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 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.

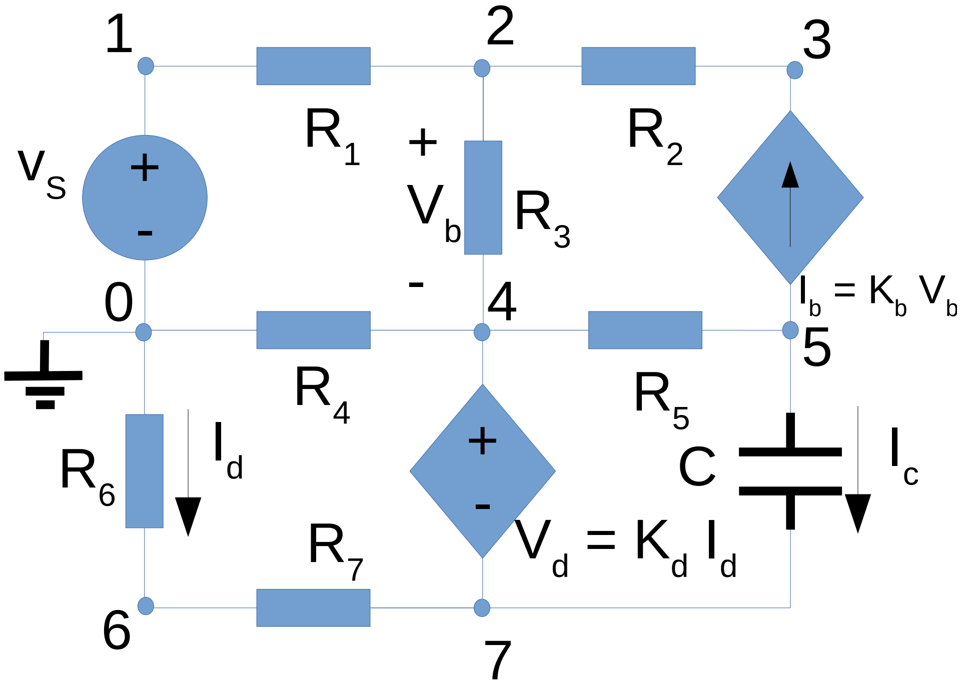


Figure 1: Circuit in study.

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 potencial 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.

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

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

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

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

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

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

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 [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 τ given by τ = R_{eq}C. 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 and the following two:

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

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

Name	Value [A or V or Ω 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
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 for t≤0. A plot of the solution in t∈[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

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

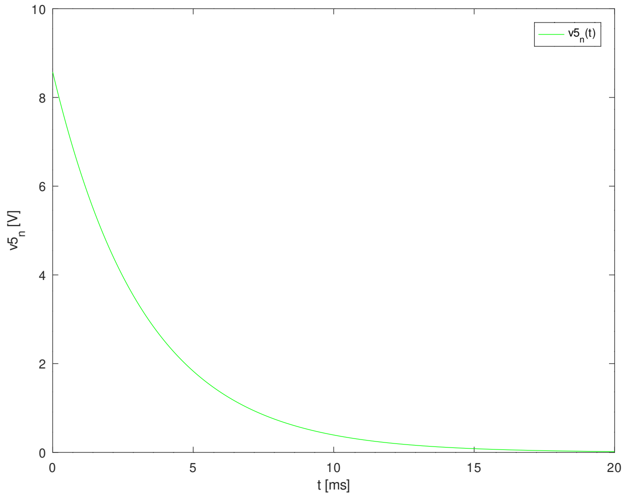


Figure 2: Natural solution for v_5

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∈[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:

