Circuit Theory and Electronics Fundamentals

Integrated Master's in Aerospace Engineering, Técnico, University of Lisbon

Laboratory 2 Report

Francisco Alves 95787, Tomás Nunes 95855, Rodrigo Sequeira 96480

April 5th, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing an independent voltage source v_s(t), a capacitor C, seven resistors R_1 to R_7, a linearly dependent current source I_b and a linearly dependent voltage source V_d. The circuit can be seen in Figure 1. The study is done in different time intervals, obtaining both the natural and the forced responses of the circuit over time making use of a certain given boundary condition, as well as plotting its response to a certain stimulus for a better understanding of the circuit's behavior. The nodal analysis method is used to obtain the circuit's parameters at certain time periods (operating points) which are the basis for further analysis.

In Section 2, a theoretical analysis of the circuit using Octave is presented. In Section 3, the circuit is analysed by simulation with Ngspice software and the results are compared to the theoretical results obtained in Section 2. Conclusions of this study can be found in Section 5.

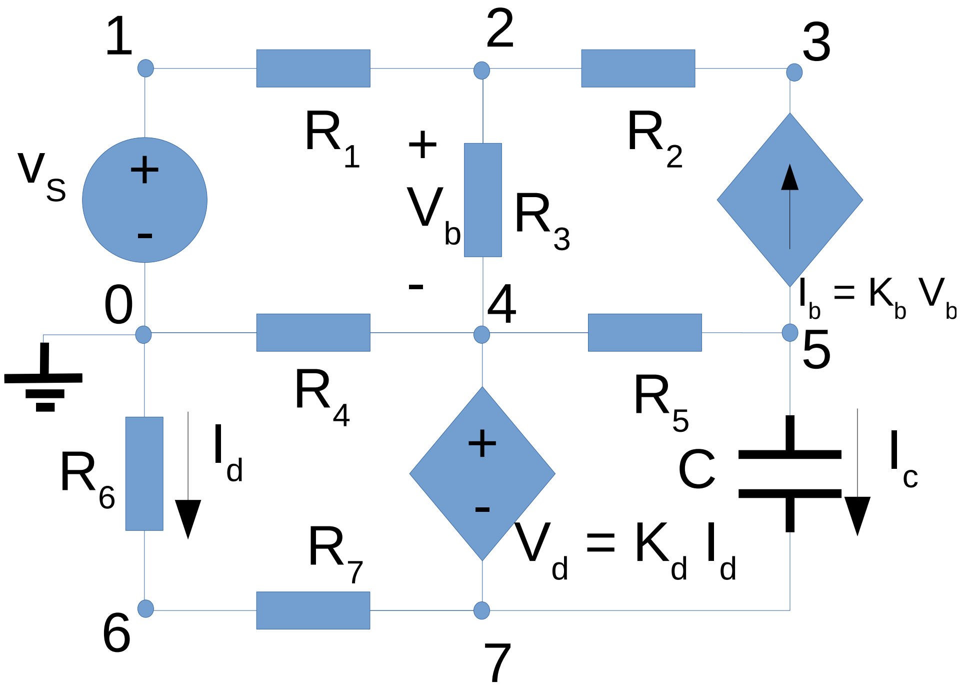


Figure 1: Circuit in study.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, in terms of each branch's current and voltage. The current flows considered in the analysis are the following: R_1 from node 1 to 2; R_2 from node 2 to 3; R_3 from node 2 to 4; R_4 from node 4 to 0; R_5 from node 4 to 5; R_6 from node 0 to 6; R_7 from node 6 to 7. Also, the potencial in Node 0 was considered to be 0V (ground).

2.1 Nodal Analysis for t¡0

In this first step, we use Nodal Analysis to determine the voltage in each node as well as the current in every branch of the circuit, when t¡0.

