

# Circuit Theory and Electronics Fundamentals 2020/2021

Integrated Masters in Aerospace Engineering, Técnico, University of Lisbon

## Second Laboratory Report

April 8, 2021

Rui Rodrigues, 95844  
Tiago Silva, 95850  
Tomás Ribeiro, 95854

## Contents

<b>1</b>	<b>Introduction</b>	<b>2</b>
<b>2</b>	<b>Theoretical Introduction</b>	<b>2</b>
<b>3</b>	<b>Theoretical Analysis</b>	<b>3</b>
3.1	Theoretical - Topic I . . . . .	3
3.2	Theoretical - Topic II . . . . .	5
3.3	Theoretical - Topic III . . . . .	5
3.4	Theoretical - Topic IV . . . . .	6
3.5	Theoretical - Topic V . . . . .	6
3.6	Theoretical - Topic VI . . . . .	7
<b>4</b>	<b>Simulation Analysis</b>	<b>9</b>
4.1	Simulation - Topic I . . . . .	9
4.2	Simulation - Topic II . . . . .	10
4.3	Simulation - Topic III . . . . .	11
4.4	Simulation - Topic IV . . . . .	11
4.5	Simulation - Topic V . . . . .	12
<b>5</b>	<b>Relative Error and Graphic Analysis</b>	<b>13</b>
5.1	Topic I . . . . .	13
5.2	Topic II . . . . .	13
5.3	Topic III . . . . .	14
5.4	Topic Theo V - Sim IV . . . . .	15
5.5	Topic Theo VI - Sim V . . . . .	15
<b>6</b>	<b>Conclusion</b>	<b>16</b>

# 1 Introduction

The objective of this laboratory assignment is to study an RC circuit containing a voltage source  $v_S$  (defined by the following sinusoidal equation:  $v_s(t) = V_s u(-t) + \sin(2\pi f t) u(t)$ ), a current-controlled voltage source  $V_D$ , a voltage-controlled current source  $I_B$  and a capacitor  $C$  connected to different fixed value resistors  $R_1, R_2, R_3, R_4, R_5, R_6$  and  $R_7$ . The circuit can be seen in Figure 1.

In Section 2, a theoretical introduction is made in order to contextualize all the main principles that sustain our analysis of the circuit. This circuit is carefully analysed according to six different theoretical analysis topics, presented in Section 3, where the results are obtained in GNU Octave. Also, in Section 4, the circuit is analysed by simulation through the use of NGSpice to simulate the electric circuit behaviour. The results of the simulation of Section 4 are then compared to the theoretical results obtained in Section 3 and the comparative results are expressed in Section 5. The conclusions of this study are outlined in the final part of the report, in Section 6.

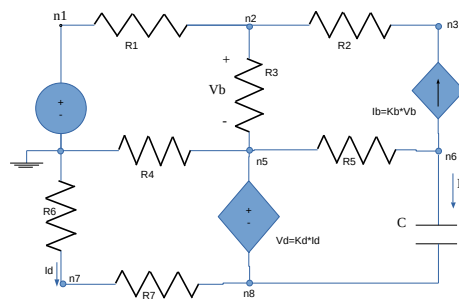


Figure 1: Second laboratory circuit.

# 2 Theoretical Introduction

The Node Voltage Method is a way to analyze a circuit. This method is based on Kirchhoff's Current Law (KCL). To apply this method, we need to define what node voltage is. When we use the term node voltage, we are referring to the potential difference between two nodes of a circuit. We select one of the nodes in our circuit to be the reference node and, therefore, all the other node voltages are measured with respect to the referenced one. This reference node is called the ground node and, as it gets the ground symbol in Figure 1, corresponds to the node between resistor  $R_1$  and voltage source  $V_A$ . The potential of the ground node is defined to be  $0V$  and the potentials of all the other nodes are measured relative to ground.

The implementation of the Node Voltage Method to analyse the circuit was done following the common sequence of steps, summarized below. We use this method to determine the voltages in all the nodes and the currents in all the branches.

- Assign a reference node.
- Assign node voltage names to the remaining nodes.
- Solve the easy nodes first, the ones with a voltage source connected to the reference node.

- Write Kirchhoff's Current Law for each node.
- Solve the resulting system of equations for all node voltages.
- Solve for any currents you want to know using Ohm's Law.

A RC circuit (also known as an RC filter or RC network) stands for a resistor-capacitor circuit. A RC circuit is defined as an electrical circuit composed of the passive circuit components of a resistor (R) and capacitor (C), driven by a voltage source or current source. Due to the presence of a resistor in the ideal form of the circuit, a RC circuit will consume energy, akin to a RL circuit or RLC circuit. This is unlike the ideal form of a LC circuit, which will consume no energy due to the absence of a resistor. Although this is only in the ideal form of the circuit, and in practice, even a LC circuit will consume some energy because of the non-zero resistance of the components and connecting wires.

### 3 Theoretical Analysis

In this section, we can find the results of each topic required in the theoretical analysis. The numeric results or graphics are presented alongside a short explanation of the interpretation of the problem. All of the results were obtained using GNU octave and the section is divided into six different subsections - Subsection 3.1, Subsection 3.2, Subsection 3.3, Subsection 3.4, Subsection 3.5 and Subsection 3.6 -, one for each topic of the theoretical analysis.

