Birzeit University
Faculty of Engineering and Technology
Department of Electrical and Computer Engineering
Circuit Analysis – ENEE2304

PSpice Assignment

Deadline for Submission of your report (via Ritaj): Tuesday 4-7-2023 (Max 23:59)

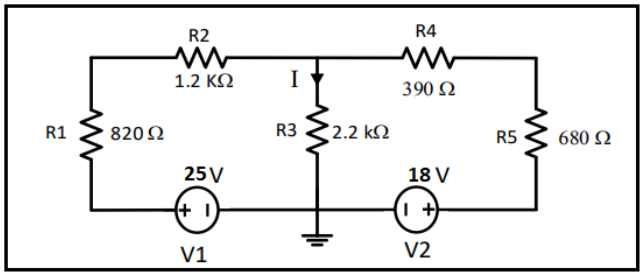
Student Name: **ID Number:**

Question # 1: Superposition Technique

Important Notes:

- ✓ The resistor R_3 must be named with the student name and ID. For example, if your name is Ahmad and your ID is 1219999 then, the resistor R_3 must be named as Ahmad_1219999. Otherwise, the problem will not be evaluated.
- ✓ The value of **R_3 not fixed**, the value depends on the **last two digits** of your ID, if your ID is 1219999 then R3=9.9 kΩ (it is 9.9 kΩ **Not** 99 kΩ).

For the circuit:



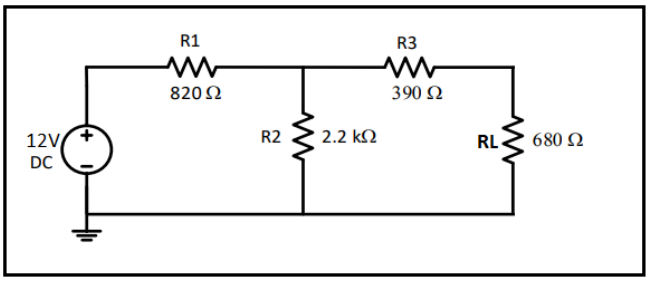
1. Use Pspice software to simulate the circuit and get the voltage across and the current through the resistor R3.
2. Apply superposition theorem to get the voltage across and the current through the resistor R3. You have to show all the results of simulation.
3. Compare the results obtained from step 1 and step 2.

Question #2: Thevenin's Theorem & Maximum Power Transfer

Important Notes:

- ✓ The resistor R_L must be named with the student name and ID. For example, if your name is Ahmad and your ID is 1219999 then, the resistor R_L must be named as Ahmad_1219999. Otherwise, the problem will not be evaluated.
- ✓ Also, note that on the simulation window, below the plot, your name and ID (name of the component R_L) must appear as seen in the example at the end of the assignment.
- ✓ The value of **R_L not fixed**, the value depends on the **last three digits** of your ID, if your ID is 1219999 then R3=999 Ω (it is 999 Ω **Not** 999 kΩ).

For the circuit:



1. Use Pspice software to simulate this circuit and get the voltage across and the current through the resistor RL (xxx (last three digits) Ω).
2. Using DC sweep, set RL as a parameter that varies from 50 Ω to 1.5 kΩ and **plot** the power dissipated by RL as it varies (plot the power of RL versus the value of RL). With the help of cursors on Pspice simulation window, approximate at which value of RL the power maximizes)
3. Use Pspice software to calculate $R_{thevenin}$ seen by the resistor RL. **Use V_{oc} and I_{sc} method only**. You have to show all the simulation results when getting V_{oc} and I_{sc}.
4. Compare the value of RL at P_{max} obtained from step 2 and the value of $R_{thevenin}$ obtained from step 3.
5. Build and then simulate the Thevenin equivalent circuit with the load resistor RL and show the voltage across and the current through the resistor RL.
6. Compare the results obtained from step 1 and step 5.

Question # 3: Sinusoidal Steady State Analysis

Important Notes:

- ✓ The resistor R must be named with the student name and ID. For example, if your name is Ahmad and your ID is 1219999 then, the resistor R must be named as Ahmad_1219999. Otherwise, the problem will not be evaluated.
- ✓ Also, note that on the simulation window, below the plot, your name and ID (name of the component R) must appear as seen in the example at the end of the assignment.

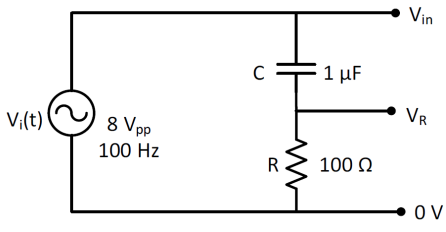


Fig. 3.1 Capacitive circuit

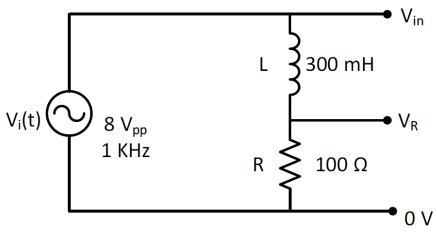


Fig. 3.2 Inductive circuit

For the circuit shown in Fig. 3.1:

1. Use PSPICE to do transient analysis of the circuit, show $V_{in}(t)$ and $V_R(t)$ on one plot (you may need to use different Y-axes).

