TÉCNICO LISBOA

Circuit Theory and Electronics Fundamentals

Engineering Physics

Lab 2: Circuit Analysis

Beatriz Rosalino (96514), Mariana Ribeiro (96552), Sofia Guerreiro (96567)

8 April, 2021

Contents

1 Introduction

The main goal of this work is to analyze an RC circuit and study its various responses to a voltage source that changes over time. First, we studied the behavior of the circuit when the voltage imposed on the capacitor was constant and non zero (Section 2.1), then we studied the natural response of the capacitor (voltage source imposing voltage equal to 0) in Section 2.3, the forced response (voltage source imposing sinusoidal voltage) in Section 2.4, over time. We also studied the circuit for different frequencies of the sinusoidal signal, plotting the voltages at the capacitor and the nodes of its terminals as functions of frequency Section 2.6.

Our circuit (Figure 1) consists of 7 resistors, 2 voltage sources - 1 independent, and 1 current controlled dependent one, 1 independent current source and 1 capacitor.
The voltage provided by the independent source follows the equation:

$$v_s(t) = \begin{cases} V_s, & if \quad t \leq 0 \\ sin(2\pi f t), & if \quad t>0 \end{cases} \tag{1}$$

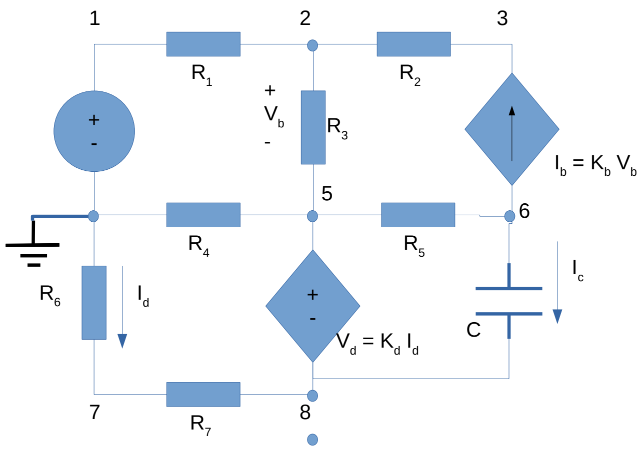

Figure 1: Circuit

We wrote a system of equations for each theoretical analysis, from Kirchhoff and Ohm's Laws. To get the solutions and plots for these analysis, we used *octave*, which solved our systems of equations efficiently and gave us all currents and voltages for all branches. We then used *ngspice* to get a simulation of this circuit, expecting to obtain the same results. We compared the results from different methods in the conclusion (Section 5).

2 Theoretical analysis

2.1 Operating point analysis for t<0

For t<0, from equation (1) we can see that $v_s(t) = V_s$. That means the voltage source drives constant voltage V_s. We consider that enough time has passed for the capacitor to fully charge from this constant voltage, so the voltage at nodes 6 and 8 are the same and there is no current flowing through the capacitor (it now acts as an open circuit). Removing the part of the circuit that is an open circuit we get a ”new” circuit, that is easy to analyse:

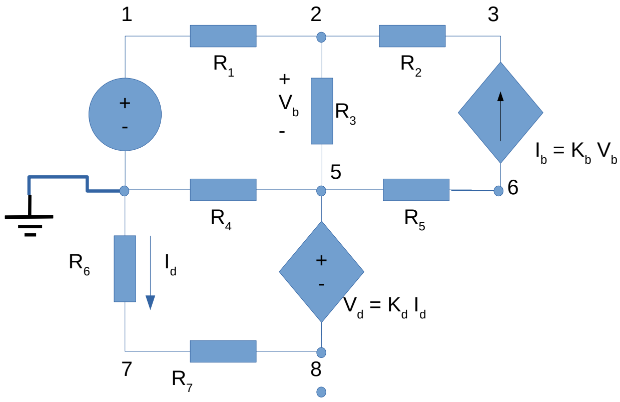

Figure 2: Circuit for t<0

In this circuit, there are 8 nodes, one of which is a ground node (the one that connects R_4, the voltage source and R_6. Therefore, to solve the circuit we must find 7 independent equations. To discover the equations for each node we need to use Kirchhoff’s Current Law, which states that the sum of all the currents entering and leaving a node must be equal to zero. For simplicity instead of using directly the currents we will also use Ohm’s Law and we will consider that all currents are leaving the nodes.
The equations for the 7 nodes are:

$$V_1 = V_s \tag{2}$$

$$\frac{V_2 - V_1}{R_1} + \frac{V_2 - V_3}{R_2} + \frac{V_2 - V_5}{R_3} = 0 \tag{3}$$

$$V_5 - V_2 + \frac{V_3 - V_2}{R_2 K_b} = 0 \tag{4}$$

$$\frac{V_5}{R_4} + \frac{V_5 - V_6}{R_5} + \frac{V_8 - V_7}{R_7} + \frac{V_5 - V_2}{R_3} = 0 \tag{5}$$

$$\frac{V_3 - V_2}{R_2} + \frac{V_6 - V_5}{R_5} = 0 \tag{6}$$

$$V_7 + R_6 \left(\frac{V_7 - V_8}{R_7} \right) = 0 \tag{7}$$

$$V_8 - V_5 + K_d \left(\frac{V_8 - V_7}{R_7} \right) = 0 \tag{8}$$

The next step is to solve the system of equations:

$$
\begin{pmatrix}
1 & 0 & 0 & 0 & 0 & 0 & 0 \\
-\frac{1}{R_1} & \frac{1}{R_1}+\frac{1}{R_2}+\frac{1}{R_3} & -\frac{1}{R_2} & -\frac{1}{R_3} & 0 & 0 & 0 \\
0 & -1-\frac{1}{R_2K_b} & \frac{1}{R_2K_b} & 1 & 0 & 0 & 0 \\
0 & -\frac{1}{R_3} & 0 & \frac{1}{R_3}+\frac{1}{R_4}+\frac{1}{R_5} & -\frac{1}{R_5} & -\frac{1}{R_7} & \frac{1}{R_7} \\
0 & -\frac{1}{R_2} & \frac{1}{R_2} & -\frac{1}{R_5} & \frac{1}{R_5} & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 1+\frac{R_6}{R_7} & -\frac{R_6}{R_7} \\
0 & 0 & 0 & -1 & 0 & -\frac{K_d}{R_7} & 1+\frac{K_d}{R_7}
\end{pmatrix}
\begin{pmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix}
=
\begin{pmatrix} V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}
\tag{9}
$$

2.2 Determination of Equivalent Resistor(R_{eq})

Just as suggested, we made $V_s = 0$ and 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 section 2.1. Then, we did once again the nodal analysis, with the purpose of obtaining values for I_x and V_x, since we need this to get R_{eq} for further analysis. To get this resistance, on the capacitor terminals, we need to use Thévenin's theorem. We are considering the capacitor, so, to get Thévenin's equivalent, we need to have an open circuit at the capacitor terminals. The rest of the circuit will be a voltage source V_{Th} in series with a resistor of $R_{eq} = R_{Th}$ so the voltage drop across R_{eq} is $V_{Th} - 0 = V_{Th}$. In this way, V_{Th} will be equal to the open-circuit voltage, which is the voltage at the capacitor terminals. So to get Thévnin's equivalent circuit, we susbtitute the capacitor with a voltage source of voltage V_{Th}. This voltage source will have to have voltage V_x to guartee the necessary continuity of voltage in the capacitor, from t<0.

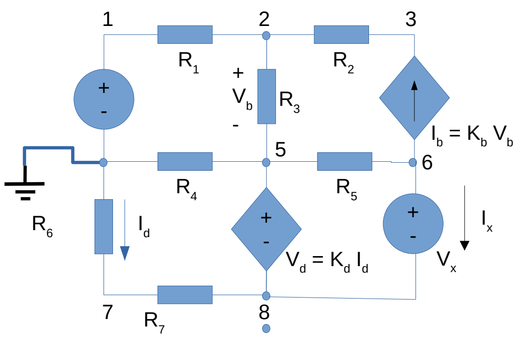

Figure 3: Circuit for t=0

Now, we only need 6 equations, because the GND node and node 1 become the same node.

