

# 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>2</b>
<b>2</b>	<b>Theoretical Analysis</b>	<b>2</b>
2.1	Nodal Analysis for $t < 0$	2
2.2	Equivalent resistance $R_{eq}$	4
2.3	Natural Solution	5
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 $t < 0$	8
3.2	Operating point for $v_S(0) = 0$	9
3.3	Natural response	10
3.4	Natural and forced responses for $f = 1\text{kHz}$ and given $v_S(t)$	11
3.5	Frequency response	12
<b>4</b>	<b>Comparison between tables from Ngspice and Octave</b>	<b>15</b>
4.1	$t < 0$	15
4.2	Equivalent resistance	15
<b>5</b>	<b>Conclusion</b>	<b>16</b>
<b>6</b>	<b>Attachments</b>	<b>16</b>
6.1	Natural solution $v_5$	16
6.2	Final solution	17
6.3	Frequency response	17

# 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 voltage dependent current source  $I_b$  and a linearly current 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 in Section 4. Conclusions of this study can be found in Section 5.

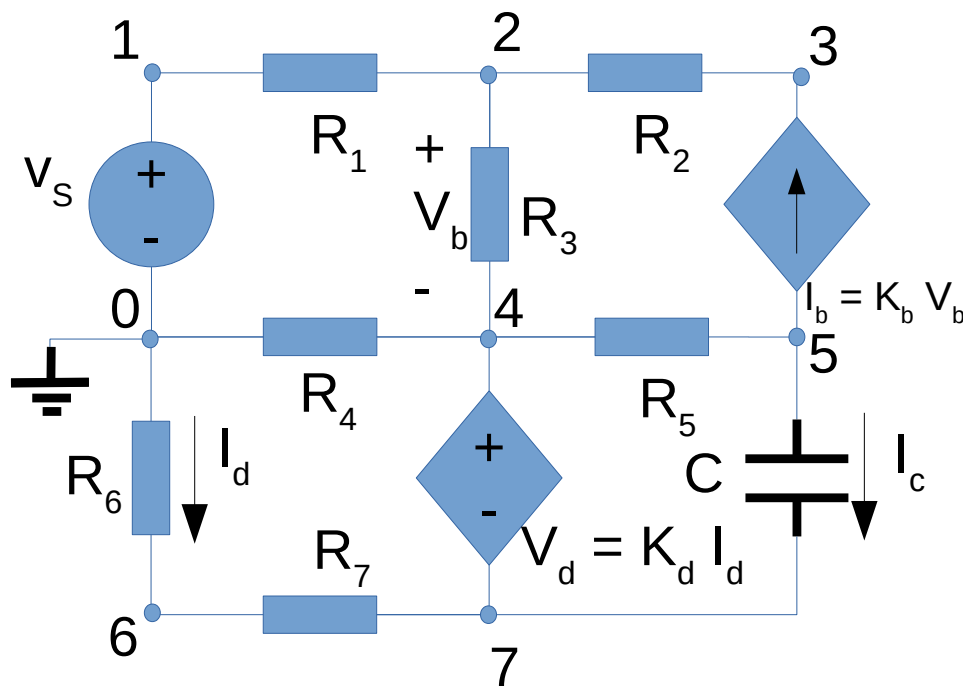


Figure 1: Circuit in study.

## 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). Node 8 is an auxiliary node which is going to be needed in the simulation.

### 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  $t < 0$ . We considered the voltage source  $v_s$  to

be connected for a long time so the voltages are already constant. Since  $i_C = C \frac{dv_C}{dt}$  we can assume  $i_C = 0$ .

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 1. Moreover, the equations relative to nodes 0, 0, 2, 3 and 5, respectively, are

$$(V_0 - V_4)/R_4 + (V_0 - V_6)/R_6 + (V_1 - V_2)/R_1 = 0 \quad (1)$$

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

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

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

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

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 [A 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 Ampere 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 5 and 7, respectively - which correspond to nodes 6 and 8 from the laboratory paper). 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 by 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 used this procedure because to obtain the equivalent resistor we need to turn off the independent sources and due to the presence of dependent sources we replaced the capacitor with a independent voltage source with value  $V_x$  to ensure the continuity of the capacitor's voltage function.

Also, this corresponds to the initial calculations for computing the natural solution.

