TÉCNICO
LISBOA

Circuit Theory and Electronics Fundamentals

Técnico, University of Lisbon

First laboratoy report

Pedro Vilas, nº8631
José Machado, nº95812
Pedro Tomé, nº93151
Group 68
March 22, 2021

Contents

1 Introduction

In this laboratory assignment our objective is to study the circuit represented in **Figure 1**, a circuit containing a capacitor and a sinusoidal voltage source v_s which will be our main focus.

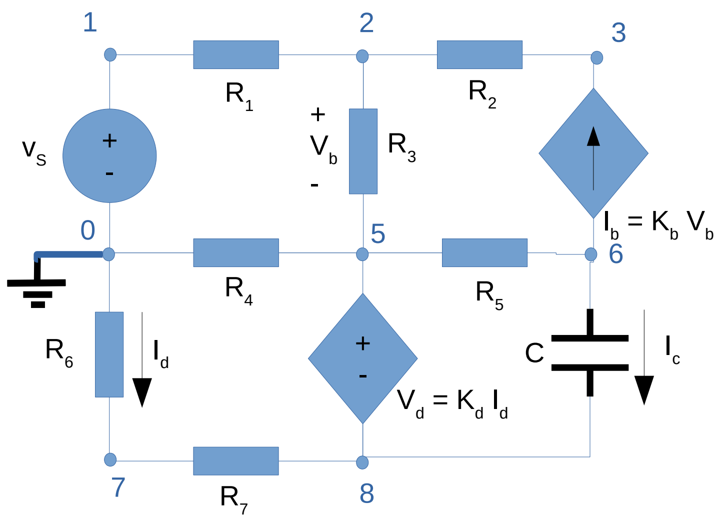

Figure 1: Circuit in study

In this circuit, as we can see, we also have a linearly dependent voltage and current source. The circuit also contains 7 resistors. We have 8 nodes numbered from 0 to 8 arbitrarily, and it was considered that *node 0* was the ground node. The voltage-controlled current source I_b has a linear dependence on Voltage V_b, of constant K_b. The current-controlled voltage source V_d has a linear dependece on current I_d, of constant K_d.These componentes that offer a linear dependence (the voltage V_b and the control current I_d) can be obtained nowing that the voltage V_b is the voltage drop at the ends of resistor R_3 and the control current I_d is the current that passes through the resistor R_6.

The following equation describes how the sinuoidal voltage from source v_s varies with time:

$$v_s(t)=V_s u(-t)+\sin(2\pi f t)u(t)$$

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

In this report we start by making a theoretical analysis of the circuit (Section 2), using the nodal analysis the circuit is analised for $t < 0$ and the equivalent resistence R_{eq}, as seen from the capacitor terminals, is obtained. In this section both the natural and forced solutions for V_6 are also determined as well as the frequency responses for V_c, V_s and V_6. In Section 3, we make a virtual simulation of the circuit using Ngspice. An operating point analysis is used to analyse the circuit when $t < 0$ and another one to determine the time constant. At the end of this section we take a look to node 6 to determine the natural and forced responses on this node using a transient analysis. We also perfome a frequency analysis on node 6. To complete this report we make a final conclusion (Section 4) where we compare the theoretical results to the results obtained by the simulation.

2 Theoretical Analysis

In this section we start by showing the circuit **Figure 2**, which will be analysed in theory.

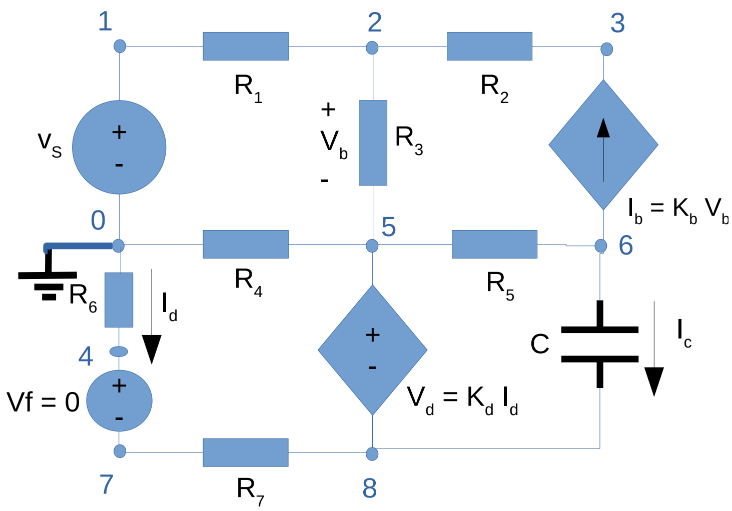

Figure 2: Diagram of the circuit considered for the computations and simulations

2.1 Analysis for $t < 0$

For $t < 0$ no current passes through the capacitor, and therefore this component behaves like an open circuit, so in this subsection we start by applying the nodal method to the circuit in order to determine the voltage in all nodes and the current on all branches. The nodal method aplies KVL. In below all equations related to nodal method are presented:

