Circuit Theory and Electronics Fundamentals

Department of Electrical and Computer Engineering, Técnico, University of Lisbon

Laboratory Assignment T2

MEAer

Group 15

Dar'ya Alekseyevna Tsukanova, nº 95777
Duarte Ferro Lopes, nº 95783
Rafaela Magalhães Chaffilla, nº 95840

April 7^{th}, 2021

Contents

1 Introduction

For this laboratory assignment, our objective was to study a circuit containing 11 components. These include seven resistors, one capacitor, one current source (voltage-controlled), a current-controlled voltage source and an independent sinusoidal voltage source. This independent source has a behavior based on the following equation:

$$V_s(t) = V_s u(-t) + sin(2\pi f t)u(t) \tag{1}$$

We studied the circuit, shown in Figure 1 1, using a structured approach, based on the nodal method, as explained in the theory classes.

To analyze and study the given information we first proceeded to identify the eight nodes present in the circuit. Furthermore, we extracted from the Python script the values for the resistors and the capacitor, as well as the values of the constants related with the independent source and the initial value (linked to t¡0).

Through the report, in section 2, we present a theoretical analysis of the circuit, which is divided in six different subsections, each of them corresponding to a different step given in the guide to solve this problem. In addition, in section 3, we run a simulation of the circuit, using NGSpice, whose results are then compared to the theoretical results obtained in Section 2. Finally, the conclusions of this study are presented in Section 4.

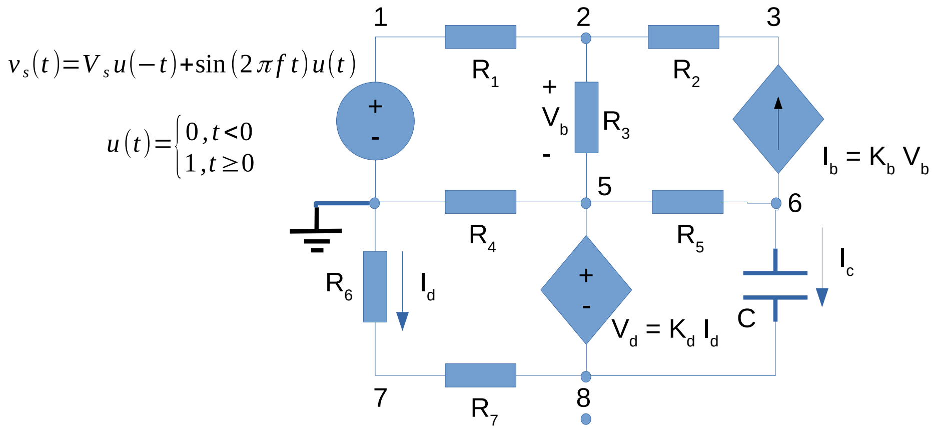


Figure 1: Circuit that will be analyzed in this lab.

2 Theoretical Analysis

In this section, we proceed to the analysis of the circuit shown in Figure 1. Unlike the circuit analyzed in the previous laboratory class, which presented a constant behaviour, the circuit that we shall now examine responds differently in function of time and the frequency of the sinusoidal voltage source. This is due to the presence of the sinusoidal voltage source and the capacitor in the circuit.

In order to determine the requested voltages and the currents in the circuit, no matter what the instant or the applied frequency, we shall follow a method based on six steps, as described in the laboratory guide.

2.1 Step 1: Circuit Analysis for $t < 0$

The first step is to analyze the circuit in the interval $t < 0$. For that, it is requested that we use specifically the Nodal Method, which consists in applying the Kirchhoff Current Law

(KCL) to the nodes that are not connected to voltage sources, using additional equations for nodes related by voltage sources. We must also note that the behaviour of the voltage source is described by the function:

$$V_s(t) = V_s u(-t) + sin(2\pi f t)u(t) \tag{2}$$

, where u(t) is defined as u(t) =

$$\begin{cases} 0, t < 0 \\ 1, t \geq 0 \end{cases} \tag{3}$$

Hence, for $t < 0$, $V_s(t) = V_s$, which is a constant. As the current ($i(t)$) in the capacitor is defined by

$$i_C(t) = C\frac{dv_C}{dt}. \tag{4}$$

, we conclude that $i(t)$ is 0 for $t < 0$, that is, the capacitor behaves like an open circuit in this interval of time.

So, at this point, the circuit can be treated like a simple circuit containing only resistors and a constant voltage source. We can, then, apply the Nodal Method to determine the voltage in all the nodes of the circuit, hence obtaining the following matricial equation:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G_1 & -G_1-G_2-G_3 & G_2 & 0 & G_3 & 0 & 0 & 0 \\ 0 & -K_b-G_2 & G_2 & 0 & K_b & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & K_dG_6 & -1 \\ 0 & -K_b & 0 & 0 & K_b+G_5 & -G_5 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & G_6+G_7 & -G_7 \\ G_1 & -G_1 & 0 & 0 & -G_4 & 0 & G_7 & -G_7 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{5}$$

By solving this equation, we obtained the results expressed in the Table 1

Name	**Value [A or V]**
V_1	5.1479314917
V_2	4.9226299786
V_3	4.4455628818
V_4	0.0000000000
V_5	4.9552367516
V_5	4.9552367516
V_6	5.6912536442
V_7	-2.0583731648
V_8	-3.0705988750
I_a	0.0002242195
I_b	-0.0002346776
I_c	0.0000000000
I_d	0.0009813848

Table 1: Step 1: Solutions given by the Nodal Method. *I* stands for currents, which are expressed in Ampere; *V* represents voltages, which are expressed in Volt.

