Circuit Theory and Electronics Fundamentals

Lab 1 - Circuit Analysis Methods

Aerospace Engineering

Laboratory Report

March 24, 2021

Eva Claro, 95785

Miguel Isidoro, 95834

Pedro Braz, 95837

Contents

1 Introduction

This report is being made for the subject of Circuit Theory and Electronics Fundamentals and is related to the 2^{st} laboratory being its objective to study an RC circuit containing seven resistors (from R_1 to R_7), one sinusoidal voltage source (v_s), one capacitor (C), one current controlled voltage (V_d) source and one voltage controlled current source (I_b). The four elementary meshes are named after the current to which they are attributed, and the nodes are named after the numbers attributed to them, being V_0 the ground node.

The current controlled voltage source V_d is calculated by multiplying K_d with the current I_d, whereas the voltage controlled current source I_b can be determined by multiplying K_b with the voltage source V_b.

The display of this circuit, as well as the equations used to determine the value of v_s, can be seen in Figure 1.

In Section 2 the circuit will be analysed theoretically with the aid of Octave, analysing firstly the circuit for t¡0 using the nodal method, calculating the equivalent resistence R_eq as seen from the capacitor terminals, determining the natural and forced solution for V_6 with the previous results, and finishing with the calculation of the frequency response for V_c, V_s and V_6 and the study of these results.

Secondly, in Section 3 it will be simulated the circuit using ngspice, with the aim of validating the results previously obtained by doing operating point, transient and frequency analysis.

Following with both results from Section 2 and Section 3 being compared and commented. (ONDE? ISTO É, EM QUE SECÇÃO).

The conclusions of this study are outlined in Section 5.

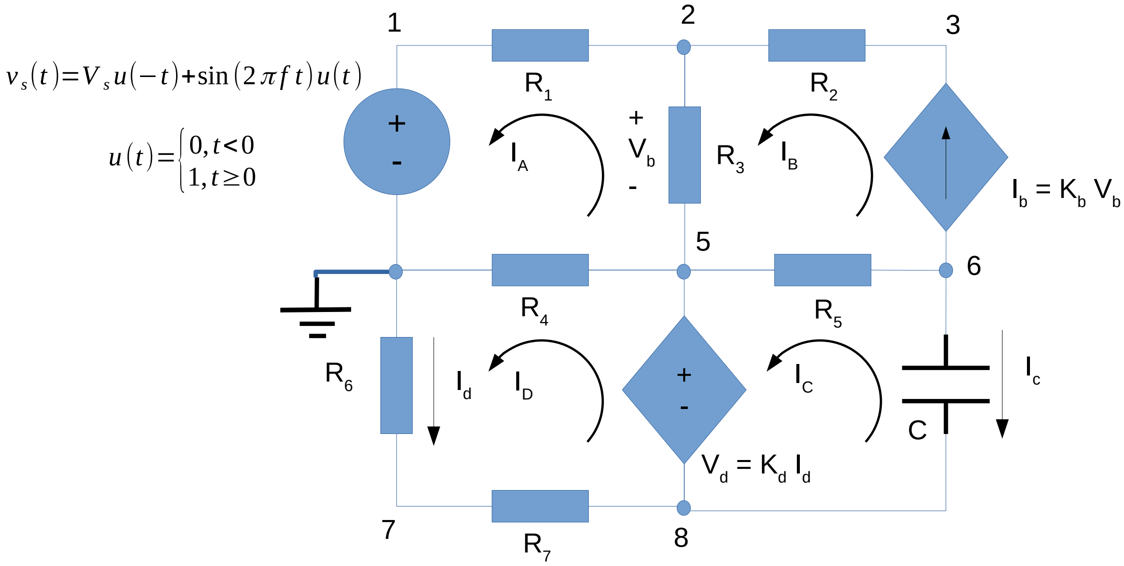

Figure 1: Circuit in analysis

Where:

Name	Value [A or V]
R1	1.034315078330000e+03
R2	2.028530907310000e+03
R3	3.146205063300000e+03
R4	4.034385474550000e+03
R5	3.121700422140000e+03
R6	2.071163796460000e+03
R7	1.015977530930000e+03
Vs	5.156959346000000e+00
C	1.014556835690000e-06
Kb	7.149794119600000e-03
Kd	8.125936425850000e+03

Table 1: Results obtained by mesh analysis method with octave

The units of the elements whose name starts with R (the resistors) are $k\Omega$ (kiloohm), V_s is expressed in V (volts) and C is given in uF (microfarad) EVA, NÃO SEI POR LETRAS GREGAS. While K_b is given in mS (milisiemens), K_c is also given in $k\Omega$.

These values where obtained using the Python script using the lowest student number on our group - 95785.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, with the Nodal Analysis Method, which uses node voltages as the circuit variables.

