

## Instituto Superior Técnico, University of Lisbon

### Integrated Master in Aerospace Engineering

#### Circuit Theory and Electronics Fundamentals

Joana Matos (95799), Ricardo Abreu (95842), Vasco Emídio (95856)

#### Second Laboratory Report

April 7, 2021

## Contents

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>Theoretical Analysis</b>	<b>2</b>
2.1	Voltages in Nodes and Currents in Branches . . . . .	3
2.2	Equivalent Resistance . . . . .	4
2.3	Point 3 . . . . .	5
2.4	Point 4 . . . . .	5
2.5	Point 5 . . . . .	7
2.6	Point 6 . . . . .	7
<b>3</b>	<b>Simulation Analysis</b>	<b>9</b>
3.1	Operating Point Analysis . . . . .	9
3.2	Transient Analysis . . . . .	10
3.3	Frequency Analysis . . . . .	12
3.3.1	Magnitude Response . . . . .	12
3.3.2	Phase Response . . . . .	12
<b>4</b>	<b>Conclusion</b>	<b>13</b>

## 1 Introduction

The objective of this laboratory assignment is to study a circuit containing a AC voltage source  $v_s$ , seven resistors, a voltage-controlled current source  $I_b$ , a capacitor  $C$  and a current-controlled voltage source  $V_d$ . The components of this circuit are distributed by 4 elementary meshes and 8 nodes, as seen in Figure 1.

In order to analyse the circuit, the following data were obtained by running the supplied Python script:

*Units for the values:* V, mA, uF, kOhm and mS

*Values:*

R1 = 1.00147062639

R2 = 2.0078068512

R3 = 3.11269704405

R4 = 4.10609573471

R5 = 3.02670672634

R6 = 2.01292455078

R7 = 1.02905244808

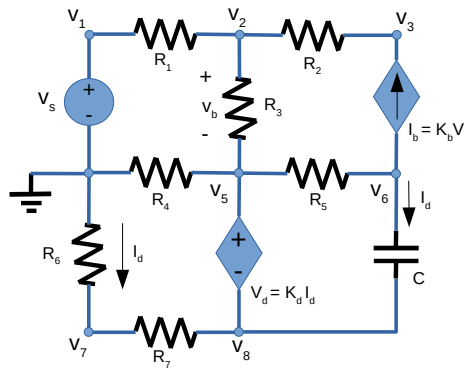
Vs = 5.24842063411

C = 1.04086013403

Kb = 7.21413591579

Kd = 8.01455113996

In Section 2, a theoretical analysis of the circuit, performed on Octave, is presented. In Section 3, the circuit is analysed by simulation, using NGSpice, and the results are compared to the theoretical results obtained in Section 2. The conclusions of this study are outlined in Section 4.



**Figure 1:** Circuit which will be analysed during this laboratory assignment.

## 2 Theoretical Analysis

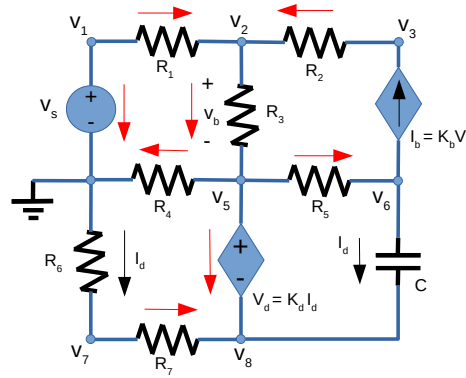
In this section, the RC circuit shown in Figure 1 is analysed theoretically. We will begin by analyzing the circuit by applying the nodal method to determine the voltages in all nodes and currents in all branches for  $t < 0$ .

In order to do this laboratory, we will not only use the node method, but also apply the Thévenin/Norton Theorem as well as what was lectured about capacitors. A RC circuit is composed by resistors and capacitors and it may driven by voltage or current sources wich will produce different responses. A capacitor is an electrical component that behaves according to the following differential equations:

$$q(t) = C \cdot v(t) \Leftrightarrow \frac{d \cdot v(t)}{dt} = C \cdot \frac{d \cdot q(t)}{dt} \Leftrightarrow i(t) = C \cdot \frac{d \cdot v(t)}{dt} \quad (1)$$

Thereafter, the current of a capacitor is not proportional to the voltage in its terminals, but rather to the voltage variation rate.

