BIRZEIT UNIVERSITY

Faculty of Engineering & Technology

Electrical & Computer Engineering Department

ENEE2304

PSPICE Project

Name: Baraa Nassar

ID: 1210880

Instructor: Dr. Mahran Quraan

Sec: 3

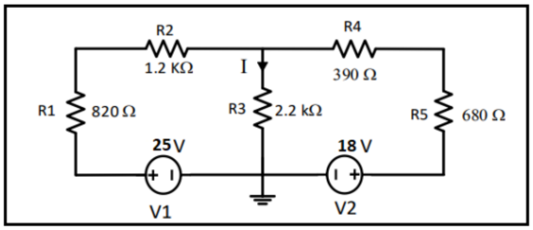
Problem # 1:

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.

The circuit obtained by Pspice simula on accompanied with the value of each component is shown in figure 1.1 below.

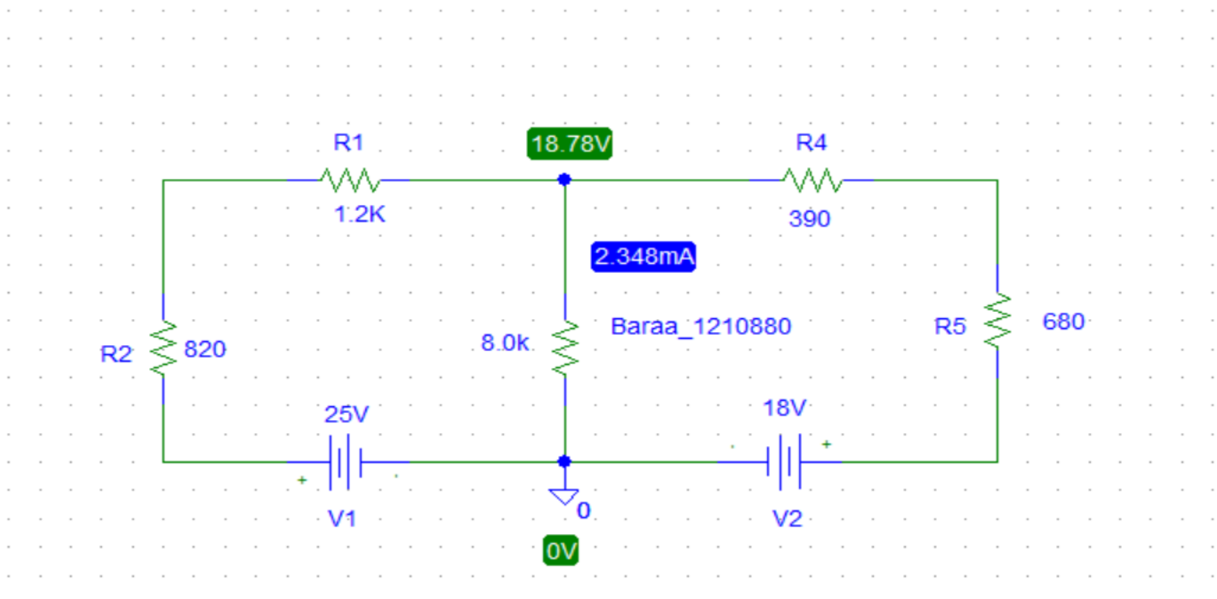


Figure 1.1 (Pspice simulation for problem 1a)

Voltage across the resistor R3 = 18.78V

Current through the resistor R3 = 2.348mA

Before you dive into the other questions, you should be familiar with some of the theories behind circuit analysis, namely, for now, The Superposition theorem.

Superposition theorem states that for any circuit containing only linear circuit elements (IE: a linear system), the net response (voltage or current) caused by independent sources in any branch or around terminals of a component is equal to the individual sum of the contribution of each independent source taken alone. More formally stated :

$L(a_1x_1 + a_2x_2 \cdots a_nx_n) = a_1L(x_1) + a_2L(x_2) + \cdots a_nL(x_n)$.

This theory is a special case of the more general result in mathematics, The superposition principle, readers striving for more knowledge will be pointed to the reference.

We are now ready to examine the second part of this problem; using superposition to get the outputs of this circuit.

We need to do 2 things, first of all, we need to turn off all independent sources and leave only one on, current sources are replaced with an open circuit (so as to kill its contribution) , voltage sources are replaced with a short circuit. In this example, we only have voltage sources. We begin by killing the one on the left, and deriving the current and voltage in the center branch. Getting $v'1$ = 10.82v, $i'1$ =1.353 mA.

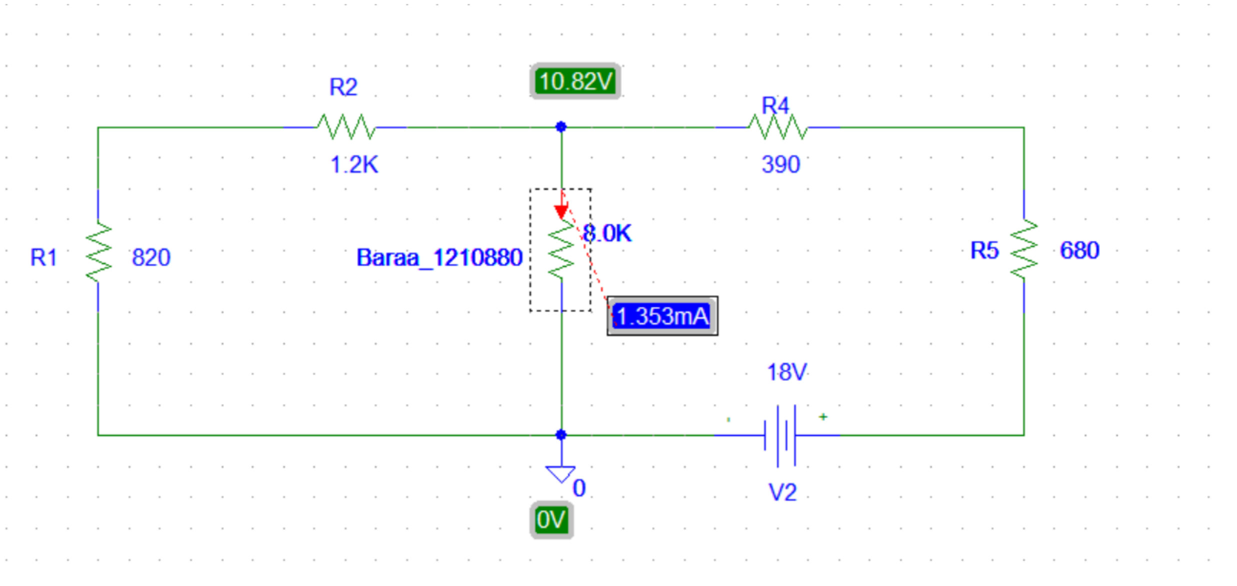


Figure 1.2 shows the circuit after killing the voltage source on the left

Now we need to kill the second voltage source and extract the results as in part 1. Doing so we have $v''1 = 7.961v$, $i''1 = 0.995mA$.