For node 2, we get:

$$\frac{V_2}{R_1}+\frac{V_2-V_3}{R_2}+\frac{V_2-V_5}{R_3}=0 \Leftrightarrow V_2\left(\frac{1}{R_1}+\frac{1}{R_2}+\frac{1}{R_3}\right)-\frac{V_3}{R_2}-\frac{V_5}{R_3}=0 \tag{10}$$

For node 3:

$$\frac{V_3-V_2}{R_2\times K_b}+V_5-V_2=0 \Leftrightarrow V_2\left(-\frac{1}{R_2\times K_b}-1\right)+\frac{V_3}{R_2\times K_b}+V_5=0 \tag{11}$$

For node 5, we get:

$$\frac{V_5}{R_4}+\frac{V_5-V_2}{R_3}+\frac{V_5-V_6}{R_5}+i_d=0 \tag{12}$$

Where i_d is the current passing trough the dependent voltage source, V_d, ($i_d=-(I_d+I_x)$).
For node 6, we get:

$$I_b+\frac{V_6-V_5}{R_5}+I_x=0 \Leftrightarrow I_x=\frac{V_2-V_3}{R_2}+\frac{V_5-V_6}{R_5} \tag{13}$$

And also:

$$V_6-V_8=V_x \tag{14}$$

For node 7:

$$-I_d+\frac{V_7-V_8}{R_7}=0 \Leftrightarrow \frac{V_7}{R_6}-\frac{V_7-V_8}{R_7}=0 \Leftrightarrow V_7\left(\frac{1}{R_6}+\frac{1}{R_7}\right)-\frac{V_8}{R_7}=0 \tag{15}$$

Combining equations ((12)), ((13)) and noticing that $I_d=\frac{V_7-V_8}{R_7}$:

$$V_2\left(-\frac{1}{R_2}-\frac{1}{R_3}\right)+\frac{V_3}{R_2}+V_5\left(\frac{1}{R_3}+\frac{1}{R_4}\right)-\frac{V_7}{R_7}+\frac{V_8}{R_7}=0 \tag{16}$$

Knowing that $I_d=\frac{V_d}{K_d}$ and $I_d=\frac{V_7-V_8}{R_7}$, we get the final equation:

$$V_8\left(\frac{1}{R_7}-\frac{1}{K_d}\right)-\frac{V_7}{R_7}+\frac{V_5}{K_d} \tag{17}$$

The next step is to solve the following system of equations:

$$\begin{pmatrix} \frac{1}{R_1}+\frac{1}{R_2}+\frac{1}{R_3} & -\frac{1}{R_2} & -\frac{1}{R_3} & 0 & 0 & 0 \\ -\frac{1}{R_2K_b}-1 & \frac{1}{R_2K_b} & 1 & 0 & 0 & 0 \\ -\frac{1}{R_2}-\frac{1}{R_3} & \frac{1}{R_2} & \frac{1}{R_3}+\frac{1}{R_4} & 0 & -\frac{1}{R_7} & \frac{1}{R_7} \\ 0 & 0 & 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 & \frac{1}{R_6}+\frac{1}{R_7} & -\frac{1}{R_7} \\ 0 & 0 & \frac{1}{K_d} & 0 & -\frac{1}{R_7} & \frac{1}{R_7}-\frac{1}{K_d} \end{pmatrix}\begin{pmatrix} V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix}=\begin{pmatrix} 0 \\ 0 \\ 0 \\ V_x \\ 0 \\ 0 \end{pmatrix} \tag{18}$$

Finally, we can calculate I_x using equation (13), the equivalent resistance R_{eq} and the time constant, τ:

$$R_{eq}=\frac{V_x}{I_x} \tag{19}$$

$$\tau=R_{eq}\times C \tag{20}$$

2.3 Natural Solution

The natural solution only depends on initial charge (voltage) and on the time constant. The voltage source V_s needs to be turned off to get the natural response of the system, so $V_s = 0$ and $V_{6n}(t)$ becomes

$$V_{6n}(t) = Ae^{-\frac{t}{\tau}} \tag{21}$$

With the results obtained in section 2.2, we get both $A = V_{6n}(0) = V_x$,and the time constant τ. We will plot the results in the time interval [0, 20] ms.

