Integrated Masters in Aerospace Engineering, Técnico, University of Lisbon

Circuit Theory and Electronics Fundamentals

Laboratory Report 2

Grupo 21:

95791, Francisco Carvalho

95805, João Matias

95846, Simão Gonçalves

April 2021

Índice

1 Introduction **3**

2 Theoretical Analysis **5**
2.1 Nodal Method for t $<$ 0 . . . 5
2.2 Determination of the resistance R_{eq} . . . 5
2.3 Natural Solution V_{6n}(t), in the interval [0,20]ms . . . 6
2.4 Forced Solution V_{6f}(t), in the interval [0,20]ms . . . 7
2.5 Final Solution V_6(t), in the interval [0,20]ms . . . 8
2.6 Frequency response for V_c(f), V_6(f) and V_s(f) . . . 9
2.6.1 Magnitude . . . 9
2.6.2 Phase . . . 9

3 Simulation Analysis **10**
3.1 Circuit Analysis for $t \leq 0$ . . . 10
3.2 R_{eq} Calculus, and Circuit Analysis for $t = 0$ . . . 11
3.3 Circuit Analysis for $t \geq 0$ (Natural Solution) . . . 12
3.4 Circuit Analysis for $t \geq 0$ (Natural and Forced Solution) . . . 13
3.5 Frequency Responses . . . 14
3.5.1 Frequency Responses - Amplitude . . . 14
3.5.2 Frequency Responses - Phase . . . 15

4 Conclusion **16**

1 Introduction

For the second experimental activity in the Circuit Theory and Electronics Fundamentals course, we were given a RC circuit to analyse. This circuit is constituted by a sinusoidal voltage source v_s, dependent voltage V_d and current I_b sources, a capacitor C and 7 resistors (fig. **??**).

The theoretical analysis (Section 2) adresses, sequencially, the nodal analysis, then the calculus of R_{eq}. With these results, the natural response of the circuit, the forced response are computed, and then superimposed.

The simulation analysis (Section 3) was made to validate the results obtained in section 2, operating point, transient and then frequency analysis were made in Ngspice. The conclusions of this study are outlined in the final section (4).

The voltage source varies in time as it follows:

$$v_s(t) = V_s u(-t) + sin(2\pi f t)u(t) \tag{1}$$

where

$$u(t) = \begin{cases} 0, & t < 0 \\ 1, & t \geq 0 \end{cases} \tag{2}$$

To obtain the initial data, we ran a python script provided by our teacher, which generated the following data:

R1	1.048998e+03 kOhm
R2	2.055568e+03 kOhm
R3	3.060188e+03 kOhm
R4	4.168980e+03 kOhm
R5	3.073950e+03 kOhm
R6	2.042810e+03 kOhm
R7	1.037563e+03 kOhm
Vs	5.114229e+00 V
C	1.011550e-06 uF
Kb	7.338555e-03 mS
Kd	8.258180e+03 kOhm

Table 1: Initial Data generated by the python script. The variables are expressed in V, mA, mS, kOhm, uF.