## 2.1 Voltages in Nodes and Currents in Branches



**Figure 2:** Circuit with each current direction arbitrarily assigned, to assist in the analysis of the circuit by the Nodal Method.

For  $t < 0$ ,  $-t > 0$ . Because of that  $u(t)=0$  and  $u(-t)=1$ . So:

$$V_S(t < 0) = V_s \quad (2)$$

That means that the voltage source drives constant voltage  $V_s$ , in other words, the voltage doesn't varies in time. Consequently, the current of the capacitor is null:

$$I_c = 0 \quad (3)$$

To determine the nodal voltages, we use the nodal method.

We first start by calculating the values of the conductances of the various resistors:

$$G_i = 1/R_i \quad (4)$$

Then we determine the KCL equations in nodes not connected to voltage sources and another additional equations for nodes related by voltage sources.

After doing that we can now obtain the matricial equation wich will allow us to determine de nodal voltages:

$$\begin{bmatrix} 1 & -0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G_1 & -G_1 - G_2 - G_3 & G_2 & G_3 & 0 & 0 & 0 & 0 \\ 0 & G_2 + K_b & -G_2 & -K_b & 0 & 0 & 0 & 0 \\ 0 & -G_3 & 0 & G_3 + G_4 + G_5 & -G_5 & -G_7 & G_7 & 0 \\ 0 & -K_b & 0 & G_5 + K_b & -G_5 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -G_6 - G_7 & -G_7 & 0 \\ 0 & 0 & 0 & 0 & 0 & G_6 \cdot K_d & -1 & 0 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \quad (5)$$

The solution of this matricial equation is determined by Octave:

Node	Voltage[V]
V <sub>1</sub>	5.248421e+00
V <sub>2</sub>	5.033187e+00
V <sub>3</sub>	4.581561e+00
V <sub>5</sub>	5.064366e+00
V <sub>6</sub>	5.745178e+00
V <sub>7</sub>	-2.050083e+00
V <sub>8</sub>	-3.098131e+00

**Table 1:** Nodal Voltages Values

Knowing these voltages, it is also possible to determine branch currents, using Ohm's Law or Kirchoff's Laws. Note that:

$$I_S = I_1 \quad (6)$$

$$I_{Vd} = -I_7 \quad (7)$$

$$I_b = K_b \cdot (V_2 - V_5) \quad (8)$$

Name	Value [A or V]
$I_{R1}$	-2.149179e-04
$I_{R2}$	-2.249348e-04
$I_{R3}$	-1.001695e-05
$I_{R4}$	1.233378e-03
$I_{R5}$	-2.249348e-04
$I_{R6}$	1.018460e-03
$I_{R7}$	1.018460e-03
$I_s$	-2.149179e-04
$I_b$	-2.249348e-04
$I_{Vd}$	-1.018460e-03
$I_c$	0.000000e+00

**Table 2:** Voltages and currents of some circuit components

## 2.2 Equivalent Resistance

The objective of this task is to calculate  $R_{eq}$  and  $I_x$  as seen in terminals of the voltage source substituting the capacitor. It is known that:

$$R_{eq} = \frac{V_x}{I_x} \quad (9)$$

In order to solve this task, we used the concept of the Thévenin and Norton Theorems.

The matrix equation used in the octave tool is the following:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ G_1 & -G_1 - G_2 - G_3 & G_2 & G_3 & 0 & 0 & 0 \\ 0 & G_2 + K_b & -G_2 & -K_b & 0 & 0 & 0 \\ -G_1 & G_1 & 0 & G_4 & 0 & G_6 & 0 \\ 0 & 0 & 0 & 1 & 0 & G_6 \cdot K_d & -1 \\ 0 & 0 & 0 & 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 & 0 & -G_6 - G_7 & G_7 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ V_x \\ 0 \end{bmatrix} \quad (10)$$

And the results obtained are expressed in this table:

Name	Value [A or V]
$V_1$	0.000000e+00
$V_2$	0.000000e+00
$V_3$	0.000000e+00
$V_5$	0.000000e+00
$V_6$	8.843309e+00
$V_7$	-0.000000e+00
$V_8$	0.000000e+00

**Table 3:** Voltages and currents of some circuit components

Name	Value [A or V]
$V_x$	8.843309e+00
$I_x$	2.921760e-03
$R_{eq}$	3.026707e+03

**Table 4:** Voltages and currents of some circuit components

### 2.3 Point 3

This task's objective is to compute the natural solution of the voltage in node 6. Using the results previously obtained in task 2, we obtained the value of the voltage in this node when  $t=0$  seconds [ $v_{6n}(t=0s)=V_x$ , since the result obtained for  $v_8$  is zero in the previous task].

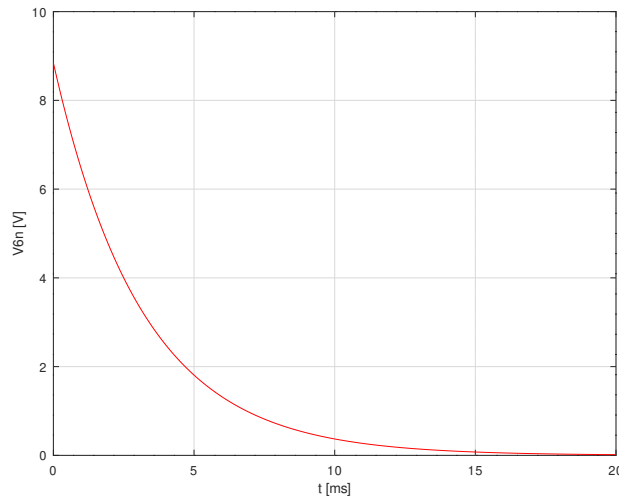
The natural solution has the following format:

$$v_n(t) = A \cdot \exp\left(\frac{-t}{RC}\right) \quad (11)$$

In this particular case,  $A = v_6(t=0) = V_x$ , giving us the equation below:

$$v_{6n}(t) = V_x \cdot \exp\left(\frac{-t}{RC}\right) \quad (12)$$

The following plot was obtained using the octave tool and it shows the graph obtained for the natural solution of the voltage in node 6 and it is as expected a negative exponential:



**Figure 3:** Final response.

### 2.4 Point 4

In order to compute the forced solution for the voltage in node 6, it was utilized the same time interval considered in the previous task.

But to calculate the forced solution, we need to use the regular circuit and to do that we assumed a phasor  $V_s$  and substituted the capacitor  $C$  for its impedance,  $Z_c$ .

The phasor used was:

$$V_s = \exp\left(-i \cdot \frac{\pi}{2}\right) \quad (13)$$

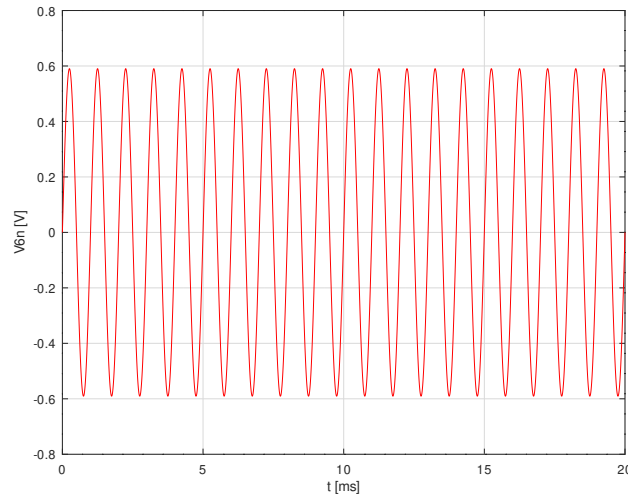
And the formula to obtain  $Z_c$  is:

$$Z_c = \frac{1}{i \cdot C \cdot \omega} \quad (14)$$

With all of that taken into consideration, the matrix equation used to calculate the forced solution in this time interval is the following:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G_1 & G_1 & 0 & G_4 & 0 & G_6 & 0 \\ G_1 & -G_1 & -G_2 & -K_b & 0 & 0 & 0 \\ 0 & G_2 + Kb & -G_2 & -K_b & 0 & 0 & 0 \\ 0 & -Kb & 0 & Kb + G_5 & -G_5 - \frac{1}{Z_c} & 0 & \frac{1}{Z_c} \\ 0 & 0 & 0 & 0 & 0 & -G_6 - G_7 & G_7 \\ 0 & 0 & 0 & 1 & 0 & G_6 \cdot K_d & -1 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ V_x \\ 0 \end{bmatrix} \quad (15)$$