2.4 Forced solution

For t>0, the voltage source has the form $v_s(t) = sin(2\pi ft)$, where f represents the frequency of the sinusoidal excitation. We expect that if we allow the system to evolve for a sufficient amount of time (enough that the natural solution dies out), the voltage at node 6 (the node near the capacitor) will also tend towards a sinusoidal signal.
Since this circuit has a lot of components, it helps to use phasor notation, in which the voltages become the real or imaginary part of complex vectors (called phasors), with information about the magnitude and phase of the signal. Since our function is a sine, we will use the imaginary component of the phasors. When we do nodal analysis we can ignore the time dependance and introduce it at the end, multiplying the phasors by the term $e^{2\pi ftj}$, where f is the forced frequency (imposed by the voltage source), taking finally the imaginary combination of sine and cosine functions:

$$V_6(t) = R(V_6)sin(\omega t) + Im(V_6)cos(\omega t) \tag{22}$$

where $\omega = 2\pi f$. Since $v_s(t) = sin(2\pi ft)$, the magnitude of $V_s = 1$ and its phase is 0. Since we are using phasors, where voltages and currents are complex numbers, we also need to substitute the capacitor with its impedance (which is the resistance equivalent for phasors). The impedance of the capacitor can be expressed by $z_c = \frac{1}{2\pi fCj}$.
The new nodal equations become:

$$V_1 = V_s = 1 \tag{23}$$

$$\frac{V_2 - V_1}{R_1} + \frac{V_2 - V_3}{R_2} + \frac{V_2 - V_5}{R_3} = 0 \tag{24}$$

$$V_5 - V_2 + \frac{V_3 - V_2}{R_2 K_b} = 0 \tag{25}$$

$$\frac{V_5}{R_4} + \frac{V_5 - V_6}{R_5} + \frac{V_8 - V_7}{z_c} + \frac{V_5 - V_2}{R_3} = 0 \tag{26}$$

$$\frac{V_3 - V_2}{R_2} + \frac{V_6 - V_5}{R_5} + \frac{V_6 - V_8}{z_c} = 0 \tag{27}$$

$$V_7 + R_6\left(\frac{V_7 - V_8}{R_7}\right) = 0 \tag{28}$$

$$V_8 - V_5 + K_d\left(\frac{V_8 - V_7}{R_7}\right) = 0 \tag{29}$$

The system of equations in matrix form:

$$\begin{pmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ -\frac{1}{R_1} & \frac{1}{R_1}+\frac{1}{R_2}+\frac{1}{R_3} & -\frac{1}{R_2} & -\frac{1}{R_3} & 0 & 0 & 0 \\ 0 & -1-\frac{1}{R_2K_b} & \frac{1}{R_2K_b} & 1 & 0 & 0 & 0 \\ 0 & -\frac{1}{R_3} & 0 & \frac{1}{R_3}+\frac{1}{R_4}+\frac{1}{R_5} & -\frac{1}{R_5} & -\frac{1}{z_c} & \frac{1}{z_c} \\ 0 & -\frac{1}{R_2} & \frac{1}{R_2} & -\frac{1}{R_5} & \frac{1}{R_5}+\frac{1}{z_c} & 0 & -\frac{1}{z_c} \\ 0 & 0 & 0 & 0 & 0 & 1+\frac{R_6}{R_7} & -\frac{R_6}{R_7} \\ 0 & 0 & 0 & -1 & 0 & -\frac{K_d}{R_7} & 1+\frac{K_d}{R_7} \end{pmatrix} \begin{pmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix} = \begin{pmatrix} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix} \quad (30)$$