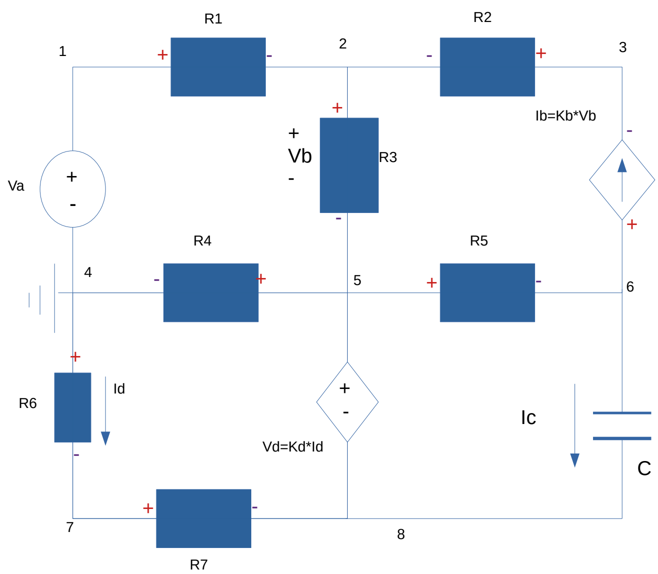


Figure 1: First Circuit

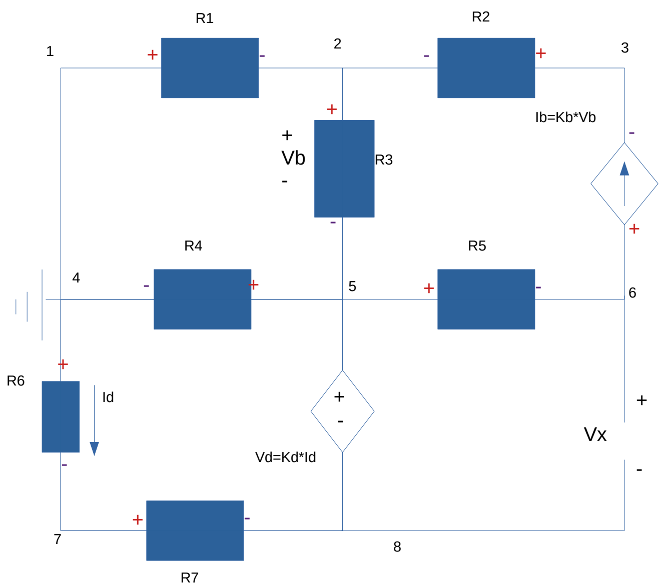


Figure 2: Second Circuit

2 Theoretical Analysis

2.1 Nodal Method for t $<$ 0

The objective of this method is to determine every node voltage. To do this we have to first number nodes arbitrarily, assign potential 0V (Ground) to one of the nodes (node 4 in this case), and then calculate all the voltages. We had 8 unknown variables, so we determined 8 independent equations. Before t $<$ 0s, $v(s) = V_s$ (constant), hence the capacitor behaves as an open-circuit (I_x = 0). The equations were rearranged in the matrix form below, in order to find the solution using Octave math tools. The circuit that we analysed in this section, was the circuit 1 of the introduction.The results are shown in the tables below.

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G_1 & G_1+G_2+G_3 & -G_2 & 0 & -G_3 & 0 & 0 & 0 \\ 0 & -K_b-G_2 & G_2 & 0 & K_b & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & -G_3 & 0 & -G_4 & G_3+G_4+G_5 & -G_5 & -G_7 & G_7 \\ 0 & K_b & 0 & 0 & -K_b-G_5 & G_5 & 0 & 0 \\ 0 & 0 & 0 & -G_6 & 0 & 0 & G_6+G_7 & -G_7 \\ 0 & 0 & 0 & K_d*G_6 & -1 & 0 & -K_d*G_6 & 1 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

V1	5.114229e+00 V
V2	4.873694e+00 V
V3	4.380386e+00 V
V4	0.000000e+00 V
V5	4.906396e+00 V
V6	5.644102e+00 V
V7	-1.935730e+00 V
V8	-2.918905e+00 V

Table 2: Theoretical Nodal Voltages expressed in V

I1	-2.293000e-04 A
I2	-2.399863e-04 A
I3	-1.068631e-05 A
I4	1.176882e-03 A
I5	-2.399863e-04 A
I6	9.475818e-04 A
I7	9.475818e-04 A
Is	-2.293000e-04 A
Ic	0.000000e+00 A

Table 3: Theoretical Nodal Voltages expressed in A

2.2 Determination of the resistance R_{eq}

In this section, the objective was to compute the equivalent resistance (R_{eq}) seen from the capacitor terminals and the time constant. For that, we use this following relation:

$$R_{eq} = \frac{V_x}{I_x} \tag{3}$$

$$\tau = R_{eq} * C \tag{4}$$

To resolve this question, we use the Thevenin and Norton theorem. So, we made Vs = 0 (independent sorce) and we replaced the capacitor with a voltage source V_x = V_6 - V_8, where V_6 and V_8 are the voltages in nodes 6 and 8 as obtained in 2.1. Vx is equivalent to Thevenin's Voltage, and Ix to Norton's Current. Finally, we ran nodal analyses to determinate the current Ix supplied by Vx. We use this procedure because the circuit is complex and has dependent sources (the dependent voltage source cannot be put equal to 0V, and the dependent current source cannot be erased from the circuit).

The equations were rearranged in the matrix form below, in order to find the solution using Octave math tools.

