Circuit Theory and Electronics Fundamentals

Department of Electrical and Computer Engineering, Técnico, University of Lisbon

March 24, 2021

Laboratory Assignment - T2

Group nº59

José Miguel Goulão - 95814
Lourenço Pacheco - 95817
André Gomes - 96352

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing:

- seven resistors (R_1-R_7)
- one voltage source (V_s)
- one capacitor (C)
- one voltage-controlled current source (I_b)
- one current-controlled voltage source (V_d)

Circuit T2 is presented in Figure 1. All components, including nodes ($N1$-$N8$) are identified with their respective names (ground is marked with its symbol).

The voltage source v_s obeys the following equations:

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

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

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

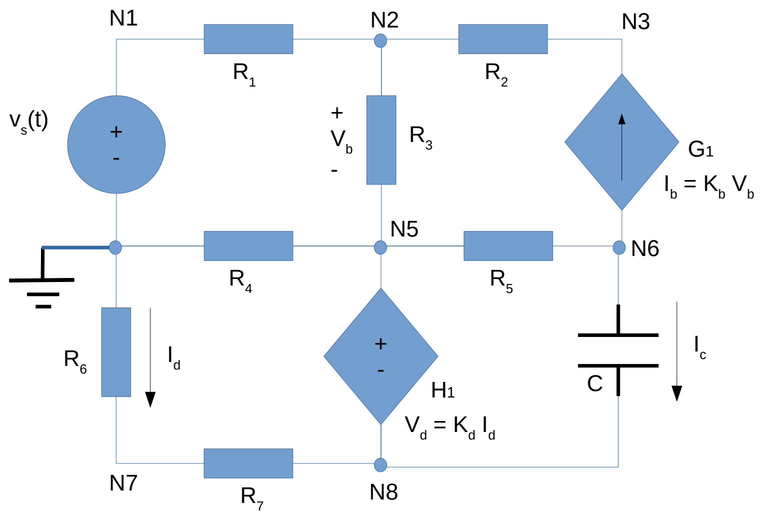

Figure 1: Circuit T2

For this laboratory assignment, the values considered for all the variables can be found on Table 1. They were obtained through a Python script that generates random values.

Name	Value
$R1$	1.00359089673
$R2$	2.04298963569
$R3$	3.02503141993
$R4$	4.05647775356
$R5$	3.07781188185
$R6$	2.01277040929
$R7$	1.01993304256
V_s	5.11402517827
C	1.03896393154
K_b	7.23768458527
K_d	8.33526265782

Table 1: Values provided by the Python sript. Units for the values: V, mA, kOhm, mS and uF

2 Theoretical Analysis

In this section, the Circuit T2 is analysed theoretically. In figure **??**, appart from all the components being identified, the assumed currents are also shown. Only the node method was used in this section. Each subsection refers to each task.

Three important equations were used: both Kirchhoff's laws (Kirchhof's current law (KCL) - eq.(2) and Kirchhoff's voltage law (KVL) - eq.(3)); Ohm's law (eq.(4)).

The algebraic sum of all the currents in any given node is zero:

$$\sum I_i = 0 \tag{2}$$

The algebraic sum of all the voltages in any given closed-loop circuit (mesh) is zero:

$$\sum V_i = 0 \tag{3}$$

The potential difference between the two nodes connected to a resistor is equal to the current that passes through the resistor multiplied by its resistance.

$$V_i = R_i I_i \tag{4}$$

$$V_1(t) = V_2(\infty) + [V_2(0) - V_2(\infty)]e^{\frac{-t}{\tau}} \tag{5}$$

2.1 Task 1)

When $t < 0$ the value of V_s is constant and so we can perform DC analysis on the circuit. We can assume that enough time has passed and so it is reasonable to assume that the circuit is in steady-state.

When performing a DC steady-state analysis on a circuit we can use the fact that the current flowing trough the capacitor is null (the circuit behaves as if the capacitor was removed).

A node analysis was performed to find the voltage value of each node and the current in each branch.

The node method uses KCL in conjunction with Ohm's law to define equations that when solved give the voltage value of each nove in relation to ground (Node 4, $V_4 = 0$).

In order to have equations that solve for the node's voltage, a relation between curent and voltage is made using Ohm's law (given a resistance between two nodes, the current that passes the resistance can be written as $I = \frac{V_2 - V_1}{R_1}$)

To simplify the equations it is useful to use the conductance G_n which is the inverse of the resistance R_n ($G_n = \frac{1}{R_n}$)

The equations used to solve the circuit were organized in matrix form.

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G1 & -(G1+G2+G3) & G2 & G3 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -G2 & G2 & 0 & 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & G3 & 0 & -(G3+G4+G5) & G5 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -G5 & G5 & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & 0 & 0 & 0 & G7 & -G7 & -1 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & -(G6+G7) & G7 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 1 & 0 & G6*K_d & -1 & 0 & 0 & 0 \\ 0 & K_b & 0 & -K_b & 0 & 0 & 0 & 0 & -1 & 0 \end{bmatrix} \begin{bmatrix} V1 \\ V2 \\ V3 \\ V5 \\ V6 \\ V7 \\ V8 \\ IH1 \\ Ib \\ Ic \end{bmatrix} = \begin{bmatrix} Vs \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

Figure 1. In adition, assume V_{Ni} to be the voltage in node Ni (every node position can also be found in Figure 1).