2.5 Final Solution

The final solution for $V_6(t)$ is the superposition the natural and forced solutions:

$$V_6(t) = V_{6n}(t) + V_{6f}(t) \quad (31)$$

For t<0, $V_6(t)$ is a constant (and so is $V_s(t)$), but in the instant t = 0, $V_6(0) = V_x$ and $V_s(0) = 0$. Then, for t>0, $V_s(t) = sin(2\pi ft)$ and $V_6(t)$, from de sections 2.3 and 2.4, is:

$$V_6(t) = R(V_6)sin(wt) + Im(V_6)cos(wt) + Aexp(-\frac{t}{\tau}) \quad (32)$$

Finally, using *Octave*, we plotted both $V_s(t)$ and $V_6(t)$ in the same figure in the interval [-5, 20] ms.

2.6 Frequency response

A transfer function is a function that computes output from input. In the case of an RC circuit such as the one being studied here, the transfer function is simply the quotient of phasors, the phasor correspondent to the voltage of the capacitor ($v_c(t)$) and the phasor correspondent to the voltage source ($v_s(t)$):

$$T(\omega) = \frac{\widetilde{v_c}}{\widetilde{v_s}} = \frac{1}{1 + j\omega RC}$$

Here,$\widetilde{v_s} = 1$.

$$T(\omega) = \widetilde{v_c} = \frac{1}{1 + j\omega RC}$$

We could then get the plot of $v_c(f)$ using this expression.We know V_8 from previous analysis, and knowing it is independent from frequency, since the voltage in node 7, not connected to the capacitor or the voltage source V_s is also independent of frequency, and V_8 is related only to this voltage by equation (28). This voltage is known from previous analysis (Section 2.4), and knowing the tansfer function, from the relation

$$V_c = V_6 - V_8$$

we can also plot V_6 as a function of frequency.

3 Simulation

In this circuit, we have a current dependent voltage source between nodes 5 and 8, so that, to sense the controlling current, I_d, between nodes GND and 7, we need to use a voltage source of voltage 0V, to sense said current. For this purpose, we placed this source between node

7 and a new node - node 9-, which connects the negative terminal of the voltage source and resistor 7, so that it is in series with resistor 6, through which I_d flows, and therefore senses that same current.

3.1 Operating Point Analysis for t<0

To simulate the response of the circuit for t<0, where the voltage source introduces to the circuit a constant voltage, V_s, the current through the capacitor is zero, so there is an open circuit between nodes 6 and 8. The results of the operating point analysis for this circuit, t<0, obtained using *ngspice*, can be seen in Table 2.

3.2 Operating Point Analysis for t=0

To get the operating point analysis for t=0, knowing $v_s(0) = 0$, we substitute the capacitor, between nodes 6 and 8, with a voltage source $V_x = V_6 - V_8$, where V_6 and V_8 are the voltage for nodes 6 and 8, respectively, obtained in the previous op analysis (for t<0). This is needed so continuity of voltage in the capacitor is guaranteed and so we can use the voltages obtained with this analysis as initial conditions in the following sections. The results of this analysis, using *ngspice* can be seen in Table 4.

3.3 Transient Analysis for Natural Response

From now on, we have the capacitor in its place, between nodes 6 and 8.
To simulate the natural response of the circuit, the source V_s was turned off (the voltage has to be 0V to get the natural response), and we used boundary conditions of V_6 and V_8(voltages in nodes 6 and 8), as the values obtained in t=0. These conditions will also be used in the latter sections. We did transient analysis, for the [0,20]ms time interval, with a 0.02ms time step. The plot of $v_6(t)$ is shown in Figure 5.

3.4 Analysis of Total Response

For this section, we now have V_s as a sine wave of frequency f=1k Hz, and amplitude 1 ($V_s = sin(2\pi ft)$). A transient analysis was done, for nodes 6 (V6) and 1 (Vs),again for the [0,20]ms time period with a 0.02ms time step. We can then see the results of the voltage introduced to the circuit (Vs) and the total response on node 6, from both this stimulus from Vs and the natural response of the system. This can be seen in Figure 8.