Defining  $V_x = V_5 - V_7$  makes sense because the voltage drop in the capacitor is a continuous function, so  $V_x(t=0^-) = V_x(t=0)$ . We considered the equations 1, 2, 3, 4, 6, 7, 8 and the following two:

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

$$I_X = (V_4 - V_5)/R_5 - K_b(V_2 - V_4) \quad (10)$$

Name	Value [A or V or $\Omega$ or s]
$I_b$	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 for  $t \leq 0$ . A plot of the solution in  $t \in [0; 20]$ ms is shown in 2. Analysing the parameters of the circuit for  $t=0$  we note that  $V_7 = 0V$  and  $V_c$  decreases, which leads to

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

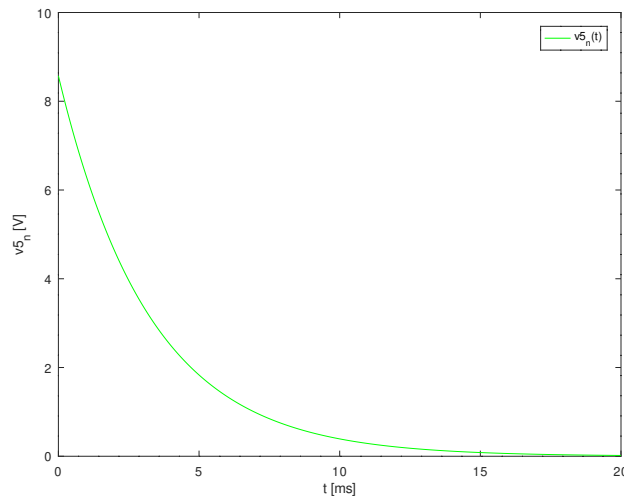


Figure 2: Natural solution for  $v_5$

## 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 3.

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

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

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

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

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

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

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

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

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

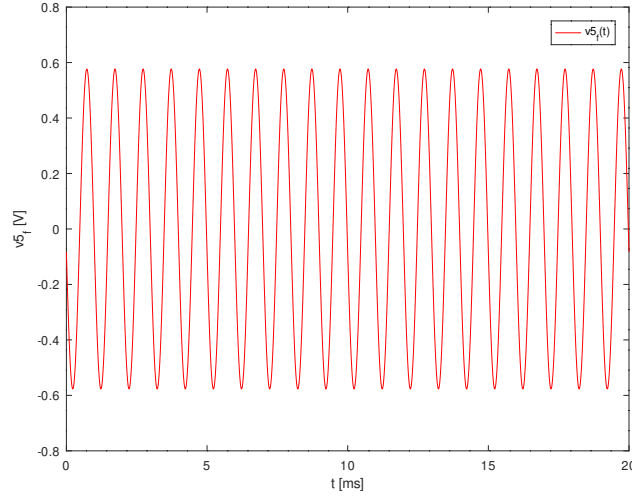


Figure 3: Forced Solution for  $v_5$

Name	Value [V]
$V_1$	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: Complex amplitudes in the nodes

## 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:

$$v_5 = v_{5f} + v_{5n} \quad (20)$$

The following plot displays  $v_5(t)$  and  $v_s(t)$  for  $t \in [-5; 20]$  ms. The values of  $t < 0$  were obtained in 2.1.

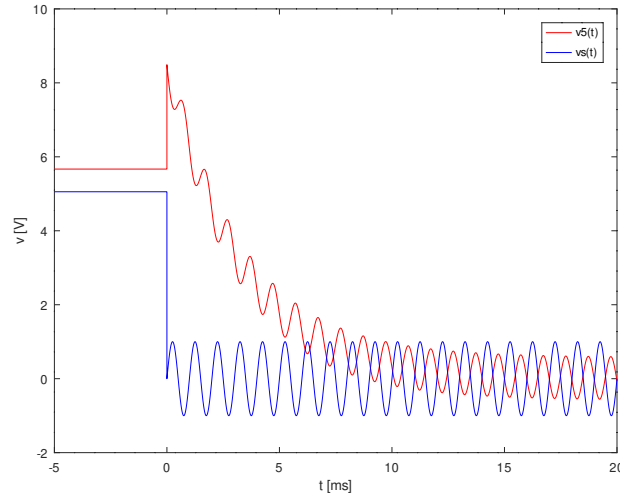


Figure 4: Final Solution for  $v_5$  and  $v_s$

## 2.6 Frequency response

In this final step we use the functions  $v_5(f)$  and  $v_7(f)$  (voltages on 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).