#### 3.1 Theoretical - Topic I

We start to apply the Node Voltage Method by starting to identify the nodes of the circuit. In this case, our circuit has 8 nodes and each one has a node voltage designated  $V_1, V_2, V_3, V_4, V_5, V_6, V_7$  and  $V_8$ , according to the related node. Then, we apply the KCL to each one of the nodes.

$$\left\{ \begin{array}{l} \text{Node1 : } V_1 = V_S \\ \text{Node2 : } I_1 + I_2 = I_3 \\ \text{Node3 : } I_B = I_2 \\ \text{Node4 : } V_4 = 0 \\ \text{Node5 : } V_5 - V_8 = V_D \\ \text{Node6 : } I_C + I_B + I_5 = 0 \\ \text{Node7 : } I_D = I_7 \\ \text{Node8 : } V_5 - V_8 = V_D \end{array} \right.$$

After that, we rewrite it into an equivalent system defining the currents in terms of node voltages.

$$\left\{ \begin{array}{l} \text{Node1 : } V_1 = V_S \\ \text{Node2 : } \frac{V_1 - V_2}{R_1} + \frac{V_3 - V_2}{R_2} = \frac{V_2 - V_5}{R_3} \\ \text{Node3 : } I_B = \frac{V_3 - V_2}{R_2} \\ \text{Node4 : } V_4 = 0 \\ \text{Node5 : } V_5 - V_8 = K_D * I_D \\ \text{Node6 : } I_C + I_B + \frac{V_6 - V_5}{R_5} = 0 \\ \text{Node7 : } I_D = \frac{V_7 - V_8}{R_7} \\ \text{Node8 : } V_5 - V_8 = V_D \end{array} \right.$$

Now, we have a system of 6 equations (note that applying KCL to node 5 and node 8 generate the same exact equation). Between these two nodes, the circuit presents a voltage source  $V_D$ . Therefore, to obtain another equation to add to the system, we need to consider a supernode that includes both node 5 and node 8 and, after that, apply KCL to the supernode we just created.

We obtain another equation:  $I_3 + I_5 + I_C + I_D = I_4$ .

Once again, we define the currents in terms of node voltages and obtain an equivalent equation:  $\frac{V_2 - V_5}{R_3} + \frac{V_6 - V_5}{R_5} + I_C + I_D = \frac{V_5}{R_4}$ .

At this point, we have a system with 7 equations. However, besides the node voltages  $V_1, V_2, V_3, V_5, V_6, V_7$  and  $V_8$ , we still have two more variables to determine its value,  $I_B$  and  $I_D$  (note that  $I_C = 0V$  when  $t < 0$ ). So, we need to find two more equations to complete our system. We have to define the missing value currents in terms of node voltages and get the two extra equations.

$$\begin{cases} I_B = (V_2 - V_5) * K_B \\ I_D = \frac{0 - V_7}{R_6} \end{cases}$$

This takes us to our final system of equations, with 10 equations to find the values of 10 variables.

$$\left\{ \begin{array}{l} V_1 = V_S \\ \frac{1}{R_1} V_1 + (-\frac{1}{R_1} - \frac{1}{R_2} - \frac{1}{R_3}) V_2 + \frac{1}{R_2} V_3 + \frac{1}{R_3} V_5 = 0 \\ I_B - \frac{1}{R_2} V_3 + \frac{1}{R_2} V_2 = 0 \\ V_5 - V_8 - K_D * I_D = 0 \\ I_B + \frac{1}{R_5} V_6 - \frac{1}{R_5} V_5 = 0 \\ I_D - \frac{1}{R_7} V_7 + \frac{1}{R_7} V_8 = 0 \\ I_D + \frac{V_2 - V_5}{R_3} + \frac{V_6 - V_5}{R_5} - \frac{V_5}{R_4} = 0 \\ I_D + \frac{1}{R_3} V_2 + (-\frac{1}{R_3} - \frac{1}{R_4} - \frac{1}{R_5}) V_5 + \frac{1}{R_5} V_6 = 0 \\ I_B - -K_B * V_2 + K_B * V_5 = 0 \\ I_D + \frac{V_7}{R_6} = 0 \end{array} \right.$$

We can transform this system of equations and put it into a matrix form, ready to be solved in GNU Octave.

