Circuit Theory and Electronics Fundamentals

Aerospace Engineering, Técnico, University of Lisbon

2^{nd} Laboratory Report

João Martins 95806
Gonçalo Martins 95793
António Lopes 95771

March 25, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a RC circuit containing 7 resistors ($R1toR7$), 1 sinusoidal voltage source (V_s), two dependent sources (one current (G_b) and one voltage (H_c)) and a capacitor (C_a) by determining how the voltages varie with the frequencie and time. In order to do that, natural and forced responses of the nodes are going to be needed. The scheme of the circuit we're going to analyse and the initial given data can be seen in the figure and table below.

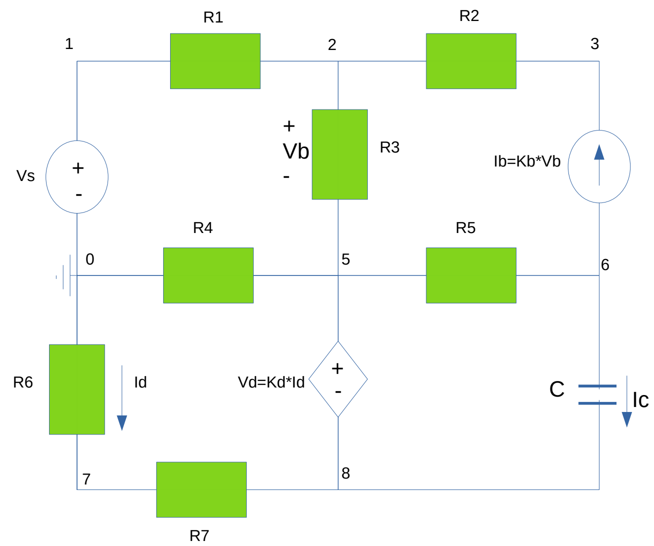

Figure 1: Picture of the circuit in analysis

The sinusoidal voltage source can be defined as

$$v_s(t) = V_{su}(-t) + sen(2\pi ft)u(t) \tag{1}$$

and,

$$u(t) = \begin{cases} 0, & t < 0; \\ 1, & t \geq 0. \end{cases} \tag{2}$$

On the first hand, in section 2, a theoretical analysis of the circuit is presented. Firstly, the circuit is resolver for $t < 0$, then R_eq is calculated by replacing the capacitor with a voltage source $V_x = V(6) - V(8)$. With it, the natural and forced responses of the circuit are calculated

R1	1.006765e+03 Ohm
R2	2.033032e+03 Ohm
R3	3.033913e+03 Ohm
R4	4.003128e+03 Ohm
R5	3.131011e+03 Ohm
R6	2.093899e+03 Ohm
R7	1.017744e+03 Ohm
Vs	5.223200e+00 V
C	1.035361e-06 F
Kb	7.272764e-03 S
Kd	8.170065e+03 Ohm

Table 1: Initial data given by the Python script. Units in kOhm, V, F, S and A

and then superimposed to obtain the final solution. Finally, the frequency response is also studied.

On the other hand, in section 3 we'll simulate the same circuit in order to confirm the results seen in the previous section 2. For that similar analysis are made: for $t < 0$, natural and forced responses and frequencies responses.

Through out the report, a comparison of the results obtain in the section 3 and the section 2 is made, reporting the relative errors encountered.

The conclusions of this study are outlined in section 4.

2 Theoretical Analysis

In this section, the circuit shown in Figure 2 is analysed theoretically, considering equations 1 and 2

and

R1	1.006765e+03 Ohm
R2	2.033032e+03 Ohm
R3	3.033913e+03 Ohm
R4	4.003128e+03 Ohm
R5	3.131011e+03 Ohm
R6	2.093899e+03 Ohm
R7	1.017744e+03 Ohm
Vs	5.223200e+00 V
C	1.035361e-06 F
Kb	7.272764e-03 S
Kd	8.170065e+03 Ohm

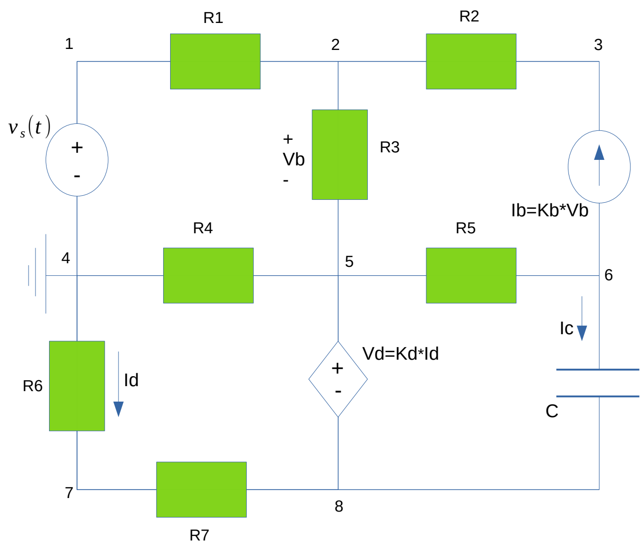

Figure 2: Circuit analysed during the lab assignment

2.1 First point

In the first point, the goal is to perform node analysis for $t < 0$, to determine all node voltages.