2.2 Step 2: Circuit Analysis for $t = 0$

The second step aims to determine the value of the equivalent resistance as seen by the capacitor terminals. This is important because, after that, we can simplify the circuit to a simple V-R-C circuit, with just one voltage source, one capacitor and one resistor, whose value is the equivalent resistance. The capacitor, then, discharges through this resistor, with an associated time constant given by RC, where R is the equivalent resistance and C is the capacity of the capacitor.

In order to obtain, the equivalent resistor, we shall analyze the circuit at the instant t = 0. We add that we first need to "switch off" the independent voltage source, that is, put its voltage to zero. We must also add that we must replace the capacitor with a voltage source $V_x = V_6 - V_8$, where V_6 and V_8 are the voltages in the nodes 6 and 8, respectively, as obtained in the previous step. We run, again, the Nodal Analysis, which leads to the following equation:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G_1 & -G_1-G_2-G_3 & G_2 & 0 & G_3 & 0 & 0 & 0 \\ 0 & -K_b-G_2 & G_2 & 0 & K_b & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & K_dG_6 & -1 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 & 0 & 0 & G_6+G_7 & -G_7 \\ G_1 & -G_1 & 0 & 0 & -G_4 & 0 & G_7 & -G_7 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ V_x \\ 0 \\ 0 \end{bmatrix} \tag{6}$$

Solving the equation produces the results presented in Table 2

Name	Value [A, V, Ω or s]
V_1	0.0000000000
V_2	-0.0000000000
V_3	-0.0000000000
V_4	0.0000000000
V_5	0.0000000000
V_6	8.7618525193
V_7	0.0000000000
V_8	-0.0000000000
I_a	0.0000000000
I_b	0.0000000000
I_c	-0.0027936998
I_x	-0.0027936998
I_d	0.0000000000
R_{eq}	3136.2898714100
τ	0.0032673318

Table 2: Step 2: Solutions given by the Nodal Method.

The equivalent resistance is now determined through Ohm's Law: $R_{eq} = \frac{V_x}{I_x}$ Then, the time constant is given by τ = RC

2.3 Step 3: Natural solution for $t > 0$

For $t > 0$, the voltage source changes to a sinusoidal behaviour, defined by $v_s(t)$ = $sin(2\pi ft)$. Consequently, $i_C(t)$ is no more null for $t > 0$, because $\frac{dv_C}{dt} \neq 0$ at this point. Hence, the capacitor begins to discharge through the equivalent resistor, and its behaviour is defined by the following first order Linear Ordinary Differential Equation (LODE), extracted by applying the Kirchhoff Voltage Law to the simplified V-R-C circuit:

$$RC\frac{dv_6}{dt} + v_6(t) = v_s \tag{7}$$

, where $v_6(t)$ is the function that describes the voltage in node 6 as the time flows. The solution for this equation has two components: the natural solution and the forced solution. This third step consists in the determination of the natural solution, which consists in the solution for the homogeneous equation. This is an equation of the type $v_{6n}(t)$ = $Ae^{-\frac{t}{RC}}$. A is a constant that is simply determined by analyzing the value of $v_{6n}(t)$ at the instant $t = 0$, which allows to conclude that A = V_6, where V_6 is the value of the voltage in node 6 determined in the previous step.

In short, $v_{6n}(t) = V_6 e^{-\frac{t}{RC}}$, for $t > 0$, as shown in the next plot:

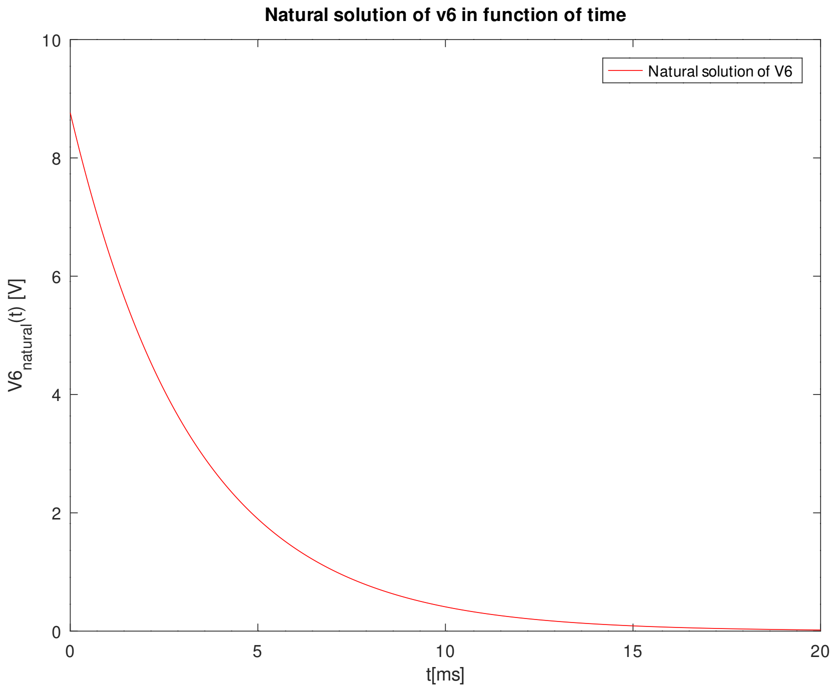


Figure 2: Natural solution for $v_6(t)$

2.4 Step 4: Forced Solution for $t > 0$ and f = 1 KHz

In this fourth step, we shall proceed to the determination of the forced solution from the previously mentioned LODE. We relied on the impedance method to achieve this goal. The application of this method consists in a series of stages, that we shall further describe:

- First, we need to replace the sinusoidal voltage source by its complex amplitude, also known as phasor. As described in the previous subsection, $v_s(t)$ is given by $sin(2\pi ft)$. Having in mind that $e^{i\phi}$ equals $cos(\phi) + isin(\phi)$ (Euler's Equation), we decided to convert $v_s(t)$ from a sine to a cosine, in order to apply the previous relation. Therefore, we rewrote, now, $v_s(t)$ as $cos(2\pi ft)$, which corresponds to the phasor $e^{-i\frac{\pi}{2}}$.
- Then, we shall replace each of the remaining components by their respective impedances. The impedance of each resistor is its resistance, and the impedance of the capacitor is given by $\frac{1}{i\omega C}$, where ω = $2\pi f$ is the radian frequency, which is the same as the source (we note that a sinusoidal excitation implies a sinusoidal forced solution at the same frequency), and C is the capacity of the capacitor.

