

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

1	Introduction	2
2	Theoretical Introduction	2
3	Theoretical Analysis	3
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
4	Simulation Analysis	8
4.1	Simulation - Topic I	8
4.2	Simulation - Topic II	9
4.3	Simulation - Topic III	10
4.4	Simulation - Topic IV	10
4.5	Simulation - Topic V	11
5	Relative Error and Graphic Analysis	12
5.1	Topic I	12
5.2	Topic II	12
5.3	Topic III	13
5.4	Topic Theo V - Sim IV	13
5.5	Topic Theo VI - Sim V	14
6	Conclusion	14

1 Introduction

The objective of this laboratory assignment is to study an RC circuit containing a voltage source V_A , a current-controlled voltage source V_C , 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 , through a sinusoidal analysis using the following equation: $v_s(t) = V_s u(-t) + \sin(2\pi f t) u(t)$. 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 following the equation already presented that describes our RC circuit. In Section 4, the circuit is analysed by simulation and the results are compared to the theoretical results obtained in Section 3. The conclusions of this study are outlined in the final part of the report, in Section 5.

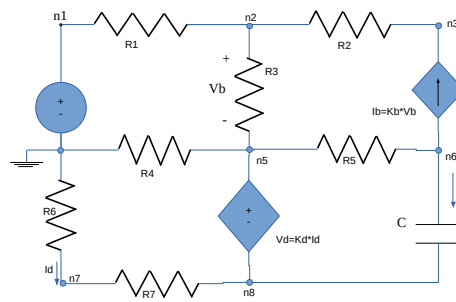


Figure 1: Second laboratory circuit.

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.

- 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.

We use this method to determine the voltages in all the nodes and the currents in all the branches.

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.

Circuits manipulate electrical signals. Signals convey energy, information or both (vital functions) and these signals can be decomposed into series or integrals of basic signals. Through this, we can use the equation presented to make a transient analysis of the circuit.

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.

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

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 (Ω) and the time constant τ 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.

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

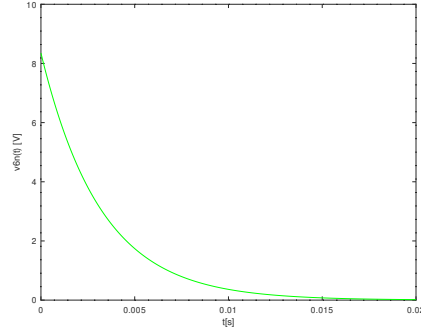


Figure 2: The natural solution, $V6_n(t)$, during time interval $[0, 20]$ ms.

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.

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

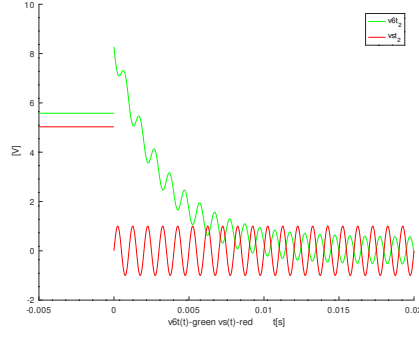


Figure 3: The final total solution, $V6(t)$, and $v_s(t)$ during time interval $[-5, 20]$ ms.

where t is expressed in seconds (s) along the x-axis and $V6(t)$, the total solution, and v_s are expressed in Volts (V) along the y-axis.

3.6 Theoretical - Topic VI

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

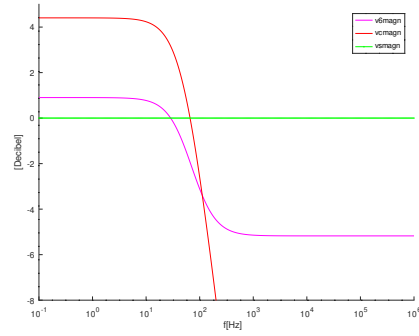


Figure 4: Magnitude of $v_s(f)$, $v_c(f)$ and $v_6(f)$ during frequency interval $[0.1, 1]$ MHz.

where 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.

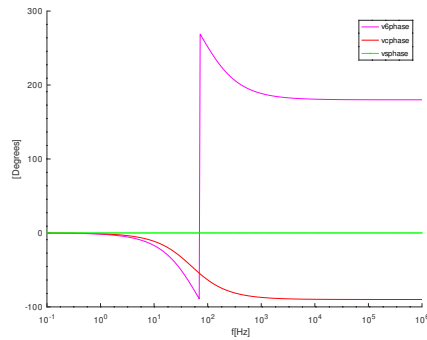


Figure 5: Phase $v_s(f)$, $v_c(f)$ and $v_6(f)$ during frequency interval $[0.1, 1]$ MHz.

where 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.