In order to do this we did KCL in each node, considering that all currents were diverging from it, which lead us to the following matrix equation

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G1 & G1+G2+G3 & -G2 & 0 & -G3 & 0 & 0 & 0 \\ 0 & -G2-Kb & G2 & 0 & Kb & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & Kd \cdot G6 & -1 & 0 & -Kd \cdot G6 & 1 \\ 0 & Kb & 0 & 0 & -Kb-G5 & G5 & 0 & 0 \\ 0 & 0 & 0 & -G6 & 0 & 0 & G6+G7 & -G7 \\ 0 & -G3 & 0 & -G4-G6 & G3+G4+G5 & -G5 & G6 & 0 \end{bmatrix} \begin{bmatrix} V1 \\ V2 \\ V3 \\ V4 \\ V5 \\ V6 \\ V7 \\ V8 \end{bmatrix} = \begin{bmatrix} Vs \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{3}$$

Solving equation 3, we get the following result

V1	5.223200e+00 V
V2	4.960945e+00 V
V3	4.406214e+00 V
V4	0.000000e+00 V
V5	4.998463e+00 V
V6	5.852787e+00 V
V7	-2.069080e+00 V
V8	-3.074760e+00 V

Knowing all node voltages, we can determine Ib, then use Ohm's law in resistors R1 and R6, and finally we can use mesh analysis to determine all branch currents.

Considering the current directions shown in figure 3, we get the following result

I1	-2.604926e-04 A
I2	-2.728588e-04 A
I3	1.236617e-05 A
I4	-1.248639e-03 A
I5	-2.728588e-04 A
I6	-9.881466e-04 A
I7	9.881466e-04 A
Is	-2.604926e-04 A
Ic	0.000000e+00 A
Id	9.881466e-04 A

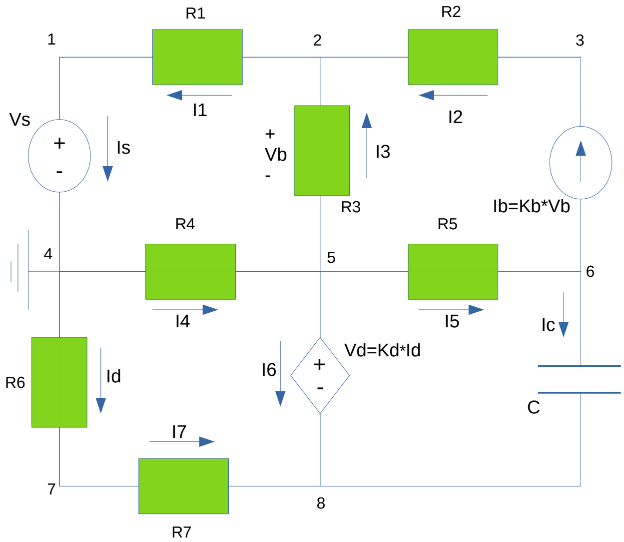

Figure 3: Current directions considered to determine all branch currents

2.2 Second Point

Now, in the second point the objective is to find the equivalent resistor seen by the capacitor, so that in the following theoretical points, we can do them more easily.

To find the equivalent resistor we'll use a very used method when our circuit is complex and has dependent sources. The method is based in Thévenin's theorem, so we must first turn off all independent sources and replace the capacitor with an independent voltage source to determine the response in the current that will flow through the voltage source. We know that this current is the same that passes through our equivalent resistor and the voltage drop will be the same as the voltage source because we're imagining a circuit containing only the voltage source and the equivalent resistor.

So, to start doing this analysis we turn off all independent sources and replace the capacitor with a voltage source, as said. This voltage source will have the value of $V_x = V(6) - V(8)$ where $V(6)$ and $V(8)$ are the voltages in node 6 and 8 found in the first point. The circuit will be as follows

Nodal analysis will be done in this circuit where we get the following matrix. Note that in all nodes, we considered divergent currents.