3.5 Frequency Analysis

To evaluate the circuit when the frequency changes, we used ac analysis, by decades - using a frequency logscale. Frequencies were considered from 0.1 Hz to 1MHz and we used 100 points to plot both the magnitude and phase of voltages of nodes 1, 6 and of the capacitor v_c, which corresponds to $V_6 - V_8$, as functions of frequency. The magnitude plot can be seen in Figure 10, and the phase plot in Figure 12.

4 Results

4.1 Analysis for t$<$0

The solution of the matrix and the respective table with the nodal analysis results from section 2.1, obtained using *octave*, can be seen below.

$$\begin{pmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix} = \begin{pmatrix} 5.233347 \\ 4.993818 \\ 4.495480 \\ 5.028594 \\ 5.796288 \\ -2.004014 \\ -3.034274 \end{pmatrix}$$

Table 1: Nodal Analysis results for t¡0

Node	Voltage (V)	R	Current(mA)
1	5.233347	R_1	-0.233449
2	4.993818	R_2	-0.244673
3	4.495480	R_3	-0.011224
4	0	R_4	1.225637
5	5.028594	R_5	0.244673
6	5.796288	R_6	-0.992188
7	-2.004014	R_7	-0.992188
8	-3.034274	I_b	-0.244673

The respective ngspice simulation results, to compare with the previous ones can be seen in Table 2.

Name	Value [A or V]
@g1[i]	-2.44673e-04
@r1[i]	-2.33449e-04
@r2[i]	-2.44673e-04
@r3[i]	-1.12237e-05
@r4[i]	1.225637e-03
@r5[i]	2.446726e-04
@r6[i]	-9.92188e-04
@r7[i]	-9.92188e-04
v(1)	5.233347e+00
v(2)	4.993818e+00
v(3)	4.495481e+00
v(5)	5.028594e+00
v(6)	5.796288e+00
v(7)	-2.00401e+00
v(8)	-3.03427e+00
v(9)	-2.00401e+00

Table 2: Operating point. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

4.2 Analysis for t=0

The solution of the matrix and the respective table with the nodal analysis results from section 2.2, obtained using *octave*, can be seen below.Here we also get I_x and R_{eq}.

$$\begin{pmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix} = \begin{pmatrix} 0.000000 \\ 0.000000 \\ 0.000000 \\ 0.000000 \\ 8.830562 \\ 0.000000 \\ 0.000000 \end{pmatrix}$$

Table 3: Nodal Analysis results

Node	Voltage (V)	R	Current(mA)
1	0.000000	R_1	0.000000
2	0.000000	R_2	0.000000
3	0.000000	R_3	0.000000
4	0	R_4	0.000000
5	0.000000	R_5	2.814401
6	8.830562	R_6	0.000000
7	0.000000	R_7	0.000000
8	0.000000	I_b	0.000000

$Ix = -2.814401mA$ $Req = 3.137634k\Omega$ $\tau = 0.003283$ The same analysis using ngspice yields the following results.

Name	Value [A or V]
@g1[i]	-2.12269e-18
@r1[i]	2.025320e-18
@r2[i]	2.122693e-18
@r3[i]	-9.73728e-20
@r4[i]	4.329577e-19
@r5[i]	2.814401e-03
@r6[i]	-4.33681e-19
@r7[i]	-8.67138e-19
v(1)	0.000000e+00
v(2)	-2.07807e-15
v(3)	-6.40146e-15
v(5)	-1.77636e-15
v(6)	8.830562e+00
v(7)	8.759452e-16
v(8)	1.776357e-15
v(9)	8.759452e-16

Table 4: Operating point. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

Here we also got, for Ix[1], -2.81440 mA.

[1]using 96514 as the generating number

4.3 t>0: Natural Response

The plot for the natural of the voltage in node 6, as a function of time can be seen in Figure 4.