This method is based on Kirchhoff's Current Law (KCL), and firstly consists in deriving equations for the current flow in nodes not connected to voltage sources, followed by writing aditional equations in nodes related to voltage sources. The node identification can be seen on Figure **??**. Moreover, the equations relative to nodes 0, 0, 2, 3 and 5, respectively, are

$$(V_0 - V_4)/r_4 + (V_0 - V_6)/R_6 + (V_1 - V_2)/R_1 = 0 \tag{1}$$

$$(V_6 - V_7)/R_7 + (V_0 - V_4)/R_4 + (V_1 - V_2)/R_1 = 0 \tag{2}$$

$$(V_2 - V_1)/R_1 + (V_2 - V_4)/R_3 + (V_2 - V_3)/R_2 = 0 \tag{3}$$

$$(V_3 - V_2)/R_2 = K_b(V_2 - V_4) \tag{4}$$

$$(V_4 - V_5)/R_5 = K_b(V_2 - V_4) \tag{5}$$

The aditional equations for the method are

$$V_4 - V_7 = K_d(V_0 - V_6)/R_6 \tag{6}$$

$$V_0 = 0 \tag{7}$$

$$V_1 = V_s \tag{8}$$

Name	Value [mA or V]
I_c	0
I_b	-2.65898e-04
I_{R1}	2.536488e-04
I_{R2}	2.658976e-04
I_{R3}	-1.22488e-05
I_{R4}	1.205310e-03
I_{R5}	-2.65898e-04
I_{R6}	9.516615e-04
I_{R7}	9.516615e-04
V_1	5.054819
V_2	4.793705
V_3	4.258198
V_4	4.831047
V_5	5.668298
V_6	-1.93423
V_7	-2.90523
V_8	-1.93423

Table 1: Nodal Analysis' variable values for t¡0, where I_j is expressed in milliampere and V_j is expressed in Volt.

2.2 Equivalent resistance R_{eq}

In this step, we use the previously computed values to calculate a voltage defined as V_x = V_5 - V_7 (voltages in nodes 6 and 8, respectively). Note that this is the voltage drop at the capacitor's terminals, which will be needed for further analysis of the circuit. Firstly, we make V_s = 0. Then, by running nodal analysis again, we can obtain the current I_x flowing through V_x. With both these values computed, we can now obtain the value of the equivalent resistance R_eq as seen be the capacitor, which is given by R_eq = V_x/I_x. Finally, we can also compute the time constant τ given by τ = R_eqC. We considered the equations **??** and the following two:

$$V_5 - V_7 = V_x \tag{9}$$

$$V_1 = V_s \tag{10}$$

Name	Value [mA or V]
I_b	0.000000
I_{R1}	0.000000
I_{R2}	0.000000
I_{R3}	0.000000
I_{R4}	0.000000
I_{R5}	-2.72282e-03
I_{R6}	-0.000000
I_{R7}	0.000000
V_1	0.000000
V_2	0.000000
V_3	0.000000
V_4	0.000000
V_5	8.573530
V_6	0.000000
V_7	0.000000
V_8	0.000000
V_X	8.573530
I_X	-2.72282e-03
R_{eq}	3.148774e+03
τ	3.239206e-03

Table 2: Equivalent resistance R_{eq} and time constant τ

2.3 Natural Solution

Now, using the previous results we can compute the natural solution v_{5n}(t) from 0 to 20ms in time, as requested. We know that the solution is of the general form

$$Ae^{-t/\tau}. \tag{11}$$

Now, we only need an initial condition to find the constant ”A” which is going to be the value of V_x, the capacitor’s voltage. A plot of the solution in t∈[0;20]ms is shown in

$$V_5 = V_c = V_x e^{t/\tau} \tag{12}$$

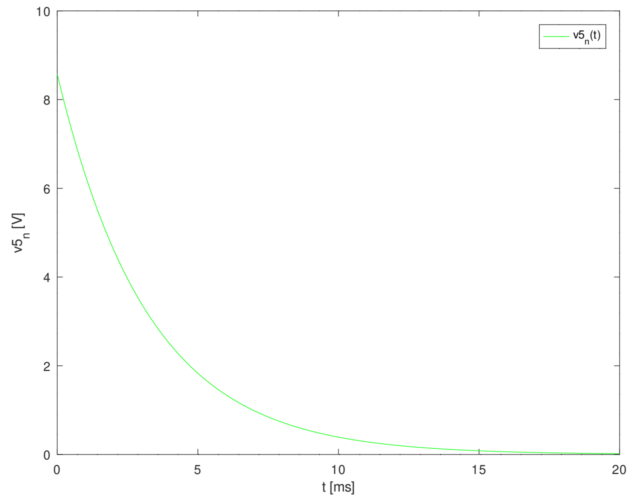


Figure 2: Natural solution V5

2.4 Forced solution

In this step, we use a phasor voltage source V_S and replace the capacitor with its impedance Z_C. Then we use the nodal method again in order to determine the phasor voltages in every node, and register the data in

The goal is to obtain the forced solution v_{5f}(t) in t∈[0;20]ms for a frequency f = 1kHz.