2.1 Node analysis for t<0

The aim of using this method is to determine every node voltage, therefore we considered the node 0 a reference node. However, due to the existence of the independent voltage source V_s and the current controlled voltage source V_d, it is useless to analyse nodes 0 and 1 (connected to V_s) and also nodes 5 and 8 (connected to V_d) since nodes connected to voltage sources can't be analysed. So, this means that we can only analyse nodes 2,3,6 and 7.

In order to determine all the unknown node voltage values, it is necessary to have eight linearly independent equations. Before t = 0s, v_s is constant, which means that the capacitor is assumed to also be constant and fully charged, behaving like an open-circuit, therefore I_c = 0.

Four of the needed equations are given by the nodal analysis: Node 2

$$(V_3 - V_2)G_2 - (V_2 - V_5)G_3 - (V_2 - V_1)G_1 = 0 \tag{1}$$

Node 3

$$(V_2 - V_5)K_b - (V_3 - V_2)G_2 = 0 \tag{2}$$

Node 6

$$(V_2 - V_5)K_b - (V_6 - V_5)G_5 = 0 \tag{3}$$

Node 7

$$-V_7G_6 - (V_7 - V_8)G_8 = 0 \tag{4}$$

We still need another four equations. For this reason, we can use these two trivial equations

$$V_0 = 0 \tag{5}$$

$$V_1 = V_s \tag{6}$$

We can also use the fact that $V_5 - V_8 = V_d = K_dI_d$ and I_d, according the Ohm's Law, is equal to $G_7(V_0 - V_7)$ which means

$$-V_7G_6K_d = V_5 - V_8 \tag{7}$$

At last, since there was still missing an equation, we considered a super node containing the branch that includes v_s

$$(V_2 - V_1)G_1 + V_5G_4 + V_7G_6 = 0 \tag{8}$$

The system of equations that will be solved in form of matrix and with the assitance of Octave is:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & G_1 & -G_1 - G_2 - G_3 & G_2 & G_3 & 0 & 0 & 0 \\ 0 & 0 & K_b + G_2 & -G_2 & -K_b & 0 & 0 & 0 \\ 0 & -G_1 & G_1 & 0 & G_4 & 0 & G_6 & 0 \\ 0 & 0 & K_b & 0 & -K_b - G_5 & G_5 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & G_6 - G_7 & G_7 \\ 0 & 0 & 0 & 0 & 1 & 0 & K_dG_6 & -1 \end{bmatrix} \begin{bmatrix} V_0i \\ V_1i \\ V_2i \\ V_3i \\ V_5i \\ V_6i \\ V_7i \\ V_8i \end{bmatrix} = \begin{bmatrix} 0 \\ V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{9}$$

Name	Value [V]
V0i	0.000000000000000e+00
V1i	5.156959346000000e+00
V2i	4.904491610977624e+00
V3i	4.386308256622717e+00
V5i	4.940219576984384e+00
V6i	5.737650461673599e+00
V7i	-2.030644972553098e+00
V8i	-3.026746617887047e+00

Table 2: Nodal voltage values (t¡0)

By using Ohm's Law, we calculate the values of the currents passing in the resistors. The value of the current in v_s is simmetrical to I_R1 and the value of the current in V_d is simmetrical to I_R6.

Name	Value [A or V]
Ib	-2.554476012603980e-04
IR1i	-2.440917089113781e-04
IR2i	-2.554476012603928e-04
IR3i	-1.135589234901462e-05
IR4i	-1.224528396740627e-03
IR5i	-2.554476012603919e-04
IR6i	9.804366878292505e-04
Ivsi	-2.440917089113781e-04
IVdi	-9.804366878292496e-04

Table 3: Branch and voltage sources current values (t¡0)

2.2 Determining R_{eq}

Analysing the circuit now for t = 0, we notice that v_s equals 0 (short circuit), thus V_1 is also null. In order to determine the equivelant resistence (R_eq)of the circuit seen from the capacitors terminals and the time constant (much needed for the following subsections), we replace the capacitor with a voltage source $V_x = V_6 - V_8$ and do another nodal analysis to determine the current supplied by V_x, which will be called I_x. These two values determine R_eq with the equation: The system of equations that will be solved is:

$$R_eq = V_x/I_x \tag{10}$$

We use the values V_6 and V_8 from the previous subsection since the voltage drop at the terminals of the capacitor needs to be a continuous function, this means that there cannot be a sudden energy discontinuity in the capacitor. This is the most efficient procedure to determine the equivelant resistence in such a complex circuit, with this reasoning being based on the usage of the $Thevenin$ and $Norton$ theorems, where V_x is equivalent to $Thevenin's$ voltage and I_x is $Norton's$ current.

For this nodal analysis, we used almost the same equations that were used in the previous subsection, except, of course, the trivial one for V_1, since now $V_1 = 0$, and we replaced the node 6 equation with the new equation $V_x = V_6 - V_8$.