V1	0.000000e+00 V
V2	0.000000e+00 V
V3	0.000000e+00 V
V4	0.000000e+00 V
V5	-5.941780e-17 V
V6	8.563007e+00 V
V7	0.000000e+00 V
V8	2.970890e-17 V
Ix	-2.785669e-03 A
Req	3.073950e+03 Ohm
tau	3.109455e-03

Table 4: Theoretical results expressed in V, A and Ohm

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G_1 & G_1+G_2+G_3 & -G_2 & 0 & -G_3 & 0 & 0 & 0 & 0 \\ 0 & -K_b-G_2 & G_2 & 0 & K_b & 0 & 0 & 0 & 0 \\ 0 & -G_1 & 0 & 0 & -G_4 & 0 & -G_6 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 & 0 \\ 0 & K_b & 0 & 0 & -K_b-G_5 & G_5 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 & G_6+G_7 & -G_7 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & K_d*G_6 & -1 & 0 & -K_d*G_6 & 1 & 0 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \\ I_x \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ V_x \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

The circuit that we analysed in this section was the circuit 2 of the introduction. Also, we expose in a table the results of the node´s voltage, R_{eq}, I_x and the time constant (τ).

2.3 Natural Solution V_{6n}(t), in the interval [0,20]ms

The purpose of this task was calculate the natural response of the circuit, specifically we calculate the voltage on node 6 over time. The natural response tells us what the circuit does as its internal stored energy (the initial voltage on the capacitor) is allowed to dissipate. It does this by ignoring the forcing input. For the initial condition we use the capacitor voltage for t $<$ 0 (V_x).

So, to calculate this natural solution, we eliminate the independent voltage source (v_s(t) = 0). Therefore, we have an equivalent circuit described by the equivalent resistor and the capacitor, as we can see below. The capacitor is discharging, and the energy is being dissipated by the resistor. Using KVL, we obtained the following differential equation:

$$\frac{dV}{dt} + \frac{V}{C*R_{eq}} = 0 \tag{5}$$

We know that V_c = V_6 - V_8 = V_x, and using the initial solution we obtain the natural solution for the capacitor:

$$V_{6n} = V_c * e^{-\frac{t}{C*R_{eq}}} \tag{6}$$

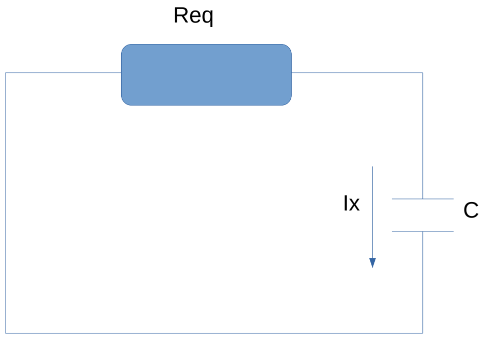


Figure 3: Third Circuit

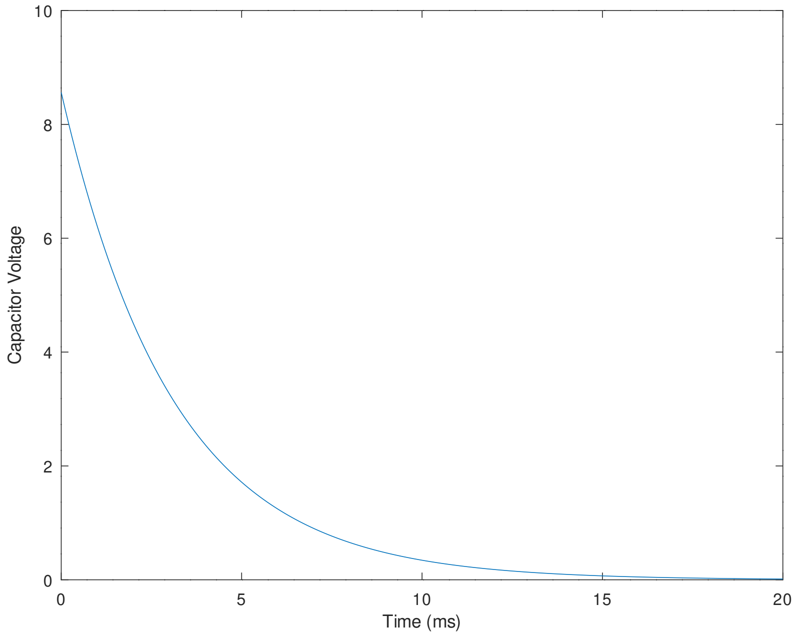


Figure 4: Natural Solution $v_{6n}(t)$, $t \in [0, 20]$ ms

2.4 Forced Solution V_{6f}(t), in the interval [0,20]ms

In a forced circuit the voltages in the nodes will have the same frequency as the source. Consequently, the voltage in the node will be something like $V_{node} = V_{node_{max}} * cos(wt + \phi_{node})$. In order to determine the voltages in the nodes, we have to determine their amplitudes and phases. In other words, we have to calculate their complex amplitude. In complex terms, the voltage in the nodes will be in the form $V_{node} = V_{node_{max}} * e^{wt} * e^{\phi_{node}})$.