The nodal analysis is equivalent to the previous ones in terms of equations, but instead of voltages we use phasors:

$$(\tilde{V}_2 - \tilde{V}_1)/R_1 + (\tilde{V}_2 - \tilde{V}_4)/R_3 + (\tilde{V}_2 - \tilde{V}_3)/R_2 = 0 \tag{13}$$

$$(\tilde{V}_0 - \tilde{V}_4)/R_4 + (\tilde{V}_0 - \tilde{V}_6)/R_6 + (\tilde{V}_1 - \tilde{V}_2)/R_1 = 0 \tag{14}$$

$$(\tilde{V}_6 - \tilde{V}_7)/R_7 + (\tilde{V}_0 - \tilde{V}_4)/R_4 + (\tilde{V}_1 - \tilde{V}_2)/R_1 = 0 \tag{15}$$

$$(\tilde{V}_3 - \tilde{V}_2)/R_2 = K_b(\tilde{V}_2 - \tilde{V}_4) \tag{16}$$

$$(\tilde{V}_5 - \tilde{V}_7)/Z_c + (\tilde{V}_5 - \tilde{V}_4)/R_5 + K_b(\tilde{V}_2 - \tilde{V}_4) = 0, Z_c = 1/(jwc) \tag{17}$$

$$\tilde{V}_4 - \tilde{V}_7 = K_d(\tilde{V}_0 - \tilde{V}_6)/R_6 \tag{18}$$

$$\tilde{V}_1 = \tilde{V}_s \tag{19}$$

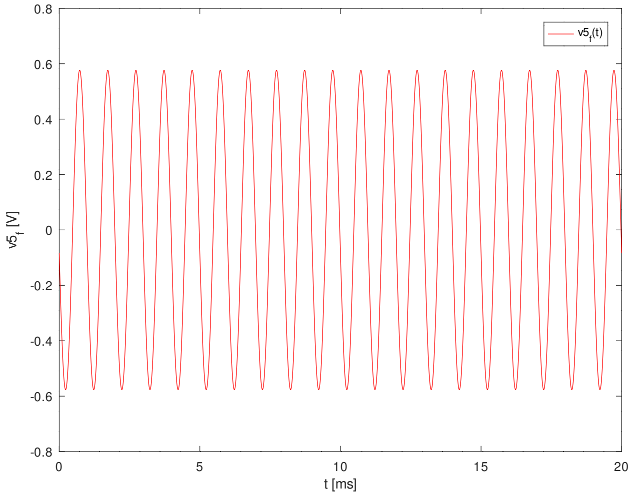


Figure 3: Forced Solution V5

Name	Value [VIVA]
V_1	1.000000
V_2	0.948344
V_3	0.842404
V_4	0.955731
V_5	0.576684
V_6	0.382650
V_7	0.574745
V_8	0.382650

Table 3: Sem legenda por agora

2.5 Final solution

Now we are able to describe the final solution v_5(t) after converting the forced solution to a real time function. The solution is the sum of the contributions of the natural and forced responses of the circuit:

$$V_5 = V_{5f} + V_{5n} \tag{20}$$

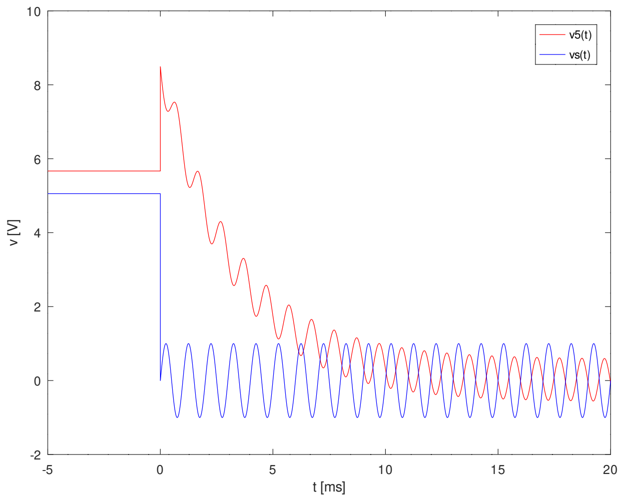


Figure 4: Final Solution V5 and Vs

2.6 Frequency response

In this final step we use the functions v_5(f) and v_7(f) (voltages of nodes 5 and 7, respectively, in terms of frequency, f) using a logarithmic scale for frequency (dB) and expressing the phase in degrees, with f∈[0.1;1]kHz.

Defining v_C(f) = v_5(f) - v_7(f), we can then plot different functions v_S(f), v_C(f) and v_6(f) as seen in 5 (amplitude) and in 6 (phase, in degrees).

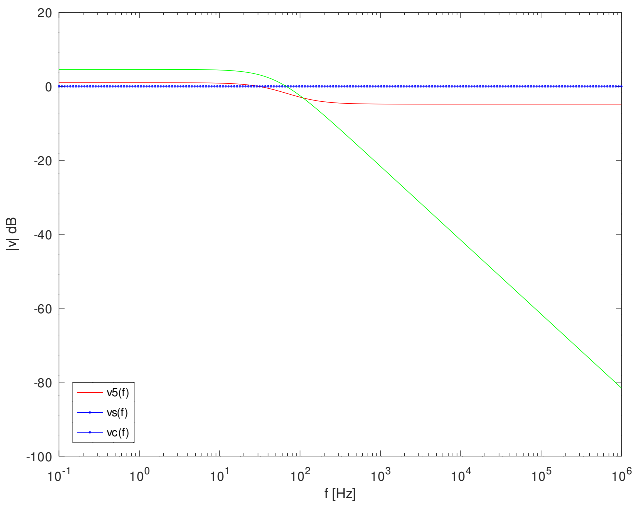


Figure 5: Frequency responses

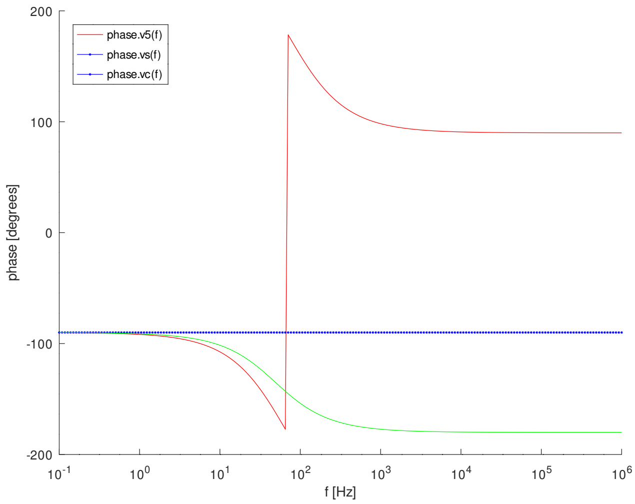


Figure 6: Phase in degrees

3 Simulation Analysis

3.1 Operating point for t ¡ 0

Table 4 shows the simulated operating point results for the circuit analysis when t ¡ 0. Once again, current flows are the ones referred in 2 and node 0 is considered to have 0V potential.

A new voltage source V_{aux} with voltage 0V (so it doesn't affect the circuit) was added to the circuit between components R_6 and R_7 so that Ngspice could simulate and calculate the current value in that branch. This current is needed since the voltage source v_S(t) depends on its value and as expected, this current's value is the same as the one that flows through R_6 and R_7. Adding a new voltage source led to the creation of node 8 between R_6 and V_{aux}. Also, as predicted, voltage V_8 is the same as voltage V_6 since the voltage in V_{aux} is 0V.