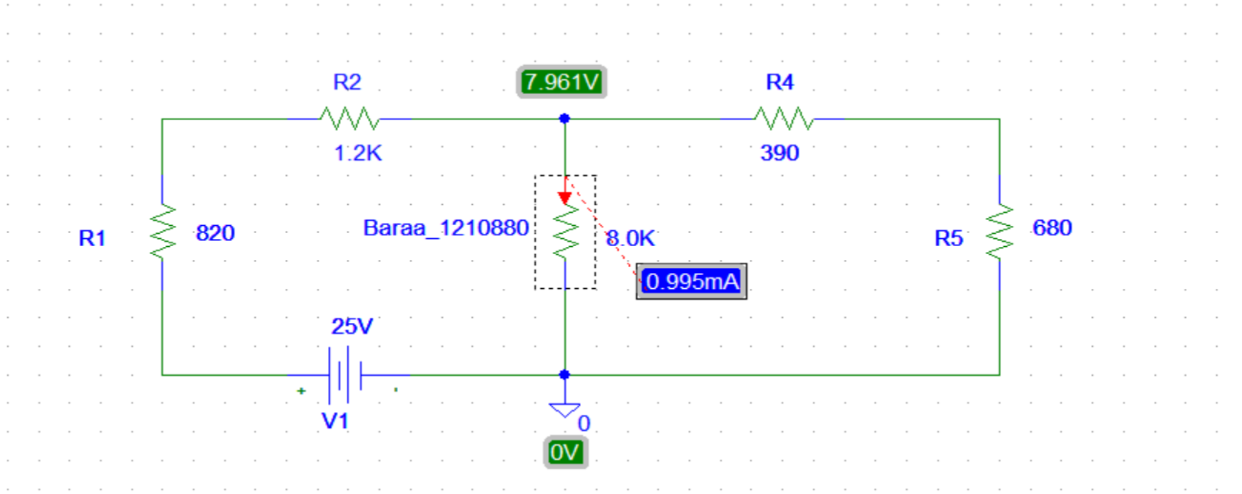


Figure 1.3 shows the original circuit with the second source disabled

As stated previously, the voltage and current of this branch is the sum of all contributions of the individual sources. Hence $v = v' + v'' \gg v = 10.82 + 7.961 \approx 18.781v$

$i = i' + i'' \gg i = 1.353 + 0.995 = 2.348mA$.

Both results are approximately equal to the original, as per the superposition theorem.

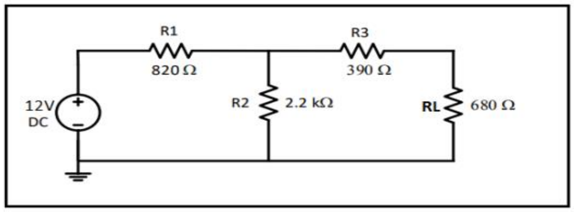
Problem #2:

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.

The circuit for Pspice simulation along with the value of each component is shown in the figure below.

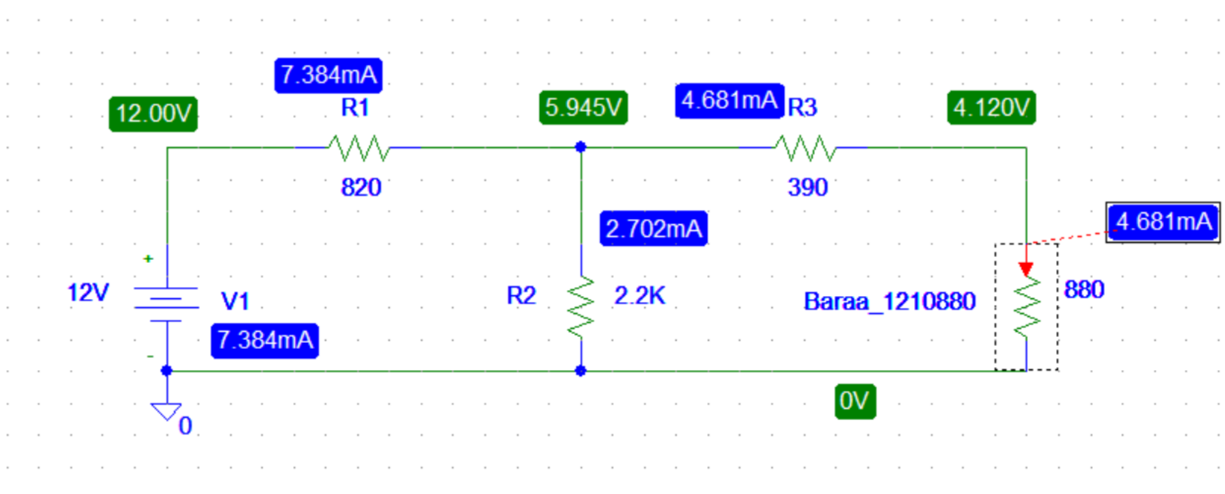
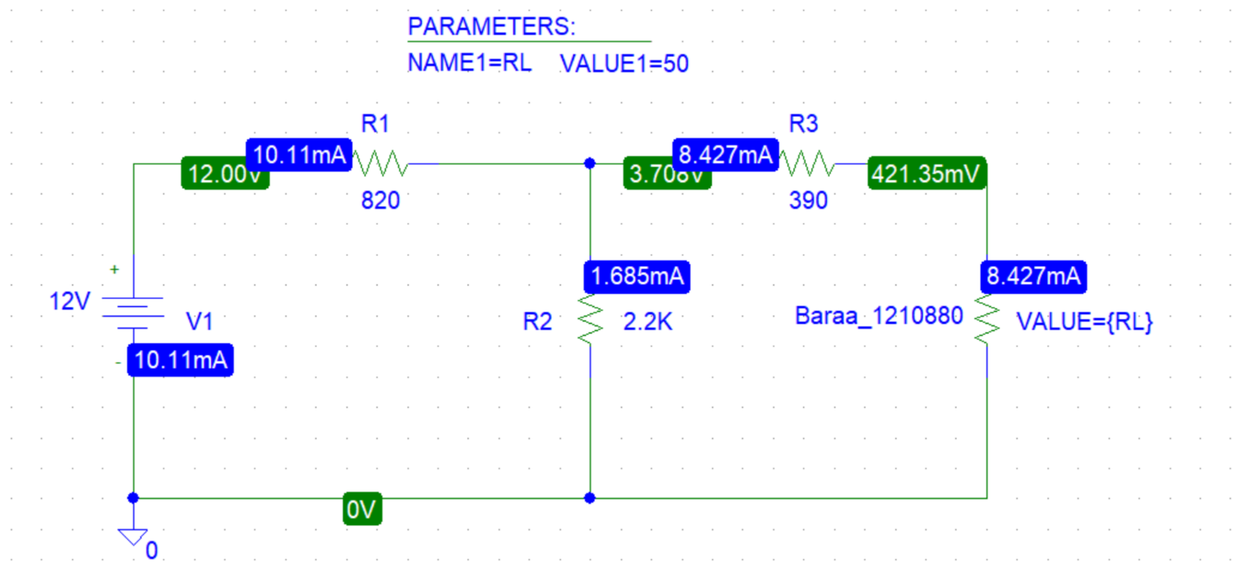


Figure 2.1 shows the simulation for the circuit in problem 2.1

Voltage across the resistor RL (Baraa_1210880) = 4.120V
Current through the resistor RL (Baraa_1210880) = 4.681mA

For the second part of this problem, the graph of the power dissipated by RL (the one with my name) for this circuit with respect to the resistance of RL is to be shown below. This is done so we would be able to determine the the value of RL for which power is maximum (this point is pinned on the graph). It is also almost equal to Rth as we will illustrate later. The value of the maximum resistance here is 1K.