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

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

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

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

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

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

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

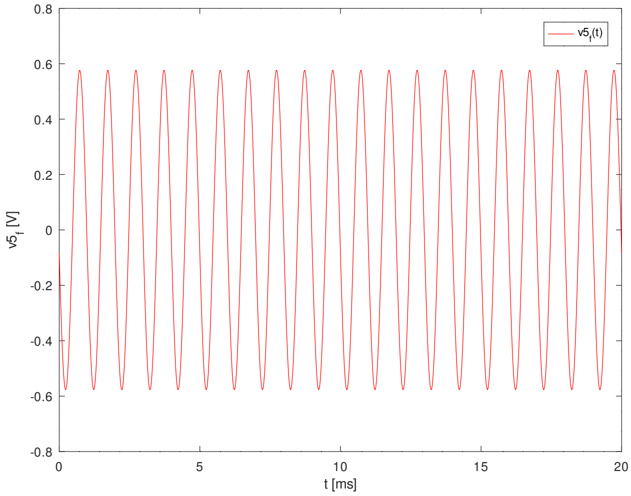


Figure 3: Forced Solution for v_5

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:

$$v_5 = v_{5f} + v_{5n} \tag{20}$$

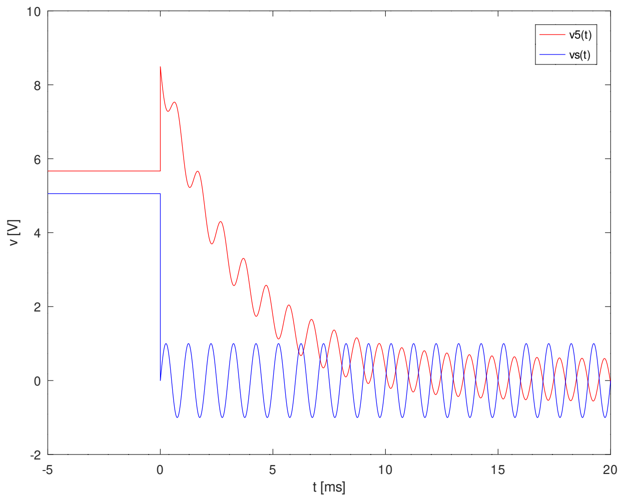


Figure 4: Final Solution for v_5 and v_s

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∈[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).

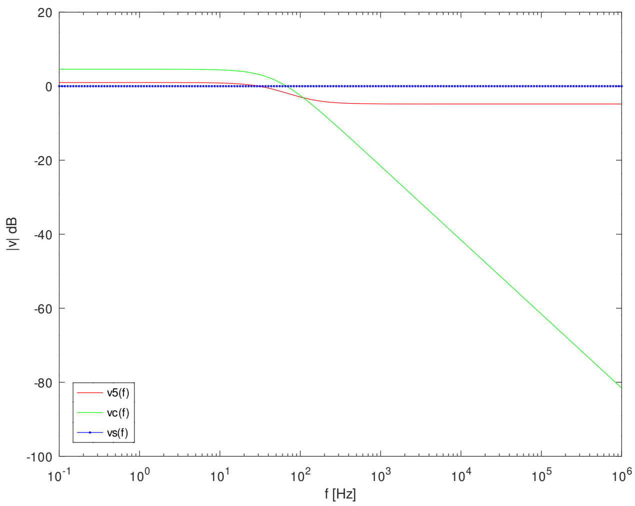


Figure 5: Magnitude in dB v_5(f) (red), v_s(f) (blue), v_c(f) (green)

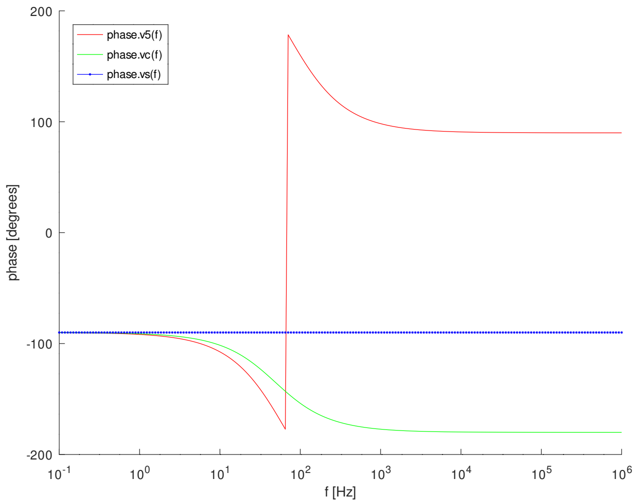


Figure 6: Phase in degrees v_5(f) (red), v_s(f) (blue), v_c(f) (green)

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_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 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, Ngspice needs to know V_5 and V_7 so that it can compute the capacitor's charge.