Name	Value [A or V]
@c[i]	0.000000e+00
@gb[i]	-2.65898e-04
@r1[i]	2.536488e-04
@r2[i]	2.658976e-04
@r3[i]	-1.22488e-05
@r4[i]	1.205310e-03
@r5[i]	-2.65898e-04
@r6[i]	9.516615e-04
@r7[i]	9.516615e-04
v(1)	5.054819e+00
v(2)	4.793705e+00
v(3)	4.258198e+00
v(4)	4.831047e+00
v(5)	5.668298e+00
v(6)	-1.93423e+00
v(7)	-2.90523e+00
v(8)	-1.93423e+00

Table 4: Operating point for t ¡ 0. A variable proceeded by [i] is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

3.2 Operating point for v_S(0) = 0

Table 5shows the simulated operating point values for the circuit when t = 0 and v_S(0) = 0V.

Name	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]	-2.72282e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
v(1)	0.000000e+00
v(2)	0.000000e+00
v(3)	0.000000e+00
v(4)	0.000000e+00
v(5)	8.573530e+00
v(6)	0.000000e+00
v(7)	0.000000e+00
v(8)	0.000000e+00
Ix	-2.72282e-03
Vx	8.573530e+00
Req	3.148774e+03

Table 5: Operating point for v_S(0) = 0. A variable proceeded by [i] is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

3.3 Natural response

In this step, the natural response of the circuit with a given boundary condition is simulated using Ngspice's transient analysis function. The plot for t∈[0;20]ms can be seen in **??**.

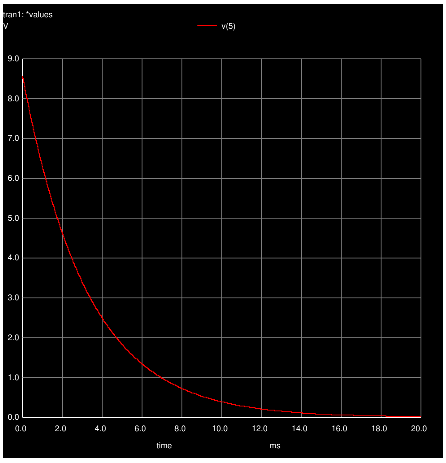


Figure 7: Natural solution V5

3.4 Natural and forced responses for f = 1kHz and given v_S(t)

The plot for v_S(t) and the circuit's response can be seen in 8.

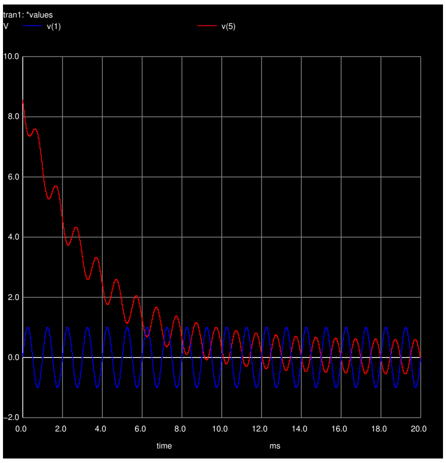


Figure 8: Stimulus (blue) and response (red) on node 5

3.5 Frequency response

For f∈[0.1Hz;1MHz] the frequency response in node 5 is simulated. The functions v_S(f) and v_5(f) are plotted in a logarithmic scale (dB) in 9, and the phases can be seen in degrees in 10.

