



Faculty of Engineering and Technology
Department of Electrical and Computer Engineering

ENEE2304, CIRCUIT ANALYSIS Semester II 2022

Course project

Instructors: Dr. Wasel Ghanem, Dr. Hakam Shehadeh and Dr. Alhareth Zyoud

Project submission deadline: 16/6/2022 23:55 on ITC

Project discussion: To be defined later

Instructions:

- Each Student should work individually
- Solve the two questions in this assignment using PSPICE 9.1 student version.
- Project submission must be via Moodle only. Your report must include a clear snapshot (use snipping tool) of the circuit's schematic in PSPICE.
- Some of the resistors' values in the questions are given as (X, Y, and Z); the values of these terms depend on your university ID Number. The last three digits of your university ID number correspond to X, Y, and Z respectively. In case X or Y or Z corresponds to zero in your ID number, make its value 5 instead of zero.
- In each circuit, name the targeted component with your name as shown below.

- In the circuit shown in figure 1, in case you want to show the voltage across the resistor R1. So instead of R1 you need to write your as shown in figure 2. Therefore, once you run the simulator, your name will appear in the figure as shown in figure 3

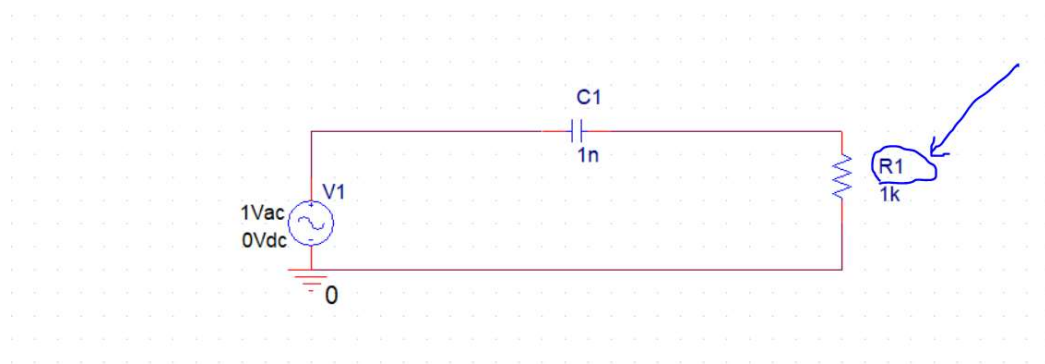


Figure 1

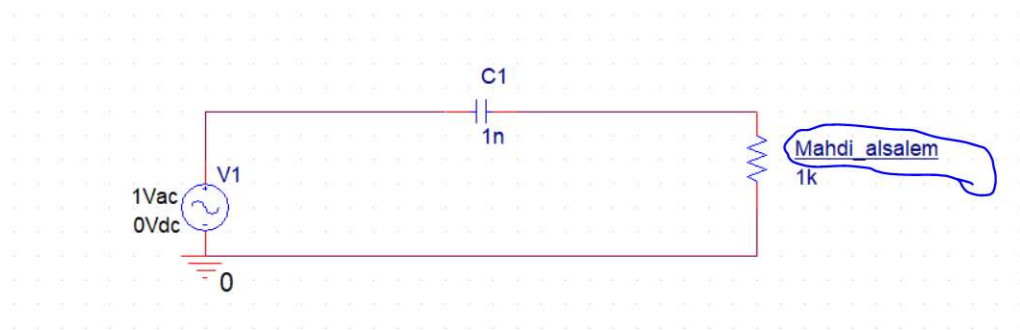


Figure 2

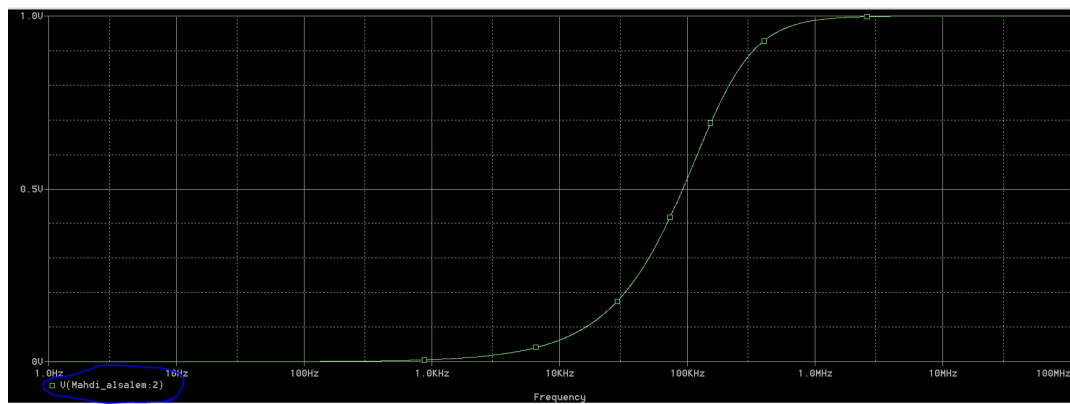
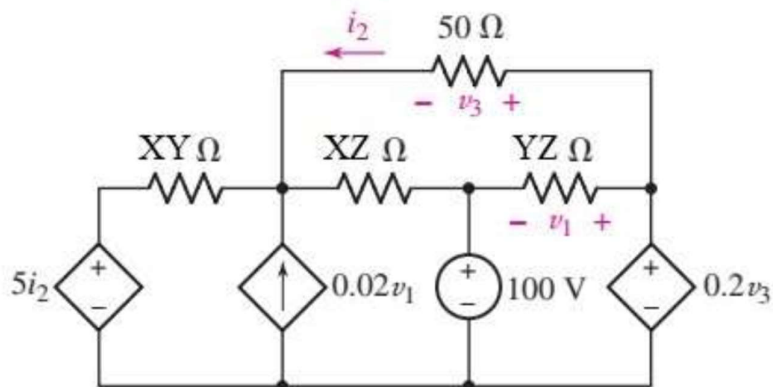


Figure 3

Question 1:

Construct a PSPICE schematic for the circuit shown in the figure below. Simulate the schematic and show voltages at each node and current in each branch.

**Question 2:**

The circuit shown in Figure below has two inputs, $v_s(t)$ and $i_s(t)$, and one output, $v(t)$. When inputs are given by $V_s(t) = V_m \sin 6t$ V and $i_s(t) = I_m$ A the output will be $v_o(t) = A \sin(6t + \theta) + B$ V

3Linearity requires that A be proportional to V_m and that B be proportional to I_m . Consequently, we can write $A = k_1 V_m$ and $B = k_2 I_m$, where k_1 and k_2 are constants yet to be determined.

- Use PSpice to determine the value of k_1 by simulating the circuit, using $V_m = 1$ V and $I_m = 0$.
- Use PSpice to determine the value of k_2 by simulating the circuit, using $V_m = 0$ V and $I_m = 1$.
- Knowing k_1 and k_2 , specify the values of V_m and I_m that are required to cause $v_o(t) = 5 \sin(6t + \theta) + 5$ V. Simulate the circuit, using PSpice to verify the specified values of V_m and I_m .
- Determine the average power delivered by $v_s(t)$ using Pspice.