- Now, it is time to solve the circuit by applying, once more, the Nodal Method (as requested in the laboratory guide).
- Finally, we convert the obtained phasor voltages to real time functions, multiplying the phasor by $e^{i\omega t}$ and extracting the real part (the only one that we are interested on).

This procedure led to the following matricial equation:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G_1 & -G_1-G_2-G_3 & G_2 & 0 & G_3 & 0 & 0 & 0 \\ 0 & -K_b-G_2 & G_2 & 0 & K_b & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & K_dG_6 & -1 \\ 0 & K_b & 0 & 0 & -K_b-G_5 & G_5+i\omega C & 0 & -i\omega C \\ 0 & 0 & 0 & 0 & 0 & 0 & G_6+G_7 & -G_7 \\ G_1 & -G_1 & 0 & 0 & -G_4 & 0 & G_7 & -G_7 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} e^{-i\frac{\pi}{2}} \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{8}$$

The obtained results are presented in Table 3

Name	Value [A or V]
$\tilde{V_1}$	0.0000000000 + -1.0000000000 i
$\tilde{V_2}$	-0.0000000000 + -0.9562345549 i
$\tilde{V_3}$	-0.0000000000 + -0.8635629454 i
$\tilde{V_4}$	0.0000000000 + 0.0000000000 i
$\tilde{V_5}$	0.0000000000 + -0.9625685112 i
$\tilde{V_6}$	-0.0827105335 + 0.5924434542 i
$\tilde{V_7}$	-0.0000000000 + 0.3998447081 i
$\tilde{V_8}$	-0.0000000000 + 0.5964723657 i

Table 3: Step 4: Solutions given by the Impedance Method.

We notice that all the phasor voltages except $\tilde{V_6}$ are in phase with the voltage source. This is due to the fact that the resistors are linear components, and that their impedances matches their resistances.

2.5 Step 5: Total Solution

The fifth step consists on computing the total solution for the stated LODE, by just summing the natural solution to the forced solution, obtained in the previous subsections. However, we first need to convert the phasor $\tilde{V_6}$, obtained in the previous subsection, to a real time function, by applying the last stage described for the Impedance Method. This function is, then, defined as $v_{6f}(t) = |\tilde{V_6}|cos(\omega t - \phi_6)$, where $|\tilde{V_6}|$ is the absolute value of the phasor and ϕ_6 is its phase.

Therefore, $v_{6total}(t) = v_{6n}(t) + v_{6f}(t) = V_6 e^{-\frac{t}{RC}} + |\tilde{V_6}|cos(\omega t - \phi_6)$. The total solution, for the interval [-5, 20] ms, is depicted in the following plot:

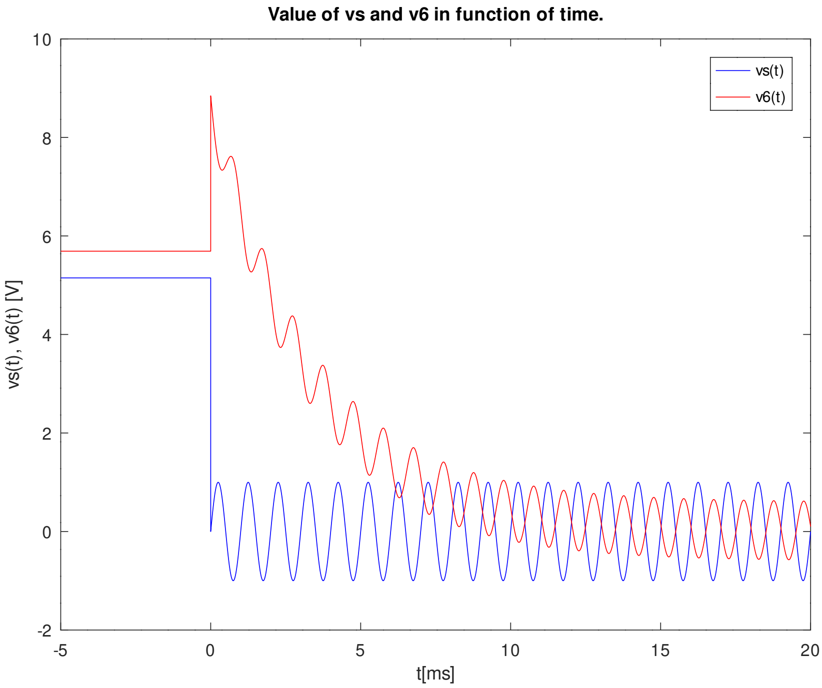


Figure 3: Total solution for $v_6(t)$

2.6 Step 6: Frequency Responses

The sixth and final step of the theoretical analysis changes the focus of our study from time to frequency. We are now examine how the circuit behaves in function of the frequency (f) of the sinusoidal voltage source. To do so, we must reapply the method described in Step 4 to collect the phasor voltages for different values of frequency, spanning 0,1 Hz to 1 MHz. As requested, we extracted the phasor voltages $\tilde{V_6}$, $\tilde{V_8}$ and $\tilde{V_C} = \tilde{V_6} - \tilde{V_8}$, this last one corresponding to the phasor voltage in the capacitor. The magnitudes and phases of the phasors $\tilde{V_s}$, $\tilde{V_6}$ and $\tilde{V_C}$ are presented in the following plots:

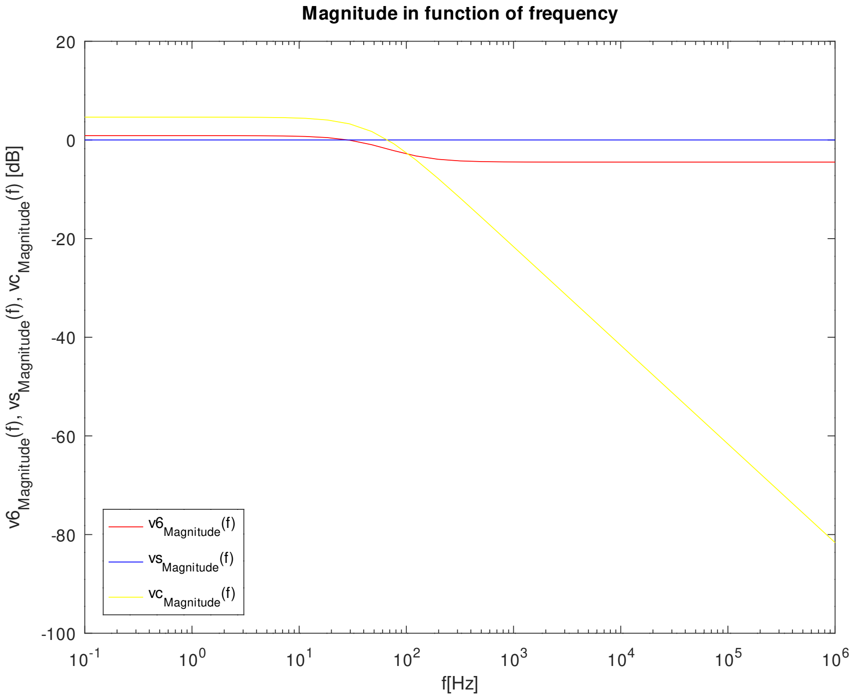


Figure 4: Variation of Magnitude of $\tilde{V_{6s}}$, $\tilde{V_{66}}$ and $\tilde{V_{6C}}$ with frequency

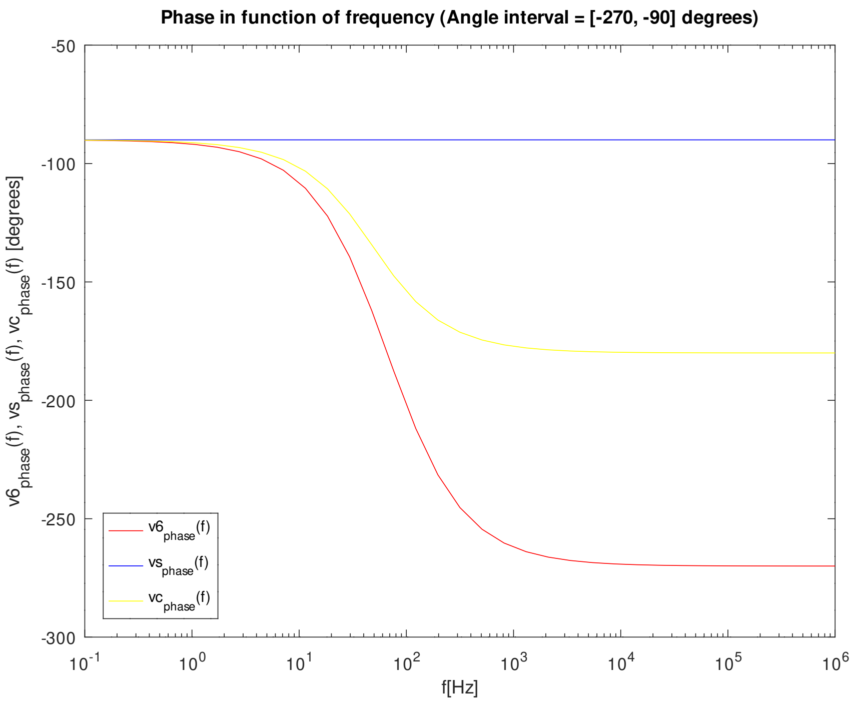


Figure 5: Variation of Phase of $\tilde{V_s}$, $\tilde{V_6}$ and $\tilde{V_C}$ with frequency

Concerning the phase plot, while the phase of the source remains constant and equal to -90 degrees, the phases of the phasors $\tilde{V_6}$ and $\tilde{V_C}$ tend to reduce. When frequency tends to infinity, the phase of $\tilde{V_6}$ converges to -270 degrees. On the other hand, the phase of $\tilde{V_C}$ converges to -180 degrees. The phase of $\tilde{V_8}$ is the same as the phase of $\tilde{V_s}$, as shown in the table provided in Step 4. Therefore, the phase of $\tilde{V_C}$ is the phase of $\tilde{V_6}$ atenuated by 90 degrees (in terms of absolute value). That's why the difference between the phases of $\tilde{V_6}$ and $\tilde{V_C}$ is 90 degrees when frequency tends to infinity.