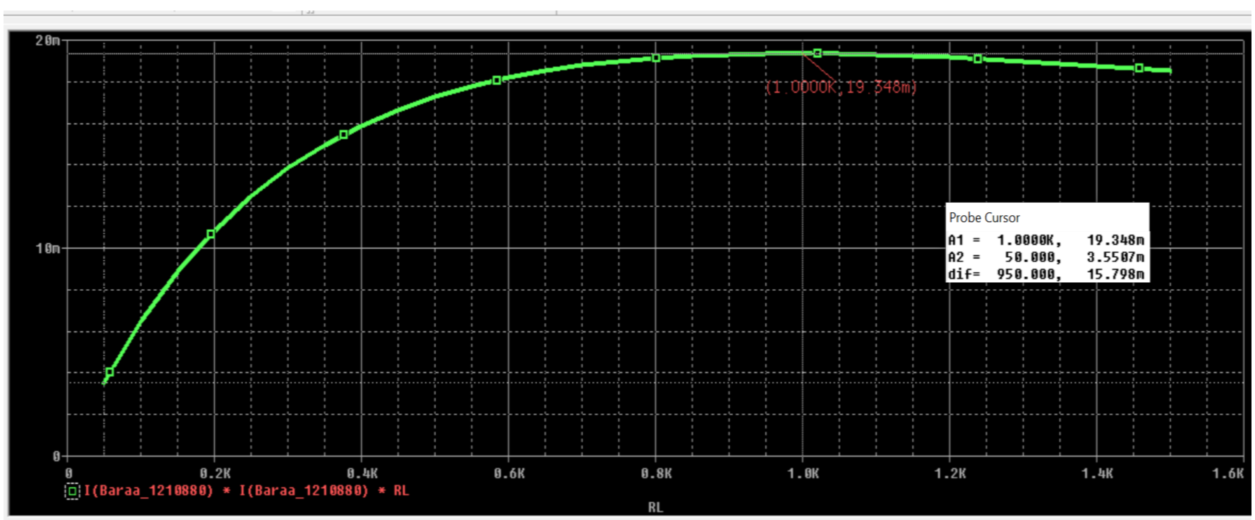


Figure 2.2 shows the graph of the power of RL with respect to value of RL

Before we continue any further, you need to be familiar with the basics of Thevenin's theorem, we will state the definition here and illustrate it with the problem in question. Thevenin's theorem states that any circuit, whether constituting of independent or dependent sources can be exchanged with one only containing a single source, be it current or voltage.

In other words, any circuit in existence can be simplified to the following form shown in figure 2.3.

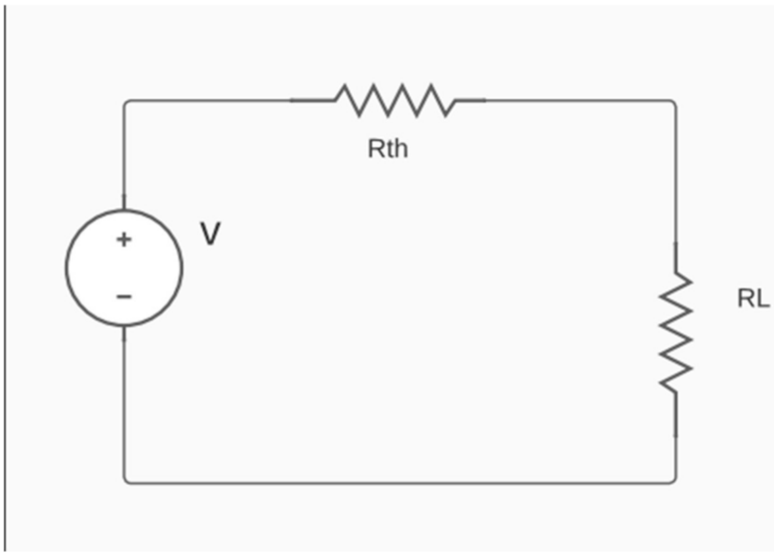


Figure 2.3 illustrates thevenins theorem

How to calculate V and Rth is a simple problem and will be illustrated in the following example.

To get the value of the voltage source we seek, we replace RL with an open circuit, then calculate the voltage across the terminals of $R\text{L}$. In this example, we get Voc = 8.742v.

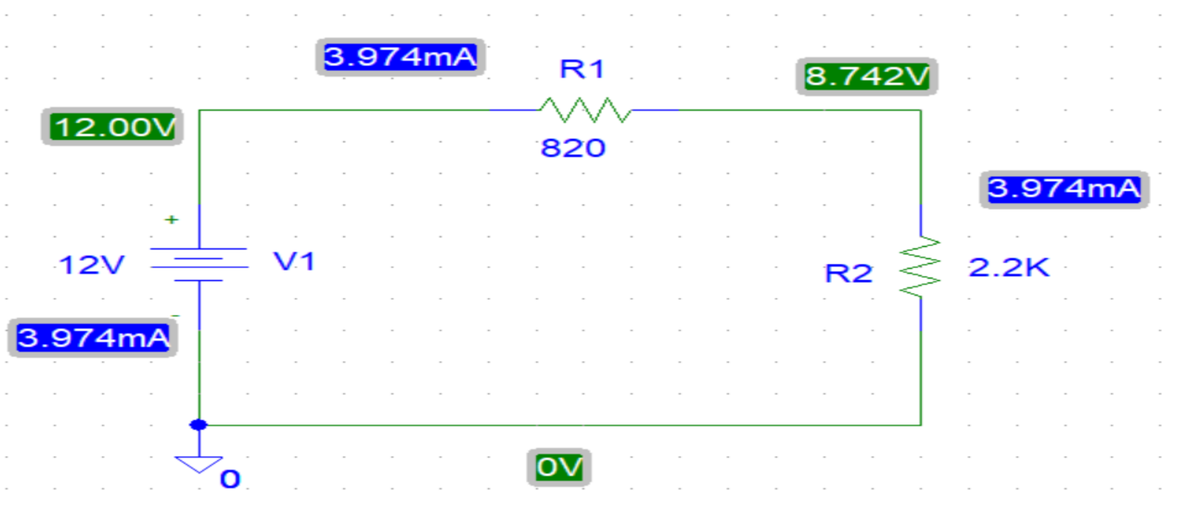


Figure 2.4 shows the original circuit with RL being replaced with an open circuit.

Then to calculate Rth, all we need is to determine the current that passes through the circuit replacing RL with a short circuit, the parameter obtained is called isc. In this example, we have isc= 8.854mA. Thus Rth= Voc/isc =8.742v/ 8.854mA=987.35Ω.

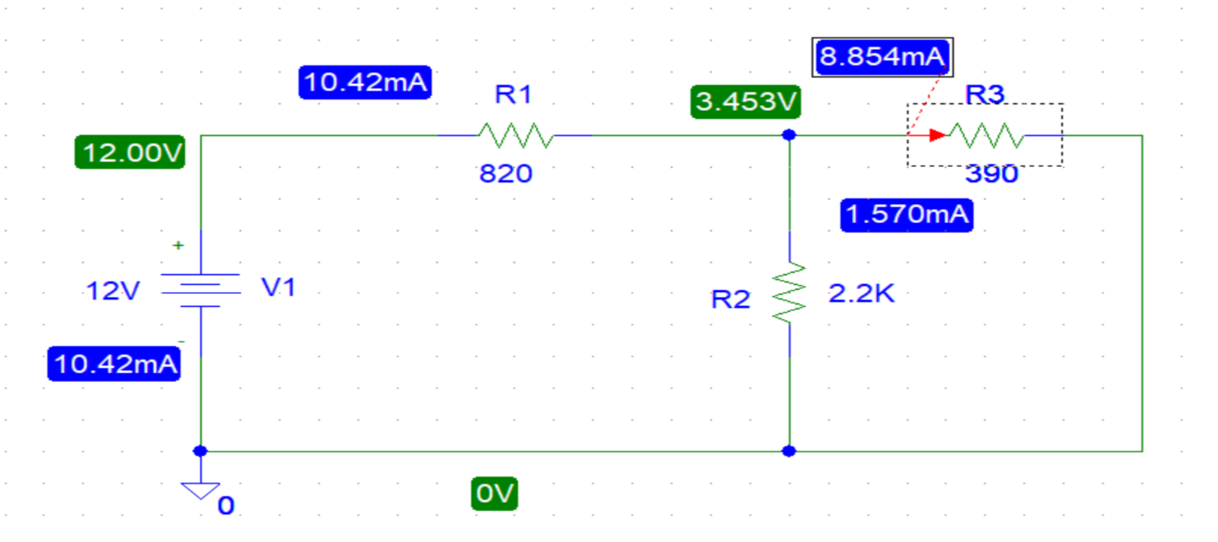


Figure 2.5 shows the value of isc after shorting the circuit out.