The result obtained can be seen in the plot below:



**Figure 4:** Final response.

The phasor amplitude values and phases were also computed in this task and can be consulted in the Table (X):

Name	Value [A or V]
$V_1$	1.000000e+00
$V_2$	9.589907e-01
$V_3$	8.729409e-01
$V_5$	9.649315e-01
$V_6$	5.902978e-01
$V_7$	3.906094e-01
$V_8$	5.902978e-01

**Table 5:** Amp

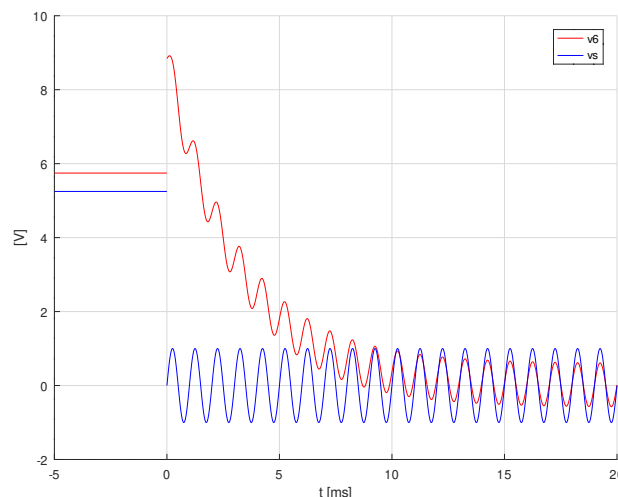
Name	Value [A or V]
$V_1$	-1.570796e+00
$V_2$	-1.570796e+00
$V_3$	-1.570796e+00
$V_5$	-1.570796e+00
$V_6$	1.570802e+00
$V_7$	1.570796e+00
$V_8$	1.570796e+00

**Table 6:** Voltages and currents of some circuit components

## 2.5 Point 5

The total solution for the node 6 for a frequency,  $f = 1000$  Hz was calculated by superimposing the forced and the natural solution for  $t$  maior ou igual 0. For  $t$  menor 0, the voltage in node 6 is the one calculated in the first task. This gives us the following function for  $v_6$ :

The total solution was plotted together with  $v_s$  and this is the result obtained:



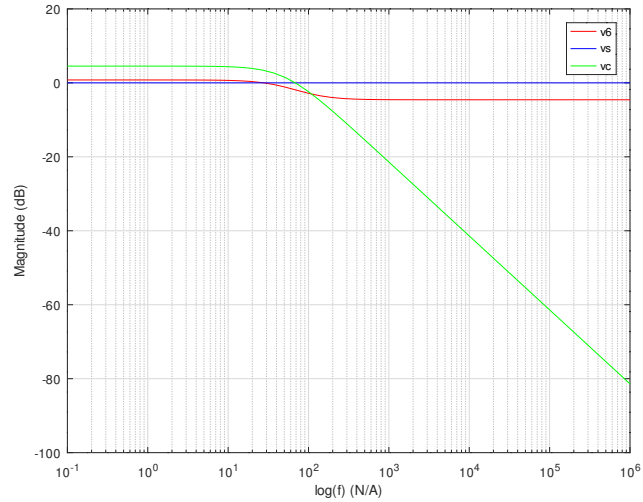
**Figure 5:** Final response.

It is worth noticing that  $v_6(t)$  is not continuous.

## 2.6 Point 6

The last assignment is to study how the amplitude, in decibels, and the phase, in degrees, of  $v_6$ ,  $v_s$  and  $v_c = v_6 - v_8$  vary with different frequencies. In order to achieve this objective, we plotted these functions together twice, in one of them we study the variance of the amplitude (Figure X) and in the other the variance of the phase (Figure Y). To obtain the intervals we used the logspace in the range of 0,1 Hz to 1 MHz.

