

# **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**

<b>1</b>	<b>Introduction</b>	<b>2</b>
<b>2</b>	<b>Theoretical Analysis</b>	<b>3</b>
2.1	Task 1) . . . . .	3
2.2	Task 2) . . . . .	5
2.3	Task 3) . . . . .	5
2.4	Task 4) . . . . .	5
2.5	Task 5) . . . . .	5
2.6	Task 6) . . . . .	5
<b>3</b>	<b>Simulation Analysis</b>	<b>6</b>
3.1	Task 1) . . . . .	6
3.2	Task 2) . . . . .	6
3.3	Task 3) . . . . .	7
3.4	Task 4) . . . . .	7
3.5	Task 5) . . . . .	7
<b>4</b>	<b>Conclusion</b>	<b>7</b>

# 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:

$$v_s(t) = V_s u(-t) + \sin(2\pi f t) u(t) \quad (1)$$

$$u(t) = 0, t < 0 \quad (2)$$

$$u(t) = 1, t \geq 0 \quad (3)$$

In Section 2, a theoretical analysis (using two distinct methods) 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.

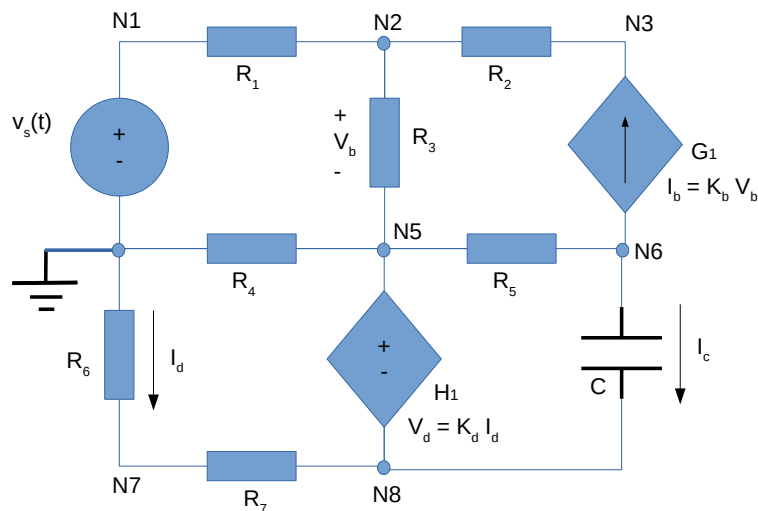


Figure 1: Circuit T2

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

## 2 Theoretical Analysis

In this section, the Circuit T2 is analysed theoretically. In figure ??, apart 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 (Kirchhoff's current law (KCL) - eq.(4) and Kirchhoff's voltage law (KVL) - eq.(5)); Ohm's law (eq.(6)).

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

$$\sum I_i = 0 \quad (4)$$

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

$$\sum V_i = 0 \quad (5)$$

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.

$$V_i = R_i I_i \quad (6)$$

### 2.1 Task 1)

Similarly to the previous subsection, for the node method, ground was considered to be where it is identified in Figure 1. In addition, assume  $V_{Ni}$  to be the voltage in node  $Ni$  (every node position can also be found in Figure 1).

The node method uses KCL in conjunction with Ohm's law to define equations that when solved give the voltage value of each node in relation to ground (Node 0,  $V_0 = 0$ ). In this circuit we defined six equations that equate the currents entering a particular node with the currents leaving said node.

In order to have equations that solve for the node's voltage, a relation between current 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  $R_n$  ( $G_n = \frac{1}{R_n}$ )