To obtained this solutions, we rearranged the equations in the matrix form below, in order to find the solution using Octave math tools.

$$w = 2 * \pi * f \tag{7}$$

$$V_C = \frac{1}{j * w * C} \tag{8}$$

$$\begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\ G_1 & -G_1 - G_2 - G_3 & G_2 & 0 & G_3 & 0 & 0 & 0 \\ 0 & K_b + G_2 & -G_2 & 0 & -K_b & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & G_3 & 0 & G_4 & -G_3 - G_4 - G_5 & G_5 + 1/Z_C & G_7 & -G_7 - 1/Z_C \\ 0 & K_b & 0 & 0 & -K_b - G_5 & G_5 + 1/Z_C & 0 & -1/Z_C \\ 0 & 0 & 0 & -G_6 & 0 & 0 & G_6 + G_7 & -G_7 \\ 0 & 0 & 0 & K_d * G_6 & -1 & 0 & -K_d * G_6 & 1 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

The tables below contain the amplitudes and the phases of the differents nodes,that allows us to compute the complex amplitude of every node voltage. So, $V_6 = A * cos(wt + \phi)$, and you can see the amplitude and the phase below.

V1	1.000000e+00 V
V2	9.529674e-01 V
V3	8.565095e-01 V
V4	0.000000e+00 V
V5	9.593618e-01 V
V6	5.727806e-01 V
V7	3.784988e-01 V
V8	5.707419e-01 V

Table 5: Amplitudes of nodal voltages in V

Phase 1	0.000000e+00 Radians
Phase 2	5.945769e-16 Radians
Phase 3	1.711774e-15 Radians
Phase 4	0.000000e+00 Radians
Phase 5	5.280845e-16 Radians
Phase 6	-2.991803e+00 Radians
Phase 7	-3.141593e+00 Radians
Phase 8	-3.141593e+00 Radians

Table 6: Phases of nodal voltages in Rad

2.5 Final Solution V_6(t), in the interval [0,20]ms

The total response of a circuit can be seen as the sum of the natural and forced response, using the principle of superposition. These two solution were calculated in the section 2.3 and 2.4.

The final solution for V_6(t) is:

$$V_{6f} = \begin{cases} V_6, & t < 0 \\ V_c * e^{-\frac{t}{C*R_{eq}}} + A * cos(wt + \phi), & t \geq 0 \end{cases} \tag{9}$$

The final solution for V_s(t) is:

$$V_{sf} = \begin{cases} V_s, & t < 0 \\ sin(2\pi ft), & t \geq 0 \end{cases} \tag{10}$$

In the following plot you can see this evolutions:

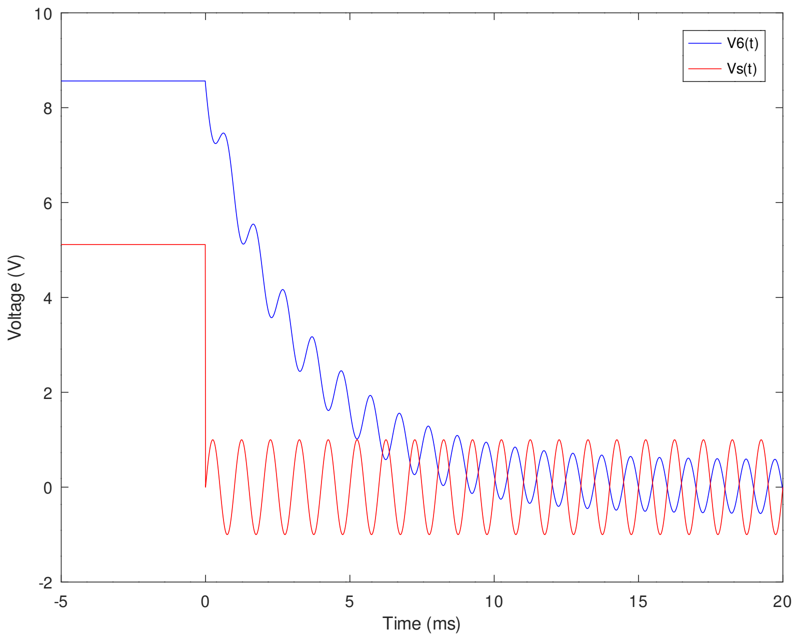


Figure 5: Final Solution

2.6 Frequency response for V_c(f), V_6(f) and V_s(f)

In this section we determine the frequency responses $V_c(f) = V_6(f) - V_8(f)$ and V_6(f), for frequency range 0.1Hz to 10MHz. To the magnitude response we use a logscale in order to provide a much better plot fit and a better great visualization for users. The unit of the magnitude is the decibel, which is common used for sound waves. To solve this problem we use the equations rearranged in the matrix in 2.4, in a cycle.

2.6.1 Magnitude

The magnitude is in dB, the absolute values were converted ($X_dB = 20log10(X)$). The frequencies were put in a logarithmic scale. The magnitude of V_s is constant despite the variation of the frequency of the signal. Since its amplitude is 1, as one can observe in the graphics shown at the end of the section, the plot shows a constant horizontal line, with the value zero (0=log10(1)). On the other hand, as the frequency is increasing, the magnitudes of V_6 and V_c decrease. As it is expected in a RC circuit, due to the capacitor's impedance ($Z_C = 1/j * w * C$), the value of V_c changes.

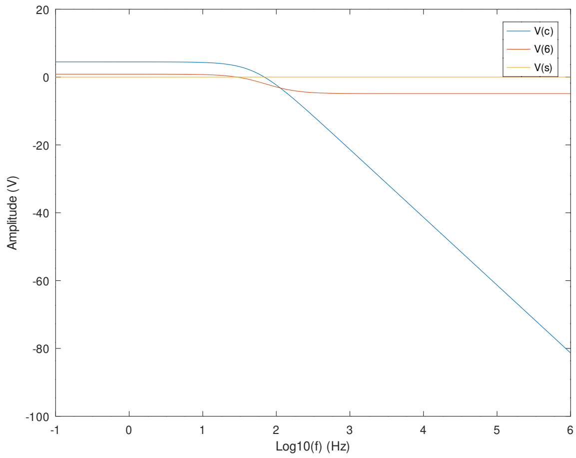


Figure 6: Amplitude

2.6.2 Phase

The angles are in degrees. As we expected the phase of V_s is 0, and the phase of V_c tends to -90° and V_6 to -180°, as the frequency is increasing. In the plot below we can see the evolution.

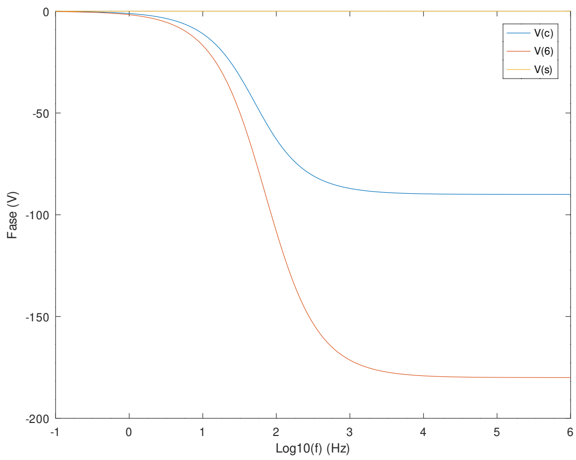


Figure 7: Phase

3 Simulation Analysis

In order to understand the values obtained in our simulation, it is important to note a couple of things:

- Node 4 is considered to be ground, therefore, its' voltage is not shown in the table of results, but is, obviously, 0V.
- To compute the voltage drop through the dependent voltage source (V_d), we needed to obtain the current value through resistor R6. However, Ngspice doesn't allow us to input Resistor R_6 current in the computation. Hence, in order to solve this problem, we introduced a voltage source with no voltage drop in series with the R_6 resistor. This way, we managed to obtain the current through it, and therefore we were able to compute the dependent voltage source V_d.

