

# 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

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>Theoretical Analysis</b>	<b>2</b>
2.1	Nodal Analysis for $t \geq 0$	2
2.2	Equivalent resistance $R_{eq}$	3
2.3	Natural Solution	4
2.4	Forced solution	5
2.5	Final solution	6
2.6	Frequency response	7
<b>3</b>	<b>Simulation Analysis</b>	<b>8</b>
3.1	Operating point for $t \geq 0$	8
3.2	Operating point for $v_S(0) = 0$	9
3.3	Natural response	9
3.4	Natural and forced responses for $f = 1\text{kHz}$ and given $v_S(t)$	10
3.5	Frequency response	11
<b>4</b>	<b>comparison</b>	<b>13</b>
4.1	$t \geq 0$	13
4.2	Equivalent resistance	13
4.3	Natural solution $V_5$	13
<b>5</b>	<b>Conclusion</b>	<b>14</b>

## 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.

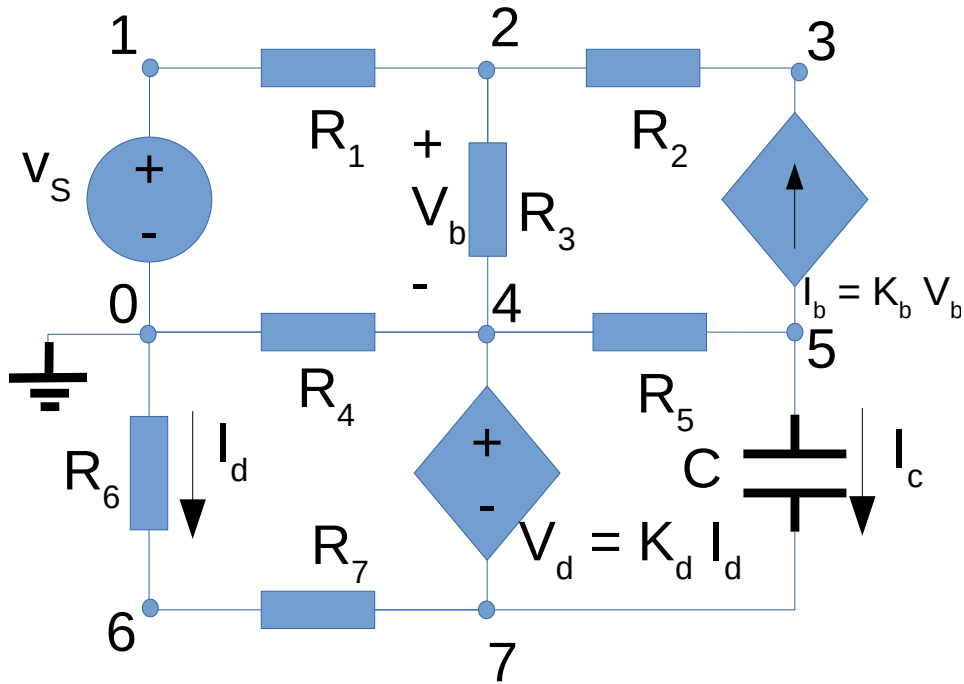


Figure 1: Circuit in study.

## 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 potential in Node 0 was considered to be 0V (ground).

### 2.1 Nodal Analysis for $t \geq 0$

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  $t \geq 0$ .

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 additional equations in nodes related to voltage sources. The node identification can be seen on Figure ???. Moreover, the equations relative to nodes 0, 2, 3 and 5, respectively, are