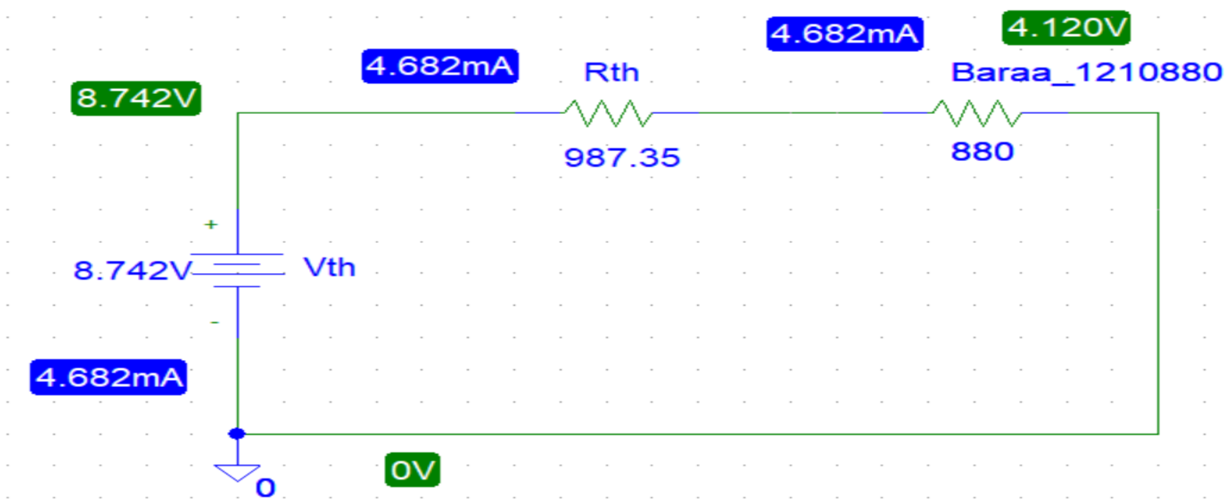


Figure 2.6 shows the equivalent circuit with Rth and Voc

```
Voltage across the resistor RL (Baraa_1210880) = 4.120V
Current through the resistor RL (Baraa_1210880) = 4.682mA
```

The values that are obtained for voltage and current through RL are identical because of Thevenin's theorem, and also due to the fact that the value of the resistor for which power is maximum is equal to Rth , hence getting the same value in both parts.

Problem #3:

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.

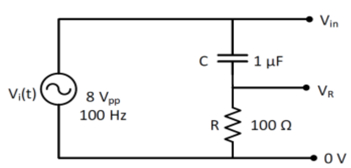


Fig. 3.1 Capacitive circuit

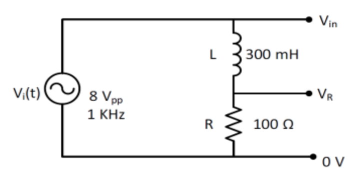


Fig. 3.2 Inductive circuit

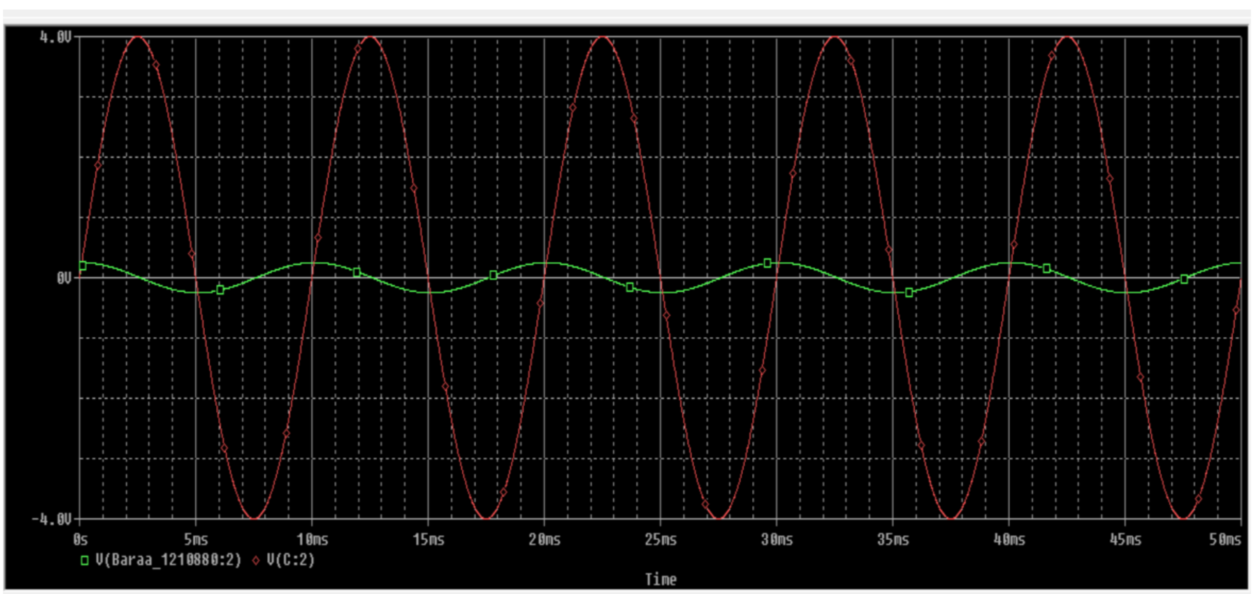
For the circuit shown in Fig. 3.1:

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).

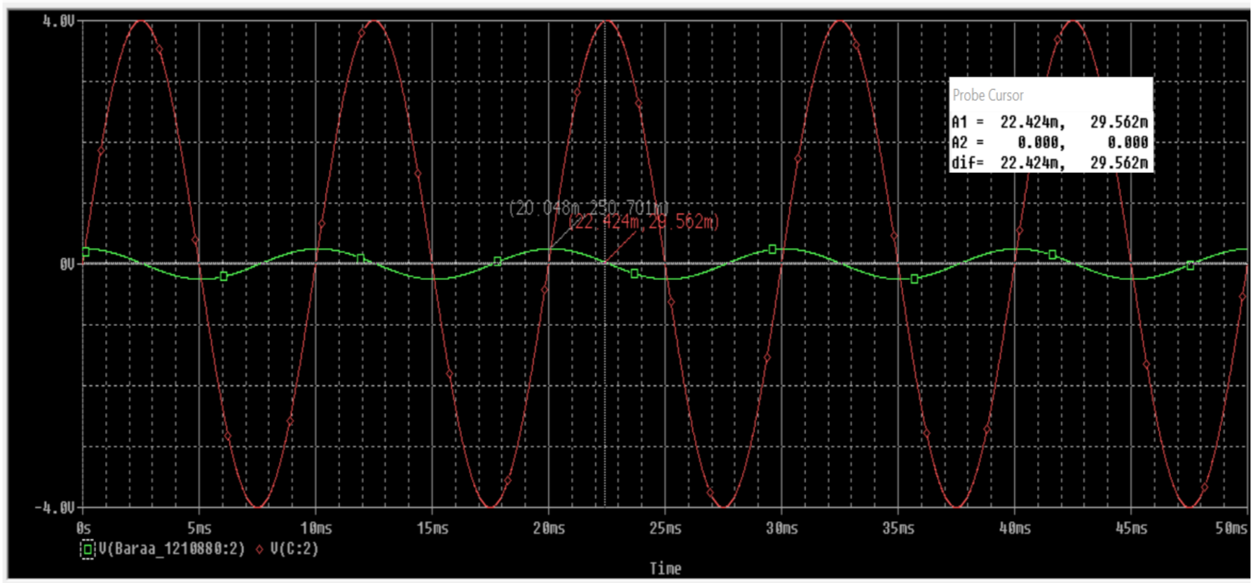
2. Use cursors to measure the time difference between the peaks of the two signals, then use the following relationship to calculate the phase shift using the measured time $\{\Delta\theta = 360° \times f \times \Delta t\}$.
3. Repeat the same procedure in the step 1and 2 above for the circuit shown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