$$V_0 = 0 \tag{1}$$

$$V_4 = V_7 \tag{2}$$

$$V_5 - V_8 = K_d \frac{V_0 - V_4}{R_6} \tag{3}$$

$$V_1 - V_0 = V_s \tag{4}$$

$$\frac{V_2 - V_1}{R_1} + \frac{V_2 - V_5}{R_3} + \frac{V_2 - V_3}{R_2} = 0 \tag{5}$$

$$\frac{V_3 - V_2}{R_2} - K_b(V_2 - V_5) = 0 \tag{6}$$

$$\frac{V_5 - V_2}{R_3} + \frac{V_5 - V_0}{R_4} + \frac{V_5 - V_6}{R_5} + \frac{V_8 - V_7}{R_7} = 0 \tag{7}$$

$$K_b(V_2 - V_5) + \frac{V_6 - V_5}{R_5} = 0 \tag{8}$$

$$\frac{V_4 - V_0}{R_6} + \frac{V_7 - V_8}{R_7} = 0 \tag{9}$$

Name	Node method
@c	0
@Gb	-2.476581e-04
@r1	2.361727724258312e-04
@r2	2.476580523080236e-04
@r3	-1.148527988219316e-05
@r4	-1.233046386675456e-03
@r5	-2.476580523080238e-04
@r6	9.968736142496250e-04
@r7	9.968736142496252e-04
v(1)	5.19420986305
v(2)	4.94841719889
v(3)	4.43185531437
v(4)	-2.08713551114
v(5)	4.98369078605
v(6)	5.76150054287
v(7)	-2.08713551114
v(8)	-3.12819500012

Table 1: A variable that starts with "@ " is of type *current* and expressed in milliampere (mA); all the other variables that start with a "V" are of the type *voltage* and expressed in Volt (V).

Name	Simulation
@c[i]	0.000000e+00
@gb[i]	-2.47658e-04
@r1[i]	2.361728e-04
@r2[i]	2.476581e-04
@r3[i]	-1.14853e-05
@r4[i]	-1.23305e-03
@r5[i]	-2.47658e-04
@r6[i]	9.968736e-04
@r7[i]	9.968736e-04
v(1)	5.194210e+00
v(2)	4.948417e+00
v(3)	4.431855e+00
v(4)	-2.08714e+00
v(5)	4.983691e+00
v(6)	5.761501e+00
v(7)	-2.08714e+00
v(8)	-3.12819e+00

Table 2: Step 1: Operating point for $t < 0$. A variable preceded by @ is of type *current* and expressed in miliAmpere; other variables are of type *voltage* and expressed in Volt.

2.2 Equivalent resistor as seen from the capacitor terminals

By replacing the independent source V_c with a short circuit ($V_s = 0$) we can switch off this source which will live us to calculate the equivalent resistance in C. We also needed to replace the capacitor with a voltage source $V_x = V_6 - V_8$ due to the presence of dependent sources. We use the V_6 and V_8 from the previous section beacause the voltage drop at the terminals of the capacitor needs to be a continuous function (there can not be an energy discontinuity in the capacitor).A nodal analysis is performed in order to determine the current I_x that is supplied by V_x, having in mind the previous allegation. At these point, with the results obtained before, we can determine R_{eq} ($R_{eq} = V_x/I_x$). All these procedures were required in order to determine the time constant τ ($\tau = R_{eq} * C$). The time constant is crucial to determine the natural and forced solutions for V_6, which will be done in the next subsections. The equations considered for these calculations were 1, 2, 3, 4, 5, 6 and the following:

$$\frac{V_1 - V_2}{R_1} + \frac{V_0 - V_4}{R_6} + \frac{V_0 - V_5}{R_4} = 0 \tag{10}$$