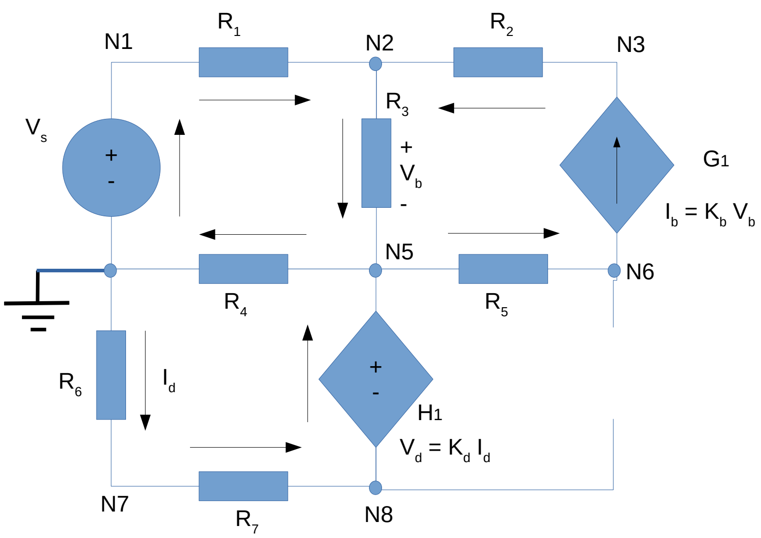

Figure 2: Circuit T2, analysed by Ngspice

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G1 & -(G1+G2+G3) & G2 & G3 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -G2 & G2 & 0 & 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & G3 & 0 & -(G3+G4+G5) & G5 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -G5 & G5 & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & 0 & 0 & 0 & G7 & -G7 & -1 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & -(G6+G7) & G7 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 1 & 0 & G6*K_d & -1 & 0 & 0 & 0 \\ 0 & K_b & 0 & -K_b & 0 & 0 & 0 & 0 & -1 & 0 \end{bmatrix} \begin{bmatrix} V1 \\ V2 \\ V3 \\ V5 \\ V6 \\ V7 \\ V8 \\ IH1 \\ Ib \\ Ic \end{bmatrix} = \begin{bmatrix} V1 \\ V2 \\ V3 \\ V5 \\ V6 \\ V7 \\ V8 \\ IH1 \\ Ib \\ Ic \end{bmatrix}$$

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G1 & -(G1+G2+G3) & G2 & G3 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -G2 & G2 & 0 & 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & G3 & 0 & -(G3+G4+G5) & G5 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -G5 & G5 & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & 0 & 0 & 0 & G7 & -G7 & -1 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & -(G6+G7) & G7 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & G6*K_d & -1 & 0 & 0 & 0 \\ 0 & Kb & 0 & -K_b & 0 & 0 & 0 & 0 & -1 & 0 \end{bmatrix} \begin{bmatrix} V1 \\ V2 \\ V3 \\ V5 \\ V6 \\ V7 \\ V8 \\ IH1_2 \\ Ib_2 \\ Ic_2 \end{bmatrix} = \begin{bmatrix} V1 \\ V2 \\ V3 \\ V5 \\ V6 \\ V7 \\ V8 \\ IH1 \\ Ib \\ Ic \end{bmatrix}$$

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G1 & -(G1+G2+G3) & G2 & G3 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -G2 & G2 & 0 & 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & G3 & 0 & -(G3+G4+G5) & G5 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -G5 & G5 & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & 0 & 0 & 0 & G7 & -G7 & -1 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & -(G6+G7) & G7 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1/Z_c & 0 & 1/Z_c & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & G6*K_d & -1 & 0 & 0 & 0 \\ 0 & K_b & 0 & -K_b & 0 & 0 & 0 & 0 & -1 & 0 \end{bmatrix} \begin{bmatrix} V1_3 \\ V2_3 \\ V3_3 \\ V5_3 \\ V6_3 \\ V7_3 \\ V8_3 \end{bmatrix} = \begin{bmatrix} V1 \\ V2 \\ V3 \\ V5 \\ V6 \\ V7 \\ V8 \\ IH1 \\ Ib \\ Ic \end{bmatrix}$$

With these 8 equations it is possible to solve the system using Octave. The results were organized in Table 7

Name	Value [A or V]
V_{N1}	5.114025e+00
V_{N2}	4.830792e+00
V_{N3}	4.226624e+00
V_{N5}	4.871651e+00
V_{N6}	5.781844e+00
V_{N7}	-1.849204e+00
V_{N8}	-2.786253e+00
$@I_b$	-2.957272e-04
$@I_c$	0.000000e+00
$@I_{R1}$	2.822201e-04
$@I_{R2}$	-2.957272e-04
$@I_{R3}$	-1.350709e-05
$@I_{R4}$	1.200956e-03
$@I_{R5}$	-2.957272e-04
$@I_d$	-9.187358e-04
$@I_{R6}$	9.187358e-04

Table 2: Values computed by Octave. Variables identified with a '@' have a corresponding value in Ampere (A). The others are expressed in Volts (V).