The system of equations that will be solved in form of matrix and with the assitance of Octave is:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & G_1 & -G_1-G_2-G_3 & G_2 & G_3 & 0 & 0 & 0 \\ 0 & 0 & K_b+G_2 & -G_2 & -K_b & 0 & 0 & 0 \\ 0 & -G_1 & G_1 & 0 & G_4 & 0 & G_6 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 & 0 & 0 & G_6-G_7 & G_7 \\ 0 & 0 & 0 & 0 & 1 & 0 & K_dG_6 & -1 \end{bmatrix} \begin{bmatrix} V_0t0 \\ V_1t0 \\ V_2t0 \\ V_3t0 \\ V_5t0 \\ V_6t0 \\ V_7t0 \\ V_8t0 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ V_x \\ 0 \\ 0 \end{bmatrix} \quad (11)$$

Name	Value [V]
V0t0	0.000000000000000e+00
V1t0	0.000000000000000e+00
V2t0	0.000000000000000e+00
V3t0	0.000000000000000e+00
V5t0	0.000000000000000e+00
V6t0	8.764397079560647e+00
V7t0	-0.000000000000000e+00
V8t0	0.000000000000000e+00

Table 4: Nodal voltage values (t¡0)

By using Ohm's Law, we calculate the values of the currents passing in the resistors.

Name	Value [A]
Ib	0.000000000000000e+00
IR1t0	0.000000000000000e+00
IR2t0	0.000000000000000e+00
IR3t0	0.000000000000000e+00
IR4t0	-0.000000000000000e+00
IR5t0	-2.807571481683833e-03
IR6t0	0.000000000000000e+00
IR7t0	-0.000000000000000e+00
Ivst0	0.000000000000000e+00
IVdt0	2.807571481683833e-03

Table 5: Branch and voltage sources current values (t=0)

Now that we have all values, we can use the Kirchhoff Current Law (KCL) in node 6 in order to compute the value of I_x

$$I_x = -K_b(V_2 - V_5) - (V_6 - V_5)G5 \quad (12)$$

With the values of V_x and I_x determined, we can calculate R_eq with 10 and the time constant value (EVA! PÕE AQUI O TAU, SFF) with the equation

$$\tau = R_eqC \quad (13)$$

Name	Value
Ixt0	-2.807571481683833e-03
Reqt0	3.121700422140000e+03
tau	3.167142502258496e-03

Table 6: I_x (in A), R_eq (in Ohm) and TAU (adimensional) values

2.3 Natural Solution with node analysis for t≥0

The aim of this section is to calculate the natural solution of $v_6n(t)$. The natural response is what the circuit does including the initial conditions (initial voltage of the capacitor) but with the input surpressed. Knowing the general solution ($v_6n(t) = V_6(+infinity) + (V_6(0) - V_6(+infinity))e^(-t/tau)$), and the fact that the capacitor begins charged but it discharges as the time passes, since it consumes energy, meaning that $V_6(+infinity) = 0$, we can write the following equation:

$$v_6n(t) = V_6(0)e^(- t/tau) \tag{14}$$

We also know that $V_x = V_6 - V_8$, and that the value of $V_8(0)$ is aproximately 0, which means that

$$v_6n(t) = V_xe^(- t/tau) \tag{15}$$

Hence, the graph of V_6n in function of the time, in the inteerval [0;20] ms is represent in 2. The result is no suprise, as it shows below, being a negative exponential graph.

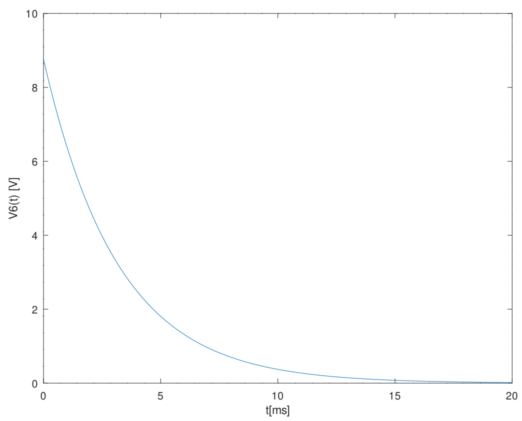

Figure 2: Natural solution $v_6n(t)$

2.4 Forced Solution with node analysis for t≥0

$$\begin{bmatrix} -G_1 & G_1+G_2+G_3 & -G_2 & -G_3 & 0 & 0 & 0 & \\ 0 & -G_2-K_b & G_2 & K_b & 0 & 0 & 0 & \\ 0 & K_b & 0 & K_b-G_5 & G_5+1/Z_c & 0 & -1/Z_c & \\ 0 & 0 & 0 & 0 & 0 & G_6+G_7 & -G_7 & \\ 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & K_dG_6 & -1 & \\ G_1 & -G_1 & 0 & -G_4 & 0 & -G_6 & 0 & \end{bmatrix} \begin{bmatrix} V_0 \\ V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 1 \\ 0 \\ 0 \end{bmatrix} \tag{16}$$