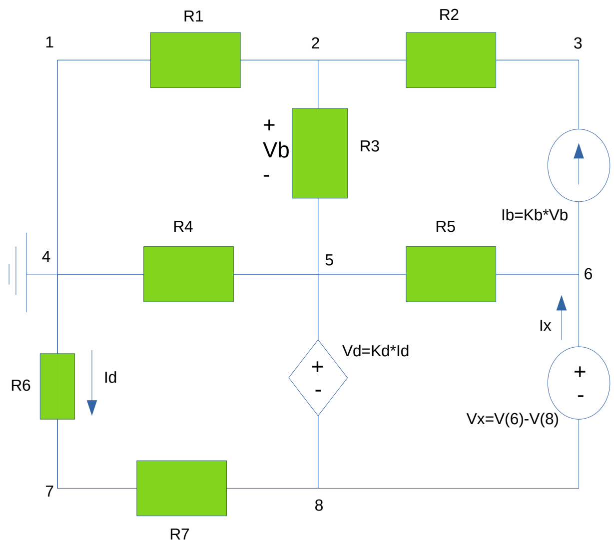

Figure 4: Circuit used in 2) to determine R_{eq}

$$\begin{bmatrix} 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ -G1 & G1+G2+G3 & -G2 & 0 & -G3 & 0 & 0 & 0 \\ 0 & -G2-Kb & G2 & 0 & Kb & 0 & 0 & 0 \\ G1+G4+G6 & -G1 & 0 & 0 & -G4 & 0 & -G6 & 0 \\ 1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & -G6 & 0 & 0 & G7+G6 & -G7 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} V_0 \\ V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ V(6) \\ 0 \\ V(8) \end{bmatrix}$$

By solving this matrix we get all node voltages and then by analysing node 6 doing KCL,

$$I_x = Kb(V_2 - V_5) - G5(V_5 - V_6) \tag{4}$$

we get I_x, and now we can easily get R_eq using Ohm's law,

$$R_{eq} = \frac{V_x}{I_x} \tag{5}$$

There's still another part of the question, finding the time constant. Since we now have a simple RC circuit the time constant τ will be $\tau = R_{eq}C$.

The following tables shows all the computed results.

V1	0.000000e+00 V
V2	8.184440e-01 V
V3	2.549645e+00 V
V4	0.000000e+00 V
V5	7.013583e-01 V
V6	5.852787e+00 V
V7	-2.069080e+00 V
V8	-3.074760e+00 V

Vx	8.927546e+00 V
Ix	2.496829e-03 A
Req	3.575554e+03 Ohm
tau	3.701990e-03 s

2.3 Third Point

For the third point, we want to determine the natural solution of the sistem and for that we'll utilise the formula given by the professor in the theoretical classes instead of resolving the differential equation manualy. The formula is

$$v(t) = v(\infty) + [v(0) - v(\infty)]e^{-\frac{t}{\tau}} \tag{6}$$

applying to the natural solution of v_6 in our circuit, we must find out all the terms of the equation 6: $\tau = R_{eq}C$ where R_{eq} is the value determined in the second point, the initial condition $v_{6n}(0)$ is equal to V_x and for $v_{6n}(\inf)$ we must know how a capacitor works, in $t < 0$ we have a voltage source Vs turned on so our capacitor is receiving charges and charging, after $t = 0$ we turn off the voltage source, since we are studying the natural solution, the capacitor discharges and in $t = \inf$ it discharges completely and since there is only dependent sources the voltage in node 6 becames zero. Knowing that, we can now write our equation for $v_{6n}(t)$

$$v_{6n}(t) = V_x e^{-\frac{t}{\tau}} \tag{7}$$

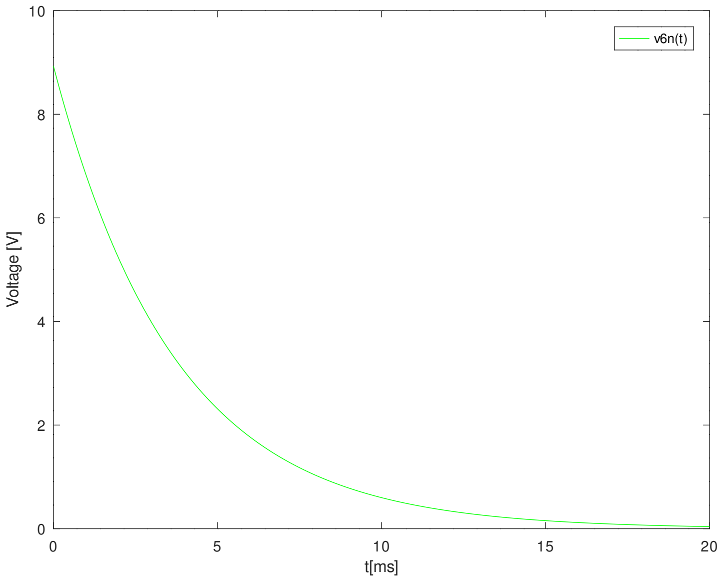

Figure 5: Natural solution $v_{6n}(t)$, $t \in [0, 20]$ms

2.4 Fourth Point

Now in this point we'll analyse the responce of v_6 to a sinusoidal signal with a frequency of 1Hz. To do this we use complex notation, because it makes it easier to calculate the amplitude and phase of its response signal. To transfer to the complex notation we simply use