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 table 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

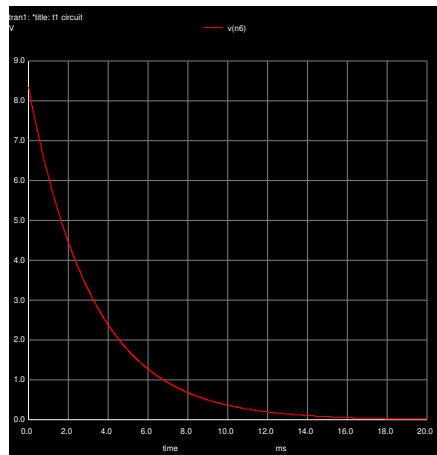


Figure 6: Natural response of the circuit

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

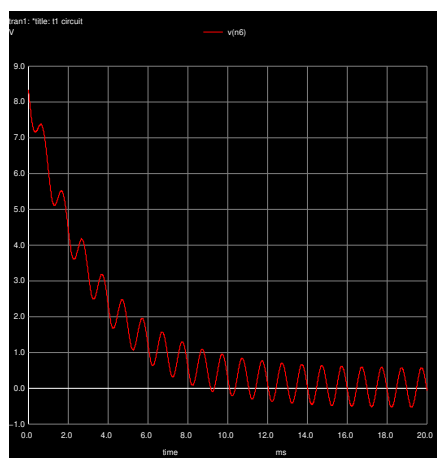


Figure 7: Total response of the circuit

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

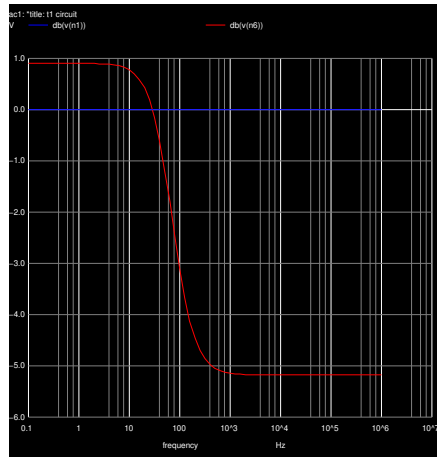


Figure 8: Frequency response for $V_s(f)$

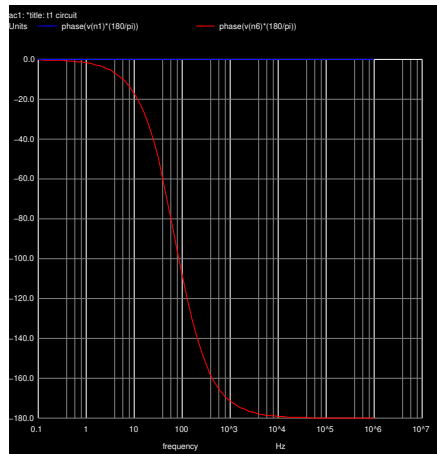


Figure 9: Frequency response for $V_6(f)$

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. By analysing both the plots, we can conclude that they are ... (FALTAM COISAS AQUI escrever análise aqui). This happens because they correspond to different nodes and different voltages.

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

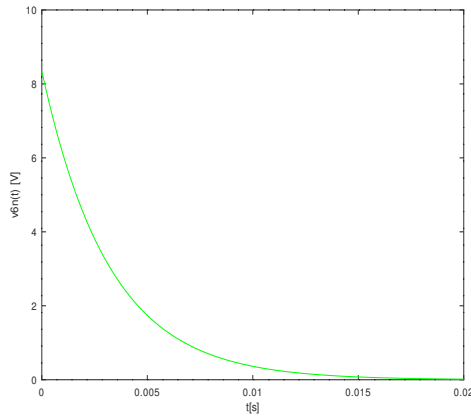
When the simulation results given by Ngspice in Section 4 are compared to the theoretical ones obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As it can be seen, the highest error in percentual value is in the order of 10^{-4} , which 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

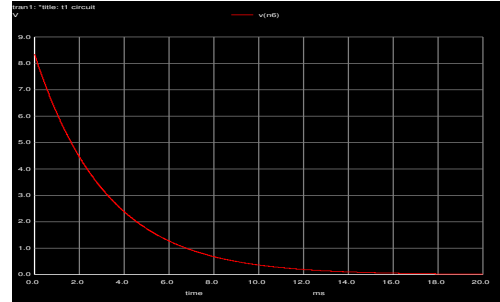
AComparative Analyses of Second Topic Node Voltages.			
Node	GNU Octave	NGSpice	Relative Error
Node 1	0.000000e+00	0.000000000000e+00	-
Node 2	0.000000e+00	-0.000000000000e+00	-
Node 3	0.000000e+00	-0.000000000000e+00	-
Node 5	0.000000e+00	0.000000000000e+00	-
Node 6	8.351528e+00	8.35152766812e+00	3.97391077415e-06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

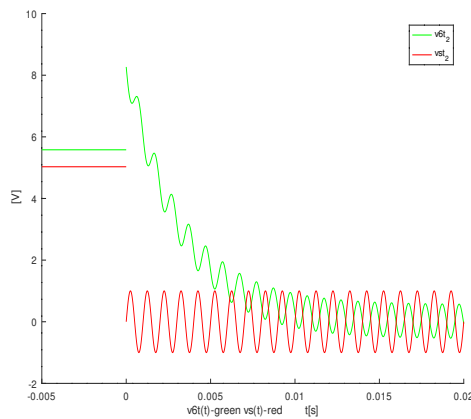


(b) The natural solution, $V_{6n}(t)$, during time interval [0, 20] ms, obtained using Ngspice.