For the circuit shown in Fig. 3.1:

Part one:



Part two:

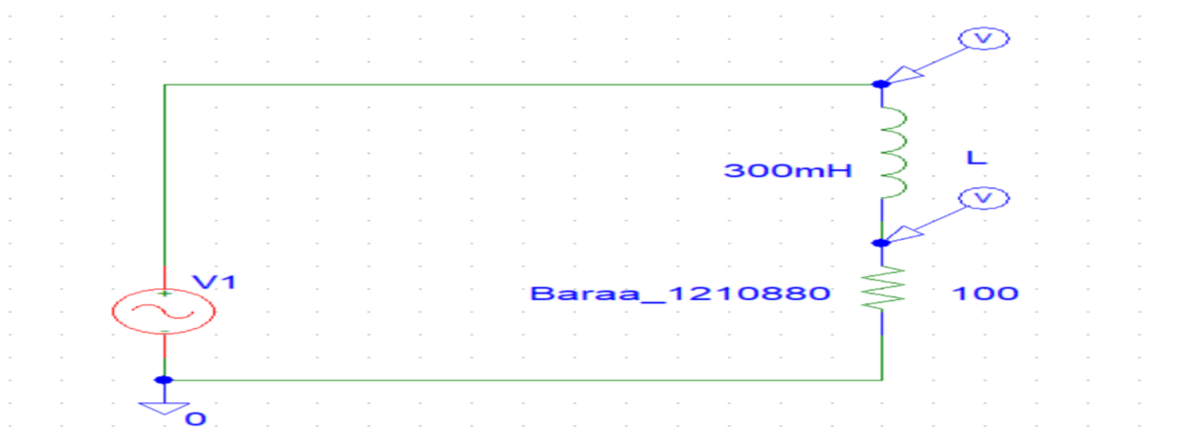


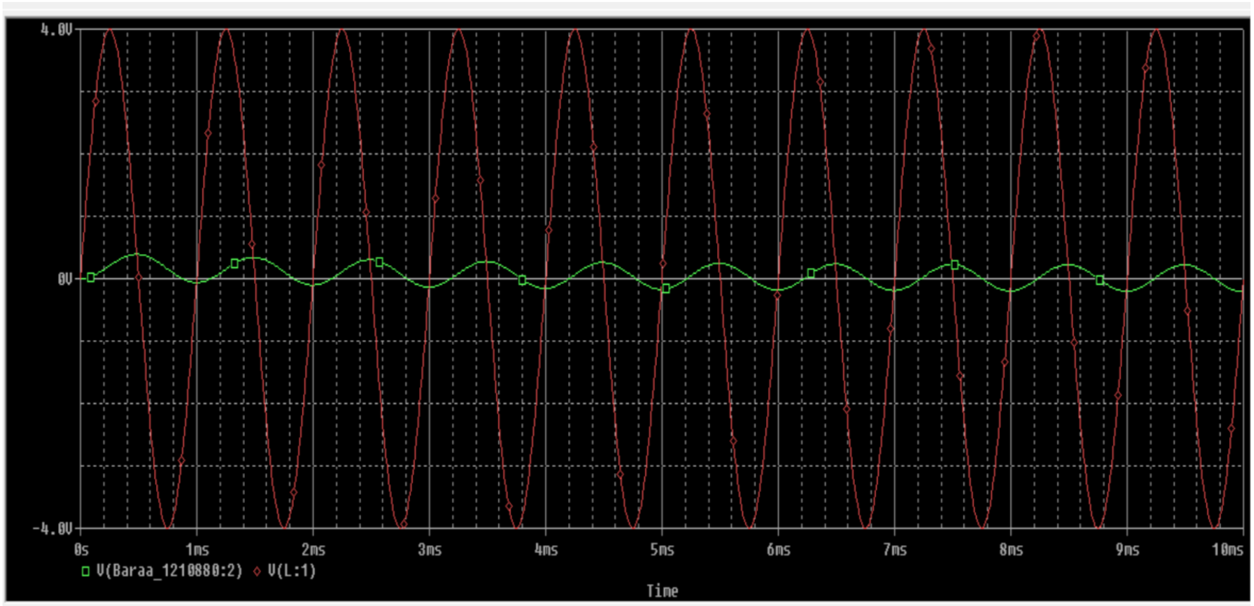
Δt= 22.424 - 20.048=2.376

Δθ=360° x f x Δt= 360° x 100 x 2.376=85m

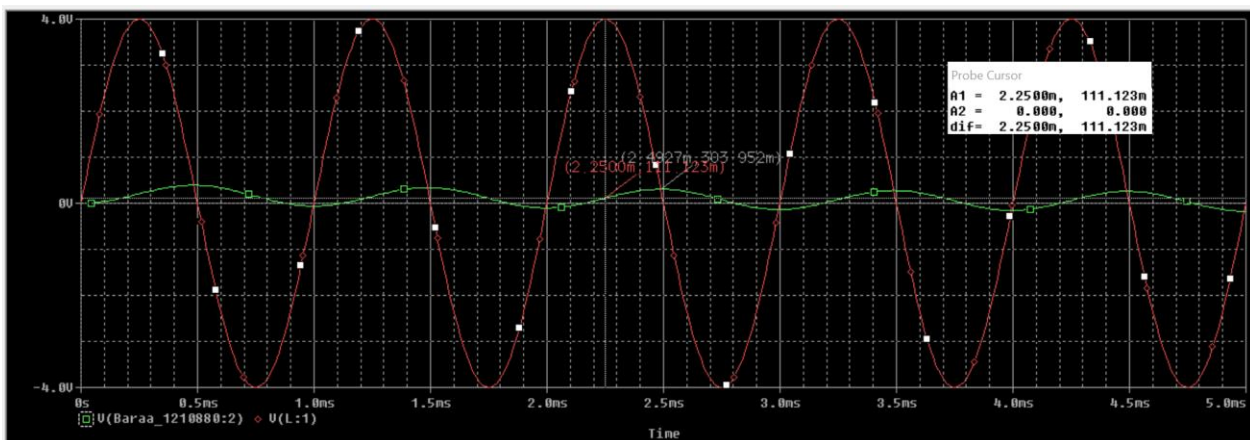
Part three: For the circuit shown in Fig. 3.2:

Part one:





Part two:



Δt= 2.4927- 2.2500=0.2427

Δθ=360° x f x Δt= 360° x 1000 x 0.2427 =87m

Part four:

Based on the previous results, we conclude that despite the difference in the frequency value, in the first case it was 100 Hz, in the second case 1000 Hz, and also in the first case it was the capacitor, and in the second case the inductor and the resistance value remained the same in both cases 100 ohms, and despite all that, the value of Δθ almost equal, in the first case it was 85m and in the second 87m.

