Circuit Theory and Electronics Fundamentals 2020/2021

Integrated Masters in Aerospace Engineering, Técnico, University of Lisbon

Second Laboratory Report

March 25, 2021

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

Contents

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.

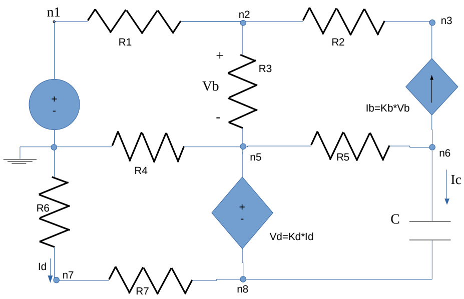

Figure 1: Second laboratory circuit.

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 obatined usig GNU octave and the section is dividid in 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.

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

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

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

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.

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

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

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

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	0.00000000000e+00
Node Voltage 2	-0.00000000000e+00
Node Voltage 3	-0.00000000000e+00
Node Voltage 5	0.00000000000e+00
Node Voltage 6	8.35152766812e+00
Node Voltage 7	0.00000000000e+00
Node Voltage 8	-0.00000000000e+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.

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

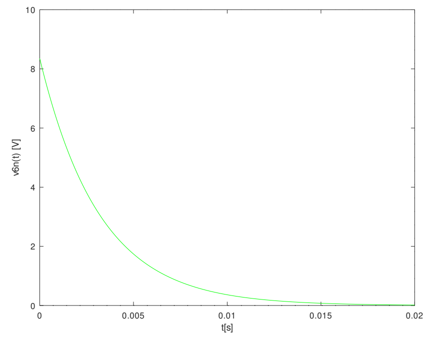

Figure 2: The natural solution, $V6_n(t)$, during time interval [0, 20] ms.

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 occors 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 respondes, natural and forced. In fact, this is the principle of superposition in action.

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

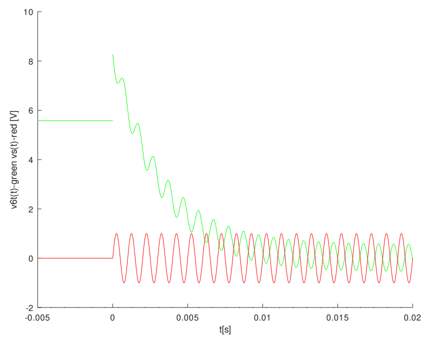

Figure 3: The final total solution, $V6(t)$, and $v_s(t)$ during time interval [-5, 20] ms.

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

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

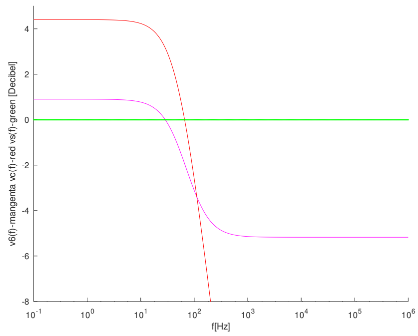

Figure 4: Magnitude of $v_s(f)$, $v_c(f)$ and $v_6(f)$ during frequency interval [0.1 , 1] MHz.

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.

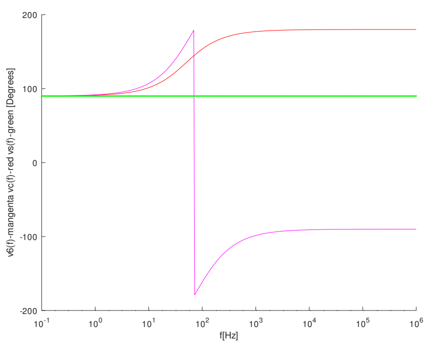

Figure 5: Phase $v_s(f)$, $v_c(f)$ and $v_6(f)$ during frequency interval [0.1 , 1] MHz.