$$K_b(V_2 - V_5) + \frac{V_6 - V_5}{R_5} + I_x = 0 \tag{11}$$

$$\frac{V_4 - V_0}{R_6} + \frac{V_7 - V_8}{R_7} = 0 \tag{12}$$

$$V_x = V_6 - V_8 \tag{13}$$

Name	Theoretical values
@Gb	0.00000000000
@r1	0
@r2	0
@r3	0
@r4	0
@r5	-2.830518214123736e-03
@r6	0
@r7	0
v(1)	0.00000000000
v(2)	0.00000000000
v(3)	0.00000000000
v(4)	0.00000000000
v(5)	0.00000000000
v(6)	8.88969554299
v(7)	0.00000000000
v(8)	0.00000000000
Ix	-0.00283051821
Vx	8.88969554299
Req	-3.140660e+03
τ	-3.270244e-03

Table 3: A variable that starts with a "V" is of type *voltage* and expressed in Volt (V). The variable R_{eq} is expressed in Ohm and the variable τ is expressed in seconds

Name	Simulation values
@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]	-2.83052e-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)	0.000000e+00
v(6)	8.889695e+00
v(7)	0.000000e+00
v(8)	0.000000e+00
Ix	-2.83052e-03
Vx	8.889695e+00
Req	-3.14066e+03

Table 4: Step 2: Operating point for $v_s(0) = 0$. A variable preceded by @ is of type *current* and expressed in miliAmpere; variables are of type *voltage* and expressed in Volt. The equivalent resistance is in Ohms

2.3 Natural solution for V_6

Now we have to obtain the natural solution for V_6. We do that by remembering that the natural solution depends on the initial charge (voltage), on R_{eq} and C and it is computed by removing all independent sources and applying KVL. In Octave, to compute the Natural solution the general formula derived in the theoretical classes was used: $V_{6n}(t) = Ae^{\frac{-t}{\tau}}$.

To resolve this formula we achieved the value of τ in the previous section and A is a constant that can be determined through the boundary conditions (when $t = 0$, $A = V_x$).

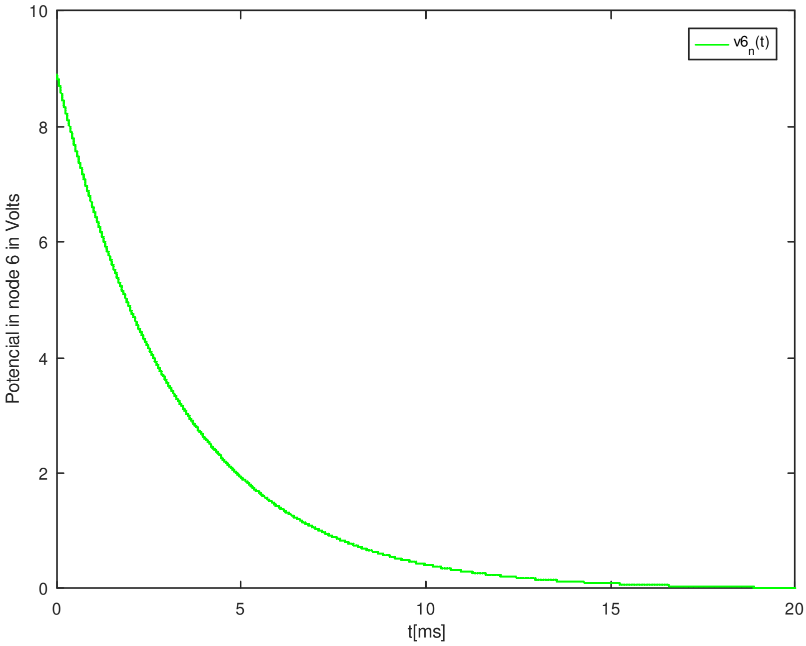

Figure 3: Natural response of V_6 as a function os time in the interval from [0,20] ms

2.4 Forced solution for V_6 with $f = 1000Hz$

In this section we determine the forced solution V_{6f} for the same period of time and for a level of frequency of 1KHz by using impedances instead of resistances and capacitances and by using a nodal analysis. It was also considered that the magnitude of the phasor representing the voltage sorce $\tilde{V}_s$ was 1 ($V_s = 1$), a result of expression **??**. By doing all of these we achieve these equations that allow us to obtain phasor voltages in all nodes:

$$Z = \frac{1}{wCj} \tag{14}$$

$$\tilde{V}_s = -j \tag{15}$$

$$\tilde{V}_0 = 0 \tag{16}$$

$$\tilde{V}_4 = \tilde{V}_7 \tag{17}$$

$$\tilde{V}_5 - \tilde{V}_8 = K_d \frac{\tilde{V}_0 - \tilde{V}_4}{R_6} \tag{18}$$