2.2 Task 2)

In this section the capacitor is replaced with a voltage source V_x with value $V_x = V_6 - V_8$ as shown in figure.... The same type of analysis was made to this circuit but with a slightly modification on some of the rows to represent the modified circuit.

Tabela————-

By performing this analysis we can compute the current I_x that flows trough V_s. Using Ohm's law we can calculate the equivente resistance R_{eq}:

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

This procedure is important to compute the circuit's time constant τ by using the equation $\tau = CR_{eq}$. The time constant in conjunction with the boundary conditions of the circuit are required to compute the natural solution of the circuit.

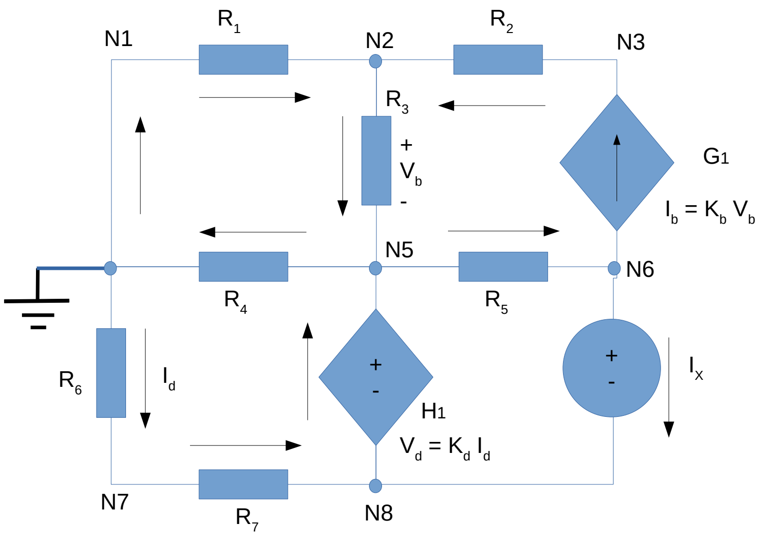

Figure 3: Circuit T2, analysed by Ngspice

2.3 Task 3)

With the circuit's time constant τ and boundary conditions calculated in the previous task we can use following equation to get the natural solution of the circuit:

$$V_{6n}(t) = V_6(\infty) + [V_6(0) - V_6(\infty)]e^{\frac{-t}{\tau}}$$

Since V_s is considered to be null on this circuit, the value of node 6 is 0 at infinity $V_6(\infty) = 0$

2.4 Task 4)

In this task the forced solution of the circuit is computed. To acomplish this objetive, we do an analysis using a phasor voltage source $V_s = 1$ and replacing C with its impedance:

$$Z_c = j\frac{1}{C\omega}$$

Doing an analysis similar to the previous ones, with a slightly modified matrix we can determine the phasor voltages in all nodes.

The following table shows the phasor magnitudes in each node.

2.5 Task 5)

In this task we compute the final total solution $v_6(t)$ with a frequncy of 1000Hz. To achieve the final result the phasors are converted to real time functions and than superimposed with the natural solution found before.

The force solution will have the form:

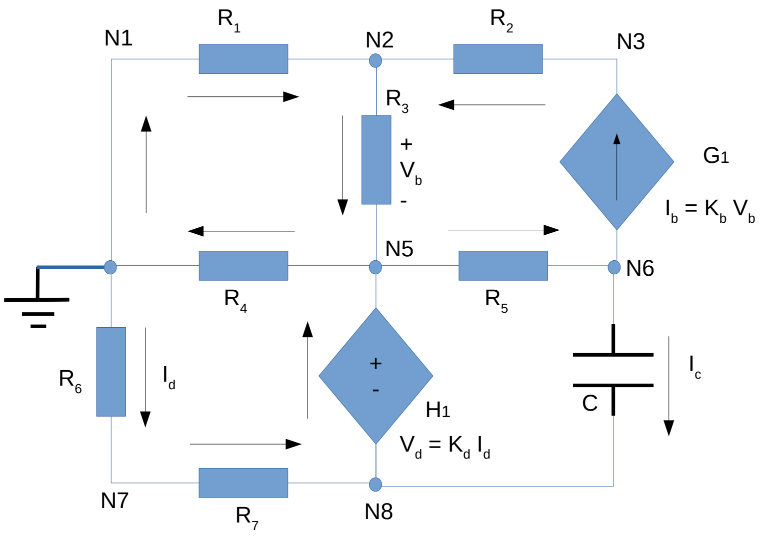

Figure 4: Circuit T2, analysed by Ngspice

$$V_{6f}(t) = V * sin(\omega t + \phi)$$

The constant ω is the angular frequency of the voltage source, V is the amplitude of the node phasor and ϕ is the phase shift of the node phasor.

The final solution will have the form:

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

The following graph plots the results computed by octave in interval [-5, 20]ms.