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

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 currets obtained by simulating the operating point for $vs(0) = 0$ and replacing the capacitor with a voltage source

For topic 2, by simulating the operating point for $vs(0) = 0$, and replacing the capacitor with a voltage source $Vx = V(6) - V(8)$, 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

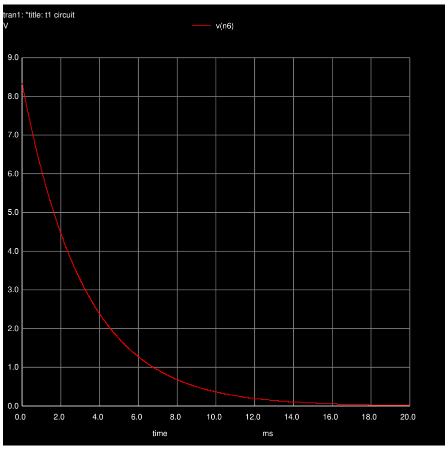

Figure 6: Natural response of the circuit

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 v6(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

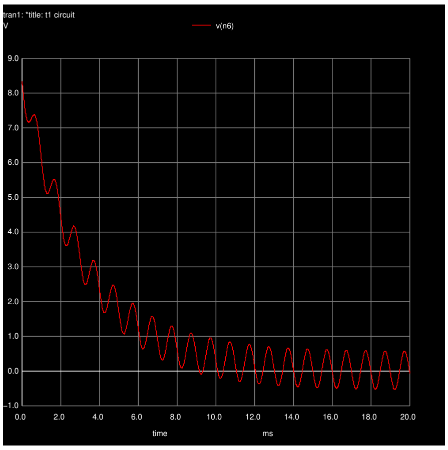

Figure 7: Total response of the circuit

Regarding topic 4, the above figure is ploted 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 $v_s(t) = V_s u(-t) + sin(2\pi f t)u(t)$ and frequency equal to 1 KH_z.

4.5 Simulation - Topic V

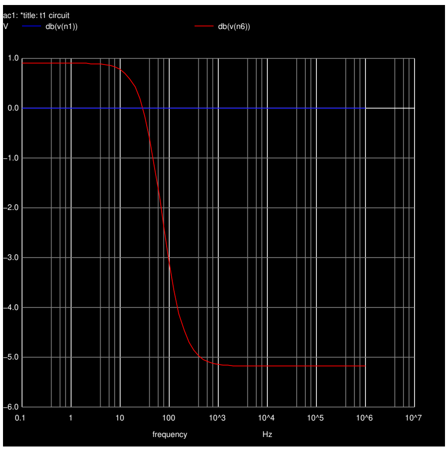

Figure 8: Frequency response for $V_s(f)$

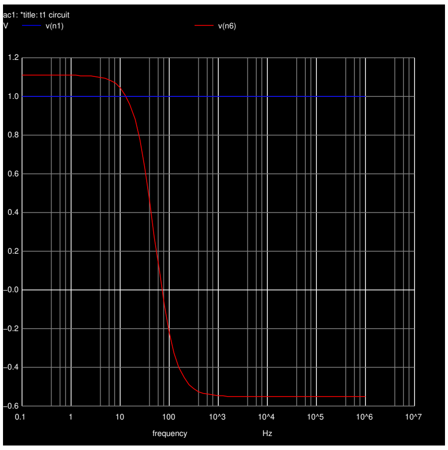

Figure 9: Frequency response for $V_6(f)$

Furthermore, in step 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 vs(f) and v6(f), as presented in the figures above. By analysing both the plots, we can conclude that they are ... (escrever análise aqui). This happens because they correspond to different nodes and diferent voltages.

5 Relative Error Analysis

5.1 Topic I

ANALISE DO ERRO COM 4 COLUNAS			
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	8.43484620282e-06
Node 4	4.817533e+00	4.81753272504e+00	5.70752955381e-06
Node 5	5.579905e+00	5.57990489781e+00	1.83143458044e-06
Node 6	-1.85471e+00	-1.85471262435e+00	1.41496768862e-04
Node 7	-2.77162e+00	-2.77162277031e+00	9.99527365397e-05

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 analised 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 matemathical 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 sucesfully and with notorious precision.