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ \frac{1}{R_1} & -\frac{1}{R_1} - \frac{1}{R_2} - \frac{1}{R_3} & \frac{1}{R_2} & \frac{1}{R_3} & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{1}{R_2} & -\frac{1}{R_2} & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & -\frac{1}{R_5} & \frac{1}{R_5} & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{R_7} & -\frac{1}{R_7} & 0 & 0 & -1 \\ 0 & -K_B & 0 & K_B & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & \frac{1}{R_3} & 0 & -\frac{1}{R_3} - \frac{1}{R_4} - \frac{1}{R_5} & \frac{1}{R_5} & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{R_6} & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \\ I_B \\ I_C \\ I_D \end{bmatrix} = \begin{bmatrix} V_S \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

These are the final values obtained with the application of the Node Voltage Method:

Nodal Analysis Voltages [in Volts]	
Node Voltage 1	5.02924600001e+00
Node Voltage 2	4.78354415384e+00
Node Voltage 3	4.28814736170e+00
Node Voltage 5	4.81753272504e+00
Node Voltage 6	5.57990489781e+00
Node Voltage 7	-1.85471262435e+00
Node Voltage 8	-2.77162277031e+00

### 3.2 Theoretical - Topic II

$$\begin{cases} v_s = 0V \\ V_x = V_6 - V_8 \end{cases}$$

To analyze an RC circuit more complex than simple series (given that we are dealing with a complex electric circuit with different dependent sources), we can convert the circuit into a Thévenin equivalent, applying Thévenin's theorem. Then, we treat the capacitor as the "load" and reduce everything else to an equivalent circuit of one voltage source and one series resistor. After that, we are able to analyze what happens over time with the universal time constant formula. Our time constant for this circuit will be equal to the Thévenin' resistance,  $R_{eq}$ , times the capacitance  $C$ .

Nodal Analysis Voltages [in Volts]	
Node Voltage 1	0.000000000000e+00
Node Voltage 2	-0.000000000000e+00
Node Voltage 3	-0.000000000000e+00
Node Voltage 5	0.000000000000e+00
Node Voltage 6	8.35152766812e+00
Node Voltage 7	0.000000000000e+00
Node Voltage 8	-0.000000000000e+00
Req, Equivalent Resistor	3094.147869
Time Constant	0.003190

where  $R_{eq} = \frac{V_x}{I_x}$  is expressed in Ohms ( $\Omega$ ) and the time constant  $\tau$  is expressed in seconds (s).

### 3.3 Theoretical - Topic III

The natural response tells us what the circuit does as its internal stored energy (the initial voltage on the capacitor, calculated in the previous topic,  $V_x$ ) is allowed to dissipate. It does this by ignoring the forcing input and considering only the initial voltage and the equivalent resistance and time constant, determined in the previous topic.

$$v_n(t) = V_x e^{-\frac{t}{\tau}} = V_x e^{-\frac{t}{RC}}$$

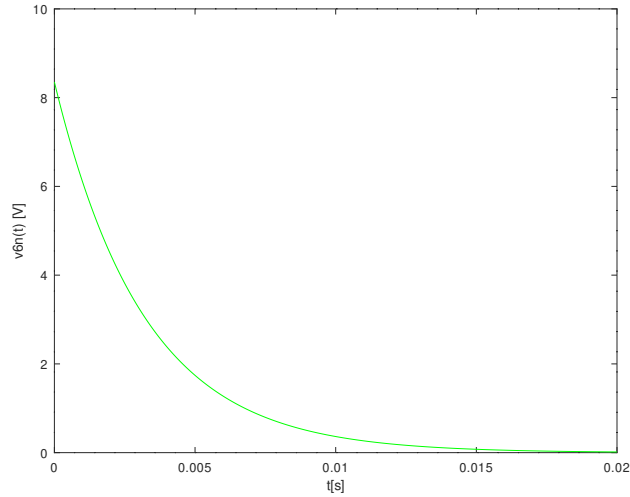


Figure 2: The natural solution,  $V6_n(t)$ , during time interval  $[0, 20]$  ms.

where  $t$  is expressed in seconds (s) along the x-axis and  $V6_n(t)$ , the natural solution, is expressed in Volts (V) along the y-axis.

### 3.4 Theoretical - Topic IV

The forced response is where the output (the voltage on the capacitor) is going to end up in the long run after all stored energy eventually dissipates. This occurs by ignoring the presence of energy storage elements (in our circuit analysis, it ignores the capacitor and its initial voltage,  $V_x$ ). As suggested, plotting the amplitude and phase shift of a sinusoid in a complex plane, we get a complex number in polar form that we can apply to the circuit analysis: a phasor voltage source,  $V_s = 1$ . Besides that, we also replaced  $C$  with its impedance  $Z_C = \frac{1}{\omega C}$ .

$$\begin{cases} f = 1kHz = 1000Hz \\ t = 20ms = 0.020s \end{cases}$$

Nodal Analysis of Phasors [in Volts]	
Phasor of Node 1	6.12323399574e-17+i-1.57079632679e+00
Phasor of Node 2	5.72250611431e-17+i-1.57079632679e+00
Phasor of Node 3	4.91453779795e-17+i-1.57079632679e+00
Phasor of Node 5	5.77793983751e-17+i-1.57079632679e+00
Phasor of Node 6	-8.26523333807e-02+i1.72076924612e+00
Phasor of Node 7	-2.34568735257e-17+i1.57079632679e+00
Phasor of Node 8	-3.46204860767e-17+i1.57079632679e+00

### 3.5 Theoretical - Topic V

The natural response considers the internal initial conditions. The forced response considers the external inputs. Given that, we get the total response by summing the two responses, natural and forced. In fact, this is the principle of superposition in action (in the mathematical sense of differential equations).