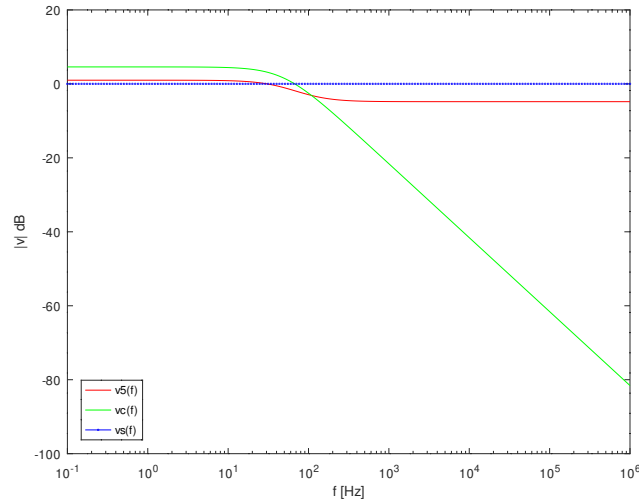


Figure 5: Magnitude in dB  $v_5(f)$  (red),  $v_s(f)$  (blue),  $v_c(f)$  (green)

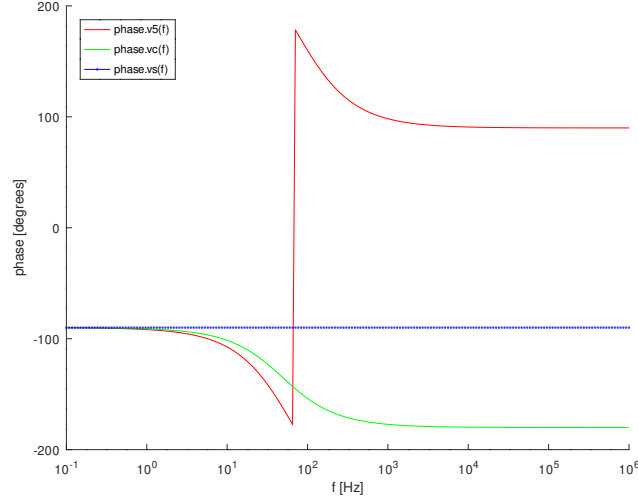


Figure 6: Phase in degrees  $v_5(f)$  (red),  $v_s(f)$  (blue),  $v_c(f)$  (green)

### 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 section 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  $v_d$  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 5 shows the simulated operating point values for the circuit when  $t=0$  and  $v_S(0) = 0V$ . The reason why this step is needed is the same as the one referred in 2.2. Also, in order to perform the transient analysis, we replaced the capacitor with an independent voltage source from node 5 to 7 with value  $V_x$ .

Name	Value [A or V or $\Omega$ ]
@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], and  $I_x$  is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt, except  $R_{eq}$  that is in Ohm.

### 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 boundary condition is  $V_5$  and  $V_7$  obtained in the previous subsection when  $t=0$ . After comparing the results we concluded that the values obtained in Ngspice matched the ones from Octave, so we used the Octave values directly in Ngspice. The plot for  $t \in [0;20]$ ms can be seen in 7.