2.6 Task 6)

In this task, the frequency response of $v_c(f) = v_6(f) - v_8(f)$ and $v_6(f)$ is determined for a frequency range of 0.1Hz to 1 MHz. For the calculation of the frequecy response a similar analysis to the one in task 4) was made for a multitude of frequencys in the set the frequency range. The following graph shows the achieved results:

Grafico———

We can see that the value of $v_c(f)$ in dB decreases with increasing frequency. This behaviour is due to the impedance of the capacitor decreasing with larger frequencies and so the phasor voltage difference tends to zero (since the magnitude is ploted in dB, when a voltage approaches 0 the dB values goes to negative values).

The value of $v_c(f)$ also decreases for the same reasons explained in subsection 2.5).

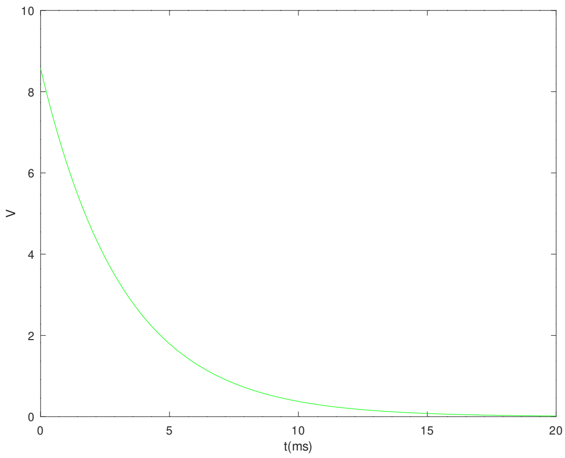

Figure 5: Plot oct - 1

3 Simulation Analysis

In this section, Circuit T2 is reproduced with the help of Ngspice (each section corresponds to each task). Ngspice is a simulator for eletronic circuits that can output a variety of results. This emulator computes the voltages in every node, as well as the potential difference between two given nodes. Apart from that, the group made use of the command *.options savecurrents* which also enables the output of the currents that pass through all branches.

With the limitation that Ngspice only provides the current in the components and not through the nodes, an aditional voltage source ($Vaux$) was added so that the current in R_6 (I_d) is known. This source (not displayed in Figure 16) has a voltage of 0V and it was implemented between R_6 and R_7. Therefore an aditional node had to be added (node $N7.$).

As previously stated, I_b is refered to as G_1. This is because, in Ngspice, a voltage-controlled current source is identified with capital 'g' (G). In the case of V_c, all current-controlled voltage source are identified with H.

3.1 Task 1)

In this subsection, the circuit is simulated when $t < 0$. There is no need for a transient analysis because $v_s(t)$=V_s (according to the data given), therefore all values are constant in time.

Table 8 shows the simulated operating point results for Circuit T2.

The three last entries in Table 8 provides the potential diference between important branches: V_b = $v(n5, n2)$ and V_d = $v(n5, n8)$.

3.2 Task 2)

In this subsection, the circuit is simulated when $t = 0$. Since we can assume that enough time has passed for the circuit to reach steady-state, the capacitor is charge and so, at the instant $t = 0$ it can replaced with a voltage source, with its value being equal to de diference between the voltages in nodes $n6$ and $n8$ (or V_x = $V(n6) - V(n8)$) obtained in subsection 3.1. Vs is also set to 0.

This step is necessary to find the boundary conditions of the circuit at $t = 0$, which will be used in the next section to calculate the natural response of the T2 circuit.

Table 4 shows the simulated operating point results for Circuit T2.

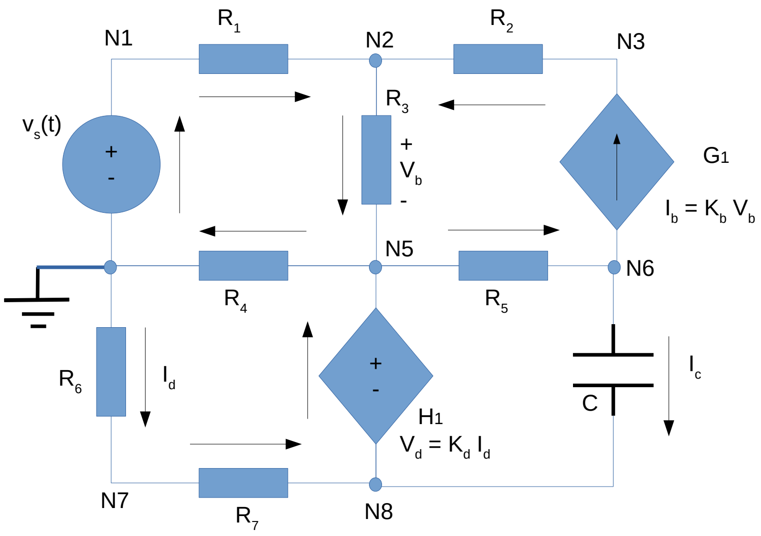

Figure 6: Circuit T2, analysed by Ngspice

3.3 Task 3)

In this subsection, the natural response of the circuit was simulated using the boundary conditions $V(n6)$ and $V(n8)$ calculated in subsection 3.2. Thus, $V_{n6}(t)$ was ploted in the interval [0;20]ms (Figure 12).

3.4 Task 4)

In this subsection, the total (natural and forced) responde on node $n6$ is simulated. The boundary conditions used are the same as subsection 3.3 and a frequency of 1kHz (f=1KHz) is considered for $v_s(t)$. Figure 14 shows the plot. It is worth noting that node $n1$ has the same value as the stimulus ($v_s(t)$), so $V(n1)$ is used instead.