$$\begin{cases} v_t = v_n + v_f \\ f = 1kHz = 1000Hz \end{cases}$$

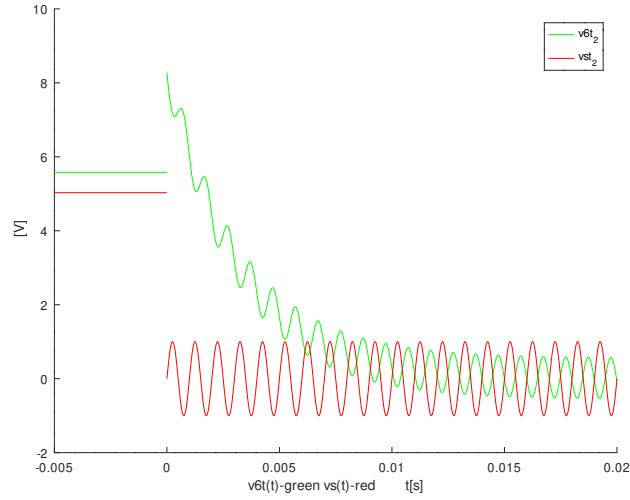


Figure 3: The final total solution,  $V_6(t)$ , and  $v_s(t)$  during time interval  $[-5, 20]$  ms.

Where  $t$  is expressed in seconds (s) along the x-axis and  $V_6(t)$ , the total solution, and  $v_s$  are expressed in Volts (V) along the y-axis.

In the graphic above, it is interesting to note that, while the voltage emitted by the source  $V_s$  is constant, the node 6 will also have a constant voltage. This is because for  $t < 0$  the circuit will essentially behave as one of continuous current. After this time mark, the circuit reveals its alternate current nature and both the voltages of the source and of node 6 will vary sinusoidally.

### 3.6 Theoretical - Topic VI

In this subsection, an analysis of the variation of the magnitude and phase of the phasors of  $V_c$ ,  $V_6$  and  $V_s$  is made in function of the frequency. The first graphic presents the study of the variation of the magnitude and the second the study of the phase. In both graphics, the frequency  $f$  in the x-axis varies from 0.1Hz to 1MHz

$$\begin{cases} f \in [0.1, 1] \text{ MHz} \\ v_c(f) = v_6(f) - v_8(f) \end{cases}$$

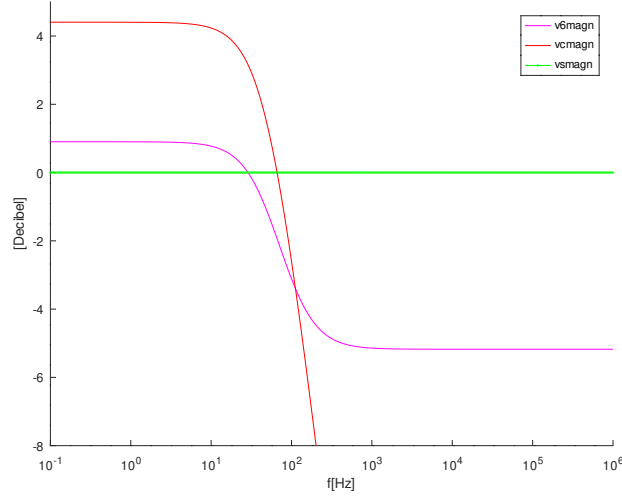


Figure 4: Magnitude of  $v_s(f)$ ,  $v_c(f)$  and  $v_6(f)$  during frequency interval [0.1 , 1] MHz.

The frequency  $f$  is expressed in Hertz (Hz) along the x-axis and the magnitude of  $v_s(f)$ ,  $v_c(f)$  and  $v_6(f)$  is expressed with a logscale decibel (dB) along the y-axis.

When looking at this graph, a couple of things are important to highlight. First,  $V_s$  will always have magnitude 0 because, given that its absolute value is 1 and we are using a logscale,  $\log(1) = 0$ . Second, we also note that the  $v_c$  graph line does not resemble the one that represents  $v_6$ . This is because the voltage of the node 8 will also vary.

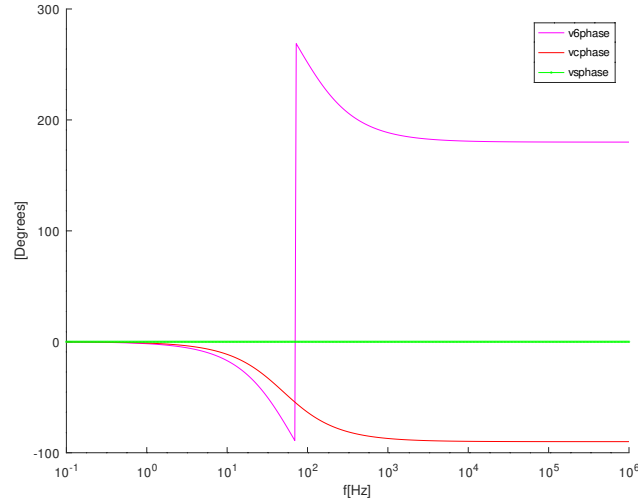


Figure 5: Phase  $v_s(f)$ ,  $v_c(f)$  and  $v_6(f)$  during frequency interval [0.1 , 1] MHz.