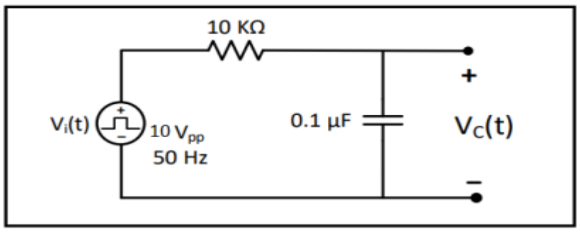
Problem #4:

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:

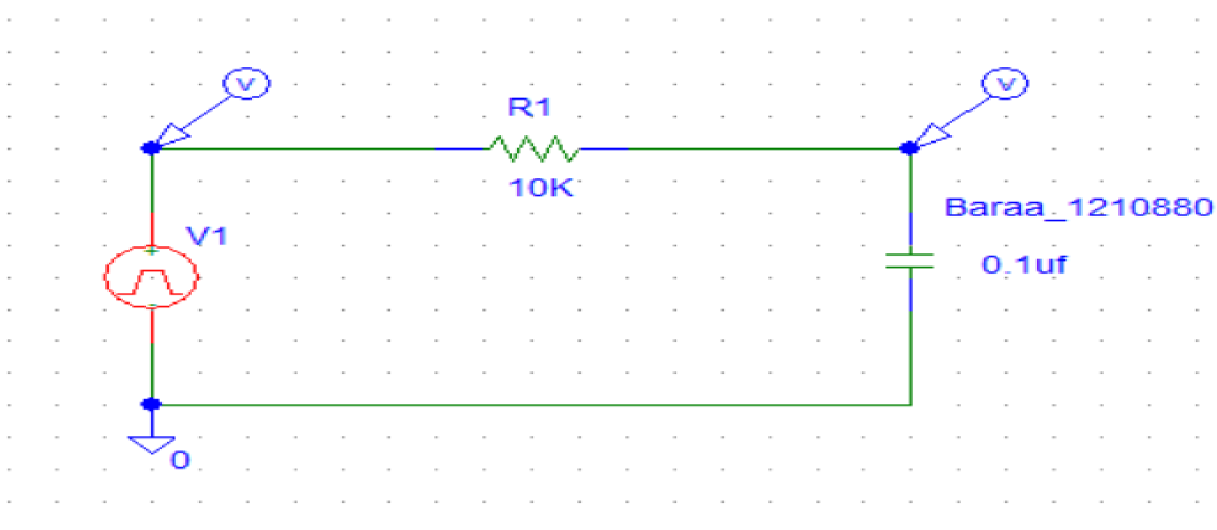


The input voltage is square signal with 10 $V_{peak\text{-}peak}$ (0 V to 10 V) and frequency of 50Hz.

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.

Part one:

The figure shown below plots both the voltage of the voltage source and the voltage across the capacitor for this circuit.



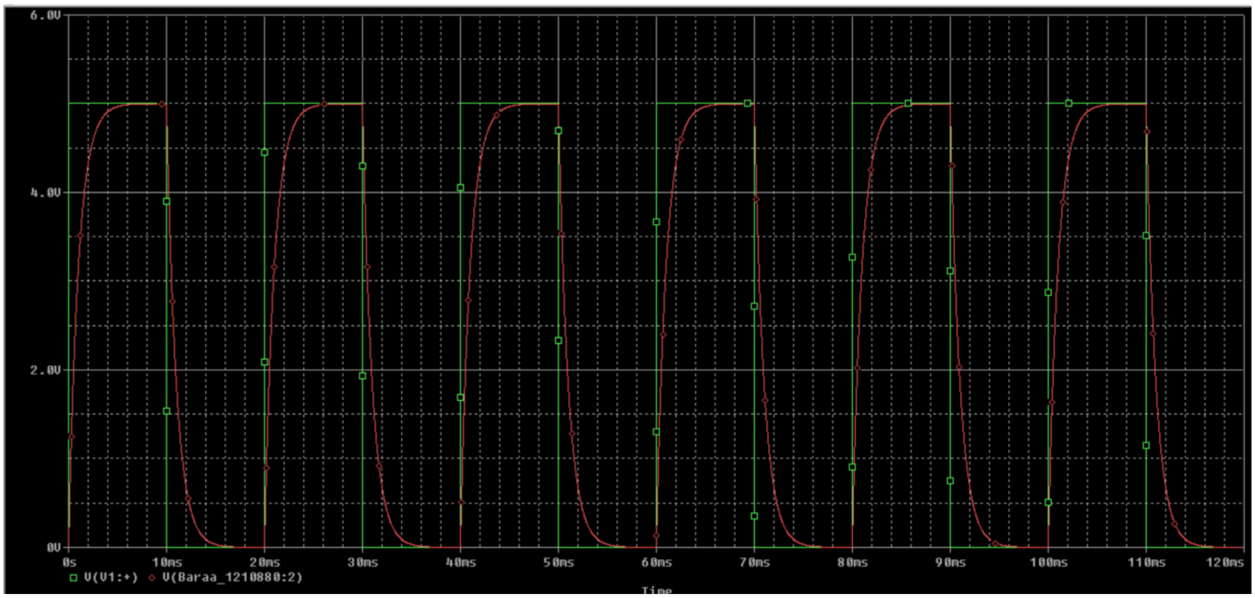
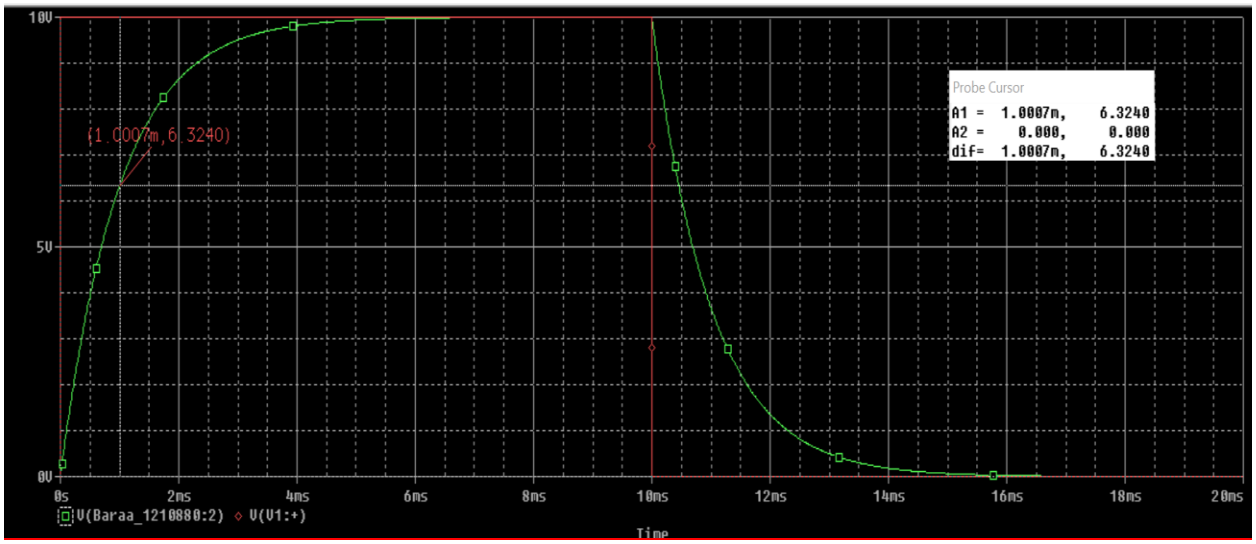


Figure 4.1 (graph of the voltage source in red, and the capacitor in green)

Part two:



τ from simulation = 1.0007s

To find time constant τ , Applying V(τ) = Vp*(1-e^-1) = 6.32V, we get 1.0004 ms. We can also use the natural response in the second half of the perion,

but applying v(τ+t0) =Vp*(e^-1), where t0 = pulse width = 10ms.