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

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

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

$$(V_3 - V_2)/R_2 = K_b(V_2 - V_4) \quad (4)$$

$$(V_4 - V_5)/R_5 = K_b(V_2 - V_4) \quad (5)$$

The additional equations for the method are

$$V_4 - V_7 = K_d(V_0 - V_6)/R_6 \quad (6)$$

$$V_0 = 0 \quad (7)$$

$$V_1 = V_s \quad (8)$$

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 milliamperes 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  $\tau$  given by  $\tau = R_{eq}C$ . We considered the equations ?? and the following two:

$$V_5 - V_7 = V_x \quad (9)$$

$$V_1 = V_s \quad (10)$$

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
$\tau$	3.239206e-03

Table 2: Equivalent resistance  $R_{eq}$  and time constant  $\tau$

### 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

$$Ae^{-t/\tau}. \quad (11)$$

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  $t \in [0;20]$ ms is shown in

$$V_5 = V_c = V_x e^{t/\tau} \quad (12)$$

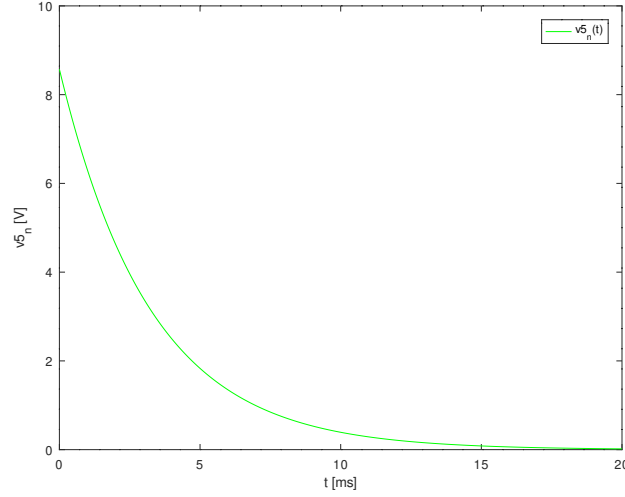


Figure 2: Natural solution V5

## 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  $t \in [0; 20]$  ms for a frequency  $f = 1$  kHz.

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

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

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

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

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

$$(\tilde{V}_5 - \tilde{V}_7)/Z_c + (\tilde{V}_5 - \tilde{V}_4)/R_5 + K_b(\tilde{V}_2 - \tilde{V}_4) = 0, Z_c = 1/(j\omega c) \quad (17)$$

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

$$\tilde{V}_1 = \tilde{V}_s \quad (19)$$

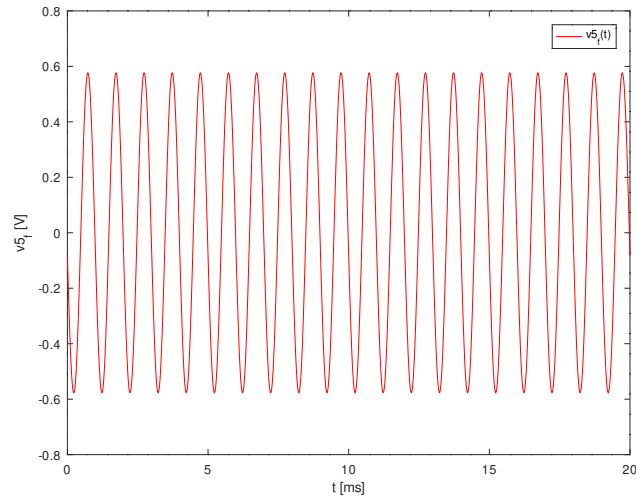


Figure 3: Forced Solution V5

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:

$$V_5 = V_{5f} + V_{5n} \quad (20)$$

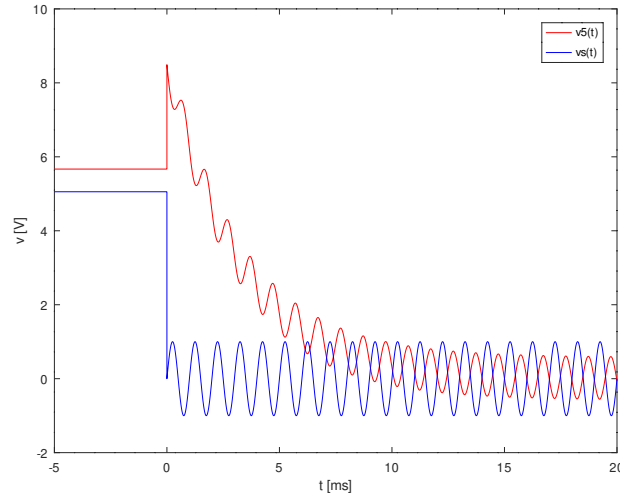


Figure 4: Final Solution V5 and Vs

## 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  $f \in [0.1; 1]$  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).