The frequency  $f$  is expressed in Hertz (Hz) along the x-axis and the phase of  $v_s(f)$ ,  $v_c(f)$  and  $v_6(f)$  is expressed in degrees along the y-axis.

In this graph, the first thing that comes to the eye is the fact that the phase of  $V_s$  will always be constant and that, once again, the  $v_c$  graph line does not resemble the one that represents  $v_6$ .



## 4 Simulation Analysis

In this section, we can find the results of each topic required in the simulation analysis. The numeric results or graphics are presented alongside a short explanation of the interpretation of the problem. All of the results were obtained using NGSpice and the section is divided into five different subsections - Subsection 4.1, Subsection 4.2, Subsection 4.3, Subsection 4.4 and Subsection 4.5, one for each topic of the simulation analysis.

Ngspice is a circuit-simulation program that makes it possible to have an accurate representation of how the circuit would behave if it was actually assembled. The different tables shown below illustrate the simulated operating point results for the circuit under analysis. As it can be seen, the tables show the values of the voltage in all nodes, the currents in all of the branches and also the currents in independent voltage sources.

Sometimes, when looking at the simulation results there is one very interesting detail which is very important: Ngspice operates with the idea that the positive current flows from the positive pole to the negative pole in all components, sources included. This explains why, in some cases, in the very same branch where a voltage source is located, the current given by Ngspice in the whole branch is the symmetric of the one given specifically in the voltage source. Another important detail about this operating point analysis of the circuit is that the node  $n_A$  and the voltage source  $V_{AB} = 0V$ , which are absent from the circuit's picture. This is because the Current-Controlled Voltage Source is dependent on current  $I_D$ , but Ngspice requires a voltage source where this current flows through, which made necessary the use of an auxiliary voltage source in series with  $R_6$  and, therefore, an auxiliary node.

### 4.1 Simulation - Topic I

Node/Component	Value [A or V]
@c1[i]	0.000000e+00
@gb[i]	-2.46392e-04
@r1[i]	2.354321e-04
@r2[i]	-2.46392e-04
@r3[i]	-1.09596e-05
@r4[i]	1.150256e-03
@r5[i]	2.463916e-04
@r6[i]	9.148235e-04
@r7[i]	9.148235e-04
n1	5.029246e+00
n2	4.783544e+00
n3	4.288147e+00
n5	4.817533e+00
n6	5.579905e+00
n7	-1.85471e+00
n8	-2.77162e+00
na	0.000000e+00

Voltages and the currents in all nodes and in all branches

For topic 1 of the simulation analysis section, we can see, in this table, the results of the simulation used to determine the voltages and the currents in all nodes and in all branches, respectively, by simulating the operating point for  $t < 0$ .

## 4.2 Simulation - Topic II

Node/Component	Value [A or V]
@gb[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]	2.699137e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
n2	0.000000e+00
n3	0.000000e+00
n5	0.000000e+00
n6	8.351528e+00
n7	0.000000e+00
n8	0.000000e+00
na	0.000000e+00

Voltages and currents obtained by simulating the operating point for  $v_s(0) = 0$  and replacing the capacitor with a voltage source

For topic 2, by simulating the operating point for  $v_s(0) = 0$ , and replacing 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 as obtained in topic 1, we obtain the results printed in tabel above. We do this step in order to use Thevenin's Theorem to simplify this complex circuit, turning it into a more simple equivalent circuit consisting of a resistance in series with a source voltage.

### 4.3 Simulation - Topic III

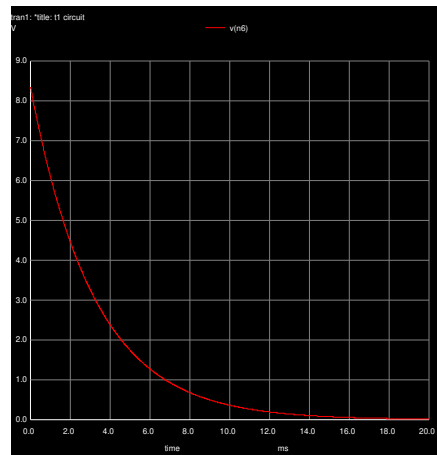


Figure 6: Natural response of the circuit.

For topic 3, by using the boundary conditions  $V(6)$  and  $V(8)$  obtained in topic 2, and by using Ngspice's transient analysis mode to get  $v_6(t)$  in the interval  $[0, 20]$  ms, we can simulate the natural response of the circuit and plot the results, as presented in the figure above.

### 4.4 Simulation - Topic IV

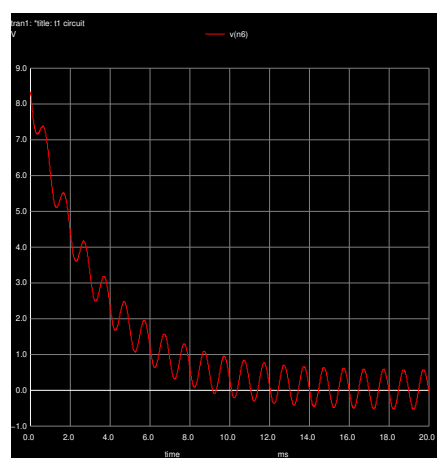


Figure 7: Total response of the circuit.