$$\tilde{V}_1 - \tilde{V}_0 = \tilde{V}_s \tag{19}$$

$$\frac{\tilde{V}_2 - \tilde{V}_1}{R_1} + \frac{\tilde{V}_2 - \tilde{V}_5}{R_3} + \frac{\tilde{V}_2 - \tilde{V}_3}{R_2} = 0 \tag{20}$$

$$\frac{\tilde{V}_3 - \tilde{V}_2}{R_2} - K_b(\tilde{V}_2 - \tilde{V}_5) = 0 \tag{21}$$

$$\frac{\tilde{V}_1 - \tilde{V}_2}{R_1} + \frac{\tilde{V}_0 - \tilde{V}_4}{R_6} + \frac{\tilde{V}_0 - \tilde{V}_5}{R_4} = 0 \tag{22}$$

$$K_b(\tilde{V}_2 - \tilde{V}_5) + \frac{\tilde{V}_6 - \tilde{V}_5}{R_5} + \frac{\tilde{V}_6 - \tilde{V}_8}{Z} = 0 \tag{23}$$

$$\frac{\tilde{V}_4 - \tilde{V}_0}{R_6} + \frac{\tilde{V}_7 - \tilde{V}_8}{R_7} = 0 \tag{24}$$

$$\tilde{V}_x = \tilde{V}_6 - \tilde{V}_8 \tag{25}$$

The complex amplitudes of the phasors are presented in **Table 5**

Name	Complex amplitude voltage
V0	0
V1	1
V2	9.526794891545152e-01
V3	8.532299293286518e-01
V4	4.018196349728130e-01
V5	9.594704329347390e-01
V6	6.039463056096701e-01
V7	4.018196349728130e-01
V8	6.022465557991893e-01

Table 5: Complex amplitudes in all nodes in Volts

2.5 Final total solution $v_6(t)$

In this section the final total solution V_6 for a frequency of 1KHz is determined by superimposing the natural and forced solutions determined in previous sections (V_6=V_{n6}+V_{6f}) In **Figure: 4** the voltage of the independent source V_s and the voltage of V_6 were plotted for the time interval of [-5,20] ms.

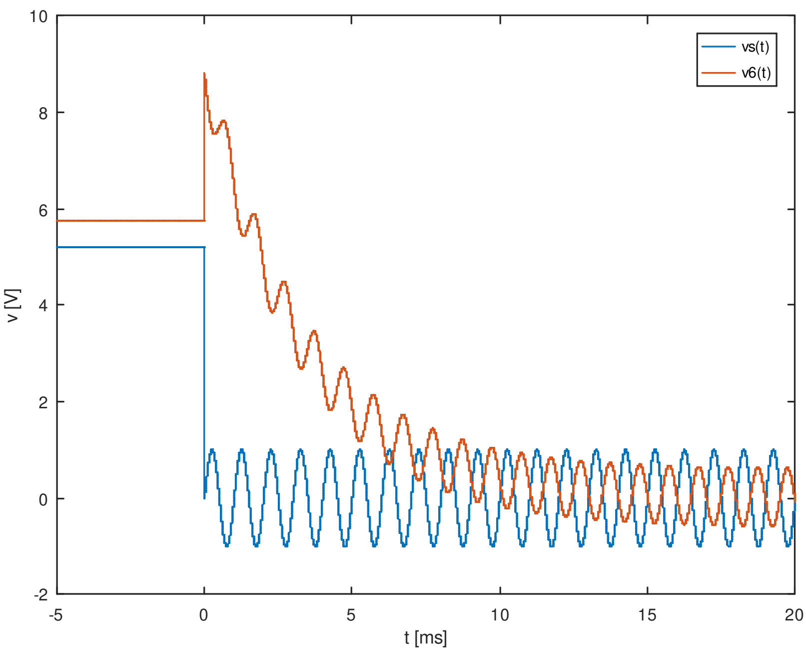

Figure 4: Voltage of $V_6(t)$ and $V_s(t)$ as functions of time from [-5,20] ms

2.6 Frequency responses $v_c(f)$, $v_s(f)$ and $v_6(f)$ for frequency range 0.1 Hz to 1 MHz

For this section, we considered $v_s(t) = sin(2\pi ft)$. As we can see, the magnitude and the phase do not depend on the frequency f. Therefore, we are to expect these values to remain constant for both these variables in the figures 5 and 6.