3.1 Circuit Analysis for $t \leq 0$

For $t < 0$, the voltage source (Va) is producing a constant voltage source of value Vs. Therefore, the capacitor behaves as an open circuit. After running the Ngspice script, we obtained the following results:

@gb[i]	-2.39986e-04
@r1[i]	-2.29300e-04
@r2[i]	-2.39986e-04
@r3[i]	-1.06863e-05
@r4[i]	1.176882e-03
@r5[i]	-2.39986e-04
@r6[i]	9.475819e-04
@r7[i]	9.475819e-04
v(1)	5.114229e+00
v(2)	4.873694e+00
v(3)	4.380386e+00
v(5)	4.906396e+00
v(6)	5.644101e+00
v(7)	-1.93573e+00
v(8)	-2.91891e+00
v(9)	-1.93573e+00

Table 7: Nodal Voltage Simulation Results. Variables expressed in V or A

V1	5.114229e+00 V
V2	4.873694e+00 V
V3	4.380386e+00 V
V4	0.000000e+00 V
V5	4.906396e+00 V
V6	5.644102e+00 V
V7	-1.935730e+00 V
V8	-2.918905e+00 V

Table 8: Theoretical Nodal Voltages expressed in V

I1	-2.293000e-04 A
I2	-2.399863e-04 A
I3	-1.068631e-05 A
I4	1.176882e-03 A
I5	-2.399863e-04 A
I6	9.475818e-04 A
I7	9.475818e-04 A
Is	-2.293000e-04 A
Ic	0.000000e+00 A

Table 9: Theoretical Nodal Voltages expressed in A

3.2 R_{eq} Calculus, and Circuit Analysis for $t = 0$

In the second simulation, the open circuit branch of the capacitor was replaced with a voltage source, with voltage $V_x = V_6 - V_8$. This step was needed to calculate the equivalent Thévenin resistance, which can be obtained through the following expression:

$$R_{eq} = \frac{V_6 - V_8}{I_x} \tag{11}$$

with I_x being the current through the voltage source previously defined V_x.

@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]	-2.78567e-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(5)	0.000000e+00
v(6)	8.563007e+00
v(7)	0.000000e+00
v(8)	0.000000e+00
v(9)	0.000000e+00

Table 10: Nodal Voltage Simulation results. Variables expressed in V

V1	0.000000e+00 V
V2	0.000000e+00 V
V3	0.000000e+00 V
V4	0.000000e+00 V
V5	-5.941780e-17 V
V6	8.563007e+00 V
V7	0.000000e+00 V
V8	2.970890e-17 V
Ix	-2.785669e-03 A
Req	3.073950e+03 Ohm
tau	3.109455e-03

Table 11: Theoretical results expressed in V, A and Ohm

3.3 Circuit Analysis for $t \geq 0$ (Natural Solution)

For the third simulation, we performed a transient analysis of the given circuit, which is similar to the one done in the previous section. The main difference is the introduction of the capacitor instead of the voltage source between nodes 6 and 8, and the fact that the voltage source is turned off. With this, we run the Ngspice script to obtain the natural solution of the circuit.

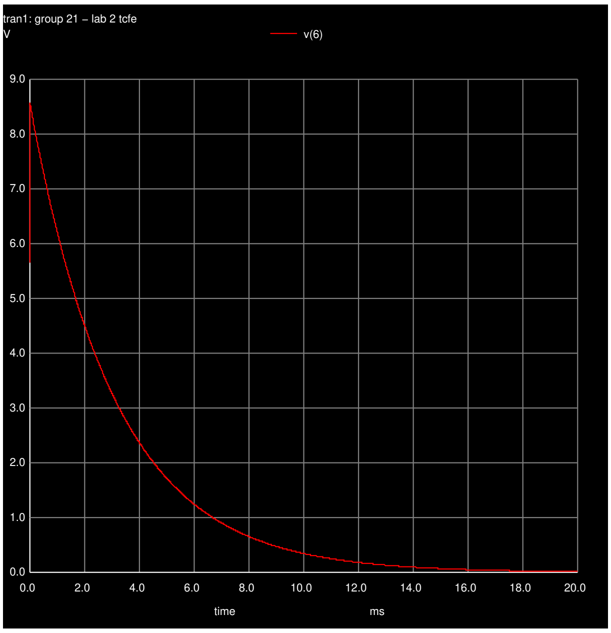


Figure 8: Natural Solution (Ngspice)