$$G_2(V_1 - V_2) + G_1(V_3 - V_2) - G_3(V_2 - V_4) = 0 \quad (7)$$

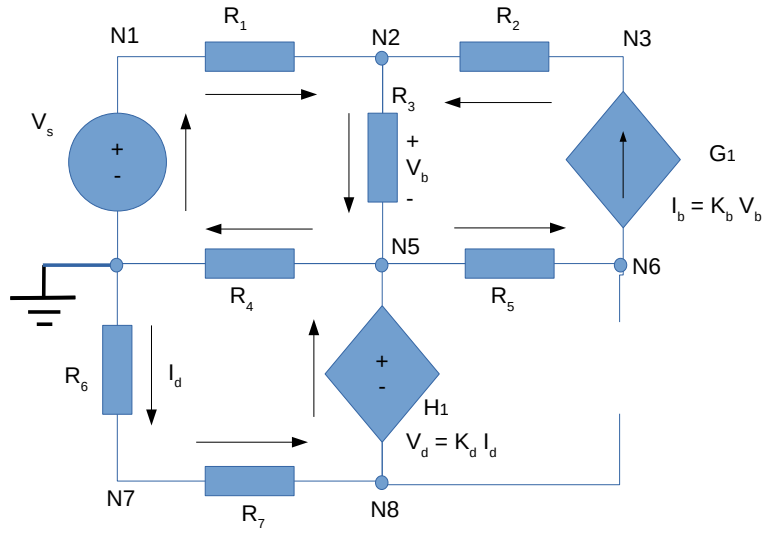


Figure 2: Circuit T2, analysed by Ngspice

$$G_2(V_1 - V_2) + K_b(V_4 - V_2) = 0 \quad (8)$$

$$G_3(V_2 - V_4) + G_5(V_5 - V_4) - G_4V_4 + I_c = 0 \quad (9)$$

$$K_b(V_2 - V_4) - G_5(V_5 - V_4) = -I_d \quad (10)$$

$$G_7(V_6 - V_7) - I_c = I_d \quad (11)$$

$$G_6(-V_6) - G_7(V_6 - V_7) = 0 \quad (12)$$

In order to solve the circuit two more equations are used to relate the voltage difference between the nodes that are connected to the voltage sources.

$$V_3 = V_a \quad (13)$$

$$K_c \frac{(-V_6)}{G_6} - (V_4 - V_7) = 0 \quad (14)$$

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G1 & -(G1 + G2 + G3) & G2 & G3 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -G2 & G2 & 0 & 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & G3 & 0 & -(G3 + G4 + G5) & G5 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -G5 & G5 & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & 0 & 0 & 0 & G7 & -G7 & -1 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & -(G6 + G7) & G7 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 1 & 0 & G6 * Kd & -1 & 0 & 0 & 0 \\ 0 & Kb & 0 & -Kb & 0 & 0 & 0 & 0 & -1 & 0 \end{bmatrix}$$

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

**2.3 Task 3)**

**2.4 Task 4)**

**2.5 Task 5)**

**2.6 Task 6)**

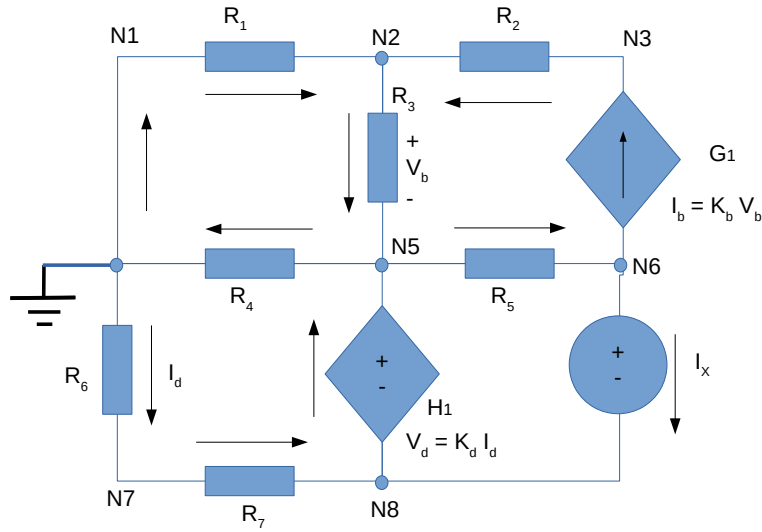


Figure 3: Circuit T2, analysed by Ngspice

### 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 electronic 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 additional voltage source ( $V_{aux}$ ) 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 additional node had to be added (node  $N7$ ).

As previously stated,  $I_b$  is referred 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 3 shows the simulated operating point results for Circuit T2.

The three last entries in Table 3 provides the potential difference 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  $t = 0$ . The capacitor is replaced with a voltage source, with its value being equal to the difference between the voltages in nodes  $n6$  and  $n8$  (or  $V_x = V(n6) - V(n8)$ ) obtained in subsection 3.1.