Name	Value [A or V or Ω]
@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 plot for t∈[0;20]ms can be seen in 7.

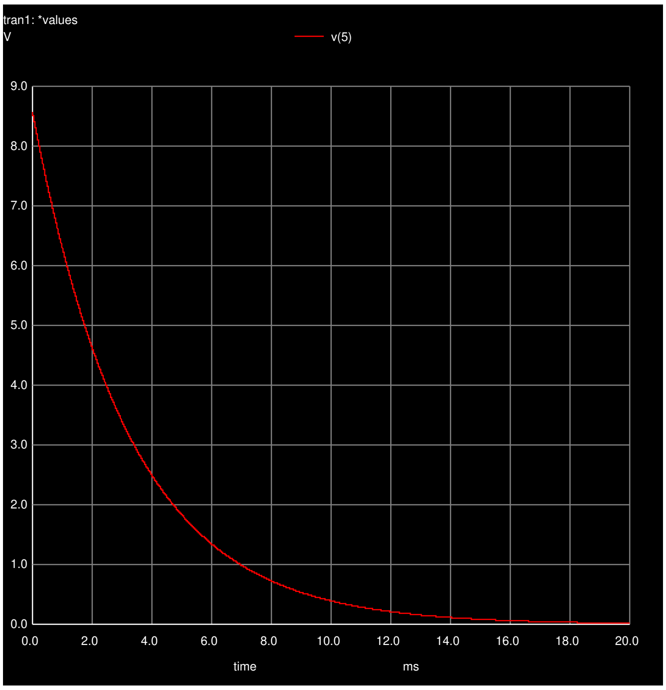


Figure 7: Natural solution for v_5

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.

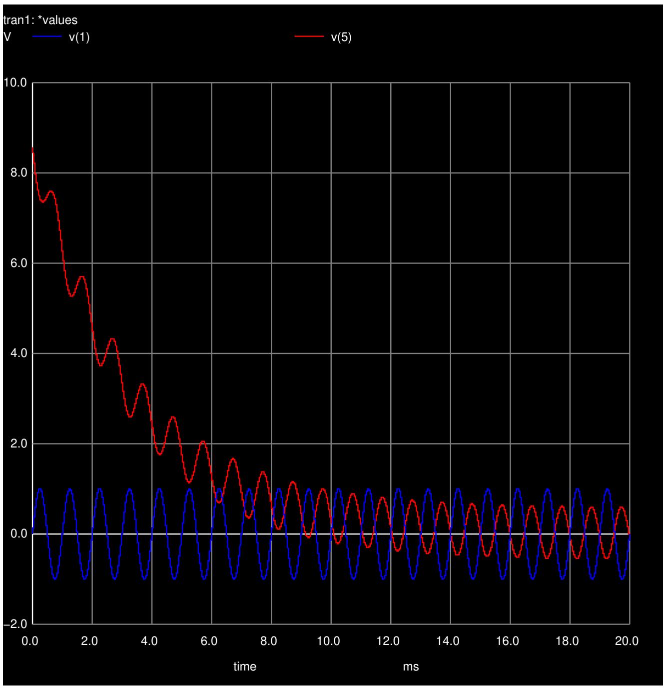


Figure 8: Stimulus (blue) and response (red) on node 5

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.