3.5 Task 5)

In this subsection, the frequency response on V_s and on node $n6$ is computed.

Since the frequency response is made by changing V_s itself, the values of magnitude and phase of V_s aren't dependent on frequency. For this reason the plot lines of these values are constant.

By contrast the magnitude and phase of V_6 change with frequency. At low frequencys V_6 is in phase with V_s but as the frequency increases, the phase shift also increases until it reaches a phase difference of 180º, meaning it's totally out of pahse with the source. The magnitude starts at a value of around 1.065dB than V_s and with increasing value it decreases until it hits a plateau at around -5.275dB.

These values are in accordance with the node voltage values one would get if the capacitor was removed or shorted, respectively. This is due to the fact that the impedance value of a capacitor follows the equation $frac1C\omega$. At low frequencies the impedance is very large and so it's almost like the connection between $N6$ and $N8$ didn't exist. In contrast, at high frequencies the impedance is almost null so the circuit behaves almost as $N6$ and $N8$ were shorted.

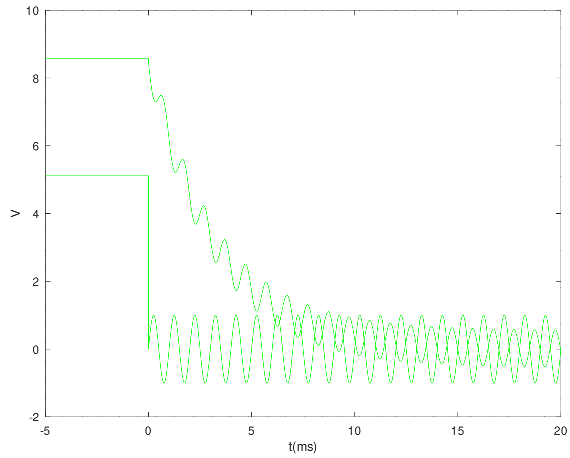

Figure 7: Plot oct - 2

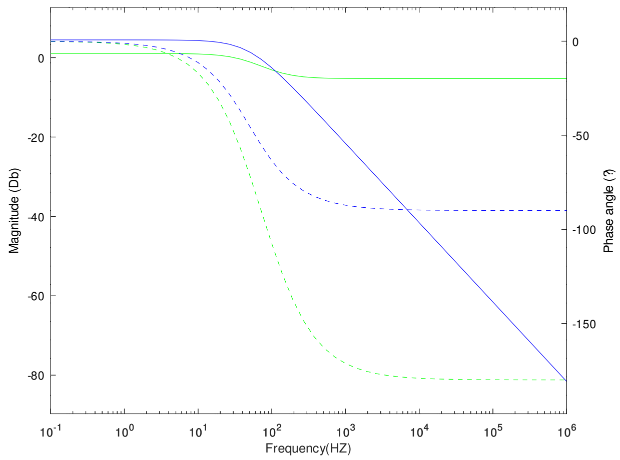

Figure 8: Plot oct - 3

4 Conclusion

For this laboratory assignment, we were given a circuit composed by resistors, one dependent current source, one independent and one dependent voltage source and had the objective of analyzing and simulating it and then compare the results obtained.

Static, transient and frequency response analyses were performed theoretically, through the node analysis and by circuit simulation, using the Octave math tool and Ngspice tool, respectively.

It is safe to say that our objective was achieved successfully. We can compare the resultes of both analysis by looking at the graphs side by side.

Task 1):

Task 2):

We can conclude that the theoretical values match the simulation ones, with relative high presicion. The only differences were generated by the limitaions of Octave and Ngspice, conserning rounding and truncating values.