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

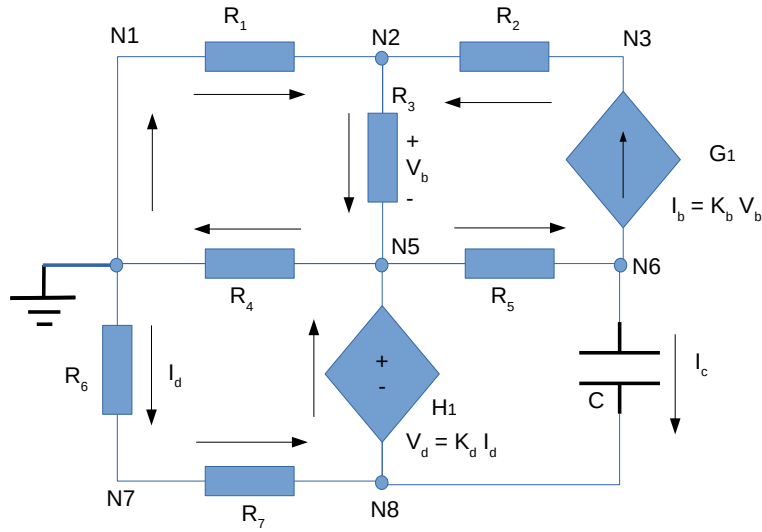


Figure 4: Circuit T2, analysed by Ngspice

### 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 plotted in the interval  $[0;20]ms$  (Figure 12).

### 3.4 Task 4)

In this subsection, the total (natural and forced) response 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 node  $n6$  is simulated.

## 4 Conclusion

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

Static analyses were performed theoretically, through mesh and node analysis and by circuit simulation, using the Octave math tool and Ngspice tool, respectively. The simulation results matched the theoretical results very precisely. So in that order it is safe to say that our objective was achieved successfully.