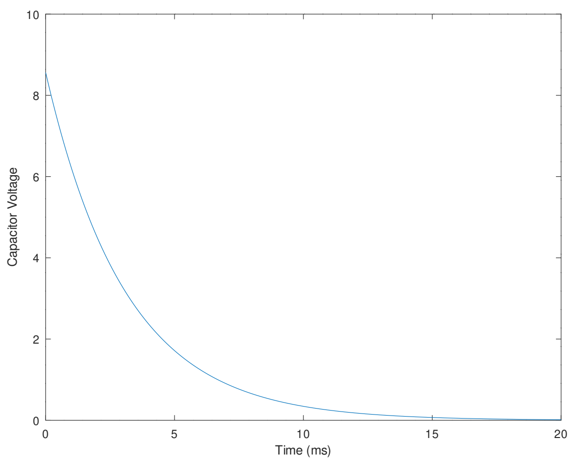


Figure 9: Natural Solution (Octave)

We can conclude that the capacitor is discharging over time, as expected, concluding that the theoretical analysis and the simulation, match.

3.4 Circuit Analysis for $t \geq 0$ (Natural and Forced Solution)

In this section we also performed a transient analysis, making a few changes to the circuit, in order to obtain a plot of the natural and forced solution. Therefore, we took in consideration the sinusoidal variation of the voltage source over time, plotting that in our computation.

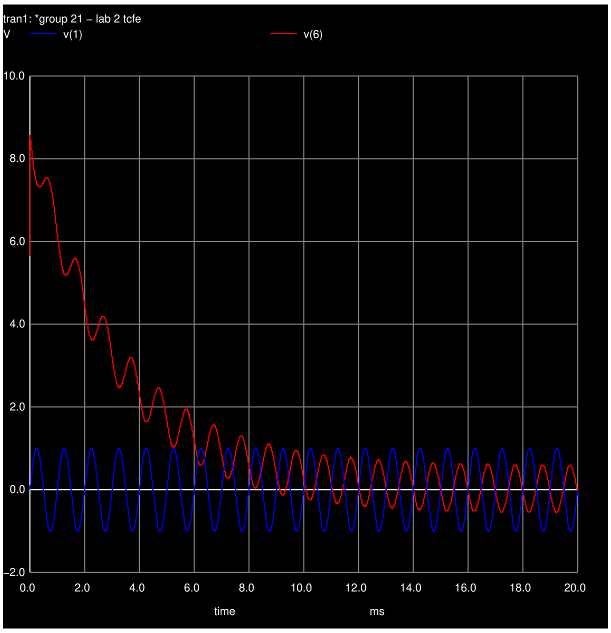


Figure 10: Final Solution (NGSpice)

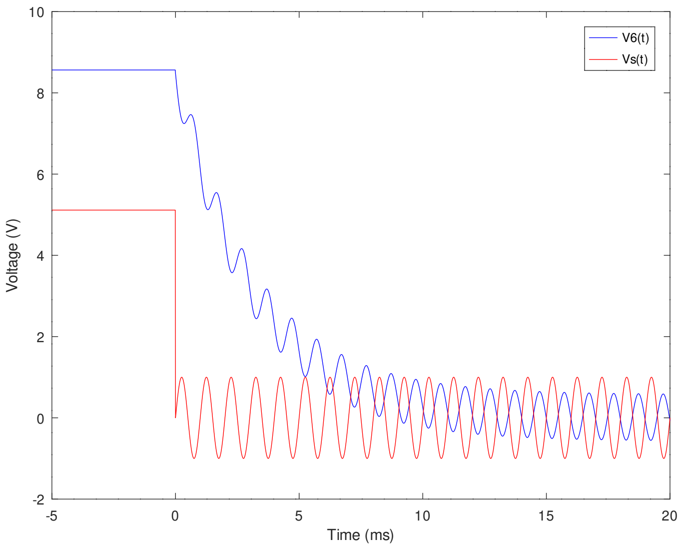


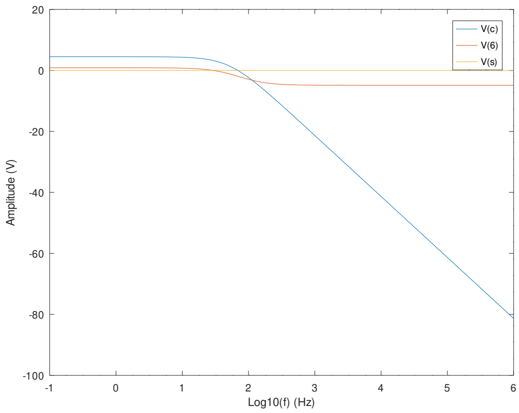
Figure 11: Final Solution (Octave)

As we can see from the pictures above, both Octave and Ngspice results match perfectly. W can conclude that the voltage in the capacitor diminishes until it has a phase difference of π, in comparison to the voltage source, as expected in the theoretical analysis.