Regarding topic 4, the above figure is plotted by simulating the total response on node 6, that is, the natural and the forced solutions, by repeating the step presented in topic 3 with  $v_s(t) = V_s u(-t) + \sin(2\pi ft)u(t)$  and frequency equal to 1  $KHz$ .

#### 4.5 Simulation - Topic V

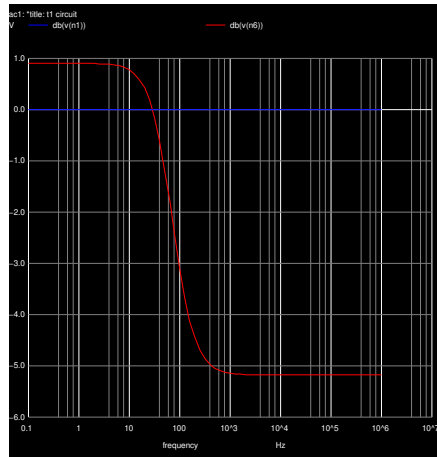


Figure 8: Magnitude of  $v_s(f)$  and  $v_6(f)$  during frequency interval [0.1 , 1] MHz.

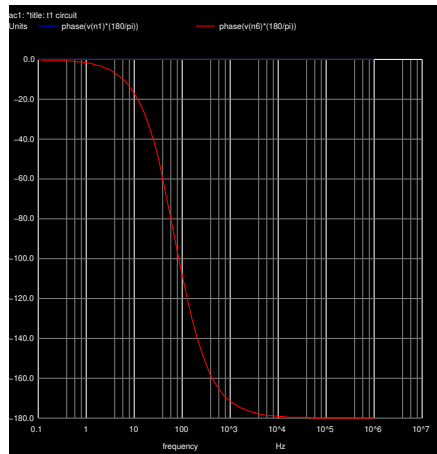


Figure 9: Phase of  $v_s(f)$  and  $v_6(f)$  during frequency interval [0.1 , 1] MHz.

Furthermore, in topic 5, we simulate the frequency response in node 6 (it is important to note that the frequency is presented in a logarithmic scale, the magnitude in dB and the phase in degrees) for the frequency range of 0.1 Hz to 1 MHz. Finally, we plot both  $v_s(f)$  and  $v_6(f)$ , as presented in the figures above. The differences between the magnitude and phase of the two voltages are explained because they correspond to different nodes.

By analysing both the plots, we can conclude that the function that describe each one of the node voltages are, in fact, similar in the representation of its magnitude and phase. As it can be seen, the magnitude of  $v_s$  is null and it remains constant till the end of the time interval. The magnitude of node voltage  $V_6$  is represented by a curve that shows variation through frequency. This magnitude is described by a descent of the node voltage values, which start with a constant value around 1 db and tends to stabilize near -5 db in the final section of the interval. As previously stated for the magnitude, also the phase of  $v_s$  is null and remains constant till the end of the time interval. On the other hand, the phase of node voltage  $V_6$  is also null in the beginning of the time interval. However, as we can see, the phase of this node voltage is represented by a curve that shows a descent variation through time. This phase can be described by this curve with an inflexion point in the middle value of the time interval that tends to stabilize near -100 degrees.

## 5 Relative Error and Graphic Analysis

### 5.1 Topic I

Comparative Analyses of First Topic Node Voltages			
Node	GNU Octave	NGSpice	Relative Error
Node 1	5.029246e+00	5.02924600001e+00	1.98854639559e-10
Node 2	4.783544e+00	4.78354415384e+00	3.21593857010e-06
Node 3	4.288147e+00	4.28814736170e+00	8.43484618211e-06
Node 4	4.817533e+00	4.81753272504e+00	5.70752955381e-06
Node 5	5.579905e+00	5.57990489781e+00	1.83143456452e-06
Node 6	-1.85471e+00	-1.85471262435e+00	1.41496768886e-04
Node 7	-2.77162e+00	-2.77162277031e+00	9.99527365557e-05

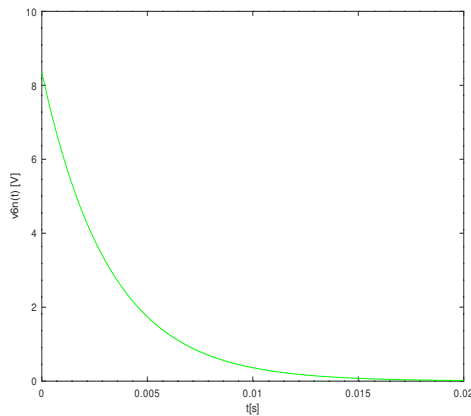
When the simulation results given by Ngspice in Section 4 are compared to the theoretical ones obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As it can be seen, the highest error in percentual value is in the order of  $10^{-4}$ , which is negligible. Such result can be explained by the fact that this is a very simple circuit with very simple components, thus not having a lot of chances to differ greatly.