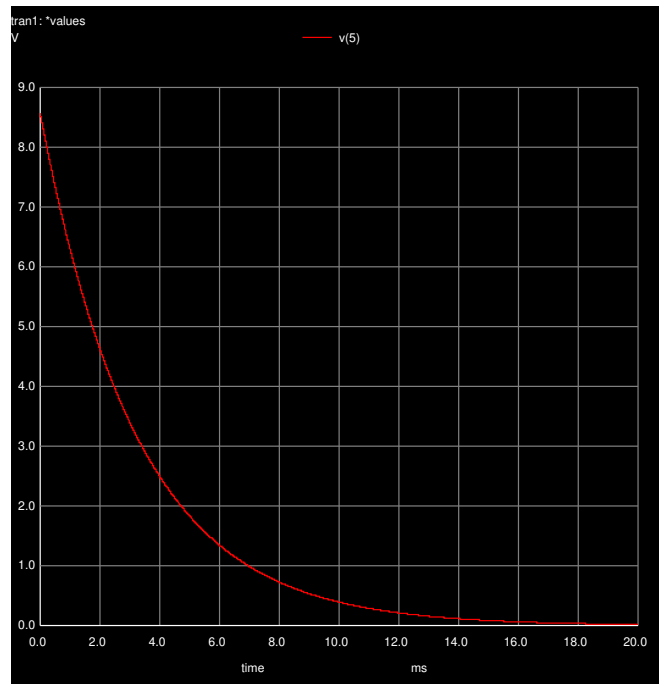


Figure 7: Natural solution for  $v_5$

### 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.

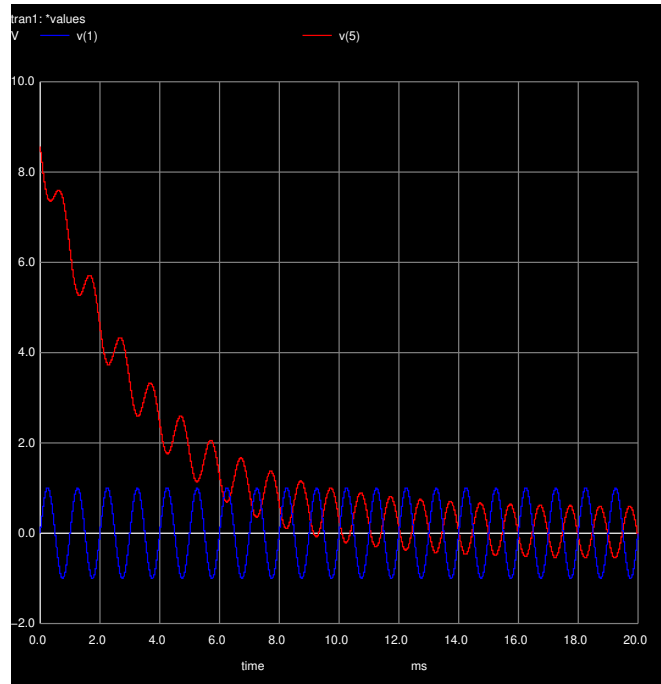


Figure 8: Stimulus (blue) and response (red) on node 5

### 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.

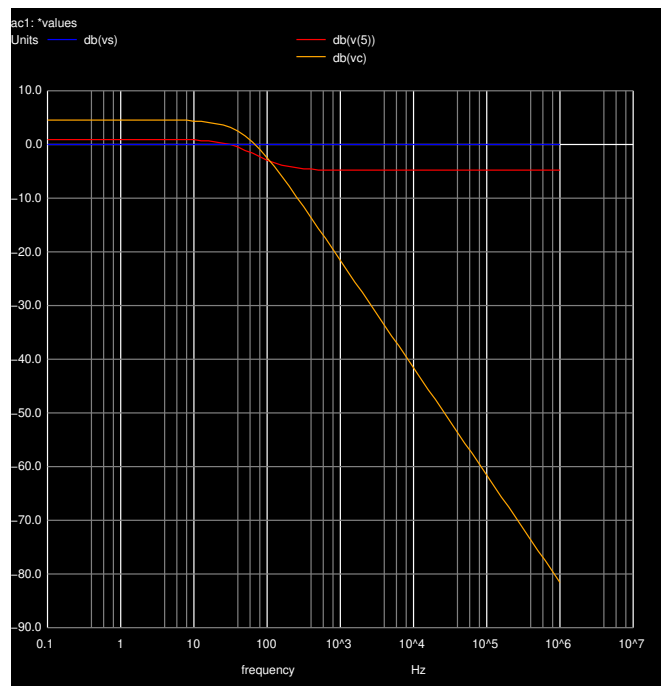


Figure 9:  $v_S(f)$ , in blue,  $v_5(f)$ , in red, and  $v_C(f)$ , in yellow

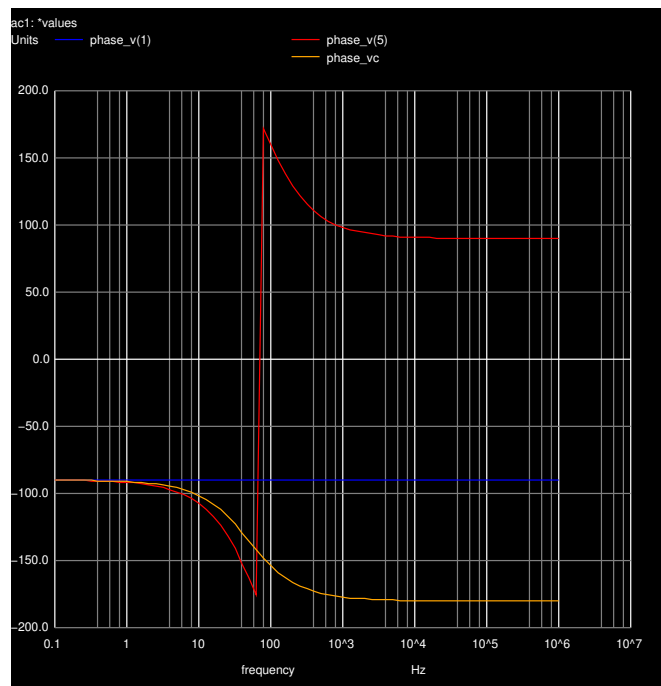


Figure 10: Phase of  $v_S(f)$ , in blue,  $v_5(f)$ , in red, and  $v_C(f)$ , in yellow

## 4 Comparison between tables from Ngspice and Octave

This section aims to confirm the values between the two analysis. The graphics obtained in both analysis can be viewed side-by-side in section 6.

### 4.1 $t < 0$

Name	Value [A 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

Here we can observe that there are no significant discrepancies between the theoretical predictions and the simulation results.

### 4.2 Equivalent resistance

Name	Value [A or V or $\Omega$ or s]	Name	Value [A or V or $\Omega$ ]
$I_b$	0.000000	@gb[i]	0.000000e+00
$I_{R1}$	0.000000	@r1[i]	0.000000e+00
$I_{R2}$	0.000000	@r2[i]	0.000000e+00
$I_{R3}$	0.000000	@r3[i]	0.000000e+00
$I_{R4}$	0.000000	@r4[i]	0.000000e+00