$$V\cos(\omega t + \varphi) = \Re(e^{\omega t - \varphi}). \tag{8}$$

Knowing that the frequency response is equal to the impulse frequency and since we now are working with the complex notation shown in 8 when we do the node analysis in each equation we can divide both terms by $e^{\omega t}$ since it is always greater than zero and in each equation we get what we call "phasors" ($\widetilde{V} = Ve^{-\varphi}$) which is the complex amplitude associated to each node that contains the information of the amplitude and phase. Another thing to take in consideration is that since we are now working phasers we will also work with impedance

$$Z = \frac{\widetilde{V}}{\widetilde{I}}. \tag{9}$$

In a resistor the impedance is $Z_R = R$, and with a capacitor the impedance is $Z_C = \frac{1}{i\omega C}$, and knowing that we can now do the node analysis to the circuit and determine a matrix that contains all equations from this analysis and determine all node phasors. The matrix is the following,

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G1 & G1+G2+G3 & -G2 & 0 & -G3 & 0 & 0 & 0 \\ 0 & -G2-Kb & G2 & 0 & Kb & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -KdG6 & -1 & 0 & -KdG6 & 1 \\ 0 & Kb & 0 & 0 & -Kb-G5 & G5+i\omega C & 0 & -i\omega C \\ 0 & 0 & 0 & -G6 & 0 & 0 & G7+G6 & -G7 \\ 0 & -G3 & 0 & -G4 & G3+G4+G5 & -G5-i\omega C & -G7 & G7+i\omega C \end{bmatrix} \begin{bmatrix} \widetilde{V_0} \\ \widetilde{V_1} \\ \widetilde{V_2} \\ \widetilde{V_3} \\ \widetilde{V_4} \\ \widetilde{V_5} \\ \widetilde{V_6} \\ \widetilde{V_7} \end{bmatrix} = \begin{bmatrix} \widetilde{V_s} \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

Considering $\widetilde{V_s}$ as 1 like it's said we resolve this matrix, and since it has complex numbers in it we'll get also complex numbers for the phasors in all nodes like we want. We can now determine its magnitude and its argument for each phasor, the following table shows exactly that,

Amplitude1	1.000000e+00 V	Fase1	0.000000e+00 graus
Amplitude2	9.497904e-01 V	Fase2	3.872773e-14 graus
Amplitude3	8.435852e-01 V	Fase3	1.942986e-13 graus
Amplitude4	0.000000e+00 V	Fase4	0.000000e+00 graus
Amplitude5	9.569733e-01 V	Fase5	2.945274e-14 graus
Amplitude6	5.905273e-01 V	Fase6	-1.718503e+02 graus
Amplitude7	3.961326e-01 V	Fase7	-1.800000e+02 graus
Amplitude8	5.886736e-01 V	Fase8	-1.800000e+02 graus

2.5 Fifth Point

Since we know from 2.4 the response amplitude and phase for all the nodes we can calculate the force solution $v_{6f}(t)$. Calling A_6 and φ_6 to the amplitude and phase of the forced response of node 6, the forced solution will be

$$v_{6f}(t) = A_6 \sin(\omega t + \varphi_6) \tag{10}$$

where $\omega = 2\pi f$ and $f = 1Hz$. Now using the superposition theorem we get the final total solution,

$$v_6(t) = v_{6n}(t) + v_{6f}(t) \Rightarrow v_6(t) = A_6 \sin(\omega t + \varphi_6) + V_x e^{-\frac{t}{R_{eq}C}} \tag{11}$$

where all the values are the ones specified in the previous points. The following plot shows v_6 and vs as functions of time.