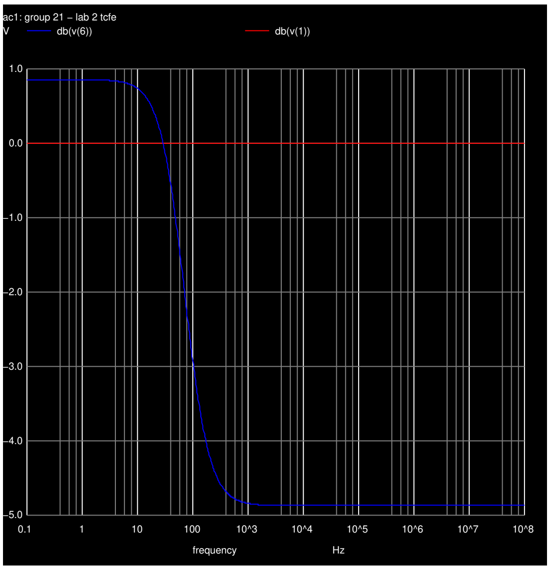
3.5 Frequency Responses

For this section, we conducted and Alternating Current (AC) Analysis. With this, we were able to study the frequency response of the circuit. It is important to note that there is no frequency variation, which means we are performing a steady-state analysis. After examination of the graphics below, it is clear that the results from Octave match the ones of Ngspice clear, where any minor difference may be explained by approximation errors.

3.5.1 Frequency Responses - Amplitude

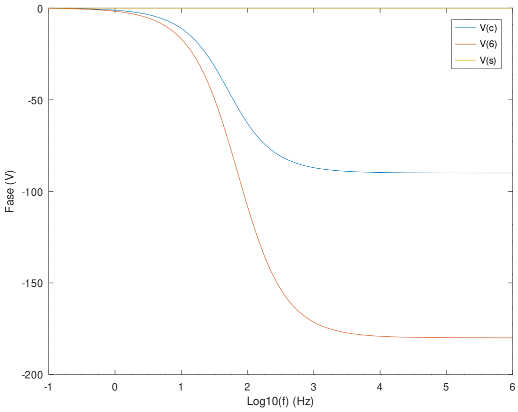


(a) Amplitude (Octave)

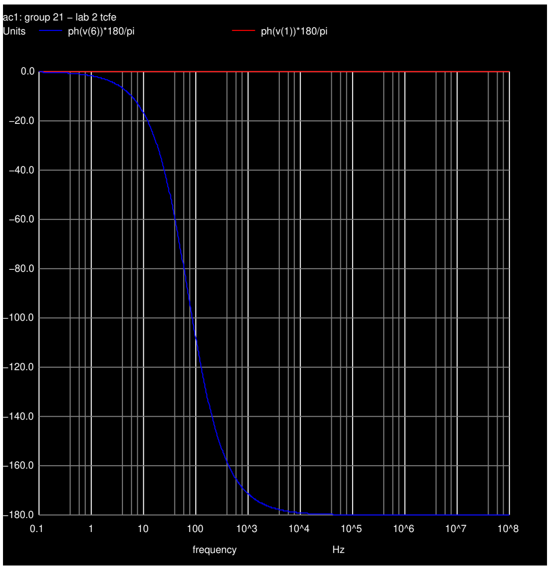


(b) Amplitude (Ngspice

3.5.2 Frequency Responses - Phase



(a) Phase (Octave)



(b) Phase (Ngspice)

4 Conclusion

All analyses have been performed both theoretically using the Octave maths tool and by circuit simulation using the Ngspice tool. As we can see the both theorical and simulation results match for every single component of the circuit. Even though we could have expected a bigger error margin due to the number of circuit components and the number of steps to get to final awnser, that was not the case. Hence, the circuit analysis proposed in the introduction was achieved.