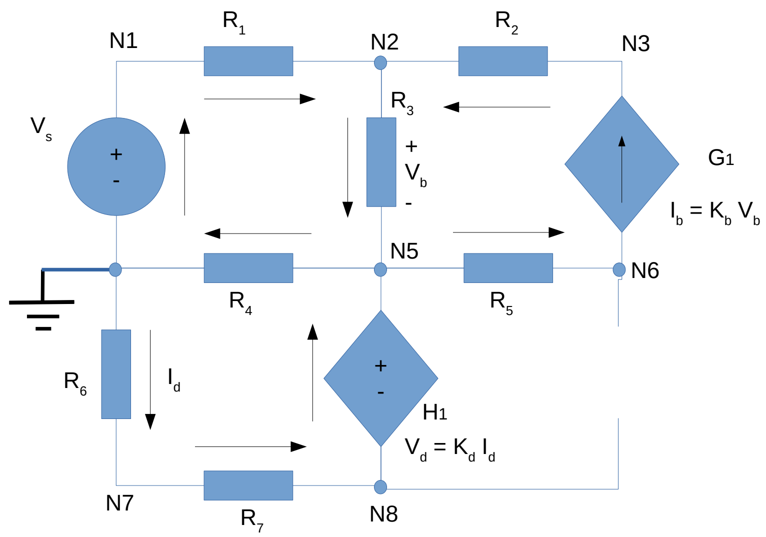

Figure 9: Circuit T2, analysed by Ngspice

Name	Value [A or V]
i(vaux)	9.187354e-04
i(h1)	-9.18735e-04
@c[i]	0.000000e+00
@g1[i]	-2.95726e-04
@r1[i]	-2.82220e-04
@r2[i]	-2.95726e-04
@r3[i]	1.350647e-05
@r4[i]	-1.20096e-03
@r5[i]	-2.95726e-04
@r6[i]	9.187354e-04
@r7[i]	-9.18735e-04
n1	5.114025e+00
n2	4.830792e+00
n3	4.226625e+00
n5	4.871649e+00
n6	5.781840e+00
n7	-1.84920e+00
n7.	-1.84920e+00
n8	-2.78625e+00
v(n5,n2)	4.085748e-02
v(n5,n8)	7.657901e+00
v(n6,n8)	8.568092e+00

Table 3: Values from Ngspice. Variables identified with a '@' or are of the type $i(...)$ have a corresponding value in Ampere (A). The others are expressed in Volts (V).

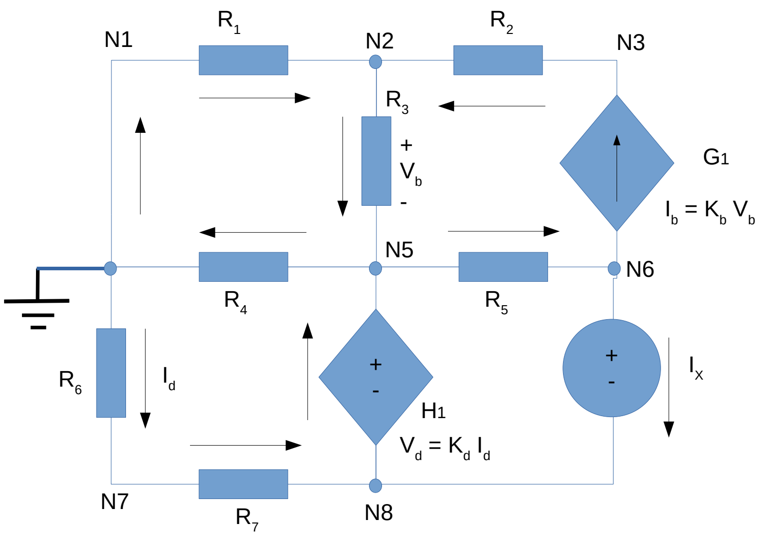

Figure 10: Circuit T2, analysed by Ngspice

Name	Value [A or V]
i(vaux)	-8.67362e-19
i(h1)	2.783826e-03
@g1[i]	-6.50208e-18
@r1[i]	-6.20511e-18
@r2[i]	-6.50208e-18
@r3[i]	2.969639e-19
@r4[i]	1.313719e-18
@r5[i]	-2.78383e-03
@r6[i]	-8.67362e-19
@r7[i]	2.995961e-20
n1	0.000000e+00
n2	-6.22740e-15
n3	-1.95111e-14
n5	-5.32907e-15
n6	8.568092e+00
n7	1.745800e-15
n7.	1.745800e-15
n8	1.776357e-15
v(n5,n2)	8.983252e-16
v(n5,n8)	-7.10543e-15

Table 4: Values from Ngspice. Variables identified with a '@' or are of the type $i(...)$ have a corresponding value in Ampere (A). The others are expressed in Volts (V).