Plot studying the amplitude variance:

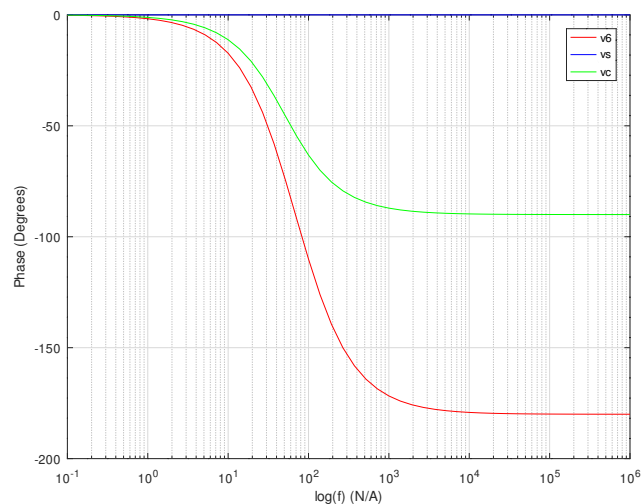


**Figure 6:** Final response.

Since the transfer function is calculated via a quotient of the output and the input, the result expected and obtained for  $v_s$  is zero. It is also noticeable that  $v_c$  rapidly decreases as the frequency shoots upwards.

Regarding  $v_6$ , it is noticeable that for higher frequencies that the functions is constant, this is due to the fact that for this kind of frequencies the capacitor acts as a shunt, meaning that only the resistors (that do not vary with the frequency) influence the variance, making it constant. For lower frequencies, it is observable that there is a slightly decrease. This is expected in a RC circuit and it is due to the impedance of the capacitor.

Plot studying the phase variance:



**Figure 7:** Final response.

It is worth noticing that, as expected, the phase variance of  $v_s$  is null. This is also due to the fact that the transfer function is calculated using a quotient between the output and the input. For  $v_6$  and  $v_c$ , it is observable that with the frequency increment the phase achieve lower values and are all negative, as expected.



### 3 Simulation Analysis

#### 3.1 Operating Point Analysis

Table 7 shows the simulated operating point results for the circuit under analysis for  $t_i0$ , which means that we will consider the voltage source  $v_s$  as a DC voltage source equal to  $V_s$ , given in the Python script. Notice that the values on the right were obtained by Octave, therefore they are the theoretical values, which we already mentioned previously but, to facilitate the comparison between the simulation and theoretical values, we mentioned them again.

Name	Value [A or V]	Node	Voltage[V]	Name	Value [A or V]
@c[i]	0.000000e+00			$I_{R1}$	-2.149179e-04
@gb[i]	-2.24935e-04	$V_1$	5.248421e+00	$I_{R2}$	-2.249348e-04
@r1[i]	2.149178e-04	$V_2$	5.033187e+00	$I_{R3}$	-1.001695e-05
@r2[i]	-2.24935e-04	$V_3$	4.581561e+00	$I_{R4}$	1.233378e-03
@r3[i]	-1.00169e-05	$V_5$	5.064366e+00	$I_{R5}$	-2.249348e-04
@r4[i]	1.233378e-03	$V_6$	5.745178e+00	$I_{R6}$	1.018460e-03
@r5[i]	-2.24935e-04	$V_7$	-2.050083e+00	$I_{R7}$	1.018460e-03
@r6[i]	1.018460e-03	$V_8$	-3.098131e+00	$I_s$	-2.149179e-04
@r7[i]	1.018460e-03			$I_b$	-2.249348e-04
v1	5.248421e+00			$I_{Vd}$	-1.018460e-03
v2	5.033187e+00			$I_c$	0.000000e+00
v3	4.581562e+00				
v4	-2.05008e+00				
v5	5.064367e+00				
v6	5.745178e+00				
v7	-2.05008e+00				
v8	-3.09813e+00				

**Table 7:** Operating point for  $t_i0$ . A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt. The values on the right are the theoretical values for the currents and voltages in the circuit.