### 5.2 Topic II

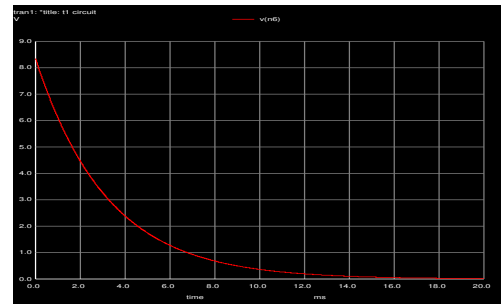
Comparative Analyses of Second Topic Node Voltages.			
Node	GNU Octave	NGSpice	Relative Error
Node 1	0.000000e+00	0.000000000000e+00	-
Node 2	0.000000e+00	-0.000000000000e+00	-
Node 3	0.000000e+00	-0.000000000000e+00	-
Node 5	0.000000e+00	0.000000000000e+00	-
Node 6	8.351528e+00	8.35152766812e+00	3.97391077415e-06
Node 7	0.000000e+00	0.000000000000e+00	-
Node 8	0.000000e+00	-0.000000000000e+00	-

When the simulation result given by Ngspice in Section 4 are compared to the theoretical one obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As it can be seen, the error in percentual value is in the order of  $10^{-6}$ , which is negligible. Such result can be explained by the fact that this is a very simple circuit with very simple components, thus not having a lot of chances to differ greatly. It is easy to understand that node voltages with a null value associated are not eligible to have a relative error analysis, so we consider that there exist an exact match between the theoretical and simulation results.

### 5.3 Topic III



(a) The natural solution,  $V_{6n}(t)$ , during time interval  $[0, 20]$  ms, obtained using GNU Octave.

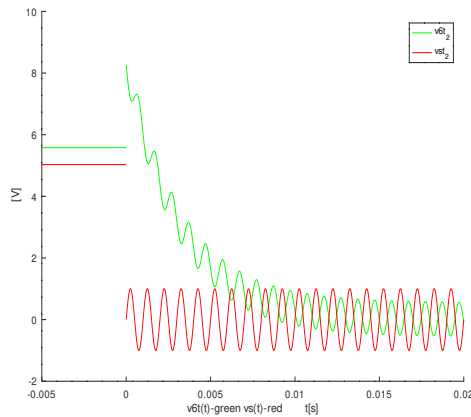


(b) The natural solution,  $V_{6n}(t)$ , during time interval  $[0, 20]$  ms, obtained using Ngspice.

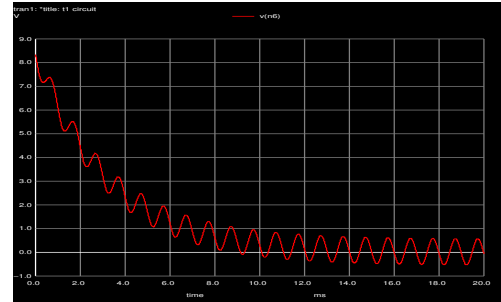
Figure 10: Comparison of theoretical and simulation analysis for the natural response.

When the simulation graphic given by Ngspice in Section 4 is compared to the theoretical one obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As it can be seen, both of the graphics describe a very similar descendent curve. The natural response of the electric circuit provokes a descent in the node voltage value, which tends to stabilize by the end of the time interval.

## 5.4 Topic Theo V - Sim IV



(a) The final total solution,  $V_6(t)$ , and  $v_s(t)$  during time interval  $[-5, 20]$  ms, obtained using GNU Octave.

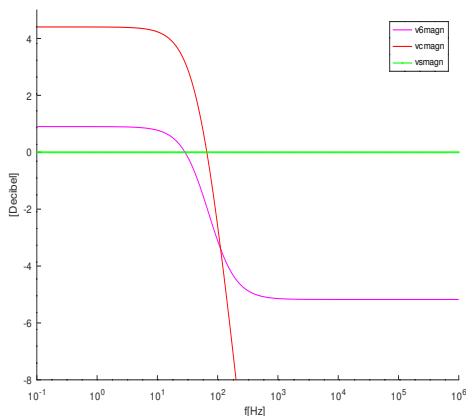


(b) The final total solution,  $V_6(t)$ , during time interval  $[-5, 20]$  ms, obtained using Ngspice.

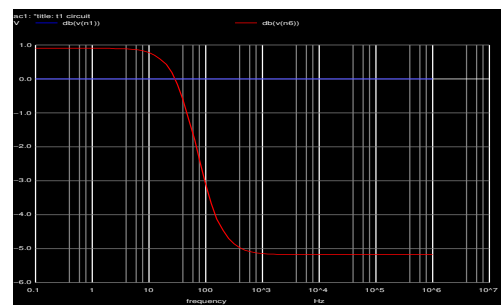
Figure 11: Comparison of theoretical and simulation analysis for the total response.

When the simulation graphic given by Ngspice in Section 4 is compared to the theoretical one obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As it can be seen, both of the graphics describe a very similar descendent curve with a certain oscillation. This variation of the node voltage along the curve seems to be constant in the two graphic representations. The total response of the electric circuit provokes a descent in the node voltage value, which tends to stabilize by the end of the time interval.