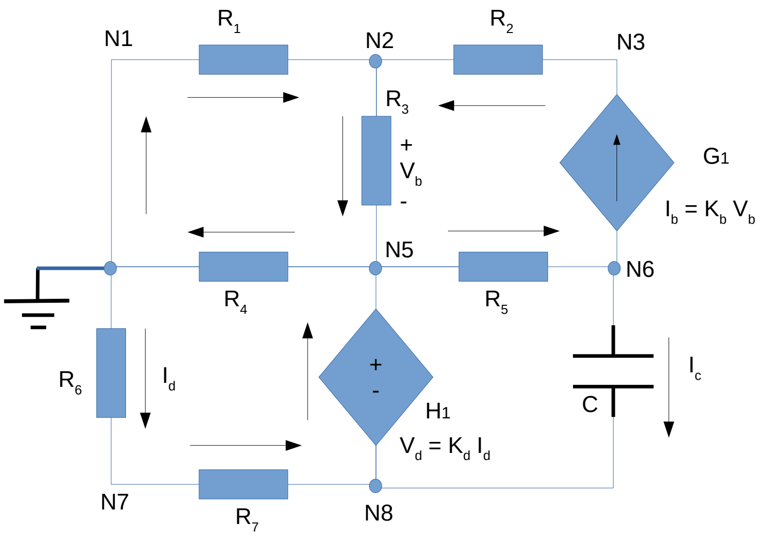

Figure 11: Circuit T2, analysed by Ngspice

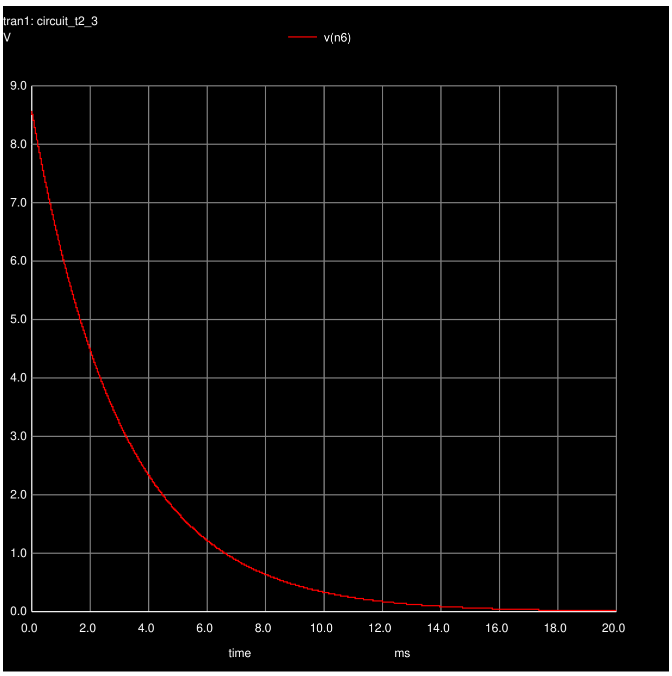

Figure 12: Transient analysis - 1: natural response on node $n6$

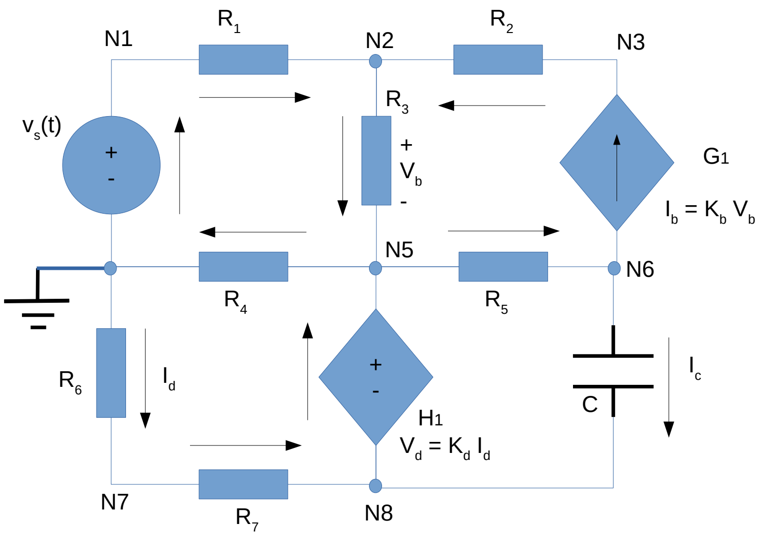

Figure 13: Circuit T2, analysed by Ngspice

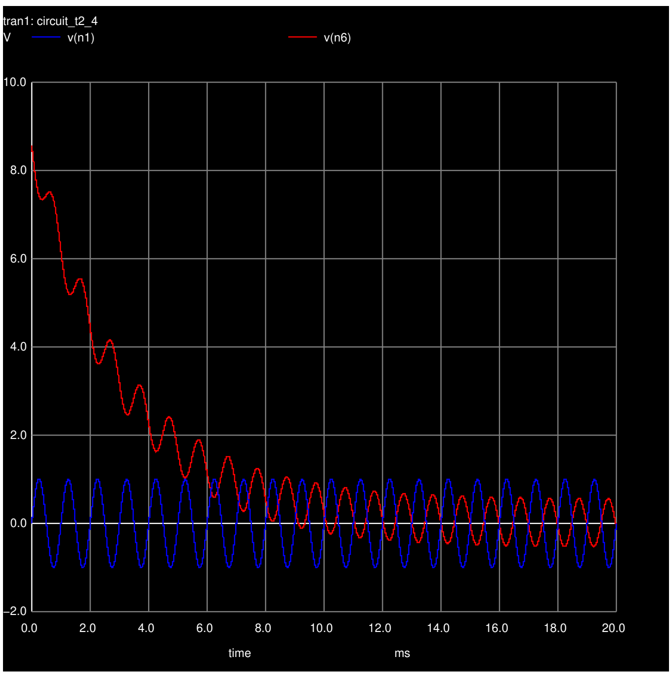

Figure 14: Transient analysis - 2: total response on node $n6$

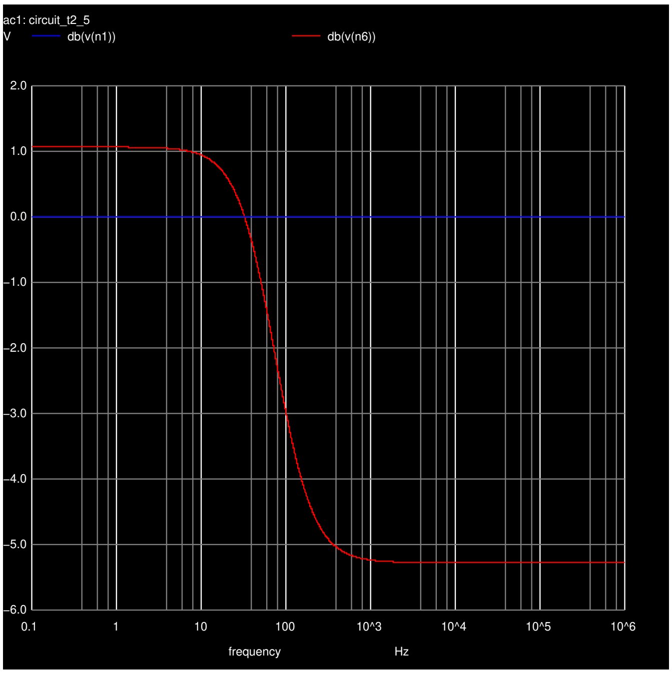

Figure 15: Frequency response - 1

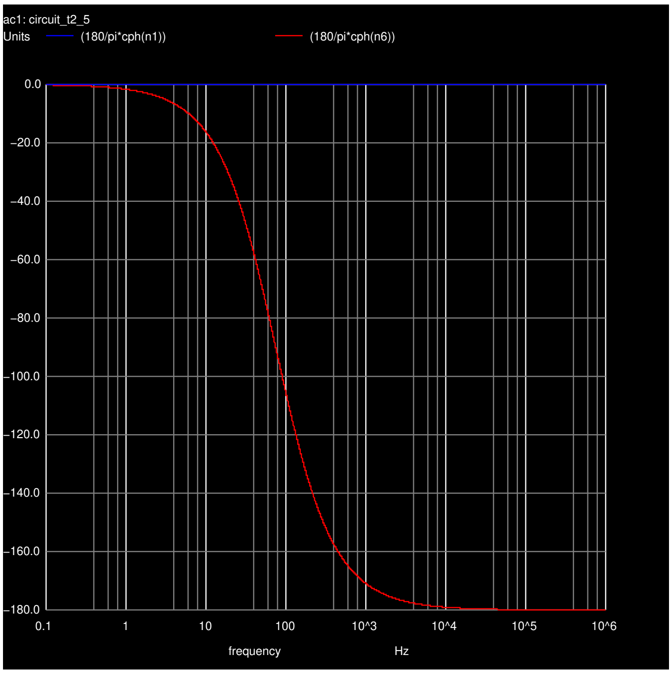

Figure 16: Frequency response - 2

Name	Value [A or V]
V_{N1}	5.114025e+00
V_{N2}	4.830792e+00
V_{N3}	4.226624e+00
V_{N5}	4.871651e+00
V_{N6}	5.781844e+00
V_{N7}	-1.849204e+00
V_{N8}	-2.786253e+00
$@I_b$	-2.957272e-04
$@I_c$	0.000000e+00
$@I_{R1}$	2.822201e-04
$@I_{R2}$	-2.957272e-04
$@I_{R3}$	-1.350709e-05
$@I_{R4}$	1.200956e-03
$@I_{R5}$	-2.957272e-04
$@I_d$	-9.187358e-04
$@I_{R6}$	9.187358e-04

Table 5: Values computed by Octave - Theoretical Task 1)

Name	Value [A or V]
i(vaux)	9.187354e-04
i(h1)	-9.18735e-04
@c[i]	0.000000e+00
@g1[i]	-2.95726e-04
@r1[i]	-2.82220e-04
@r2[i]	-2.95726e-04
@r3[i]	1.350647e-05