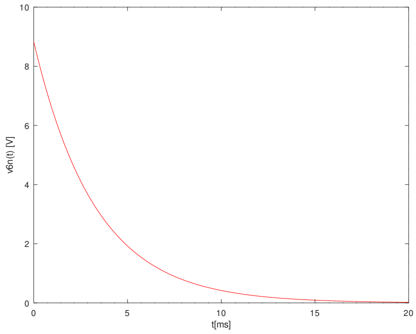

Figure 4: Natural response of V6 [0,20]ms using octave

The same plot, obtained by transient analysis using *ngspice*, is in Figure 5.

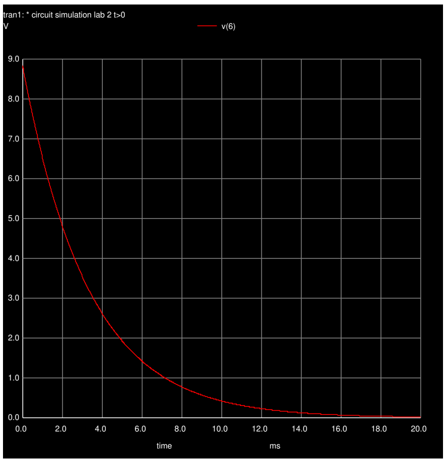

Figure 5: Natural response of V6 in [0,20]ms using ngspice

As we can see, we get the same result using both methods.

4.4 t>0: Forced Response

The phasor nodal voltages obtained can be seen in the table below.

$$\begin{pmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix} = \begin{pmatrix} 5.233347 \\ 0.954230 \\ 0.859007 \\ 0.960875 \\ -0.579796 + -0.000082i \\ -0.382932 \\ -0.579796 \end{pmatrix}$$

Table 5: Nodal Analysis results for t>0

	Voltage (V)
1	5.233347
2	0.954230
3	0.859007
4	0
5	0.960875
6	-0.579796 + -0.000082 i
7	-0.382932
8	-0.579796

The plot of the forced component of V6 can be seen in Figure 6

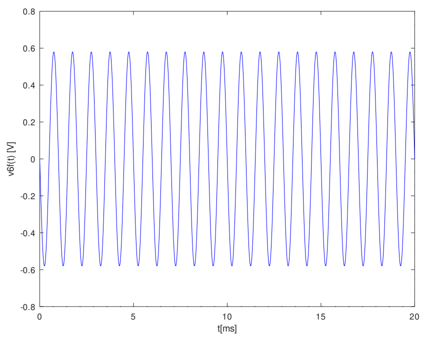

Figure 6: Force component of V6 in [0,20]ms using octave

4.5 t>0: Total Response

The plot of the full response of node 6 (with contributions from the natural and forced components) and the voltage in node 1, in the time interval [-5,20]ms, from *octave* is in Figure 7.

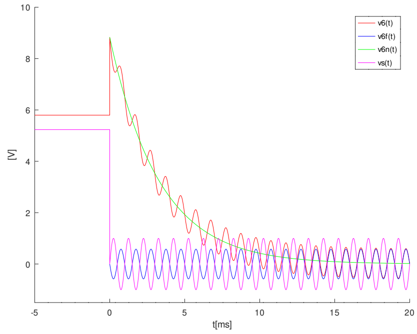

Figure 7: V6 and V1 in [-5,20]ms using octave

The same plot using ngpice in the time interval [0,20]ms, is in Figure 8

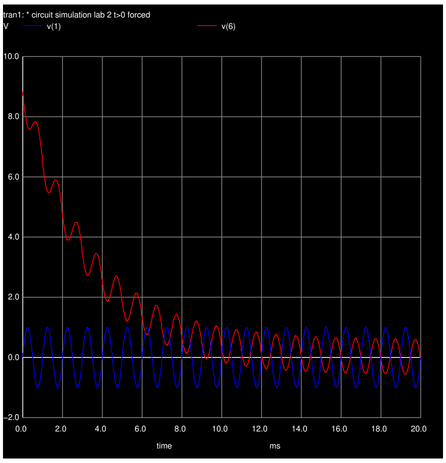

Figure 8: V6 and V1 in [0,20]ms using ngspice

As we can see, we get the same results from both analysis.