Name	Value [A or V]
Phase1	0.000000000000000e+00
Phase2	-5.224782481821870e-17
Phase3	1.727520704050529e-17
Phase5	-5.650387200837146e-17
Phase6	1.451830770421718e-01
Phase7	-5.650387200837146e-17
Phase8	-5.650387200837146e-17

Table 7: LEGENDA

Name	Value [A or V]
V1	1.000000000000000e+00
V2	9.510432954608804e-01
V3	8.505609531370399e-01
V5	9.579714024343183e-01
V6	5.888386465685786e-01
V7	3.937678845826408e-01
V8	5.869246613772022e-01

Table 8: LEGENDA

2.5 Ponto 5

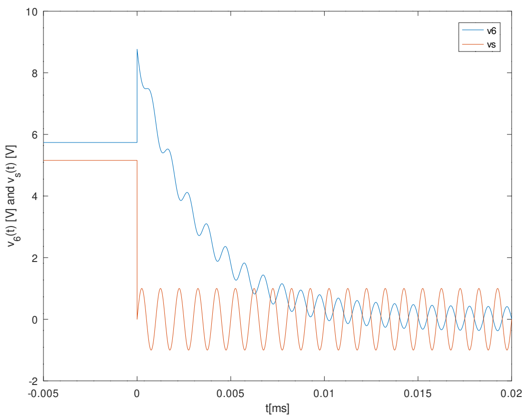

Figure 3: LEGENDA

2.6 Frequency Responses

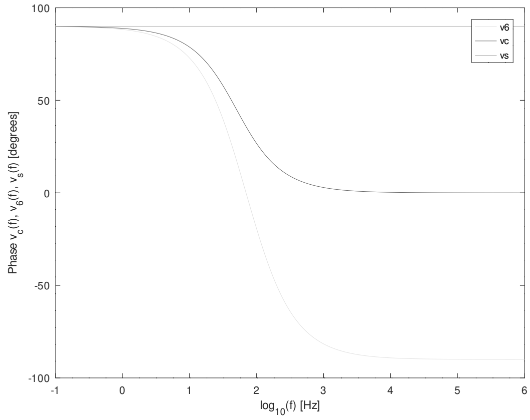

Figure 4: LEGENDA

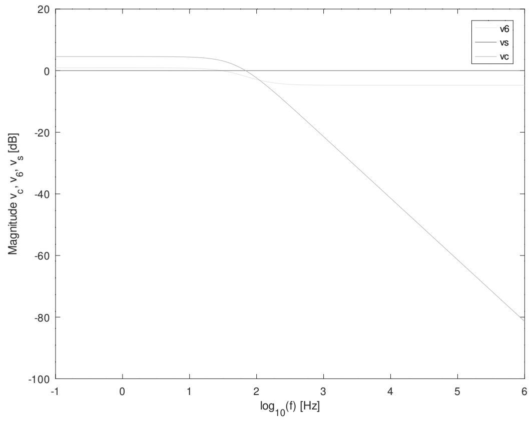

Figure 5: LEGENDA

After solving the system with Octave tools we get the Table results.In Table voltage values are identified with V and their measure is V(Volts), the remaining ones are current values so their units are A(Amperes).

3 Simulation Analysis

First of all, in this simulation is important to explain the creation of an auxiliary voltage V_{aux} (with a the same voltage of V_7) that was put between N7 and R7 as shown in Figure 6. Consequently, this led to the appearance of a node that we designated by N9 that has the same voltage as N7 (the drop voltage is 0). This was necessary because of Ngspice software requirements. After doing that ngspice was able to compute and determine all node voltages and current branches.