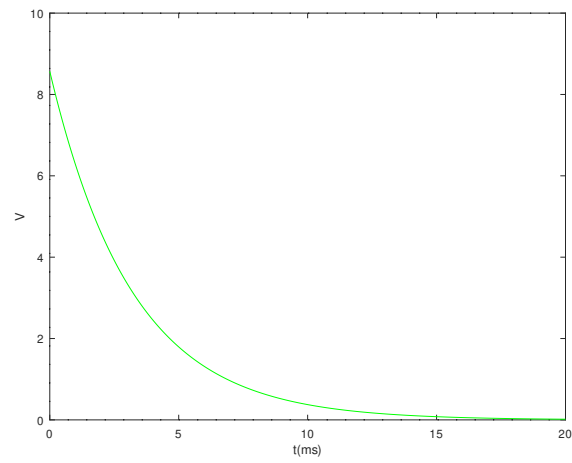


Figure 5: Plot oct - 1

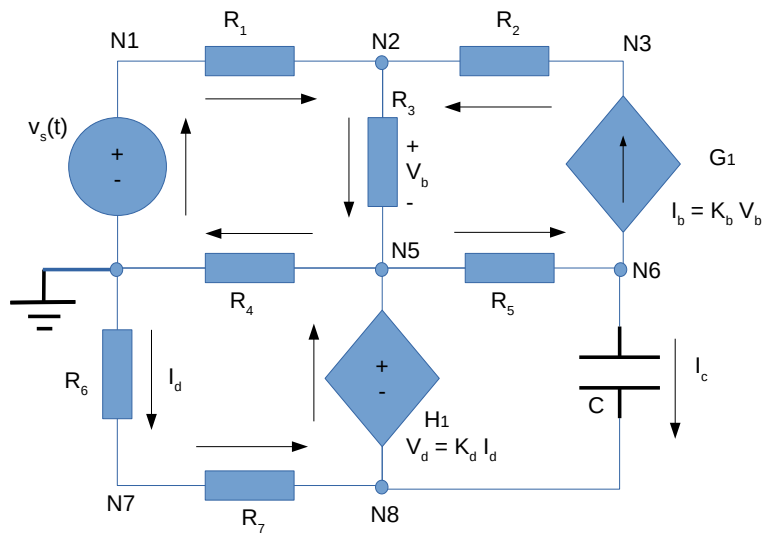


Figure 6: Circuit T2, analysed by Ngspice



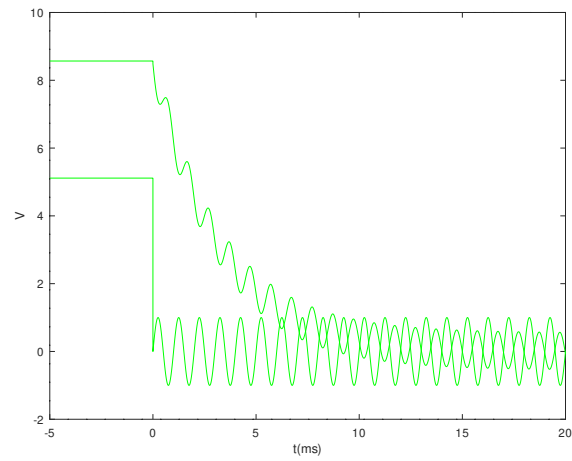


Figure 7: Plot oct - 2

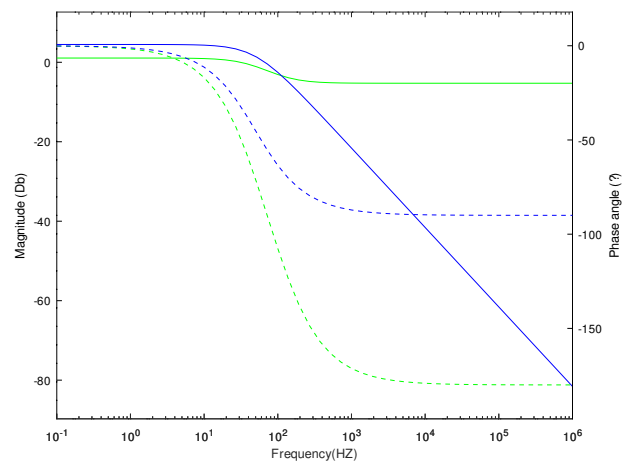


Figure 8: Plot oct - 3

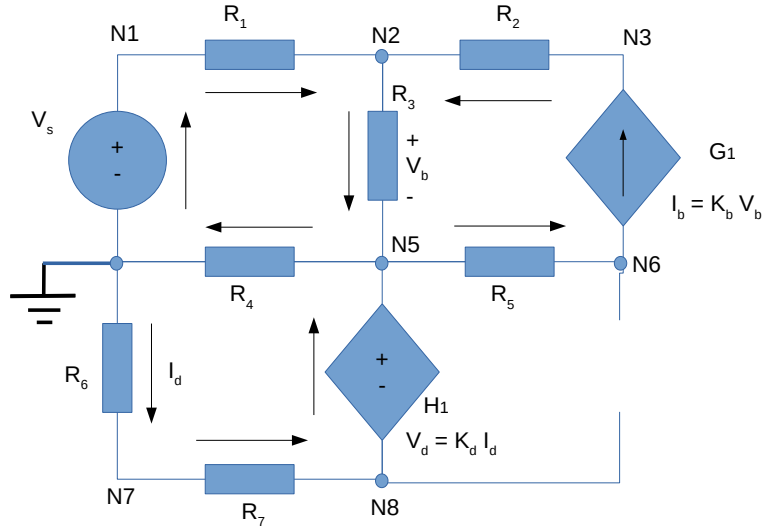


Figure 9: Circuit T2, analysed by Ngspice

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).