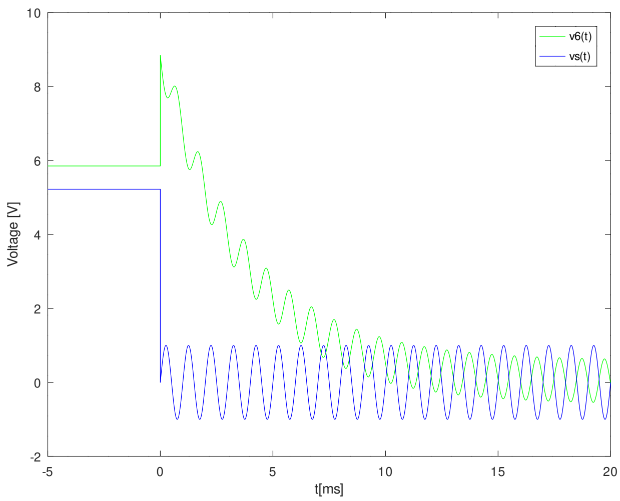

Figure 6: Solução total $v_6(t)$, $t \in [-5, 20]$ ms

2.6 Sixth Point

In this final point we want to analyse the frequency response of the magnitude and phase of $vs(f)$, $vc(f)$ and in node 6 $(v_6(f))$ for a frequency range 0.1Hz to 1MHz. This will be a very important result to have because it gives us caracteristics of power consumptions.

Normaly the plots are made in a logaritmic scale of frequency because it gives us a big density of points in areas where we want more details, also in the plot, the magnitude will be in decibels (dB) ($X_{dB} = 20\log_{10}(X)$). Note that $vc(f)$ is defined as $v_6(f) - v_8(f)$.

The following graphics show the resulting plots.

For vs, we can see that the magnitude and phase are equal to zero for all the frequency values since we are considering the phasor $vs = 1$, so the magnitude is one and the phase is zero, and since $\log_{10}(1) = 0$, that explains the value for vs. On the other hand for vc and $v6$, for the magnitude we see it decreasing when f increases, as expected since its a RC circuit and the capacitor impedance is $Z = \frac{1}{i\omega C}$. For the phase we see it go to negative values as expected.

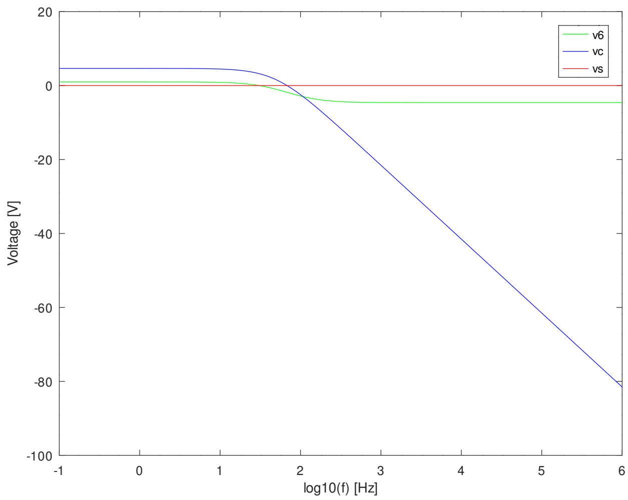

Figure 7: Magnitudes as functions of frequency, $f \in [0.1, 10^6]$ Hz

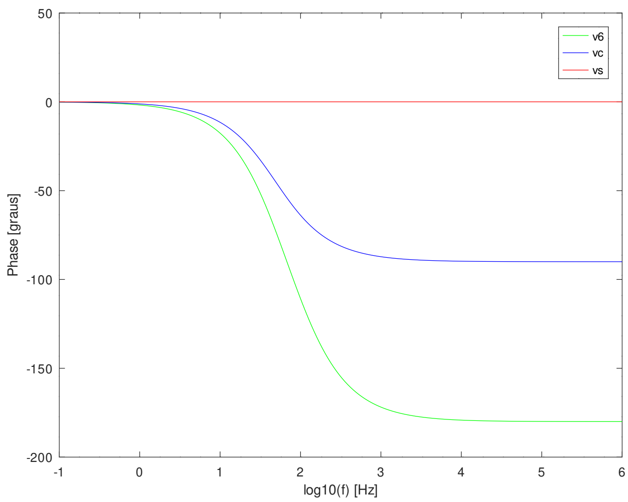

Figure 8: Phases as functions of frequency, $f \in [0.1, 10^6]$ Hz

3 Simulation

3.1 First point

With the circuit solved using the mesh and node methods, it's necesssary to solve it experimentally. In order to simulate the real conditions, which we would have encountered in the laboratory, Ngspice was used. The results obtained from this simulation will take in account the energy dissipation due to the many processes that occur such as Joule effect on conductors. Using this software, we were able to verify the results obtained from the methods already described.

In order to describe the circuit to be studied, it is necessary to follow some guidelines. Firstly, it was needed to specify , for each component, which were its positive and negative nodes. Because of that, the current directions were defined accordingly to the picture shown. As such, the electric resistances, current sources and the capacitor are going to have the positive node in the location of the current input and the negative node in the current output. The voltage sources nodes were defined so that the nodes are in accordance with the pre established nodes. the picture below, a brief representation of the interpertation of the circuit by the ngspice programme is presented.