4.6 Frequency Analysis

The plots of magnitude from frequency response in nodes 1 and 6, and in the capacitor, using *octave* can be seen in Figure 9, and using *ngspice* in Figure 10

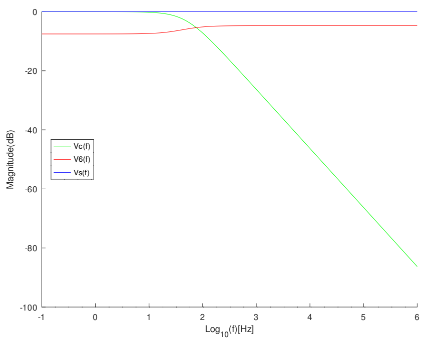

Figure 9: Magnitude of V6(f),V1(f) and Vc(f) using octave

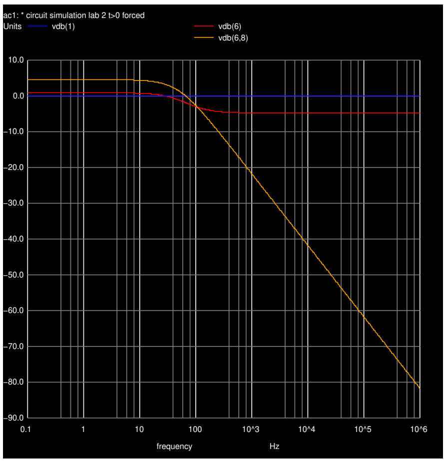

Figure 10: Magnitude of V6(f),V1(f) and Vc(f) using ngspice

The plots of phase from frequency response in nodes 1 and 6, and in the capacitor, using *octave* can be seen in Figure 11, and using *ngspice* in Figure 12

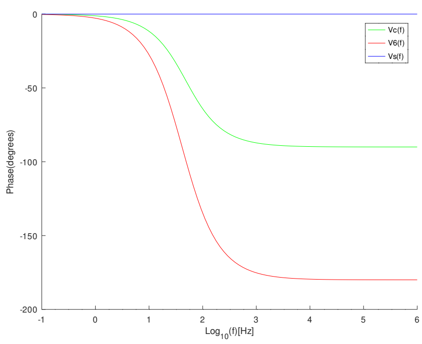

Figure 11: Phase of V6(f), V1(f) and Vc(f) in [0,20]ms using octave

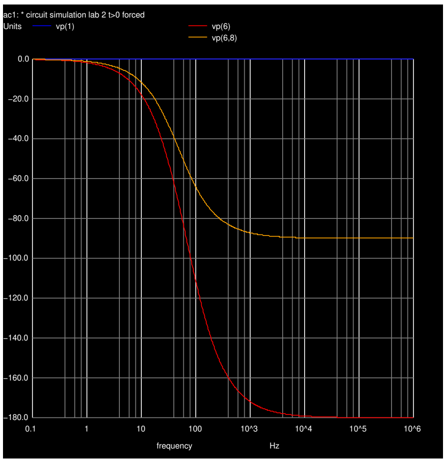

Figure 12: Phase of V6(f), V1(f) and Vc(f) in [0,20]ms using ngspice

We can see these phase plots are the same, no matter the method used.

5 Conclusion

In this laboratory assignment, the objective of analysing the RC circuit has been achieved.
When looking at the tables of currents and voltages obtained by *Octave* and *ngspice* for the analysis of the circuit when t<0 we can that they perfectly match, which is expected since all components considered are linear.
In the analysis for t=0, just one voltage is different than 0, V_6, and both methods resulted in the same value for this voltage, and for the current that goes throught R_5. In the *ngspice* simulation the rest of voltages, that were supposed to be zero (and were, in the table obtained by *Octave*), are in fact values of very small order of magnitude being practically negligible, which checks out.
In terms of all of the plots, the ones obtained by *Octave* were a perfect match for the ones obtained in *ngspice*.
Only the magnitude plot regarding V_6 had differences, when comparing both tools, since they seem to be mirrored.