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 14-2-2023 (Max 23:59)

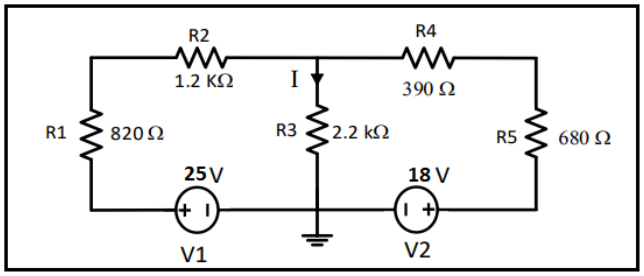
Student Name: **ID Number:**

Important notes for questions 1 and 2

- 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.

Question # 1: Superposition Technique

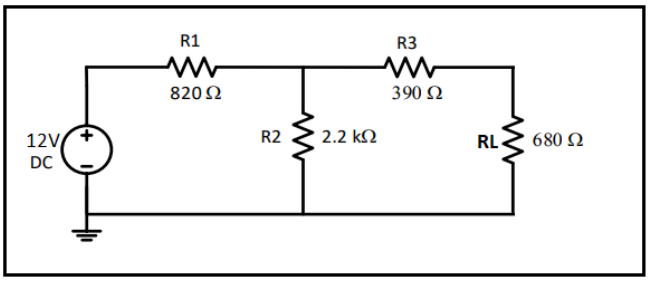
For the circuit shown below:



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

For the circuit shown below:



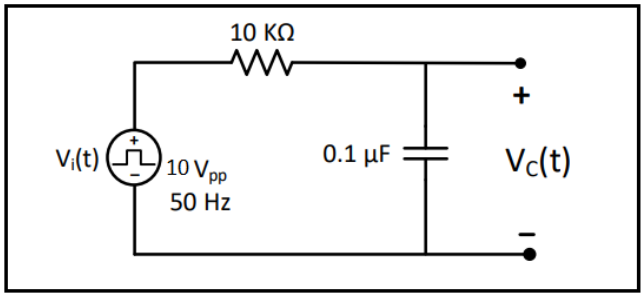
1. Use Pspice software to simulate this circuit and get the voltage across and the current through the resistor RL (680 Ω).
2. Using DC sweep, set RL as a parameter that varies from 100 Ω to 1.5 kΩ and **plot** the power dissipated by RL as it varies (plot the power of RL versus the value of RL). With 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.

Important notes for questions 3 and 4

- 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.

Question #3: First Order RC Circuit Analysis

For the circuit shown below:

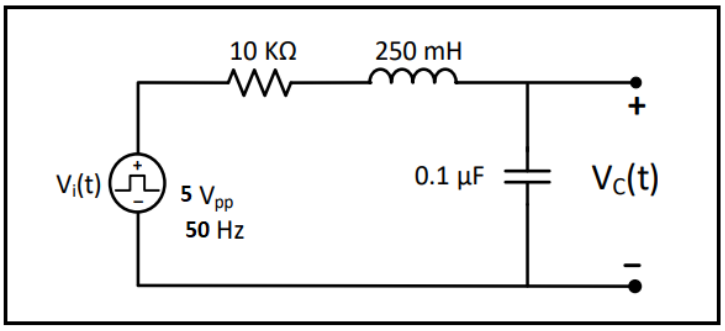


The input voltage is square signal with 10 V peak-peak (0 V to 10 V) and frequency of 50Hz.

1. Use Pspice software to plot both Vi(t) and Vc(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 #4: Second Order RLC Circuit Analysis

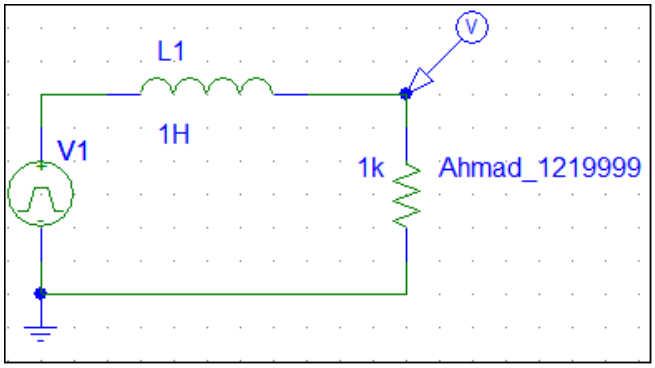
For the circuit shown below:

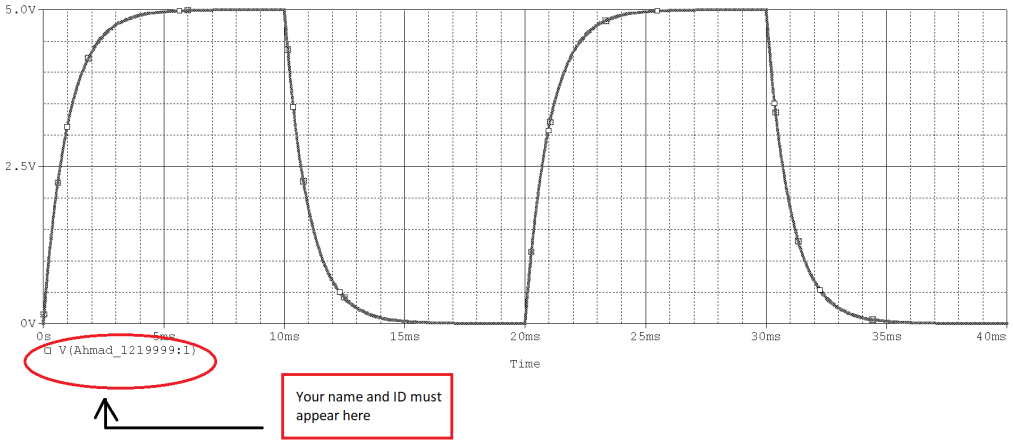


The input voltage is square signal with 5 V peak-peak (0 V to 5 V) and frequency of 50Hz.

1. Use Pspice software to plot both Vi(t) and Vc(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 ...