This circuit can serve the purpouse of a low-pass filter, that is, when the frequencies are low, the capacitor can reach the same drop down as the input in a long period of time, which means it will act aproximately as an open circuit, thus allowing a considerable potential drop from nodes 6 to 8.This means that for low frequencies the voltage in the capacitor is in phase with the voltage source.

But in the other case, when we have high level of frequencies, the capacitor only has a small time to charge up before the input changes direction, which will result in the capacitor acting like a short-circuit. Therefore, there will be close to none potential drop between nodes 6 and 8 and the capacitor and source will start to fall out of phase, for frequencies greater that the cutoff frequecy (f_c). The following formula. $f_c = \frac{1}{2.\pi.\tau}$ allow us to calculate the frequency. For the values provided this cutoff frequency is around 50Hz. This explains the steep drop in potential difference that we can see in the graph around the first and second decades. The phase difference between the capacitor voltage and the voltage source also begins to show at around this frequency as can be seen in **Figure: 6**.

In order to understand the phase and magnitude declination with the increase in frequency we simplefly the circuit to a voltage source, capacitor and equivalent resistor, which will live to the following equations:

$$V_c = \frac{V_s}{\sqrt{1 + (R_{eq}.C.2\pi.f)^2}} \tag{26}$$

$$\phi_{V_c} = -\frac{\pi}{2} + arctan(R_{eq}.C.2\pi.f) \tag{27}$$

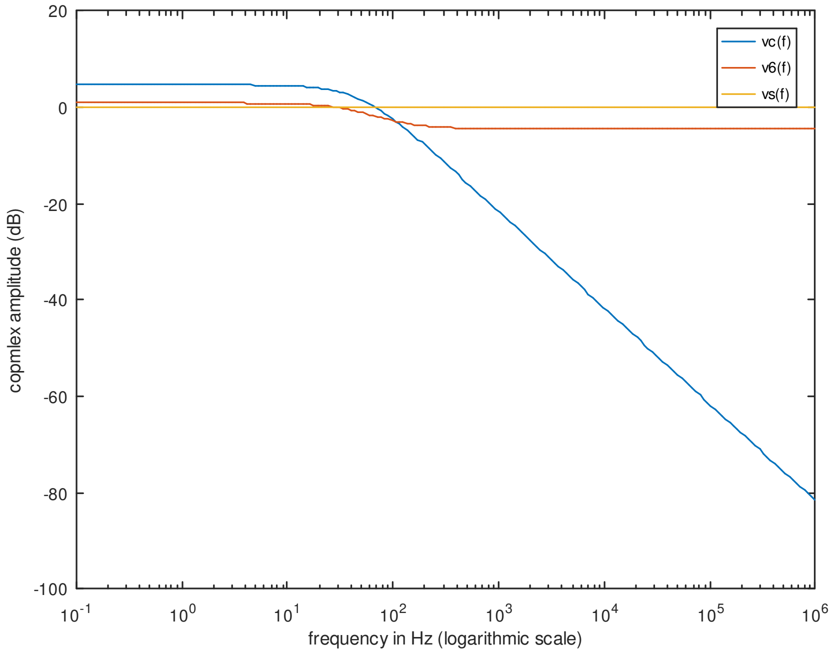

Figure 5: Graph for amplitude frequency response, in dB, of V_c, V_6 and V_s for frequencies ranging from 0.1Hz to 1MHz (logarithmic scale).

