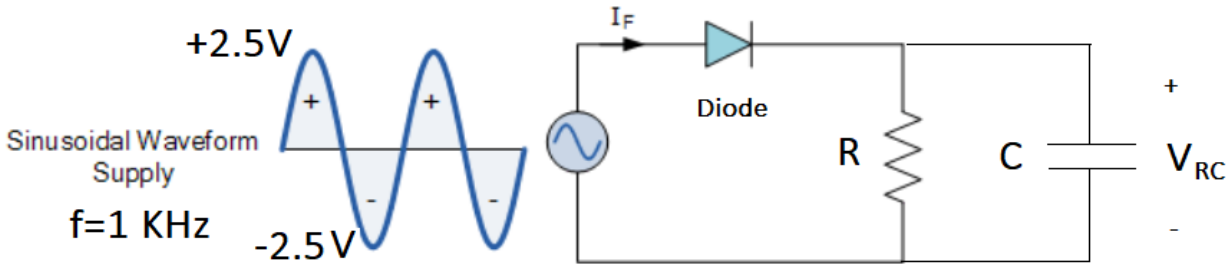

Fig 3. AC-DC rectifier using one diode, a resistor, and a capacitor.

Figure 2 shows an AC-DC rectifier using one diode and a resistor (R=10 KΩ). Taking into account the input AC voltage (5 V peak-to-peak), draw the output DC voltage V_R where the diode is 1N4148 type.

Figure 3 shows an AC-DC rectifier using one diode, a resistor (R=10 KΩ), and a capacitor (C=100 nF). Taking into account the input AC voltage (5 V peak-to-peak), draw the output voltage V_{RC} where the 1N4148 diode is used.

Start to create and simulate the circuits in LTSpice. The input voltage is an AC sinusoidal wave with a peak-to-peak voltage of 5 V (-2.5 to +2.5 V) with the frequency of 1 KHz.

- Obtain the simulation results for V_R in the circuit shown in Fig. 2. Does the simulation result match your drawings?
- Obtain the simulation results for V_{RC} in the circuit shown in Fig. 3. What is the impact of the capacitor on the output DC voltage?
- Repeat the simulations where you use a 20 KΩ resistor (instead of a 10 KΩ resistor). What is the impact of the value of the resistance on the output DC voltage?