Compared to the theoretical analysis results, we notice that both simulation results are accurate, except for the last decimal places, as a consequence of the scientific notation and the number of significant algarisms used by each program to present the results. Despite that, we realise that the values with more significant algarisms match correctly the rounded values.

Table 8 shows the simulated operating point results for the circuit under analysis for  $t=0$ , where we will consider the voltage source  $v_s=0$  and replace the capacitor for a voltage source  $V_x=V_6-V_8$ , where  $V_6$  and  $V_8$  are the voltages in nodes 6 and 8, respectively. This step will allow us to verify the theoretical results obtained for the node voltages and the current  $I_x$ , which are accurate, as we can see. Again, we show the theoretical values obtained for this step, to make it easier to compare between simulation and theoretical values.

Furthermore, we can see that for  $t_i0$ ,  $V_6$  will have a significant different voltage value than for  $t=0$ ; however, the voltage in the capacitor  $C$ , which is given by  $V_6-V_8$  remains the same, despite the different values for  $V_6$  and  $V_8$ . Therefore, we witness a discontinuity in the node voltages, that does not happen for the voltage in the capacitor - well, this happens because only the voltages in capacitors and inductors remain continuous, therefore the voltages in the resistors will have 'jumps' in their values to maintain the continuity previously mentioned.

In both tables, we can see an ninth node (node 8), which has the same voltage value as node 6. This happens because, in NGSPice, when we want to simulate circuits with current-

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]	-2.92176e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
v1	0.000000e+00
v2	0.000000e+00
v3	0.000000e+00
v4	0.000000e+00
v5	0.000000e+00
v6	8.843309e+00
v7	0.000000e+00
v8	0.000000e+00

Name	Value [A or V]
V <sub>1</sub>	0.000000e+00
V <sub>2</sub>	0.000000e+00
V <sub>3</sub>	0.000000e+00
V <sub>5</sub>	0.000000e+00
V <sub>6</sub>	8.843309e+00
V <sub>7</sub>	-0.000000e+00
V <sub>8</sub>	0.000000e+00

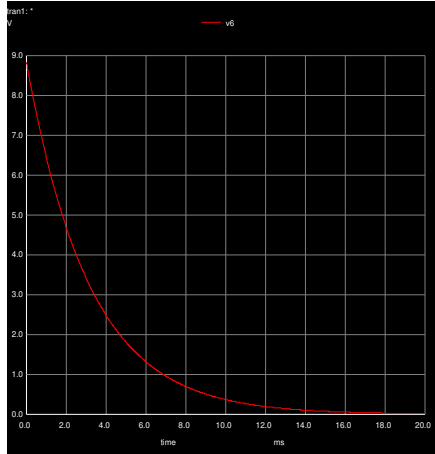
Name	Value [A or V]
V <sub>x</sub>	8.843309e+00
I <sub>x</sub>	2.921760e-03
R <sub>eq</sub>	3.026707e+03

**Table 8:** Operating point for  $t=0$ , where the capacitor  $C$  was replaced by a voltage source  $V_x = V_6 - V_8$ , with  $V_6$  and  $V_8$  obtained previously. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt. The values on the right are the theoretical values for the currents and voltages in the circuit.

dependent sources, we must add a 0V voltage source in series to a component to sense the current flowing through it. Therefore, an additional node appears in the simulated circuit.

### 3.2 Transient Analysis

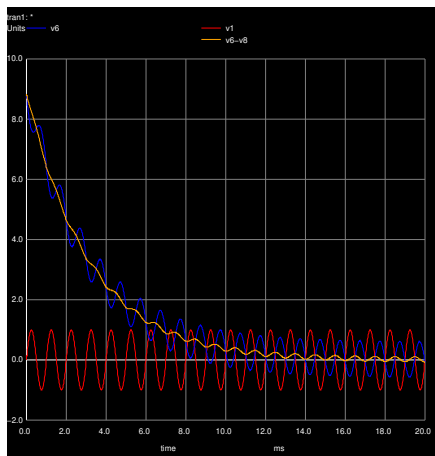
Figure ?? shows the simulated transient analysis results for the natural response of the circuit under analysis, using the boundary conditions for  $V_6$  and  $V_8$  obtained previously in the second operating point analysis performed.