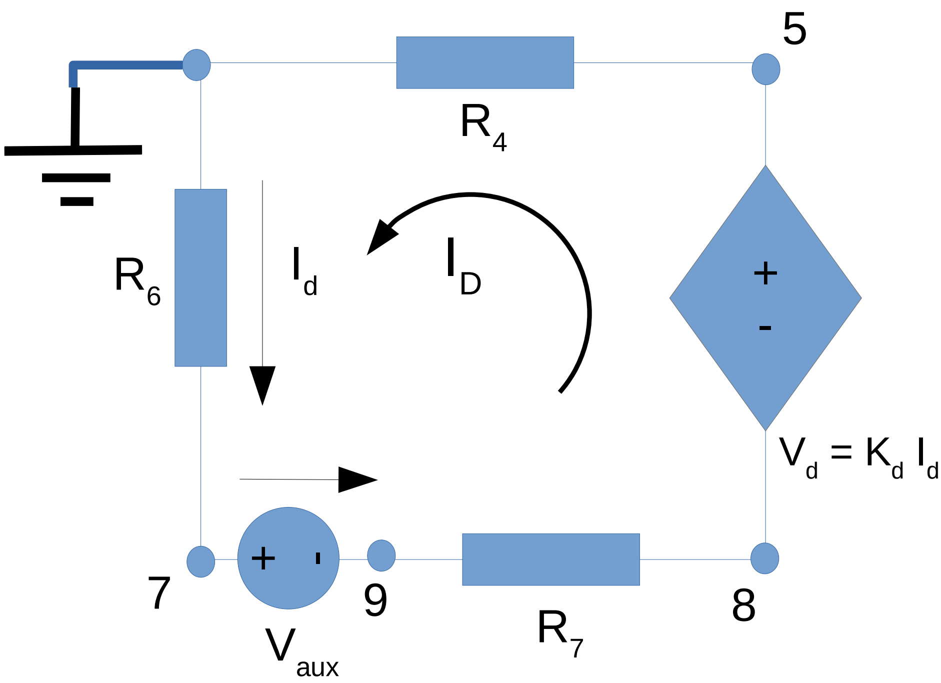

Figure 6: D Mesh with an additional voltage source

3.1 Operating Point Analysis for t<0

The Table 9 shows the simulated operating point results for the circuit described in Figure 1, considering t<0, which means $V_s(t)$=V_s.

Name	Value [A or V]
@hd[i]	-9.80437e-04
@vs[i]	-2.44092e-04
@c[i]	0.000000e+00
@gb[i]	-2.55448e-04
@r1[i]	-2.44092e-04
@r2[i]	-2.55448e-04
@r3[i]	-1.13559e-05
@r4[i]	-1.22453e-03
@r5[i]	-2.55448e-04
@r6[i]	9.804367e-04
@r7[i]	9.804367e-04
n1	5.156959e+00
n2	4.904492e+00
n3	4.386308e+00
n5	4.940220e+00
n6	5.737650e+00
n7	-2.03064e+00
n8	-3.02675e+00
n9	-2.03064e+00

Table 9: Operating point. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

3.2 Operating Point Analysis for t=0

This second part covers the simulation of the circuit for t=0. To do that the capacitor is replaced with a voltage source V_x = V_6 - V_8 using the values obtained in the previous section. This is necessary because for t$\leq$0 the voltage in the capacitor is the same. So to mantain the boundary conditions V_6 and V_8 the capacitor is replaced with the initial voltage source.

Name	Value [A or V]
@hd[i]	2.807571e-03
@vs[i]	0.000000e+00
@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]	-2.80757e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
n1	0.000000e+00
n2	0.000000e+00
n3	0.000000e+00
n5	0.000000e+00
n6	8.764397e+00
n7	0.000000e+00
n8	0.000000e+00
n9	0.000000e+00

Table 10: Operating point. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

3.3 Natural Solution

In order to study the natural solution response of the circuit in the interval [0;20]ms using the boundary conditions (V_6 and V_8) calculated before, a transient analysis was realized. Fig. 7 shows the plot of the required results.

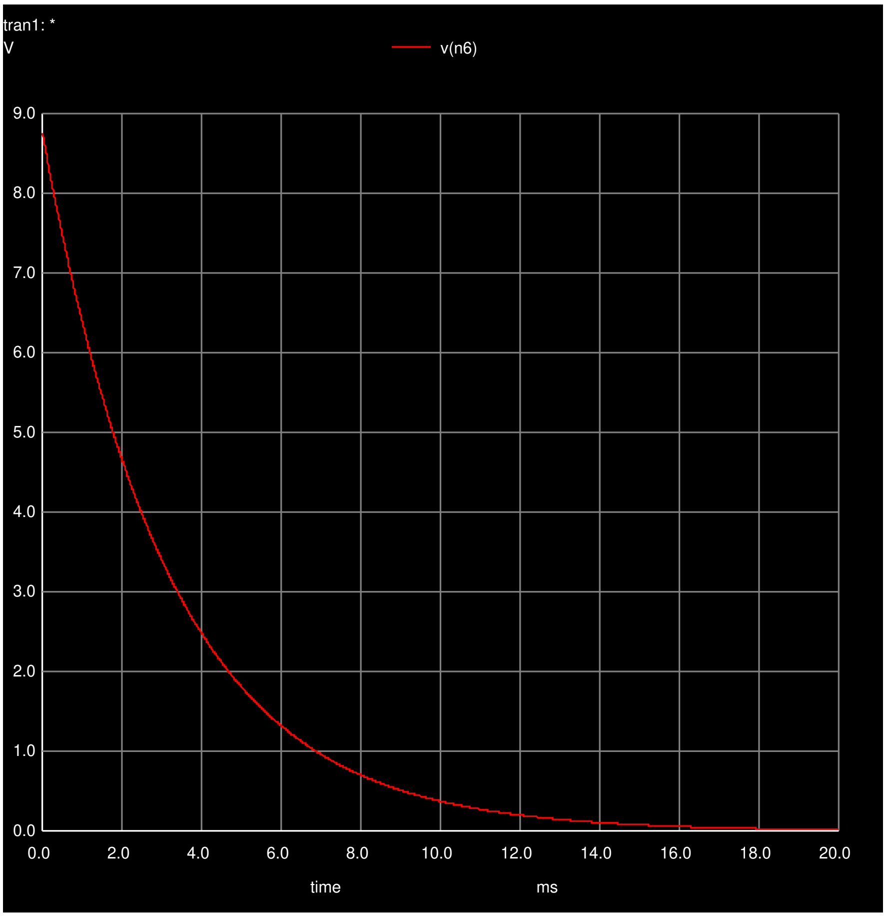

Figure 7: Natural Response of V_6

3.4 Total Solution

In the fourth section a total response of node 6 was perfomed, using the same procedure and interval of 3.3 with a initial sinusoidal voltage source $V_s(t)$ that has a frequency of 1000Hz. Fig. 8 shows the plot of the required results.