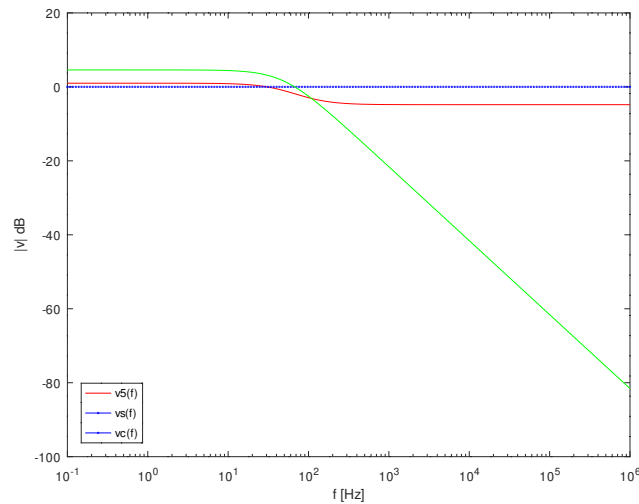


Figure 5: Frequency responses

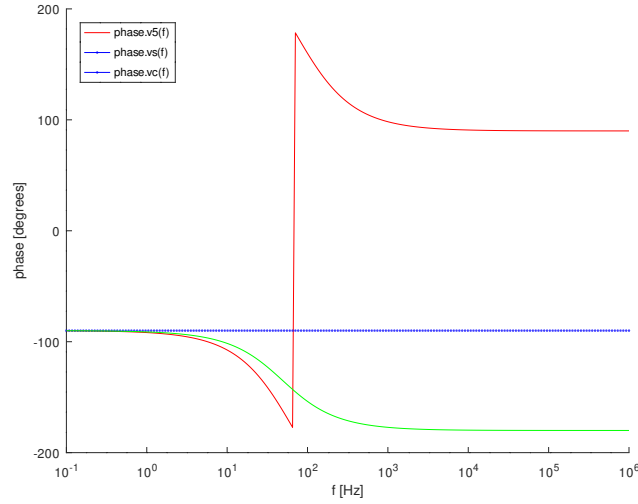


Figure 6: Phase in degrees

### 3 Simulation Analysis

#### 3.1 Operating point for $t \downarrow 0$

Table 4 shows the simulated operating point results for the circuit analysis when  $t \downarrow 0$ . 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  $t \downarrow 0$ . 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 5 shows 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 preceded 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  $t \in [0; 20]ms$  can be seen in ??.