For low frequencies, $sin(\omega t)$ is very close to 0; therefore, the circuit behaves aproximately like the voltage source had been switched off. The voltage in node 6 corresponds, then, to the natural solution, which is constant in order to the frequency. That's why the magnitude is presented, in the plot, as constant at low frequencies. Changing our focus, now, to the high frequencies, we notice that the magnitude of $\tilde{V_6}$ tends to stabilize. Therefore, the magnitude of $\tilde{V_8}$ must be progressively increasing, given that the magnitude of $\tilde{V_C}$ is negative and is increasing in absolute value.

3 Simulation Analysis

In this section we analyzed the circuits behavior for different instances according to what was asked in the "Simulation" section of the lab instructions.

To obtain the values and use them correctly in NGSpice, the variables calculated by the Python script were read by Octave and afterwards a $.mod$ file was created in order to send the values to the NGSpice's netlist. This way, if the variables given by Python were to change, that modification is ran through octave and sent directly to the NGSpice files.

The comparison of the simulated and theoretical results will be presented in Section 4

3.1 Operating Point Analysis for $t < 0$

The first step to analyze this circuit according to the steps given by the instructions was to analyze the circuit's behavior when $t < 0$, this is, the values of current through the components and voltages in the nodes of the circuit for when $vs = Vs$ but enough time has passed and the voltage through the capacitor is constant (the capacitor is fully loaded or unloaded) and the current flowing through the capacitor is zero.

To run this study we declared all the variables, and as stated above, vs being Vs, without the sinusoidal response being active yet. After that we made a $.op$ analysis that gave us the following results[1 2]:

[1] *@rn[i]* is the current flowing through R_n and *v(n)* is the voltage V_n(voltage in node n)

[2] *fic* represents a fictitious node between R_6 and GND (or V_4 according to the notation used in section 2), created in Ngspice in order to properly define the current dependent voltage source V_c (as required by NGSpice syntax). It is supposed to be in the same node as *GND*; therefore, they have the same voltage value.

Name	NGSpice Value [A or V]
@c1[i]	0.000000e+00
@gib[i]	-2.34678e-04
@r1[i]	2.242195e-04
@r2[i]	-2.34678e-04
@r3[i]	-1.04581e-05
@r4[i]	1.205604e-03
@r5[i]	-2.34678e-04
@r6[i]	9.813848e-04
@r7[i]	9.813848e-04
v(1)	5.147931e+00
v(2)	4.922630e+00
v(3)	4.445563e+00
v(5)	4.955237e+00
v(6)	5.691254e+00
v(7)	-2.05837e+00
v(8)	-3.07060e+00
fic	0.000000e+00

Table 4: Simulated results for $t < 0$. Current in Ampere [A] and Voltage in Volt [V]

3.2 Operating Point Analysis for $v_s(t) = 0$

For the second part of the simulation analysis, we made a study on the circuit's response when there was no independent voltage source. This point is important under the simulations analysis because it will provide us the values of the currents and voltages flowing through every node at point $t = 0$. If we see the function that defines $vs(t)$, we see that $vs(0) = 0$ and we also take into consideration that the voltage flowing through the capacitor has to be constant(The current flowing through the capacitor is given by the expression: $I_{capacitor} = C \cdot \frac{dv_{capacitor}}{dt}$ and in order to be able to differentiate, the voltage $v_{capacitor}$ must be differentiable in order to time).

Introducing a new variable $V_X = V_6 - V_8$ that represents the voltage going through the capacitor, we know that this value in $t = 0$ will have to be equal to the value in $t = 0^-$. Given that the values calculated in point 3.1 were constant, to calculate the difference $V_X(0^-) = V_6(0^-) = V_8(0^-)$ we can use the values obtained for $t < 0$. and with this we will obtain the initial($t = 0$) values for the voltage in each node and the current in each branch. After we discover this value, we replace the capacitor with the voltage source V_X, and with this we can calculate the values we want.

This is a very important step because we will obtain the values of the voltages that the nodes present for t=0. And considering that for $t \geq 0$ the dependent voltage source is continuous, so will be all the rest of the components, therefore the values we get in this part can be used as a boundary condition for the voltages in all the nodes, as we will see in in Sections 3.3 and 3.4.

Applying the alterations stated above, we obtained the following results:

Name	NGSpice Value [A or V]
@gib[i]	0.000000e+00
@r1[i]	0.000000e+00
@r2[i]	0.000000e+00
@r3[i]	0.000000e+00
@r4[i]	0.000000e+00
@r5[i]	-2.79370e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
v(1)	0.000000e+00
v(2)	0.000000e+00
v(3)	0.000000e+00
v(5)	0.000000e+00
v(6)	8.761853e+00
v(7)	0.000000e+00
v(8)	0.000000e+00
fic	0.000000e+00

Table 5: Simulated results for $v_s = 0$. Current in Ampere [A] and Voltage in Volt [V]

We must also note that this process gave us the equivalent resistance R_{eq} (in this case where the only resistance that has current flowing through is R_5, we quickly conclude that $R_{eq} = R_5$) that gives us the time constant $\tau = R_{eq} \cdot C$ that could be used to calculate the function of $V_X(t) = V_X(\infty) + [V_X(0) - V_X(\infty)]e^{-\frac{t}{\tau}}$.

However, for this lab we did not use this generic solution, instead we discovered the natural and forced solutions, as it was already explained in Section 2. But considering that to calculate the natural solution of V_X, we also need τ (Natural solution= $V_X(t) = Ae^{-\frac{t}{\tau}}$), the calculation of R_{eq} was necessary to be able to do the theoretical analisys.