To verify this result, τ = RC = 1*(10^4)*1*(10^-7) = 1*10^-3 s = 1ms, which almost the same as the previous one.

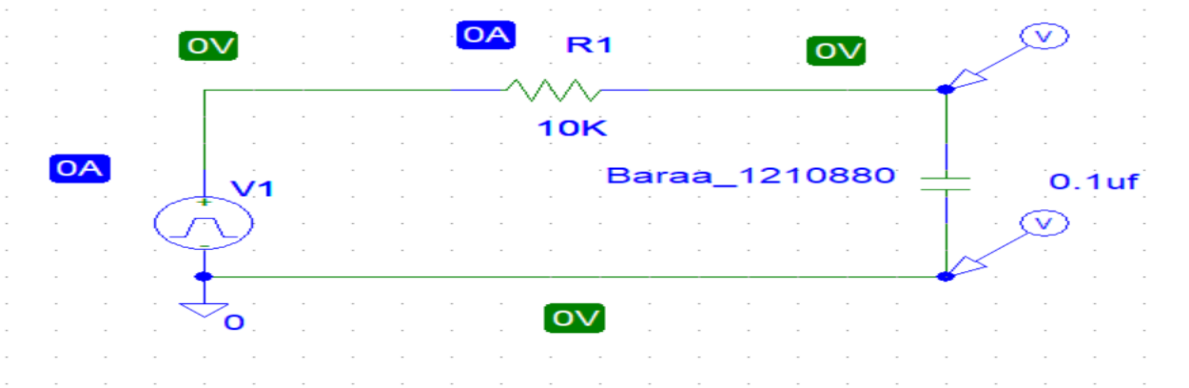


Figure 4.2 shows the simulation of the circuit

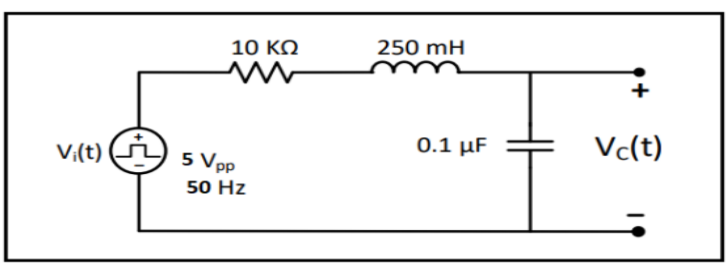
Problem #5:

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.

The figure shown below plots the voltage of the source and the capacitor of the circuit in figure 5.1

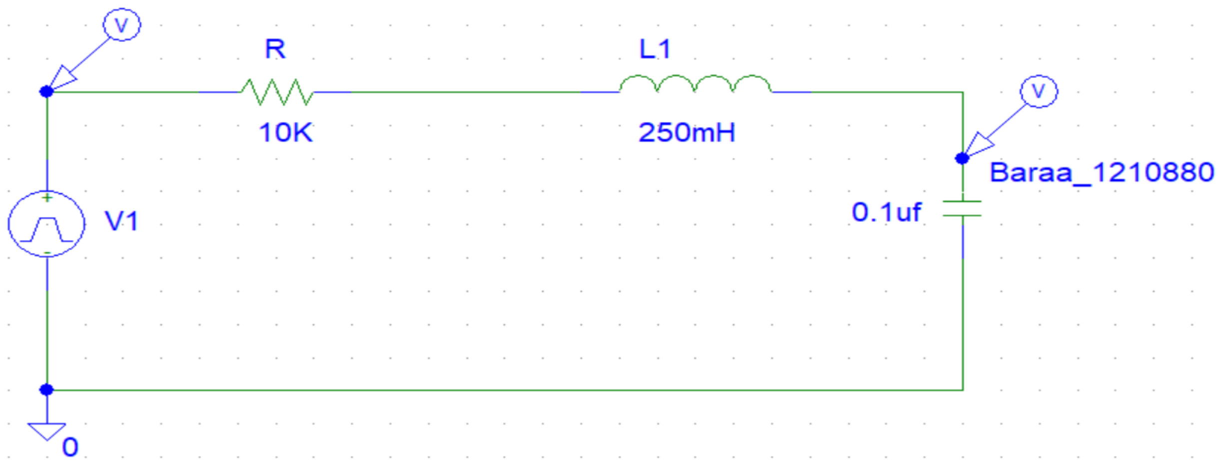


Figure 5.1 shows the circuit of problem 5

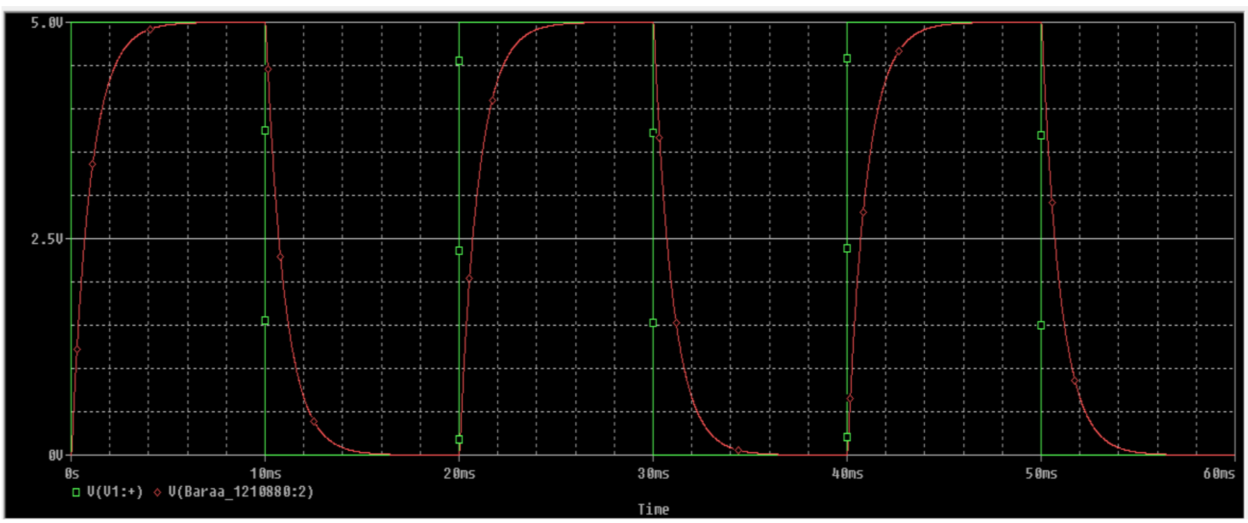
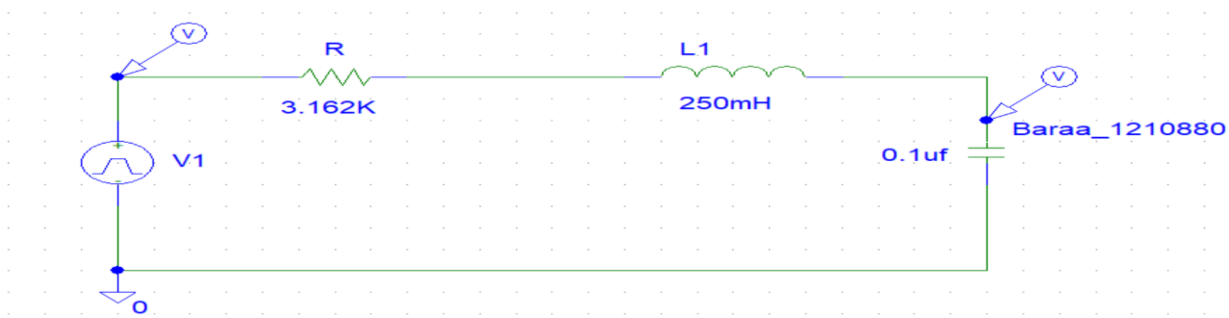