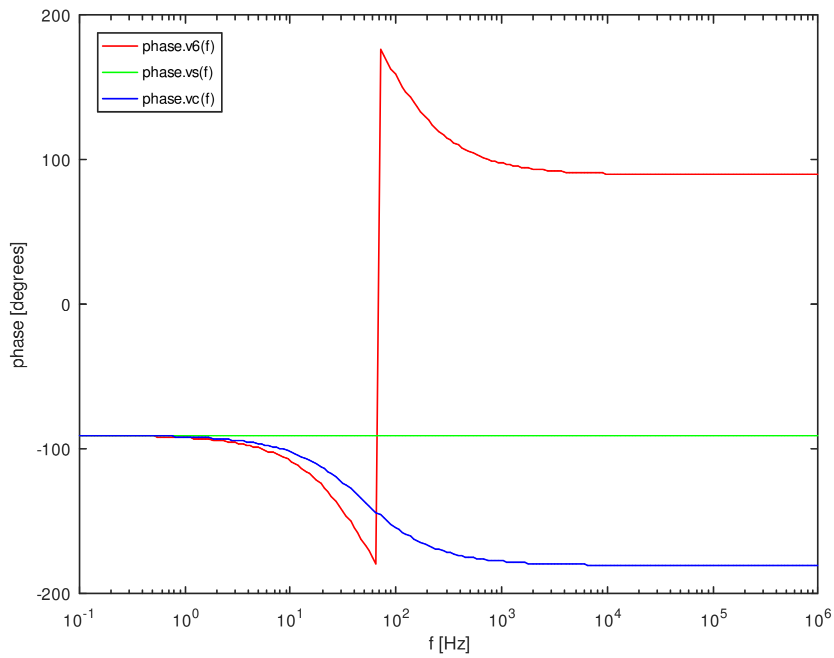

Figure 6: Graph for the phase response, in degrees of V_c, V_6 and V_s for frequencies ranging from 0.1Hz to 1MHz, displayed in a logarithmic scale. Note that the apparent peak discontinuity in the phase of V_6 is only due to the domain of the arctan function that gives the phase (angle of the phasor), and so the phase is in fact continuous.

3 Simulation Analysis

In this section we will describe the steps needed to simulate this circuit using the software Ngspice. Three types of analysis will be performed: Operating Point analysis, Transient analysis ans Frequency analysis.

The following steps in the simulations are to be conducted:

- for $t < 0$ (operating point only, in order to obtain the voltages in all nodes and the currents in all branches);
- operating point for $V_s(0) = 0$, replacing the capacitor with a 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 the previous step (this step is necessary given that we must the compute the boundary conditions that guarantee continuity in the capacitor's discharge - such may imply that the boundary conditions differ from those computed for $t < 0$);
- simulate the natural response of the circuit (using the boundary conditions V(6) and V(8) as obtained previously) using a transient analysis;
- repeating the third step, using V_s as given in **Figure 7** and f = 1kHz in order to simulate for the total response on node 6
- simulate the frequency response in node 6 for a frequency range 0.1 Hz to 1MHz.

$$v_s(t) = V_s u(-t) + \sin(2\pi f t) u(t)$$

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

Figure 7: Time step conditions

3.1 Operating Point Analysis for $t < 0$

There was a need to create a "fictional" voltage source, between node 7 and resistor 6 (providing 0V to the circuit in order not to alter the behaviour of the rest of the circuit) so as to be able to define the dependecy for the current-controlled voltage source V_d. This has no specific reason to be, other than the particularities of the Ngspice software. The circuit and nodes used for the simulation can be seen in **Figure 2**.

Table 6 shows the simulated operating point results for the circuit under analysis for $t < 0$.

Name	Value [mA or V]
@c[i]	0.000000e+00
@gb[i]	-2.47658e-04
@r1[i]	2.361728e-04
@r2[i]	2.476581e-04
@r3[i]	-1.14853e-05
@r4[i]	-1.23305e-03
@r5[i]	-2.47658e-04
@r6[i]	9.968736e-04
@r7[i]	9.968736e-04
v(1)	5.194210e+00
v(2)	4.948417e+00
v(3)	4.431855e+00
v(4)	-2.08714e+00
v(5)	4.983691e+00
v(6)	5.761501e+00
v(7)	-2.08714e+00
v(8)	-3.12819e+00

Table 6: Operating point for $t < 0$. A variable preceded by @ is of type *current* and expressed in miliAmpere; other variables are of type *voltage* and expressed in Volt.

3.2 Operating Point Analysis for $t = 0$

In this section the circuit was simulated using an operating point analysis with $V_s(0) = 0$ and with the capacitor replaced by a voltage source $V_x = V(6) - V(8)$ with these as obtained in the last step. This step was taken because we must compute the boundary conditions that guarantee continuity in the capacitor's discharge (such may imply that the boundary conditions differ from those computed for $t < 0$). In other words $V(6) - V(8)$ needs to be a continuos function in time (in this case particularly from $t < 0$ to $t = 0$), as there can not be a energy discontinuity in the capacitor ($E_C = \frac{1}{2}CV^2$). However, that does not imply that that $V(6)$ and $V(8)$ are continuos functions in time. In **Table 7** the simulation results are presented.