Apart from this, it was also necessary to establish the values of the resistance, current and voltage associated with the resistors and independent current and voltage sources. The dependent sources, Vc and Ib (Hc and Gb in ngspice because of the symbolic representation in the program), were respectively current controlled voltage source and voltage controlled current source. Because of that, kc and kb were in kOhm and mS as the script indicated. In the case of Vc, it was needed to indicate a control voltage source, Ve which, even though it doesn't exist, is going to measure the current Ic, the control current, passing through it. This voltage control has a null potential so it doesn't interfere with the rest of the circuit. The nodes of Ve were placed in the circuit so that the current input is in the positive node of the voltage source and the output of the current is in the negative node of the source. In the case of Ib, it was needed to introduce the nodes in which the potential difference is going to be read (again the positive node in first place and then the negative one).

The circuit with the considered current directions can be seen in the picture below.

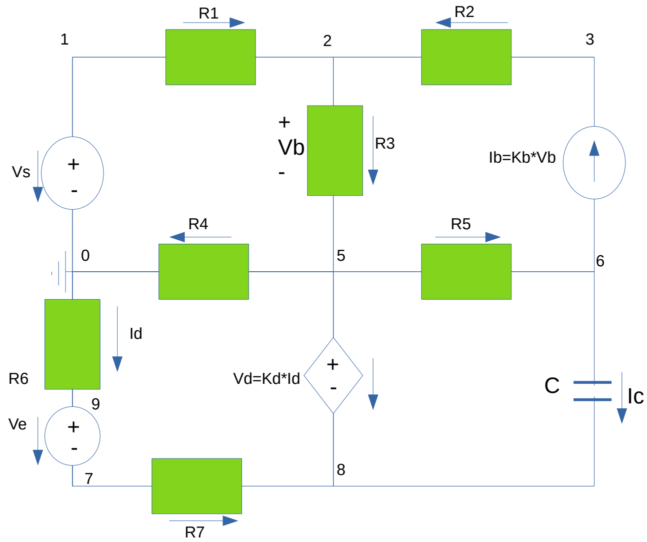

Figure 9: Picture of the circuit in analysis with current directions followed in the ngspice simulation

For the first point, it is asked to simulate the circuit when t is less than zero. As such, $vs = Vs$ and the current is DC, which means that there is no voltage variation with time in nodes 0 and 1 and, as the other sources are linear, voltage won't varie with time in the nodes. As such, the current which passes through the capacitor is going to be null as there is no variation with the potential difference in it. Runing the simulation (not transient as all intensity and potential values are constant), we obtain a table with the values of the current and voltage in all nodes confirming what was previously said. This confirmes the values calculated through octave in the theoretical analysis. Below a table with the values obtained and the errors associated is shown.

@ca[i]	0.000000e+00
@gb[i]	-2.72859e-04
@r1[i]	2.604926e-04
@r2[i]	-2.72859e-04
@r3[i]	-1.23662e-05
@r4[i]	1.248639e-03
@r5[i]	-2.72859e-04
@r6[i]	-9.88147e-04
@r7[i]	9.881466e-04
v(1)	5.223200e+00
v(2)	4.960945e+00
v(3)	4.406214e+00
v(5)	4.998463e+00
v(6)	5.852787e+00
v(7)	-2.06908e+00
v(8)	-3.07476e+00
v(9)	-2.06908e+00

Comparing the simulation with the theoretical prediction, we can compute the error associated to each value. The error values are presented in the table bellow.

@ca[i]	0.000000e+00 A
@gb[i]	2.000000e-10 A
@r1[i]	5.209852e-04 A
@r2[i]	2.000000e-10 A
@r3[i]	2.473237e-05 A
@r4[i]	2.497278e-03 A
@r5[i]	2.000000e-10 A
@r6[i]	4.000000e-10 A
@r7[i]	0.000000e+00 A
v(1)	0.000000e+00 V
v(2)	0.000000e+00 V
v(3)	0.000000e+00 V
v(5)	0.000000e+00 V
v(6)	0.000000e+00 V
v(7)	0.000000e+00 V
v(8)	0.000000e+00 V
v(9)	0.000000e+00 V

3.2 Second point

In this point, the goal is to simulate the operating point for $v_s(0) = 0$. In order to accomplish this, we replaced the capacitor with a voltage source $V_x = V(6) - V(8)$, where $V(6)$ and $V(8)$ are the voltages in nodes 6 and 8, respectively, found in point 1. This step is, as explained in subsection 2.2, based on Thévenin's theorem: we replace the capacitor with V_s, then we analyse all node voltages and branch currents; then, knowing $V(6)$, $V(8)$ and I_x, we can determine R_{eq} ($R_{eq} = \frac{V_x}{I_x}$), and compute the circuit's time constant ($\tau = R_{eq}C$).