3.3 Natural Solution of $v_6(t)$

The natural solution of a component in a circuit is the response of the element in study (voltage or current) if there was no independent voltage source active. To obtain the natural response of the voltage in node 6 we must ignore the action of vs(t). Therefore we declared Vs as being a constant voltage source of 0V. After that we put back the capacitor we had replaced by a voltage source V_X in the previous section, and finally we used the boundary conditions of v_6 obtained also in the section above. We were able to do this using the *.ic* mode that allowed us to tell NGSpice the values for certain node voltages at instance t=0.

Then we printed the transient analysis in the time interval of $[0;20]ms$ and in the image below we present the graph of the natural response of $v_6(t)$:

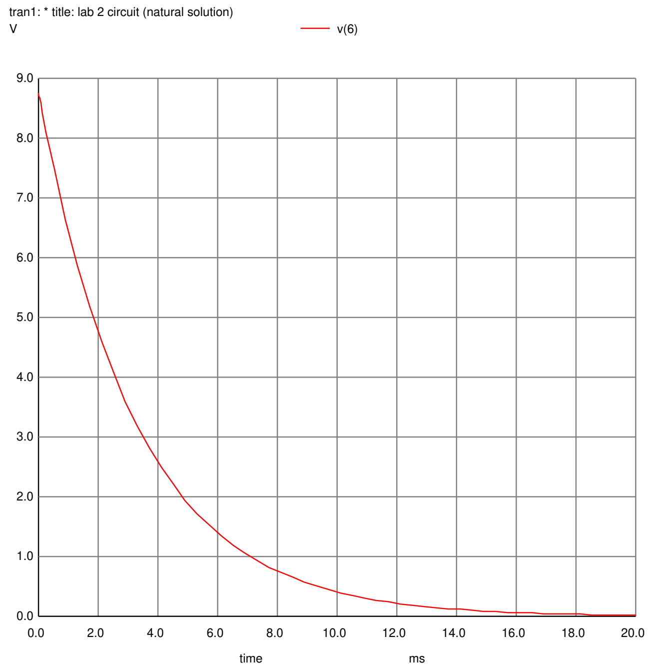


Figure 6: Natural solutions of $v_6(t)$ in Volt [V]

3.4 Total (natural+forced) solution of $v_6(t)$

In this section we finally calculated the total response of the voltage in $v_6(t)$ using once again the *.ic* feature of NGSpice and setting the same boundary conditions as previously stated, declared the voltage source with it's representative function ($v_s(t) = sin(2\pi f)$, for $t \geq 0$). Once we reconsidered the voltage source and the boundary conditions we were able to obtain the total responce in node 6. In Figure 7 we show the result of $v_6(t)$ and and the stimulus voltage $v_s(t)$.

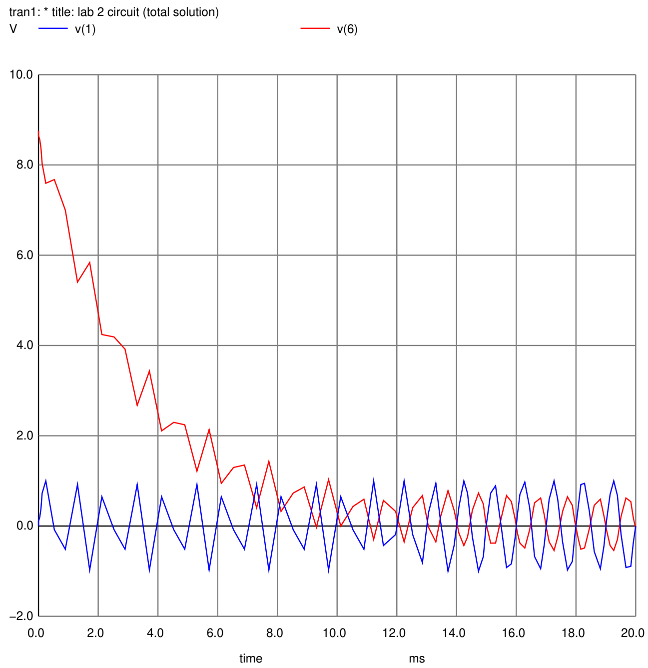


Figure 7: Total solution of $v_6(t)$ (red) and stimulus $v_s(t)$ (blue) in Volt [V]

As we expected, the two graphs are not equal. First of all, $v_6(t)$ presents a sinusoidal shape, but also an exponentially decaying shape. This happens because node 6 is associated to the capacitor, that isn't a linear component. Instead, the capacitor's associated voltage presents an exponential behavior. Given by the Superposition Theorem, every independent voltage or current source has a contribution to every voltage and current values of the circuit and if we study the circuit without the voltage source (natural response-exponential shape) and then just analyzed the circuit without the boundary conditions (forced solution-sinusoidal shape), once again according to the Superposition Theorem, the final result should be the sum of the two. And this sum will present both an exponential and sinusoidal shape such as the one seen in the figure above for $v_6(t)$. We also see that at the end, after some time has passed, the capacitor unloads, and it stops having an effect on the circuit, has it will only present a sinusoidal shape, because it is affected by the source $v_s(t)$. Additionally, we note that the the two graphs have different phases, therefore their crests and troughs are not aligned, besides having the same period and frequency. This happens, once again, because the capacitor is not a linear component, therefore it will have an impedance $Z_c = \frac{1}{i \cdot \omega \cdot C}$. This will have an effect on the phase of the voltages around the capacitor, as we can see by comparing the two solutions.