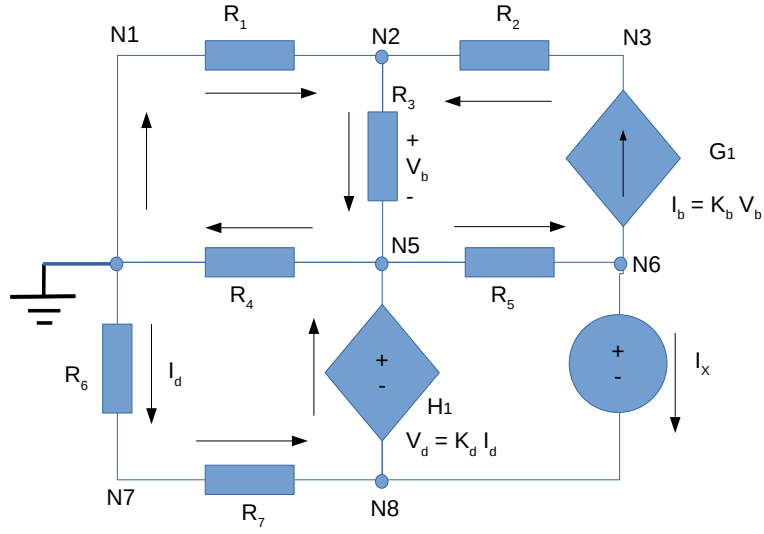


Figure 10: Circuit T2, analysed by Ngspice

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).