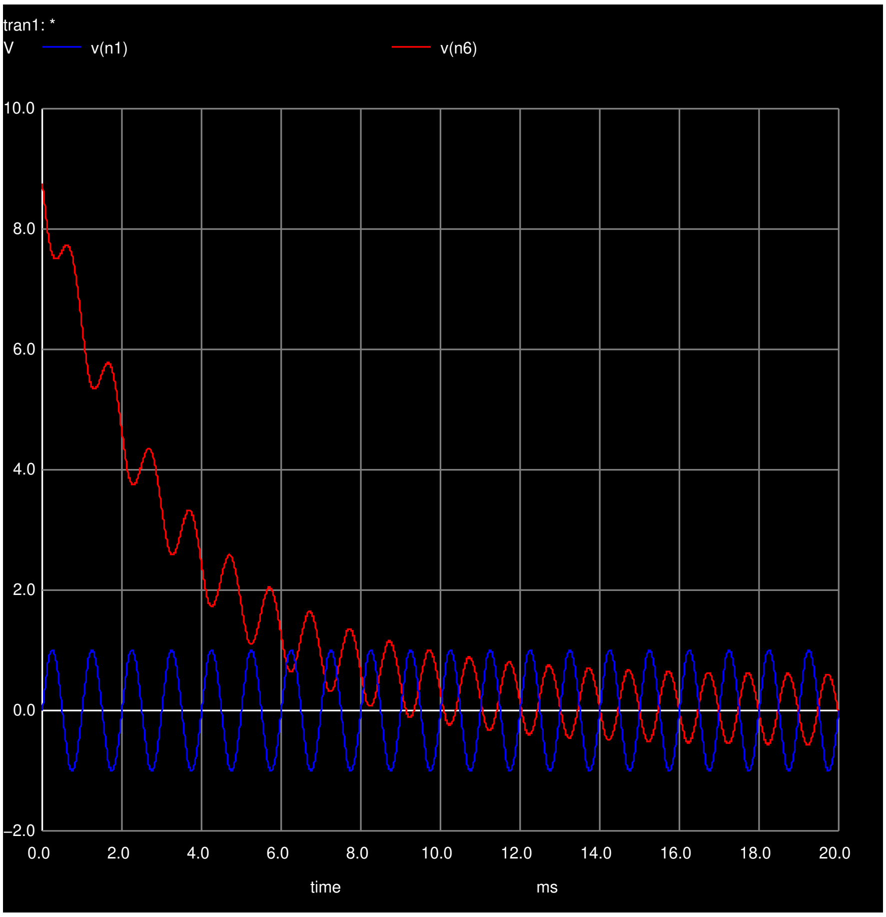

Figure 8: Total Response of V_6 and V_s

It is important to note that during the time interval considered the voltage in the capacitor diminuish until its phase differs π from the voltage source.

3.5 Frequency Responses

In this part of the chapter a small signal analysis was realized. The frequency response is simulated on node 6 for the frequency between 0.1Hz to 1Mhz. Since V_s is the source of the frequency and V_6 is an output voltage is expected to V_s to remain constant and V_6 to decrease. This can be seen in the following figures 9 and 10.