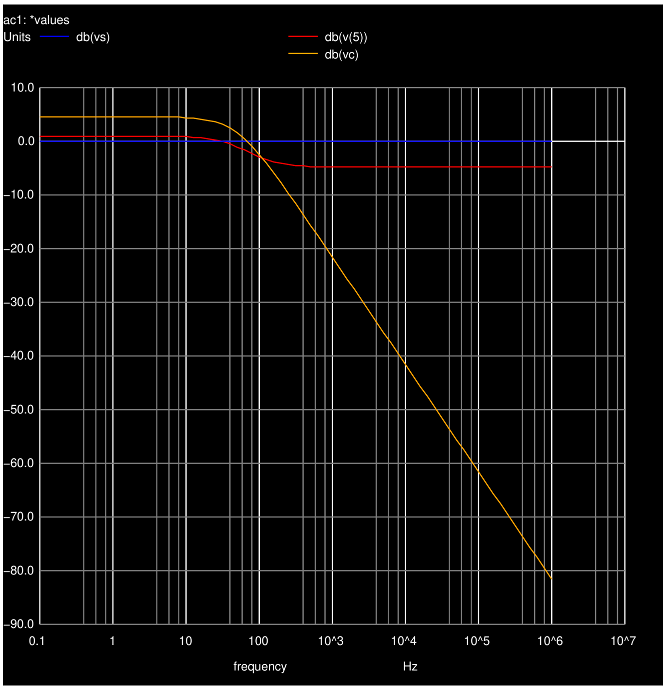


Figure 9: v_S(f), in blue, v_5(f), in red, and v_C(f), in yellow

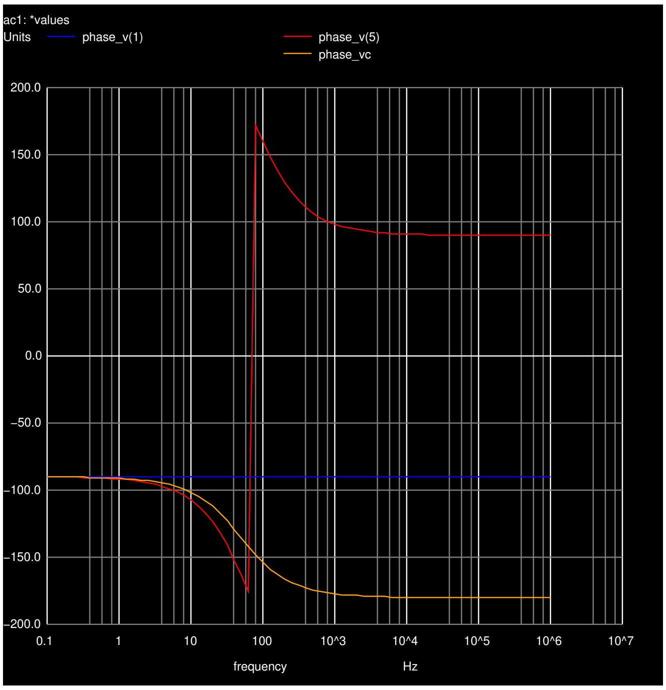


Figure 10: Phase of v_S(f), in blue, v_5(f), in red, and v_C(f), in yellow

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 Ω 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
τ	3.239206e-03

Name	Value [A or V or Ω]
@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 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