Name	Value [mA or V and Ohm]
@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]	-2.83052e-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)	0.000000e+00
v(6)	8.889695e+00
v(7)	0.000000e+00
v(8)	0.000000e+00
Ix	-2.83052e-03
Vx	8.889695e+00
Req	-3.14066e+03

Table 7: Operating point for $v_s(0) = 0$. A variable preceded by @ is of type *current* and expressed in miliAmpere; variables are of type *voltage* and expressed in Volt. The equivalent resistance is in Ohms

3.3 Natural solution for V_6 using transient analysis

In this section the natural response of the circuit in the interval [0,20] ms was studied using a transient analysis simulation. To do so the boundary conditions V(6) and V(8) obtained in the previous section were used, as well as the NgSpice directive *.ic*. These values are being obtained from the previous simulations run, and not from the theoretical previsions.

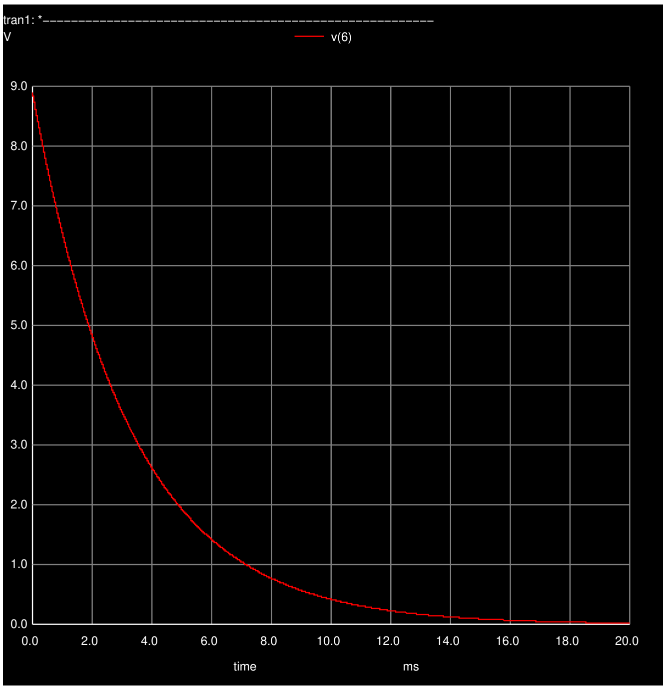

Figure 8: Simulated natural response of $V_6(t)$ in the interval [0,20] ms. The *x axis* represents the time in miliseconds and the *y axis* the Potencial in node 6 in Volts.

3.4 Total solution for V_6 using transient analysis

In this section the total response of V_6 (natural + forced) is simulated using transient analysis. This is done by repeating the previous section, but using V_s as given in **Figure 7** and f = 1kHz.

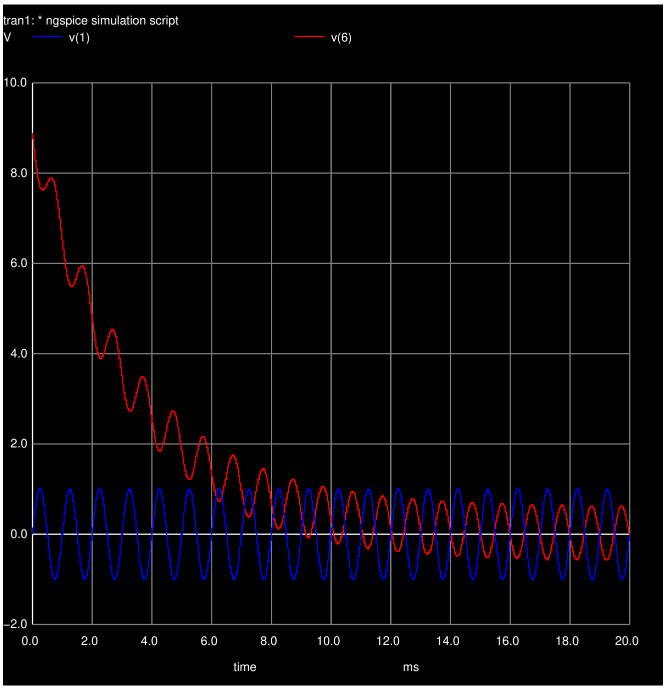

Figure 9: Simulated response of $V_6(t)$ and of the stimulus $V_s(t)$ as functions of time from [0,20] ms. The *x axis* represents the time in miliseconds and the *y axis* the Voltage in Volts.

3.5 Frequency response in node 6

In this section the frequency response in node 6 is simulated for the frequency range from 0.1 Hz to 1 MHz. The reasons of how and why $V_6(t)$ and $V_s(t)$ differ have been coverd in **subsection 2.6**.