## 5.5 Topic Theo VI - Sim V



(a) Magnitude of  $v_s(f)$ ,  $v_c(f)$  and  $v_6(f)$  during frequency interval  $[0.1, 1]$  MHz, obtained using GNU Octave

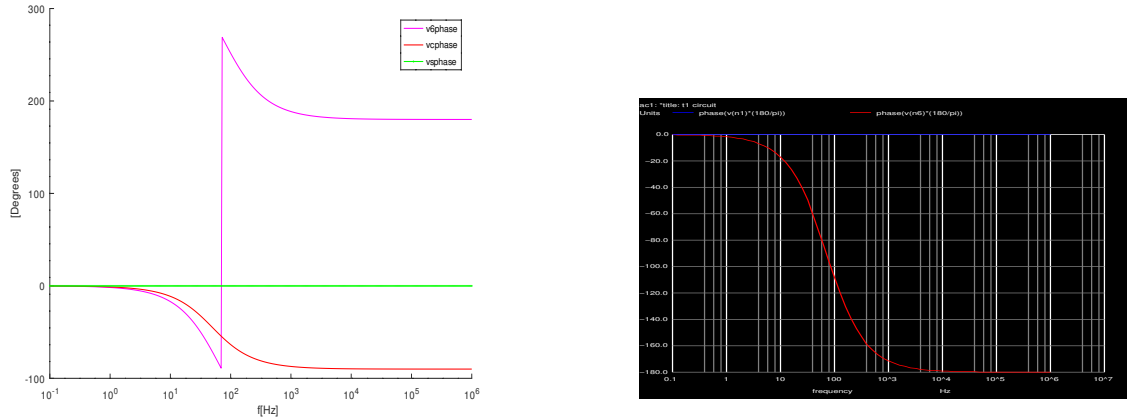


(b) Magnitude of  $v_s(f)$  and  $v_6(f)$  during frequency interval  $[0.1, 1]$  MHz, obtained using Ngspice.

Figure 12: Comparison of theoretical and simulation analysis for the nodal voltage magnitude.

When the simulation graphic given by Ngspice in Section 4 is compared to the theoretical one obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As it can be seen, the magnitude of  $v_s$  is null and it remains constant till the

end of the time interval. On the other hand, the magnitude of node voltage  $V_6$  is represented by a curve that shows variation through time. This magnitude is described by a descent of the node voltage values, which start with a constant value around 1 db and tends to stabilize near -5 db in the final section of the interval. This values can be evaluated both in theoretical and simulation graphics, which represent the exact same curve with an inflexion point in the middle value of the time interval.



(a) Phase of  $v_s(f)$ ,  $v_c(f)$  and  $v_6(f)$  during frequency interval [0.1 , 1] MHz, obtained using GNU Octave. (b) Phase of  $v_s(f)$  and  $v_6(f)$  during frequency interval [0.1 , 1] MHz, obtained using Ngspice.

Figure 13: Comparison of theoretical and simulation analysis for the nodal voltage phase.

When the simulation graphic given by Ngspice in Section 4 is compared to the theoretical one obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As previously stated for the magnitude, also the phase of  $v_s$  is null and remains constant till the end of the time interval. On the other hand, the phase of the capacitor  $V_c$  is also null in the beginning of the time interval. However, as we can see, the phase of this component is represented by a curve that shows a descent variation through time. This phase can be described by this curve with an inflexion point in the middle value of the time interval that tends to stabilize near -100 degrees. Once more, all the values can be evaluated both in theoretical and simulation graphics, that proved to be really close to each other.

It is also important to note that, even though, the  $V_6$  graph line might seem different for the two graphics, that really is not the case. What happens has entirely to do with the way angles can be written. For example, when the phase reaches -90° degrees octave then transforms it into 270 degrees, while Ngspice continues its graph line until it reaches -180 degrees.

## 6 Conclusion

In this second laboratory assignment, all the major goals of the project were achieved. We concluded with success a further interaction with a new software (Ubuntu), with a simulation platform (Ngspice), with a computational language program (GNU Octave) and with a text report editor (LaTeX). The analysis of the circuit was also finished with success through simulation and theoretical interpretation, which allowed a good comparative analysis between these two methods.

The main objective of the report was completed with the study of an RC circuit containing a voltage source  $v_s$  (defined by the following sinusoidal equation:  $v_s(t) = V_s u(-t) + \sin(2\pi ft)u(t)$ ),



a current-controlled voltage source  $V_D$ , a voltage-controlled current source  $I_B$  and a capacitor  $C$  connected to different fixed value resistors  $R_1$ ,  $R_2$ ,  $R_3$ ,  $R_4$ ,  $R_5$ ,  $R_6$  and  $R_7$ . Node voltages, magnitudes and phases were analysed both theoretically, using the Octave maths tool, and by circuit simulation, using the Ngspice tool. The simulation results matched the theoretical results precisely. This accuracy was confirmed by the mathematical calculation of relative errors, which were proved to be really small. Also, the comparative analysis of graphics plotted by both theoretical and simulation tools confirmed the similarity of the results. The reason for this perfect match is the fact that this is a straightforward circuit containing only simple and linear components (the capacitor included), so the theoretical and simulation models cannot differ. For more complex components, the theoretical and simulation models could differ. However, this is not the case of the analysis of this report, where the results are obtained successfully and with notorious precision.