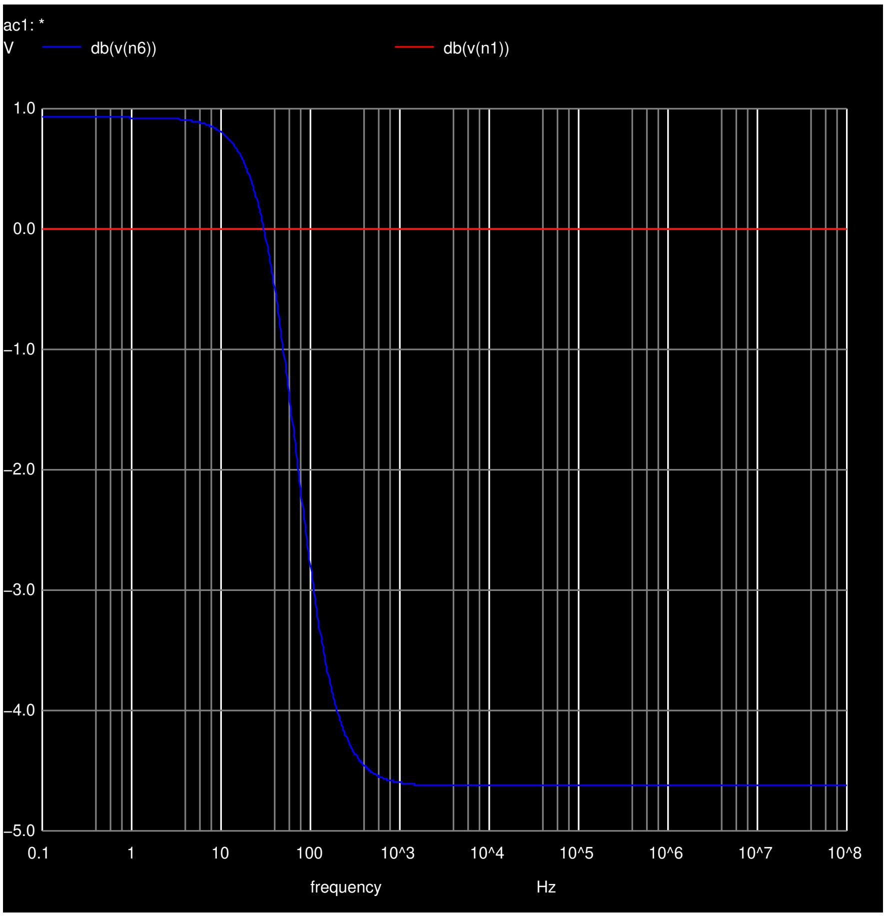

Figure 9: Magnitude response for V_6 and V_s (in dB)

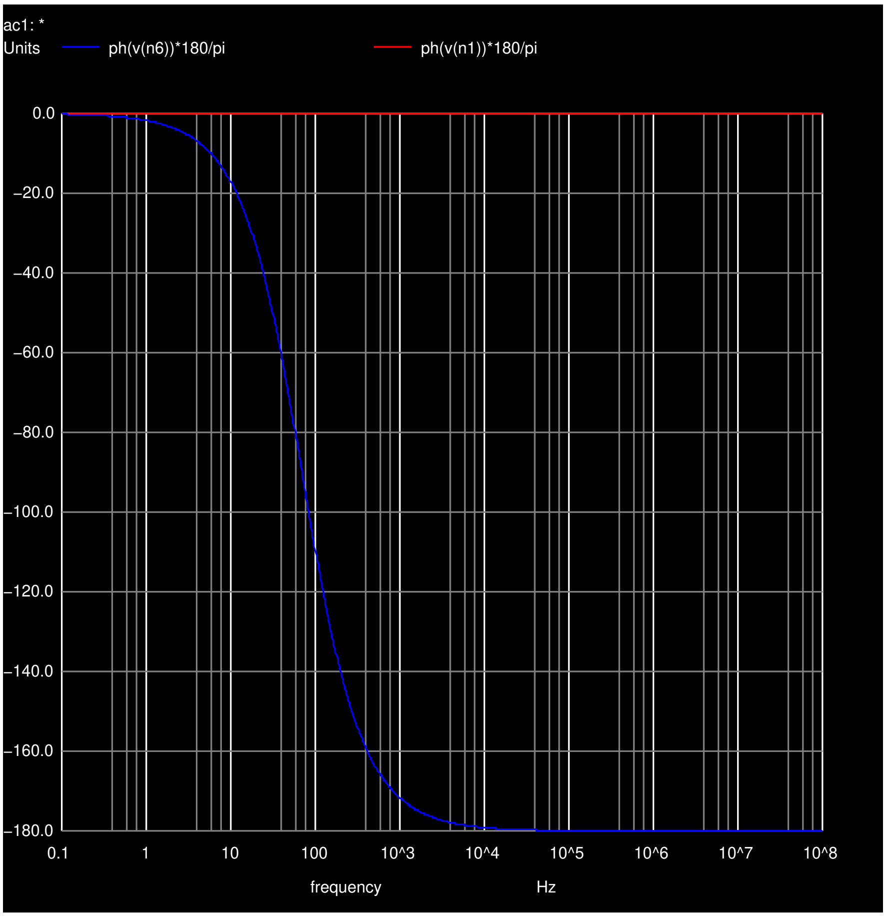

Figure 10: Phase response for V_6 and V_s (in degrees)

By observing the Table 9,Table 3 and Table 4 values it's possible to conclude that the simulated results are the same as the theoretical results.

However, was also calculated the relative errors made in order to understand the accuracy of the results.

Related to that calculations, it was noticed that in an experimental procedure, the calculation of relative errors is made by comparing experimental values and theorical values, meaning that the decimal places used in the theoretical value to be considered will be in accordance with the experimentally obtained places.