$I_{R5}$	-2.72282e-03	@r5[i]	-2.72282e-03
$I_{R6}$	-0.000000	@r6[i]	0.000000e+00
$I_{R7}$	0.000000	@r7[i]	0.000000e+00
$V_1$	0.000000	v(1)	0.000000e+00
$V_2$	0.000000	v(2)	0.000000e+00
$V_3$	0.000000	v(3)	0.000000e+00
$V_4$	0.000000	v(4)	0.000000e+00
$V_5$	8.573530	v(5)	8.573530e+00
$V_6$	0.000000	v(6)	0.000000e+00
$V_7$	0.000000	v(7)	0.000000e+00
$V_8$	0.000000	v(8)	0.000000e+00
$V_X$	8.573530	Ix	-2.72282e-03
$I_X$	-2.72282e-03	Vx	8.573530e+00
$R_{eq}$	3.148774e+03	Req	3.148774e+03
$\tau$	3.239206e-03		

Table 7: Comparison 2

The predicted value for the equivalent resistance matches the result obtained in the Ngspice simulation.

## 5 Conclusion

Comparing the results given by Nodal analysis and the Ngspice simulation it can be observed that the theoretical predictions and the actual parameters of the circuit converge to the same values. All in all, the analysis of the given circuit following the suggested steps was achieved successfully, so we can say that the theoretical model makes a good representation of how RC circuits behave in reality.

## 6 Attachments

In this section, the plots can be viewed side-by-side as an extra comparison tool.

### 6.1 Natural solution $v_5$



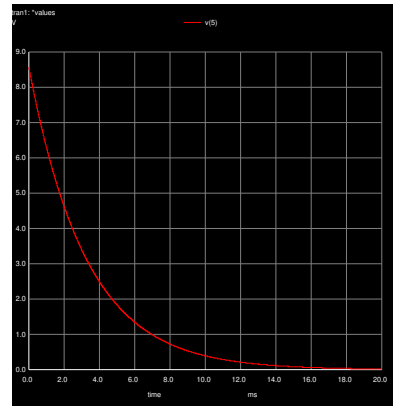
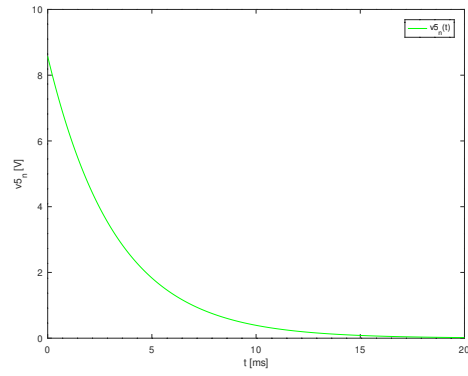