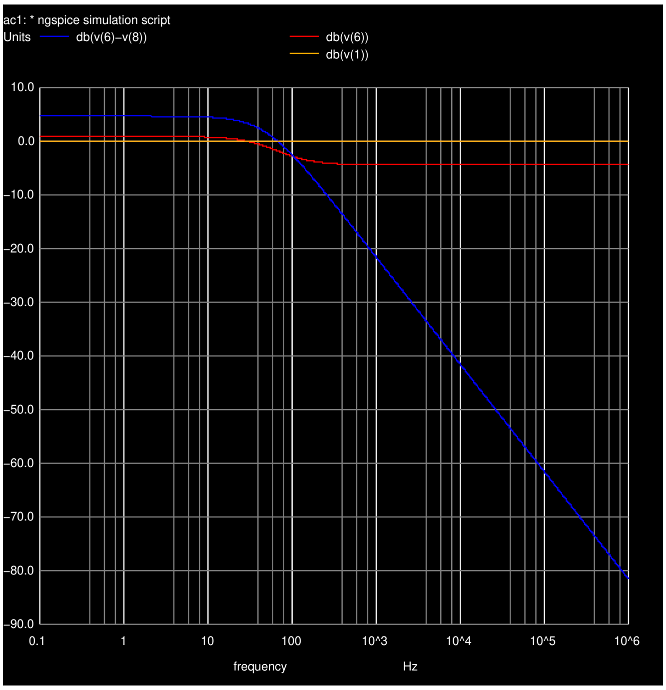

Figure 10: Magnitude of $V_s(f)$, $V_c(f)$ and of $V_6(f)$. The *x axis* represents the frequency in Hz, using a logarithmic scale and the *y axis* the magnitude in dB.

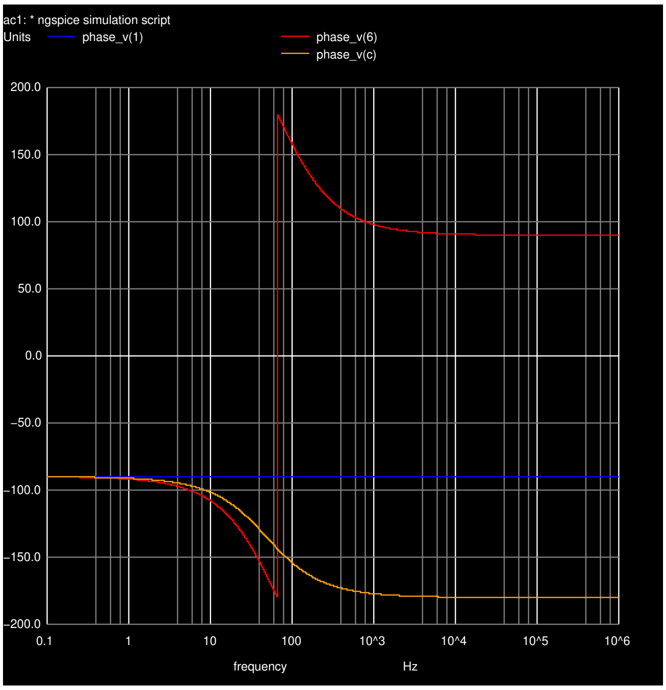

Figure 11: Phase of $V_s(f)$, $V_c(f)$ and of $V_6(f)$. The *x axis* represents the frequency in HZ, using a logarithmic scale and the *y axis* the phase in degrees.

4 Conclusion

In conclusion of our laboratory, we can say that our objective has been achieved, as we analysed the circuit that was proposed to us, which contained various resistances, a capacitor, and a sinusoidal voltage source v_s that varies in time. We have also analysed static, time and frequency, in a theoretical way, by using the Octave math tool, and by simulating the circuit using the Ngspice tool. By doing this, we have obtained results very similar in both ways, resulting in an error matching 0, in values obtained theoretically, and in few cases of the simulation, we obtained a very small and insignificant error, which we can consider to be 0.

We can conclude that the reason for this almost unexisting error, is the fact that the circuit proposed was pretty simple, having only one capacitor and some linear components, so we can attribute this small discrepancys to the model used in Ngspice for the capacitor, and to the transient boundary condition analysis. This means that the greater the complexity of the components in the circuit, the greater are the discrepancys to be expected between the simulated results, and the results obtained theoretically, due to the also greater complexity of the models used for simulating these components in Ngspice.