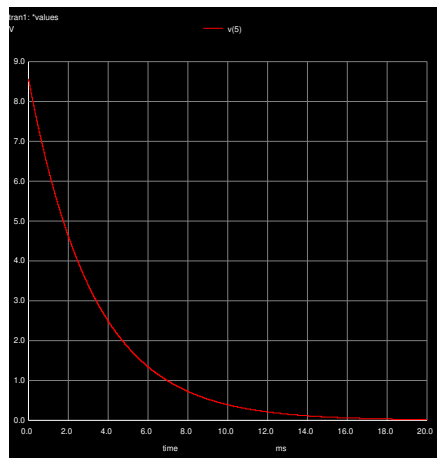


Figure 7: Natural solution V5

### 3.4 Natural and forced responses for $f = 1\text{kHz}$ and given $v_S(t)$

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

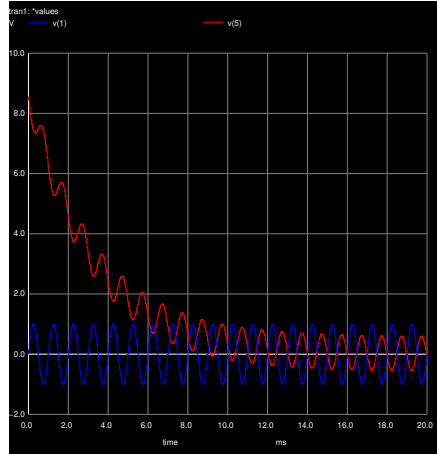


Figure 8: Stimulus (blue) and response (red) on node 5

### 3.5 Frequency response

For  $f \in [0.1\text{Hz}; 1\text{MHz}]$  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.

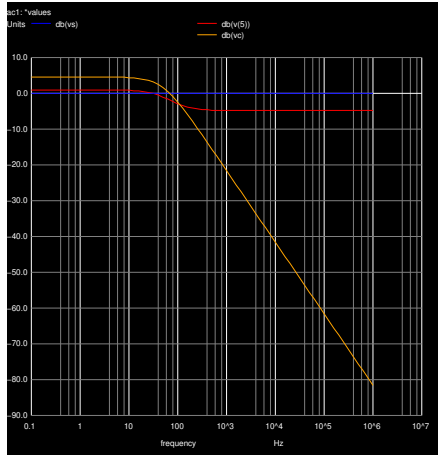


Figure 9:  $v_S(f)$ , in blue, and  $v_5(f)$ , in red

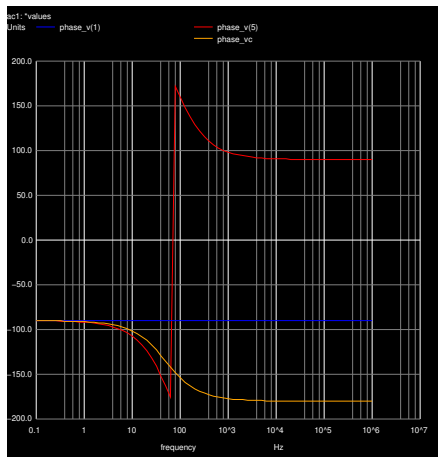


Figure 10: Phase of  $v_S(f)$ , in blue, and phase of  $v_5(f)$ , in red

## 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 $V_5$

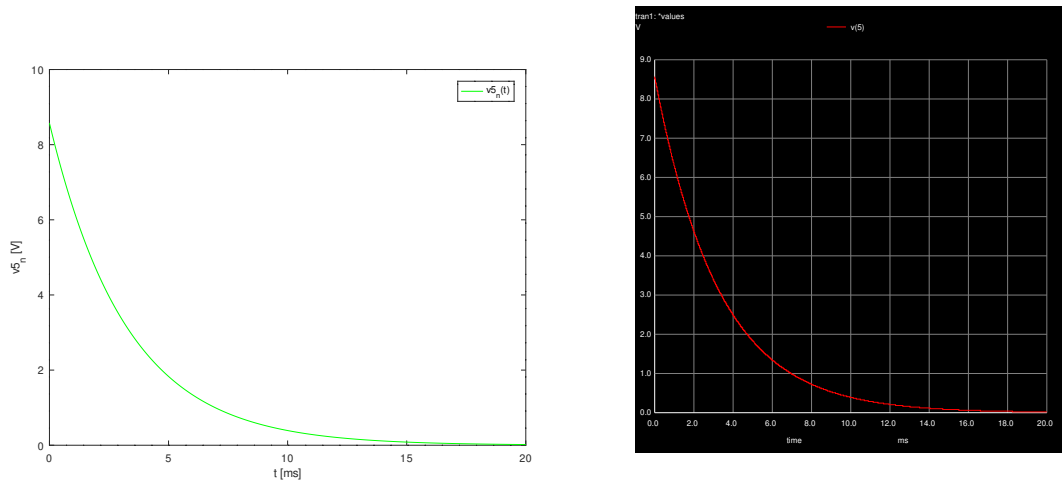


Figure 11: Natural solution V5

## 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.