Figure 11: Natural solution  $v_5$

## 6.2 Final solution

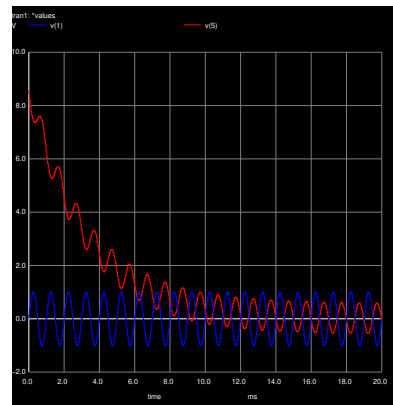
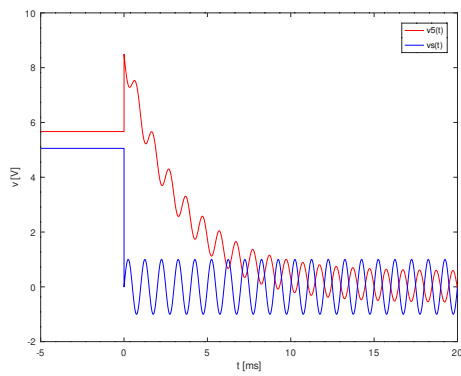


Figure 12: Final solution  $v_5$  and  $v_s$

## 6.3 Frequency response

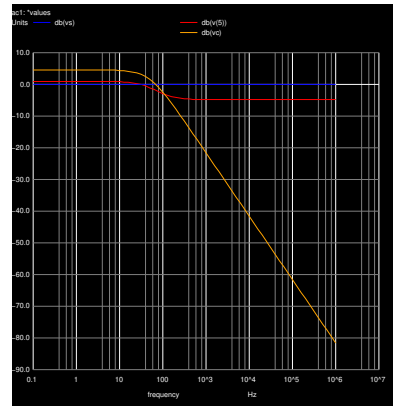
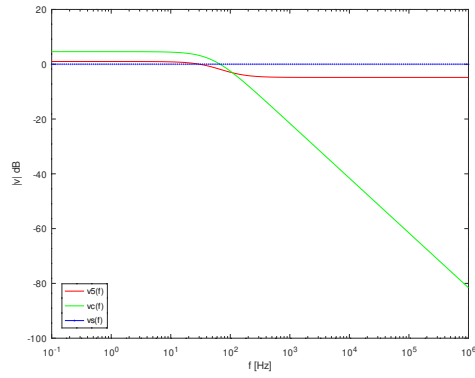


Figure 13: Magnitude in dB

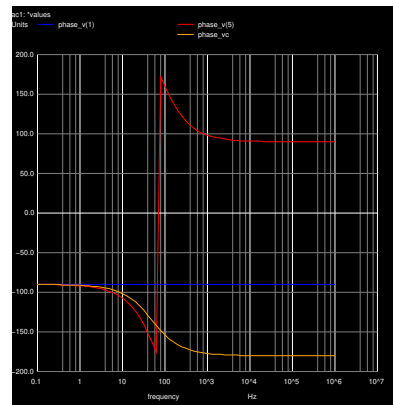
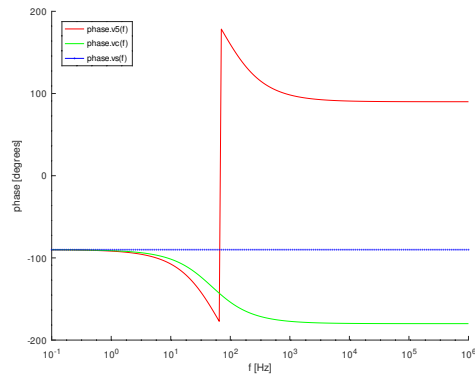


Figure 14: Phase in degrees