The results output by Ngspice are presented in the following table.

@gb[i]	-1.07836e-18
@r1[i]	1.029487e-18
@r2[i]	-1.07836e-18
@r3[i]	-4.88721e-20
@r4[i]	-2.21871e-19
@r5[i]	-1.89416e-03
@r6[i]	-2.16840e-19
@r7[i]	-4.26567e-19
v(1)	0.000000e+00
v(2)	-1.03645e-15
v(3)	-3.22879e-15
v(5)	-8.88178e-16
v(6)	5.930624e+00
v(7)	4.540420e-16
v(8)	8.881784e-16
v(9)	4.540420e-16

Comparing the simulation with the theoretical prediction, we can compute the error associated to each value. The error values are presented in the table bellow.

v(1)	0.000000e+00 V
v(2)	8.184440e-01 V
v(3)	2.549645e+00 V
v(5)	7.013583e-01 V
v(6)	7.783700e-02 V
v(7)	2.069080e+00 V
v(8)	3.074760e+00 V
v(9)	2.069080e+00 V

3.3 Third Point

In this point, the objective is to simulate the natural response of the circuit in node 6 for $t \in [0; 20]$ms, that is, the evolution of this node's voltage throught time, when all independent sources are switched off. As the used boundary condition is $V(6)$ and $V(8)$ equal to the values found in point 3.2 for this nodes, we used the command *.include* in the *Ngspice's* script so that this values are imported automatically.

The obtained result is plotted bellow.

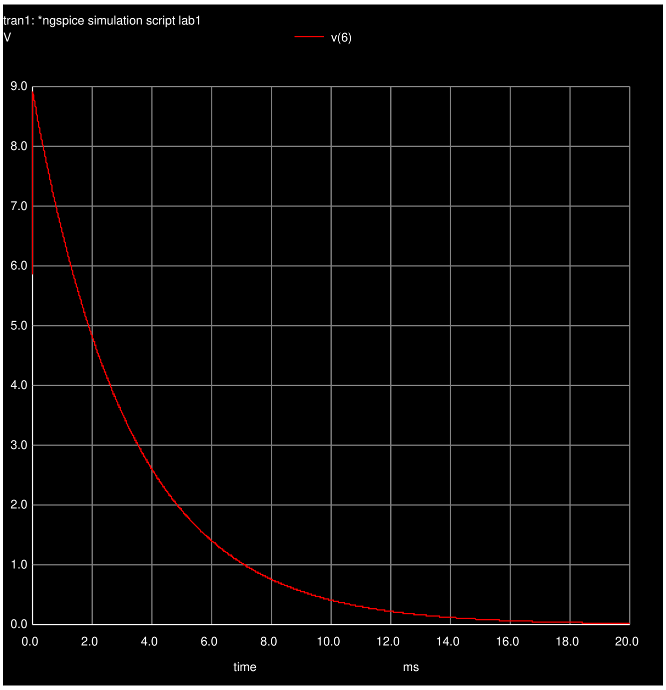

Figure 10: Natural solution $v_{6n}(t)$, $t \in [0, 20]$ms (simulation)

3.4 Fourth point

With the natural response values obtained, it's now asked to simulate the natural and forced response on node 6 now with vs as given in the script. The natural solution, as it doesn't varie with the values of the source in the circuit will be the same in this and the previous case. The forced solution is equivelent to the response of the node when vs is applied to the circuit maintining the condition that initially the values of the voltages in nodes 6 and 8 were the ones calculated in point 2 of the simulation (in order to simplify the code the values used were the ones calculated through octave in question 2). As the values in the nodes at the beggining of the transient simulation aren't the ones in which the circuit is stable, variation of the potential difference through the capacitor is going to happen. This can be seen in the ploted graph in which, v(1) varies around 0V as it is connected to the source and v(6) starts at near 6V and after 20ms is also varying around 0V. Also, as the voltage source response is given by a sinusoidal function, regular variation of the voltage in all the nodes is going to appear as also in them voltage values are given by sinusoidal funtions with the same given frequency (but different amplitudes). As seen in the graph below, regular oscilation can be seen in V(6) and v(1) even though the circuit has stabelized after 20ms.

All of this means that, variation of the potential difference through the capacitor is going to be noticeable and current isn't null. Notice that the amplitude of the current source vs is 1 and as seen in the graph amplitude is also 1 in v(1). However, the amplitude of voltage in node 6 is no longer 1 (approx 0.8V) as voltage is loss in the rest of the components of the circuit until the current reaches node 6.