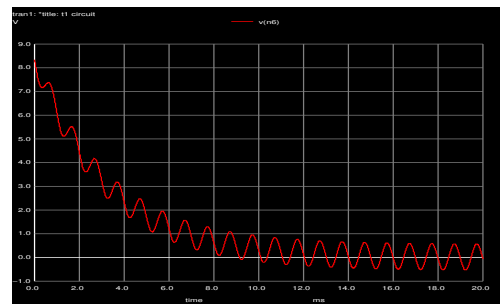
Figure 10: Comparison of theoretical and simulation analysis for the natural response.

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.



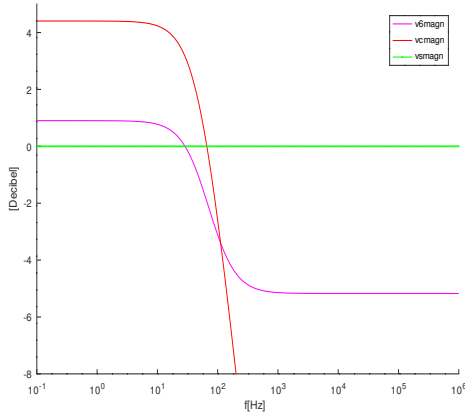
(b) The final total solution, $V_6(t)$, during time interval [-5, 20] ms, obtained using Ngspice.

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

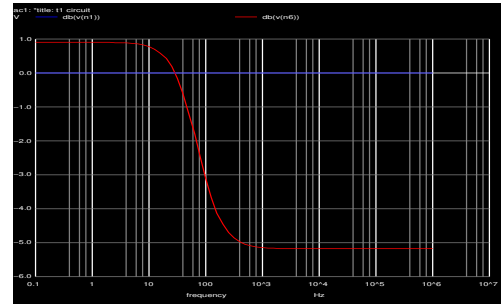
When the simulation graphic given by Ngspice in Section 4 is compared to the theoretical one obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As it can be seen, both of the graphics describe a very similar descendent curve with a certain 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

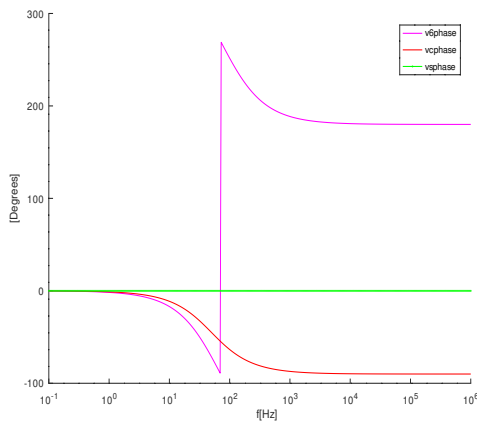


(a) Magnitude of $v_s(f)$, $v_c(f)$ and $v_6(f)$ during frequency interval [0.1 , 1] MHz, obtained using GNU Octave

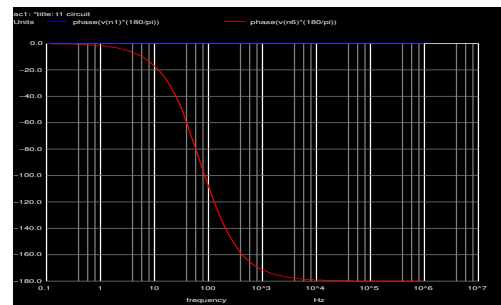


(b) Magnitude of $v_s(f)$ and $v_6(f)$ during frequency interval [0.1 , 1] MHz, obtained using Ngspice.

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



(a) Phase $v_s(f)$, $v_c(f)$ 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.

6 Conclusion

In this first laboratory assignment, all the major goals of the project were achieved. We concluded with success our first 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.

The main objective of the report was completed with the study of a circuit containing a voltage source V_A , a current-controlled voltage source V_C , a current source I_D and a voltage-controlled current source I_B connected to different fixed value resistors R_1 , R_2 , R_3 , R_4 , R_5 , R_6 and R_7 . Mesh currents and node voltage 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. The reason for this perfect match is the fact that this is a straightforward circuit containing only simple and linear components, 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.