**Figure 8:** Transient output voltage for node 6, with boundary conditions  $V_6$  and  $V_8$  obtained previously.

We can notice that the graphic for the voltage  $V_6$  is in a form of a negative exponential, which is confirmed by the expression and respective graphic obtained for the theoretical analysis, performed in Octave. Furthermore, because the theoretical analysis is performed for the interval  $[-5; 20]$ ms, we can easily notice the discontinuity for the voltage values  $v_s$  and  $V_6$ , that does not happen for the voltage in the capacitor.

Figure ?? shows the simulated transient analysis results for the circuit under analysis, where we finally consider the voltage source  $v_s$ , as a sinusoidal function of time. We perform this simulation to obtain the total solution (natural+forced).



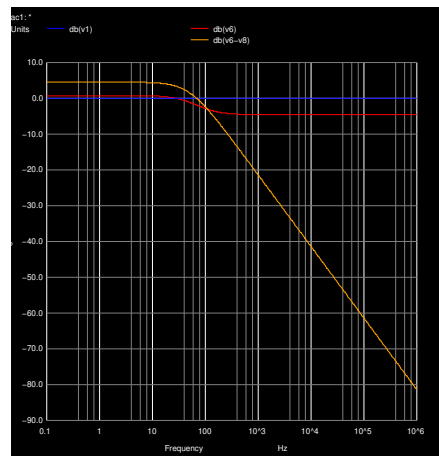
**Figure 9:** Transient output voltage for node 6, voltage source  $V_s$  and capacitor  $C$

By analysing the result of the simulation, we notice that the total solution resulted of an overlap between the natural solution (in the form of a negative exponential) and the forced solution (in a form of a sinusoidal function). Comparing to the theoretical analysis, we notice again, because of the time interval used, the discontinuity of the values in  $t=0$ , that does not happen for the voltage in the capacitor. As far as the result is concerned, the form of the theoretical solution (negative exponential+sinusoidal function) match the one obtained by NGSpice.

### 3.3 Frequency Analysis

#### 3.3.1 Magnitude Response

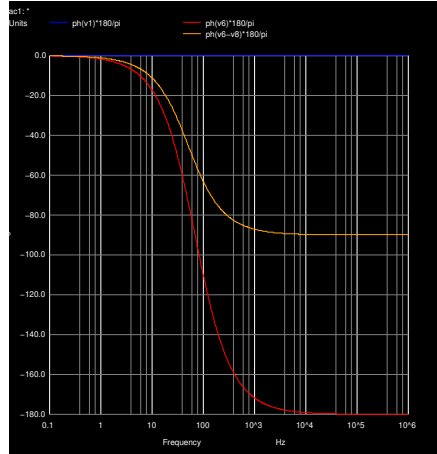
Figure 10 shows the magnitude of the frequency response for the circuit under analysis. Compared to the theoretical analysis results, one notices the following differences: describe and explain the differences.



**Figure 10:** Magnitude response for node 6, voltage source  $V_s$  and capacitor  $C$

#### 3.3.2 Phase Response

Figure 11 shows the magnitude of the frequency response for the circuit under analysis. Compared to the theoretical analysis results, one notices the following differences: describe and explain the differences.



**Figure 11:** Phase response for node 6, voltage source  $V_s$  and capacitor  $C$

## 4 Conclusion

In this laboratory assignment the objective of analysing a RC circuit by determining its natural and forced solution has been achieved. The theoretical analysis was performed with the help of the Octave math tool and the circuit simulation using the Ngspice tool. In both analysis, we had to determine the operating point results for  $t < 0$  and  $t = 0$ , and then apply the results obtained previously to plot the solutions, which were the time function of  $v_6$ ,  $v_s$  and the voltage in the capacitor  $C$  and the respective frequency functions of their magnitudes and phases. The simulation results matched the theoretical results accurately - furthermore, because we use the time interval  $[-5; 20]$ ms for the theoretical analysis, we can notice the discontinuities of some voltage values, which were previously explained in this report.