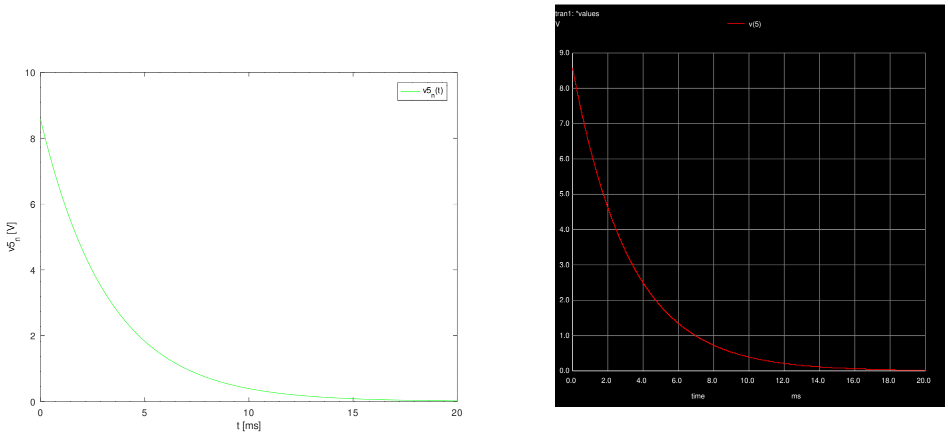


Figure 11: Natural solution v_5

6.2 Final solution

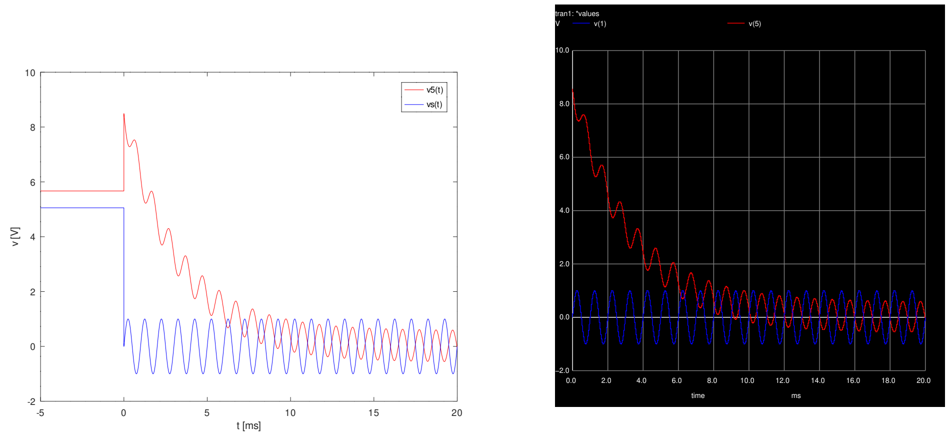
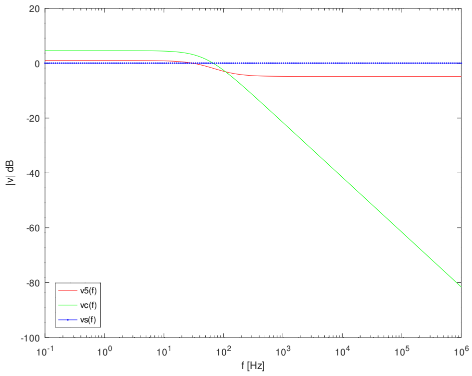


Figure 12: Final solution v_5 and v_s

6.3 Frequency response



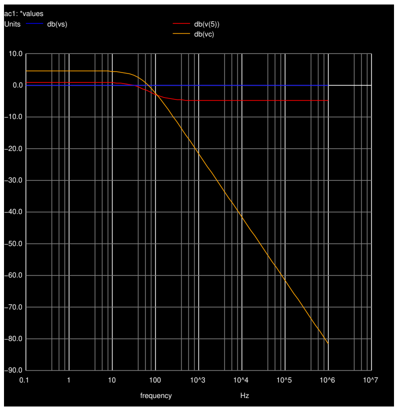
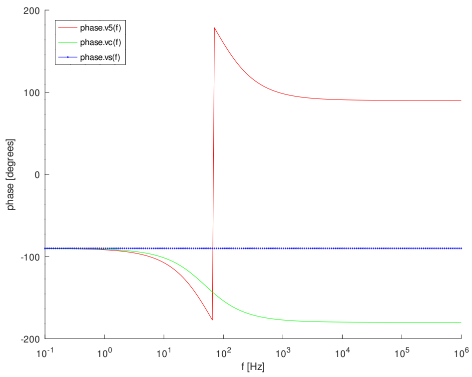


Figure 13: Magnitude in dB



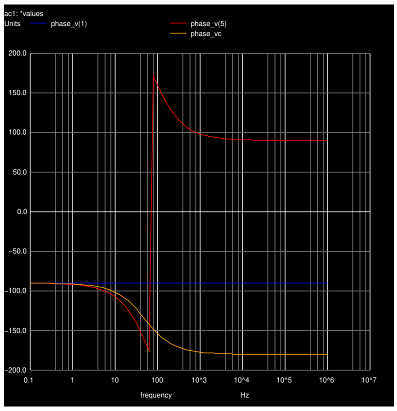


Figure 14: Phase in degrees