@r4[i]	-1.20096e-03
@r5[i]	-2.95726e-04
@r6[i]	9.187354e-04
@r7[i]	-9.18735e-04
n1	5.114025e+00
n2	4.830792e+00
n3	4.226625e+00
n5	4.871649e+00
n6	5.781840e+00
n7	-1.84920e+00
n7.	-1.84920e+00
n8	-2.78625e+00
v(n5,n2)	4.085748e-02
v(n5,n8)	7.657901e+00
v(n6,n8)	8.568092e+00

Table 6: Values from Ngspice- Simulation Task 1)

Name	Value [A or V]
V_{N1}	0.000000e+00
V_{N2}	-7.143971e-16
V_{N3}	-2.238283e-15
V_{N5}	-6.113379e-16
V_{N6}	8.568097e+00
V_{N7}	1.476238e-16
V_{N8}	-0.000000e+00
$@I_b$	-7.459097e-19
$@I_d$	7.334357e-20
$@I_{H1}$	-2.783827e-03
$@V_x$	8.568097e+00
$@I_x$	-2.783827e-03
$@R_{eq}$	3.077812e+03
$@\tau$	3.197736e-03

Table 7: Values computed by Octave - Theoretical Task 2)

Name	Value [A or V]
i(vaux)	-8.67362e-19
i(h1)	2.783826e-03
@g1[i]	-6.50208e-18
@r1[i]	-6.20511e-18
@r2[i]	-6.50208e-18
@r3[i]	2.969639e-19
@r4[i]	1.313719e-18
@r5[i]	-2.78383e-03
@r6[i]	-8.67362e-19
@r7[i]	2.995961e-20
n1	0.000000e+00
n2	-6.22740e-15
n3	-1.95111e-14
n5	-5.32907e-15
n6	8.568092e+00
n7	1.745800e-15
n7.	1.745800e-15
n8	1.776357e-15
v(n5,n2)	8.983252e-16
v(n5,n8)	-7.10543e-15

Table 8: Values from Ngspice- Simulation Task 2)