3.5 Frequency analysis

Lastly, we analyzed the variation of the phasors of voltages $v_s(t)$ and $v_6(t)$ in function of the frequency. For this we, once more, declared the values in NGSpice, and ran an *ac* analysis in the programm. The graph we obtained for the voltage's Magnitudes in function of the frequency is the following:

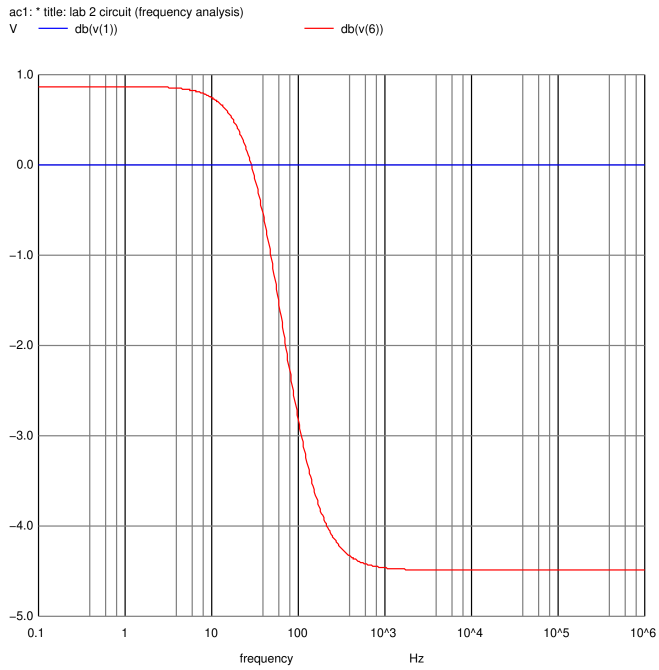


Figure 8: Graphs of the Magnitude [dB] of $v_6(f)$ (red) and $v_s(f)$ (blue).

First of all we see that the Magnitude of v_s is always 0 dB which is expected once that $v_s(t)$ is the independent voltage source and is given by $v_s(t) = 1 \cdot sin(2\pi f \cdot t)$, therefore the Magnitude does not depend on the frequency, and it should be always 1V=0dB[1]

However the same does not apply to the Magnitude of v_6. As we know and calculated in Section 2, the phasors of the voltages on the nodes are dependent on linear components such as resistors and dependent linear sources. But we also have a capacitor with an impedance $Z_c = \frac{1}{i \cdot \omega \cdot C}$. Since this is the only component that depends on the frequency value[2], the capacitor will be responsible for giving a non linear behavior to the phasors surrounding it.

Analyzing the curve for $v_6(f)$ we see that for frequencies between the values of $[0,1;10]Hz$ and $[10^4;10^6]Hz$ the Magnitude doesn't show significant change, and we could even say it stays constant, approximately. On the other hand, for an interval of $[10;10^3]Hz$ there is a rapid decrease of the value of Magnitude in dB. This has already been mentioned in subsection 2.6.

To see the influence of the frequency in the voltage's phase, we get the following graph[3]:

[1] $0dB = 20 \cdot log_{10}(1V)$
[2] $\omega = 2\pi f$
[3] The values of the NGSpice graph are shown in an interval of [-180;0] Degrees.

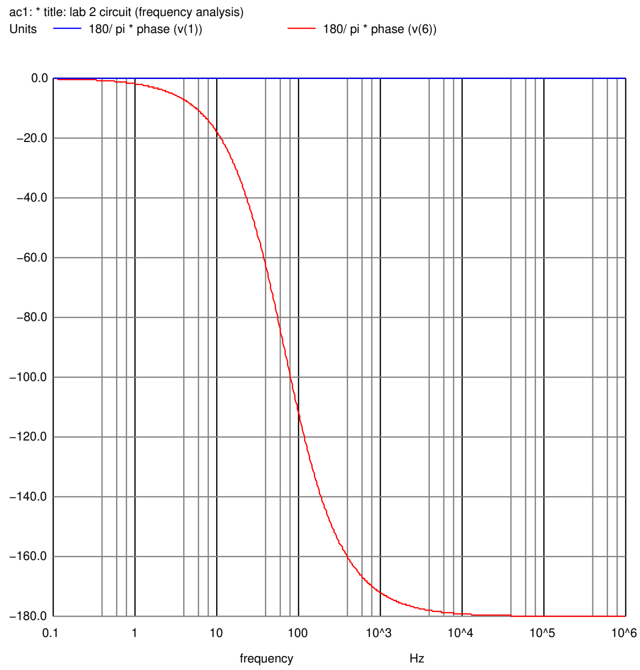
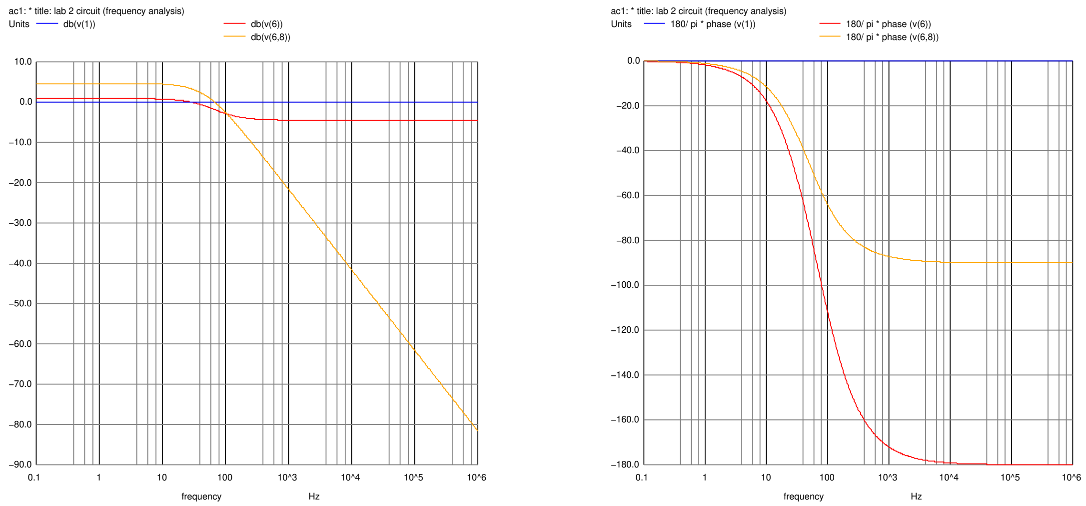


Figure 9: Graphs of the Phase of $v_6(f)$ (red) and $v_s(f)$ (blue) in Degrees

Once again, we see that the phase for v_s does not change according to the frequency. As we saw previously, this is as expected given that the frequency does not have any influence on the independent voltage source, therefore the phase remains equal to zero, considering that is it's inicial phase.