Considering that, in our case, the experimental values are obtained through NGSice, we must use theoretical values with the same number of decimal places returned by the simulation for calculating the errors, as it has a number of decimal places lower than that of the octave. By doing this calculating, all error values absolute and relative are equal to zero.

Therefore, we can see that the order of magnitude of the errors will be residual.

4 Side by Side Comparison

Name	Value [A or V]
@hd[i]	-9.80437e-04
@vs[i]	-2.44092e-04
@c[i]	0.000000e+00
@gb[i]	-2.55448e-04
@r1[i]	-2.44092e-04
@r2[i]	-2.55448e-04
@r3[i]	-1.13559e-05
@r4[i]	-1.22453e-03
@r5[i]	-2.55448e-04
@r6[i]	9.804367e-04
@r7[i]	9.804367e-04
n1	5.156959e+00
n2	4.904492e+00
n3	4.386308e+00
n5	4.940220e+00
n6	5.737650e+00
n7	-2.03064e+00
n8	-3.02675e+00
n9	-2.03064e+00

Table 11: Simulation nodal voltage results. All variables are expressed in Volt or Ampere. (Ngspice)

Name	Value [A or V]
IVdi	-9.804366878292496e-04
Ivsi	-2.440917089113781e-04
Ib	-2.554476012603980e-04
IR1i	-2.440917089113781e-04
IR2i	-2.554476012603928e-04
IR3i	-1.135589234901462e-05
IR4i	-1.224528396740627e-03
IR5i	-2.554476012603919e-04
IR6i	9.804366878292505e-04
V1i	5.156959346000000e+00
V2i	4.904491610977624e+00
V3i	4.386308256622717e+00
V5i	4.940219576984384e+00
V6i	5.737650461673599e+00
V7i	-2.030644972553098e+00
V8i	-3.026746617887047e+00

Table 12: Theoretical nodal voltage results. All variables are expressed in Volt.(Octave)

TEXTO NAO PERTENCE AQUI In this simulation is important to explain the creation of an auxiliary voltage V_b (with a voltage equal to $0V$) that was put between N6 and R7 as shown in Figure 6. Consequently, this led to the appearance of a node that we designated by N8 that has the same voltage as N6.

This was necessary because of Ngspice software requirements.

By observing the Table 9,Table 3 and Table 4 values it's possible to conclude that the simulated results are the same as the theoretical results.

However, was also calculated the relative errors made in order to understand the accuracy of the results.

Related to that calculations, it was noticed that in an experimental procedure, the calculation of relative errors is made by comparing experimental values and theorical values, meaning that the decimal places used in the theoretical value to be considered will be in accordance with the experimentally obtained places.

Considering that, in our case, the experimental values are obtained through NGSice, we must use theoretical values with the same number of decimal places returned by the simulation for calculating the errors, as it has a number of decimal places lower than that of the octave. By doing this calculating, all error values absolute and relative are equal to zero.

Therefore, we can see that the order of magnitude of the errors will be residual.

Name	Value [A or V]
@hd[i]	2.807571e-03
@vs[i]	0.000000e+00
@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]	-2.80757e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
n1	0.000000e+00
n2	0.000000e+00
n3	0.000000e+00
n5	0.000000e+00
n6	8.764397e+00
n7	0.000000e+00
n8	0.000000e+00
n9	0.000000e+00

Table 13: Simulation nodal voltage results. All variables are expressed in Volt or Ampere. (Ngspice)

Name	Value [A or V]
IVdt0	2.807571481683833e-03
Ivst0	0.000000000000000e+00
Ib	0.000000000000000e+00
IR1t0	0.000000000000000e+00
IR2t0	0.000000000000000e+00
IR3t0	0.000000000000000e+00
IR4t0	-0.000000000000000e+00
IR5t0	-2.807571481683833e-03
IR6t0	0.000000000000000e+00
IR7t0	-0.000000000000000e+00
V1t0	0.000000000000000e+00
V2t0	0.000000000000000e+00
V3t0	0.000000000000000e+00
V5t0	0.000000000000000e+00
V6t0	8.764397079560647e+00
V7t0	-0.000000000000000e+00
V8t0	0.000000000000000e+00

Table 14: Theoretical nodal voltage results. All variables are expressed in Volt.(Octave)

5 Conclusion

The objective of this laboratory assignment is to analyse the circuit and solve it. After discussing with all members of the group we can conclude that this goal was achieved.

As presented the results obtained by the Octave math tool and Ngspice simulation tool are the same. This perfect match was achieved because the circuit is not very complex, being only composed of linear components so both models (Ngspice and Octave) used the same methods to solve the circuit and therefore the results can not differ.

Also, all the components used in this circuit (resistors, branches, nodes,...) are perfect this means they don't dissipate energy by heating.This is one of the advantages of simulating rather than doing it on the laboratory, the other one being the elimination of ”humam error”. It's known that this type of error can influence the experimental results causing considerable relative errors, which in our case weren't made.

Finally, this similarity proves the efficiency and importance of the nodal and mesh methods.