Figure 5.2 shows the graph of the capacitor and voltage source for the circuit of figure 5.1

For the second part of the question, the value of the resistance was changed to $3.162K$, with the others left intact, the graph is shown below



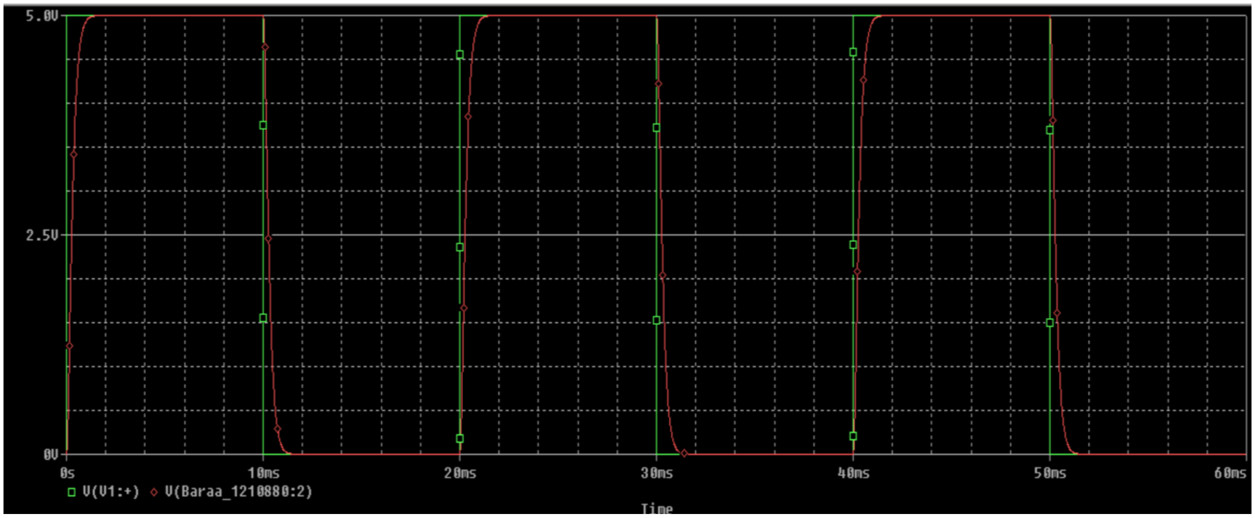
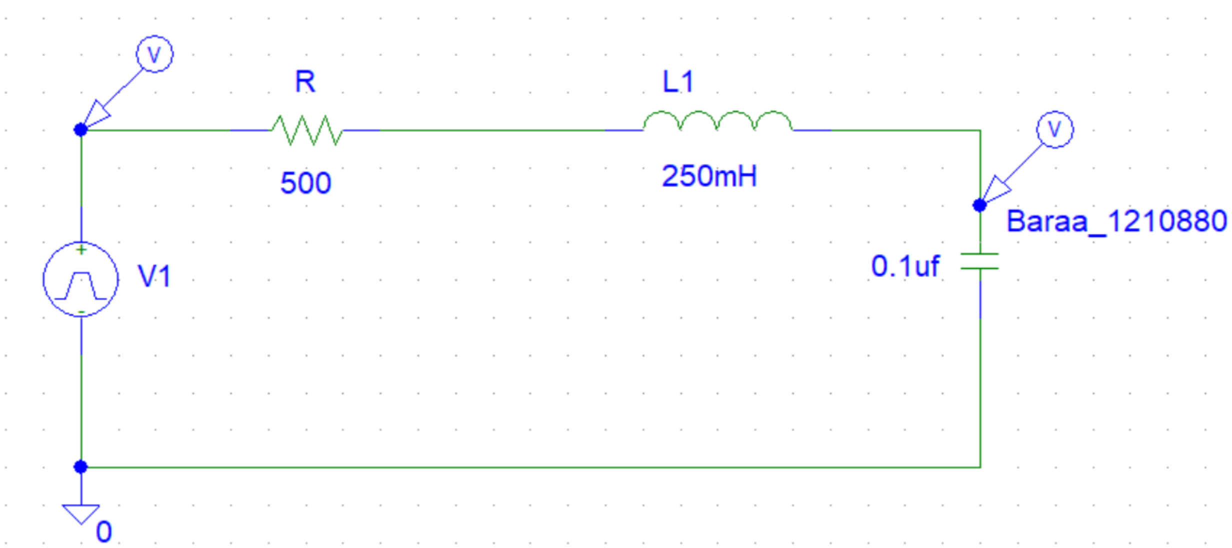


Figure 5.3 shows the plot of the resistance and the voltage source

And finally, the figure of the circuit when R = 500 is shown below :



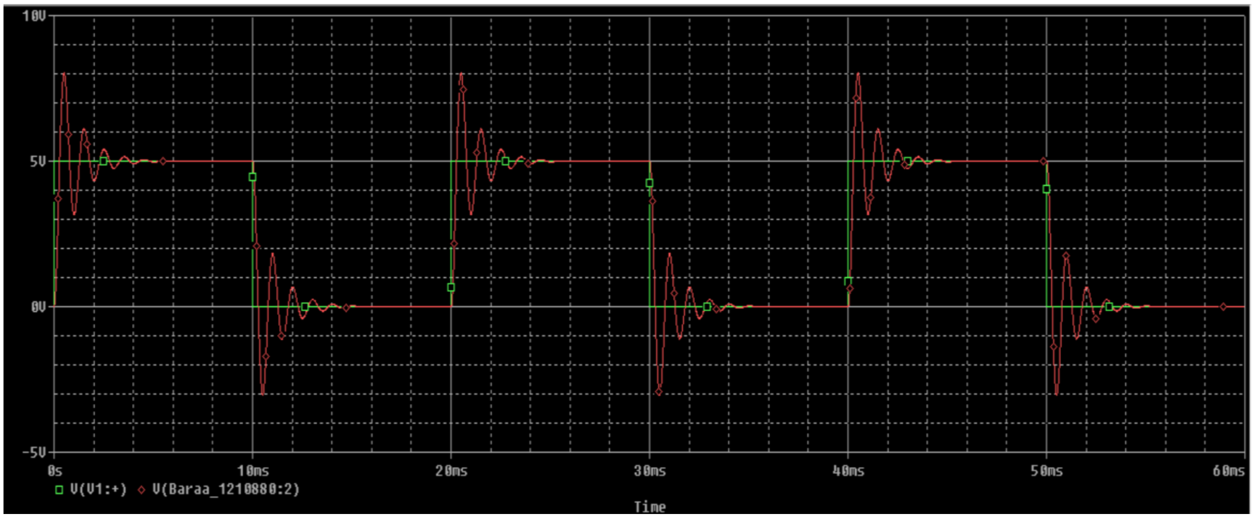


Figure 5.4 shows the same circuit as 1 and 2 with R being 500Ω

These circuits are as following respectively, the first one is overdamped, because the solutions of the second order ODE are real and distinct. In other words, we have $\alpha^2 - \omega_0^2 > 0$.

For the second one, $\alpha \approx \omega_0$, thus we say that this circuit is critically damped, and has real repeated roots. And finally, the last one is underdamped, due to the fact that $\alpha^2 - \omega_0^2 < 0$, hence there are unique complex roots of the second order ODE, as illustrated by the plot of the circuit, you can observe some oscillation due to the presence of sine and cosine in the solution.