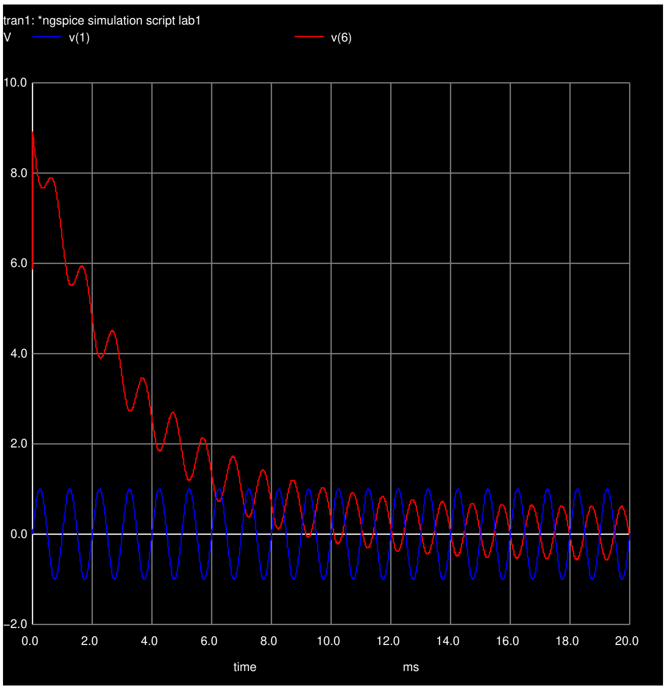

Figure 11: Voltages as functions of time

3.5 Fifth point

Finally, in the last question it was studied the frequency response in node 6 through the plot of the phase (rad) in function of the frequency and the magnitude in decibels (in which a 20db increment corresponds to 10x more amplitude) in function of the frequecy of the voltage source and thus the frequecy of the circuit. In these plots a logarithmic scale was used so that the graphics could be read more easily.

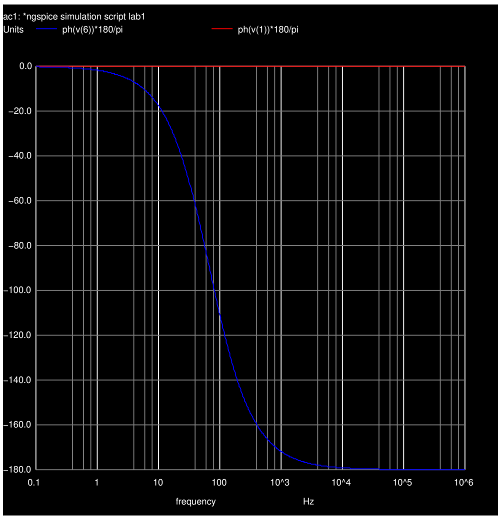

Figure 12: Phase as function of frequency

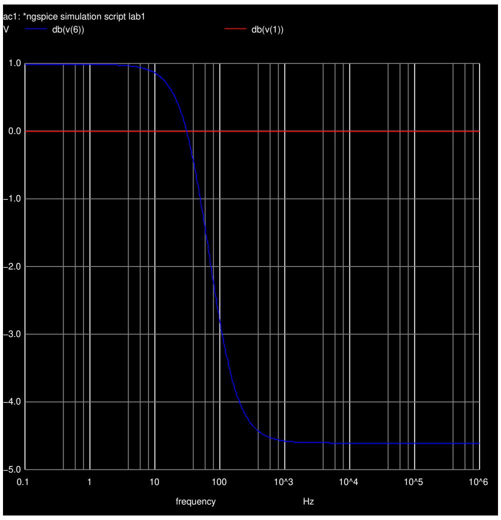

Figure 13: Voltage magnitude as function of time

In the first graphic which describes the phase of v(1) and v(6), as given by the expression intially given of v(1) when $t > 0$, its phase is null independently of the value of frequency. As we can see, the same doesn't happen to the phase of node 6. The reason why this happens can be seen in 2.6.

In the second graphic, the magnitude is expressed in db. In this case, the magnitude of V(1) is constant and null as its magnitude is always 1V and $log(1) = 0$. In the case of the magnitude of v(6) as we can see in the equation below, its magnitude varies. The reason why this happens can be seen in 2.6.

4 Conclusion

In this laboratory assignment the goal of analysing the natural, forced and frequencie response of a transient RC circuit has been achieved. As seen in the whole report, the theoretical results (obtained using Octave) matched the simulation ones (obtained using Ngspice simulator) with negligible relative errors (less than 0.05 as seen in the tables above. Even though, the match isn't perfect because of the many elements that are included in it and the non-linearity associated, it's good enough to validate the model used in the theoretical analysis. Although the match is almost perfect, we have to say that if the data were obtained in a real situation, this result would be very hard to achieve, since we had to take into acount other systhematic and accidental errors that might have occurred, as well as phenomena like joule effect in the conductors.