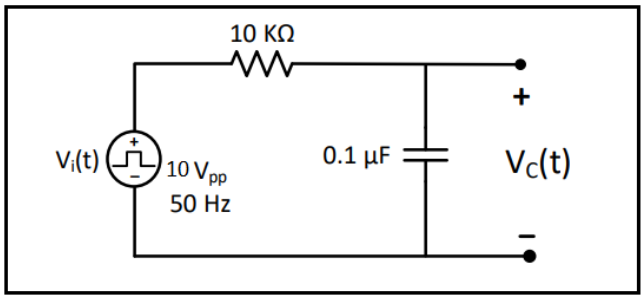
2. Use cursors to measure the time difference between the peaks of the two signals, then use the following relationship to calculate the phase shift using the measured time $\{\Delta\boldsymbol{\theta}=\mathbf{360}^\circ \mathbf{\ x\ f\ x\ } \Delta\mathbf{t}\}$.
3. Repeat the same procedure in the step 1and 2 above for the circuit shown in Fig. 3.2.
4. Compare and discuss the results obtained for the two circuits.

Question #4: First Order RC Circuit Analysis

Important Notes:

- ✓ The capacitor C must be named with the student name and the ID. For example, if your name is Ahmad and your ID is 1219999 then, the capacitor C must be named as Ahmad_1219999. Otherwise, the problem will not be evaluated.
- ✓ Also, note that on the simulation window, below the plot, your name and ID (name of the component C) must appear as seen in the example at the end of the assignment.

For the circuit:



The input voltage is square signal with 10 $V_{peak\text{-}peak}$ (0 V to 10 V) and frequency of 50Hz.

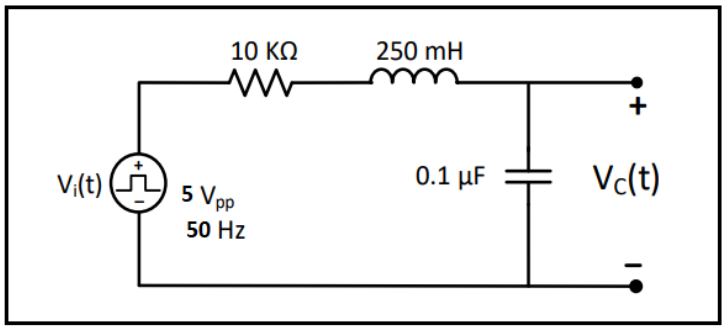
1. Use Pspice software to plot both $V_i(t)$ and $V_c(t)$ (on the same graph) for a meaningful period of time.
2. With help of cursors on Pspice simulation window, show the value of the time constant (τ). You have to show both the circuit and the simulation result.

Question #5: Second Order RLC Circuit Analysis

Important Notes:

- ✓ The capacitor C must be named with the student name and the ID. For example, if your name is Ahmad and your ID is 1219999 then, the capacitor C must be named as Ahmad_1219999. Otherwise, the problem will not be evaluated.
- ✓ Also, note that on the simulation window, below the plot, your name and ID (name of the component C) must appear as seen in the example at the end of the assignment.

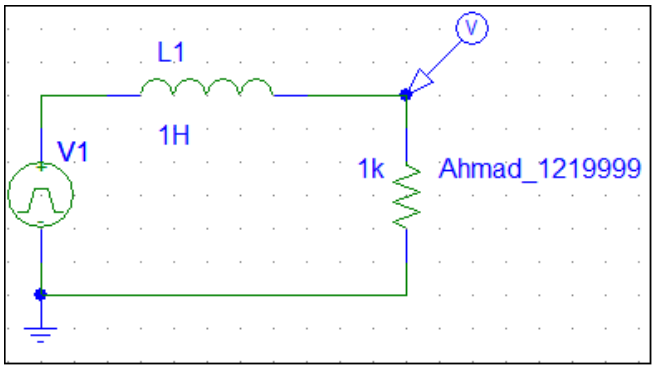
For the circuit:

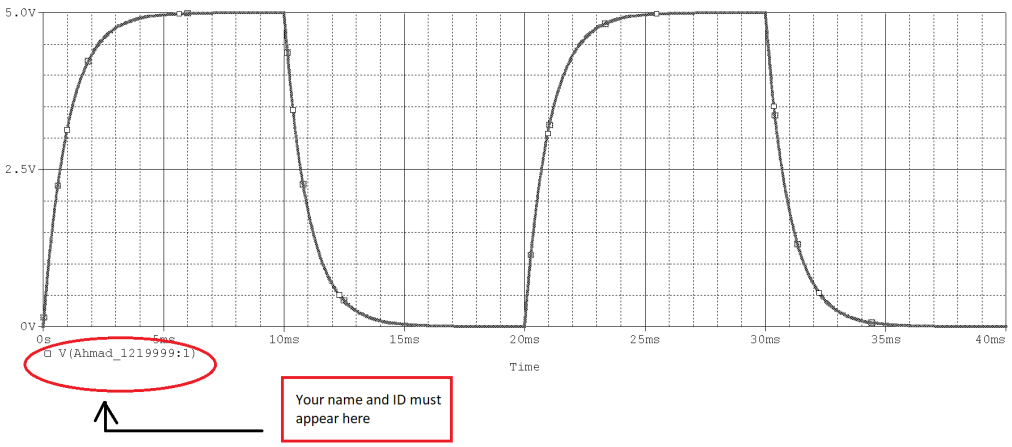


The input voltage is square signal with 5 $V_{peak\text{-}peak}$ (0 V to 5 V) and frequency of 50Hz.

1. Use Pspice software to plot both $V_i(t)$ and $V_c(t)$ (on the same graph).
2. Change the Value of R to 3.162 kΩ, repeat step 1.
3. Change the Value of R to 500 Ω, repeat step 1.
4. Comment on each result: is it over-damping, critical-damping, or under-damping response.

Example of a circuit and the simulation result





... With Best Wishes ...