Differently, the phase of v_6 does show a dependency on the frequency, showing a decaying curve with the increase of frequency, but only until a certain point. Afterwards it seems to stabilize again. Once again this has already been justified above in subsection 2.6

In addition to the figures above we also printed the Magnitude and phase of $v_c(f)$, defined by the instructions as the voltage flowing through the capacitor, given by $v_c(f) = v_6(f) - v_8(f)$, in the same graph, so that we could later compare it to the theorical results. The graphs are as follows:



(a) Magnitude (dB)

(b) Phase (Degrees)

Figure 10: Frequency analysis on v_6 (red), v_s (blue), v_c (yellow)

4 Conclusion

To start this conclusion we will present all the tabled results we already shown in sections 2 and 3 side by side to compare them.[4]

Starting of with the study for $t < 0$, we show again the following results.

[4]By analysis of the circuit in figure 1 we know that I_a is *@r1[i]*, I_b is *@gib[i]*, I_c is *@ci[i]* and I_d is *@r6[i]*.

Name	Theoretical Value [A or V]
V_1	5.1479314917
V_2	4.9226299786
V_3	4.4455628818
V_4	0.0000000000
V_5	4.9552367516
V_5	4.9552367516
V_6	5.6912536442
V_7	-2.0583731648
V_8	-3.0705988750
I_a	0.0002242195
I_b	-0.0002346776
I_c	0.0000000000
I_d	0.0009813848

(a) Theoretical

Name	NGSpice Value [A or V]
@c1[i]	0.000000e+00
@gib[i]	-2.34678e-04
@r1[i]	2.242195e-04
@r2[i]	-2.34678e-04
@r3[i]	-1.04581e-05
@r4[i]	1.205604e-03
@r5[i]	-2.34678e-04
@r6[i]	9.813848e-04
@r7[i]	9.813848e-04
v(1)	5.147931e+00
v(2)	4.922630e+00
v(3)	4.445563e+00
v(5)	4.955237e+00
v(6)	5.691254e+00
v(7)	-2.05837e+00
v(8)	-3.07060e+00
fic	0.000000e+00

(b) Simulated

Table 6: Operating point analysis for $t < 0$

With this side by side comparison, we are assured that the results obtained by the theoretical calculations and the simulation computation are exactly the same, as expected given that this is an operating point analysis for constant and linear components, except for the capacitor, that has no effect in the circuit at the time we are doing this analysis.

For the second topic of the analysis of $v_s(t)$ these are the tables we obtained:

Name	Theoretical Value [A or V]
V_1	0.0000000000
V_2	-0.0000000000
V_3	-0.0000000000
V_4	0.0000000000
V_5	0.0000000000
V_6	8.7618525193
V_7	0.0000000000
V_8	-0.0000000000
I_a	0.0000000000
I_b	0.0000000000
I_c	-0.0027936998
I_x	-0.0027936998
I_d	0.0000000000
R_{eq}	3136.2898714100
τ	0.0032673318

(a) Theoretical

Name	NGSpice Value [A or V]
@gib[i]	0.000000e+00
@r1[i]	0.000000e+00
@r2[i]	0.000000e+00
@r3[i]	0.000000e+00
@r4[i]	0.000000e+00
@r5[i]	-2.79370e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
v(1)	0.000000e+00
v(2)	0.000000e+00
v(3)	0.000000e+00
v(5)	0.000000e+00
v(6)	8.761853e+00
v(7)	0.000000e+00
v(8)	0.000000e+00
fic	0.000000e+00

(b) Simulated

Table 7: Operating point analysis for $v_s = 0$

Once again the results we obtained were equal, as we assumed it should be, because with this being a linear component circuit under the conditions $V_s = 0$ and the capacitor being replaced with a linear voltage source, the theoretical and simulated results should be equal if the nodal method was used correctly.

Addressing the graphs we obtained, both the theoretical and simulated plots were alike for the same time intervals. The only exception we must point out is in figures 5 and 10b, where the phases of the phasors in function of frequency have the same shape but are delayed by 90 degrees. This is because, as explained in section 2, we defined the $v_s(t)$ function using cosine. To do this, we had to subtract 90 degrees[5] to the original phase of v_s, that gave us a new phase of -90 degrees, seen in figure 5. In addition, to calculate the phases with the use of the *atan* function of Octave, we defined an angle interval of $[-270; -90]$ degrees. This two modifications caused a subtraction of 180 degrees to all the phases of the circuit, for the theoretical analysis. However given that NGSpice did not recognize any cosine command, we still used a sine function to define $v_s(t)$ in the simulation section. Given this, the interval of the phases shown in Section 2 has a difference of -90 degrees.

This laboratory assignment was very important, as it was one of our first contacts with the solving of circuits containing alternate sources and non linear components, as well as to improve our skills on using NGSpice and LaTeX.

[5] $sin(\alpha) = cos(\alpha - 90)$, α in degrees