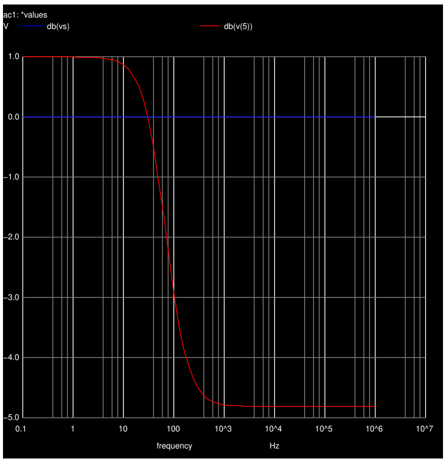


Figure 9: v_S(f), in blue, and v_5(f), in red

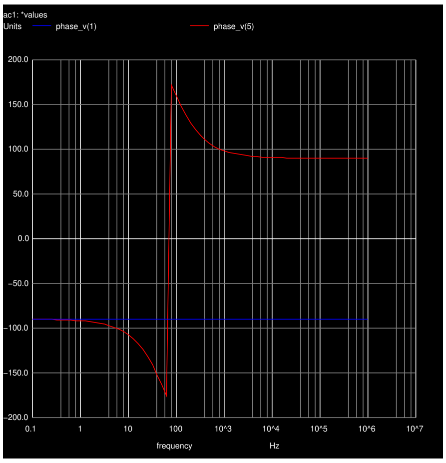


Figure 10: Phase of v_S(f), in blue, and phase of v_5(f), in red

4 comparison

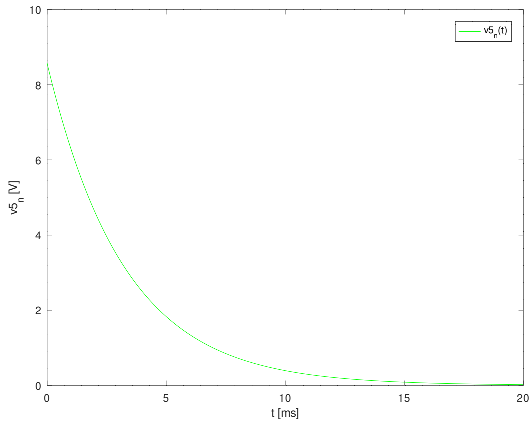
4.1 t¡0

Name	Value [mA or V]	Name	Value [A or V]
I_c	0	@c[i]	0.000000e+00
I_b	-2.65898e-04	@gb[i]	-2.65898e-04
I_{R1}	2.536488e-04	@r1[i]	2.536488e-04
I_{R2}	2.658976e-04	@r2[i]	2.658976e-04
I_{R3}	-1.22488e-05	@r3[i]	-1.22488e-05
I_{R4}	1.205310e-03	@r4[i]	1.205310e-03
I_{R5}	-2.65898e-04	@r5[i]	-2.65898e-04
I_{R6}	9.516615e-04	@r6[i]	9.516615e-04
I_{R7}	9.516615e-04	@r7[i]	9.516615e-04
V_1	5.054819	v(1)	5.054819e+00
V_2	4.793705	v(2)	4.793705e+00
V_3	4.258198	v(3)	4.258198e+00
V_4	4.831047	v(4)	4.831047e+00
V_5	5.668298	v(5)	5.668298e+00
V_6	-1.93423	v(6)	-1.93423e+00
V_7	-2.90523	v(7)	-2.90523e+00
V_8	-1.93423	v(8)	-1.93423e+00

Table 6: Comparison 1

4.2 Equivalent resistance

4.3 Natural solutin V_5



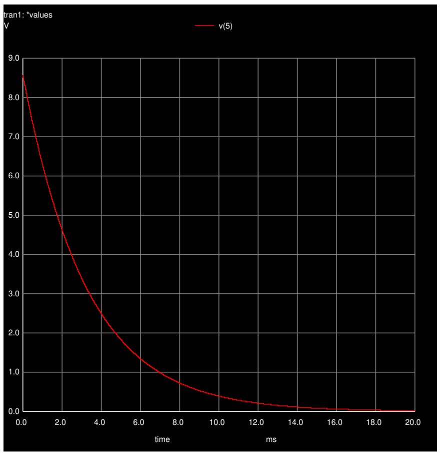


Figure 11: Natural solution V5

5 Conclusion

Comparing the results given by Nodal and Mesh analysis, it can be observed that both methods achieved the same values. Also, the results given by the Ngspice simulation are the same as those of the methods stated above. All in all, the analysis of the given circuit using the stated methods was achieved successfully. Nodal and mesh methods were performed both theoretically using the Octave maths tool and by circuit simulation using the Ngspice software. Given the fact that the circuit only has linear components and it is a simple circuit to analyse, Ngspice probably used the same models as we did in the theoretical analysis, so it makes sense that no discrepancies were detected. However, if we had been given a circuit with more complex components, the theoretical and simulation models could not match as precisely as they did in this laboratory assignment.