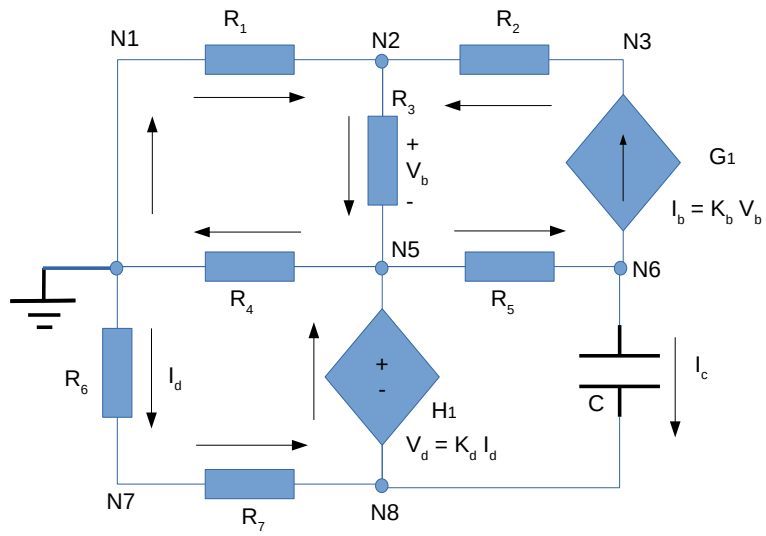


Figure 11: Circuit T2, analysed by Ngspice

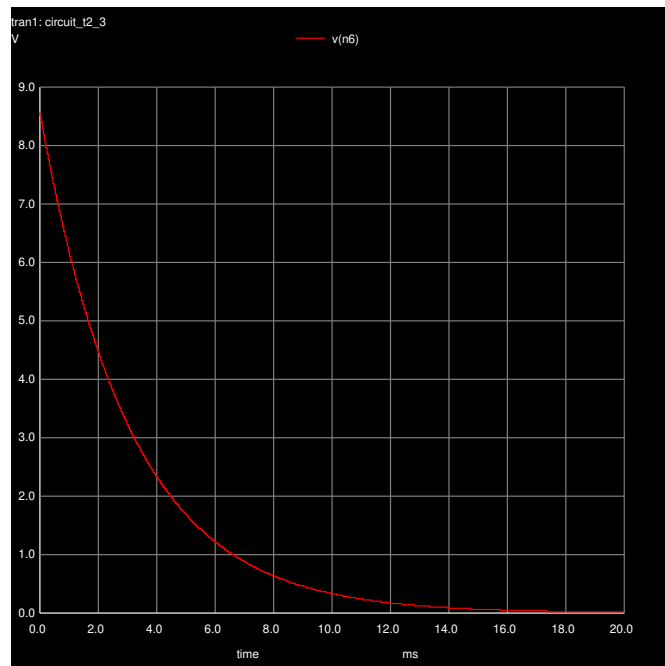


Figure 12: Transient analysis - 1: natural response on node  $n6$

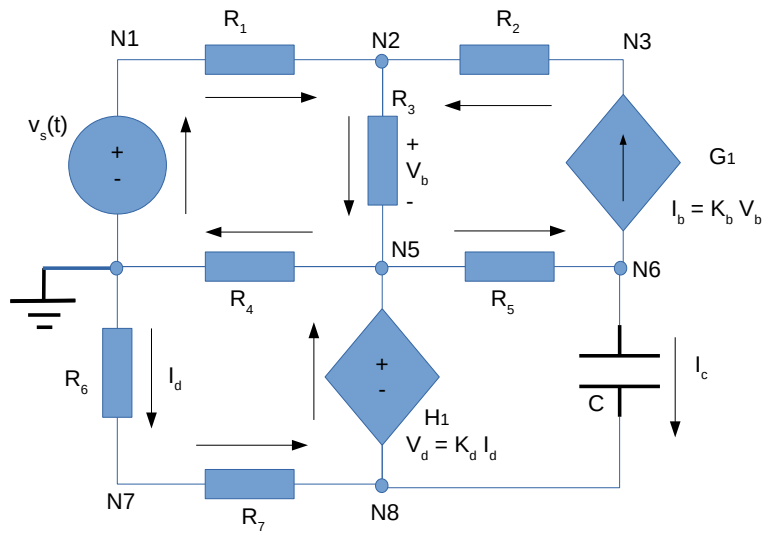


Figure 13: Circuit T2, analysed by Ngspice

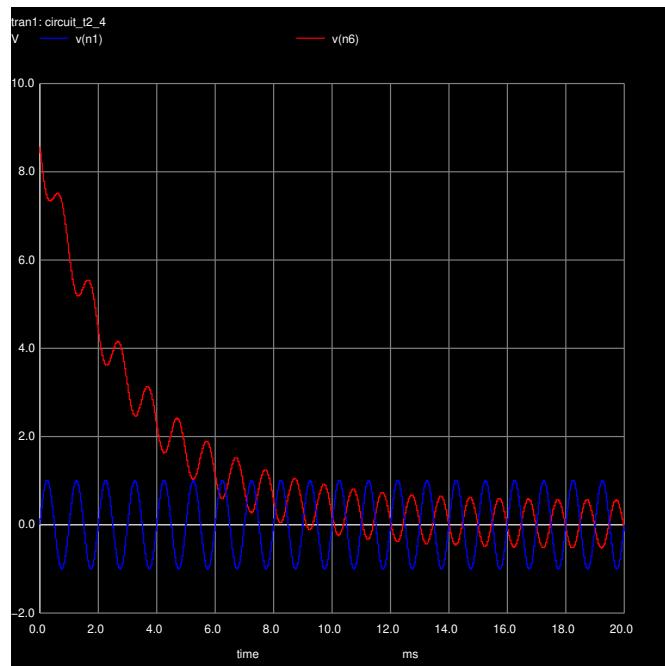


Figure 14: Transient analysis - 2: total response on node  $n6$

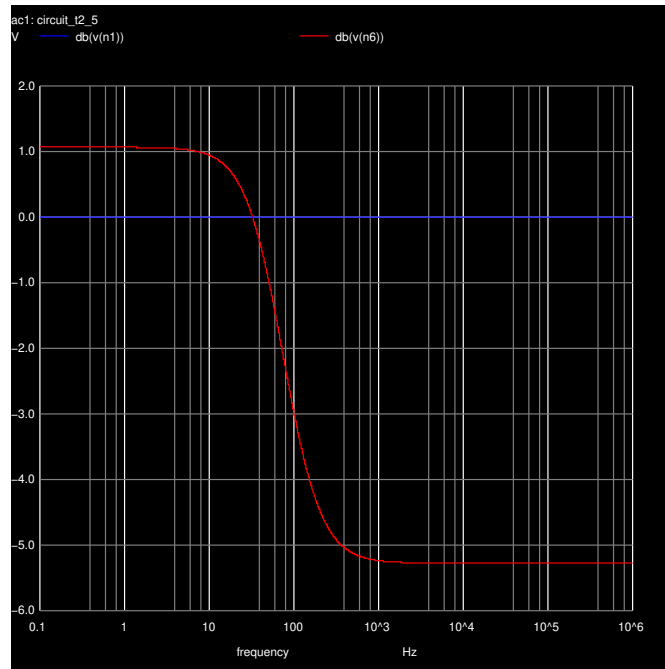


Figure 15: Frequency response - 1

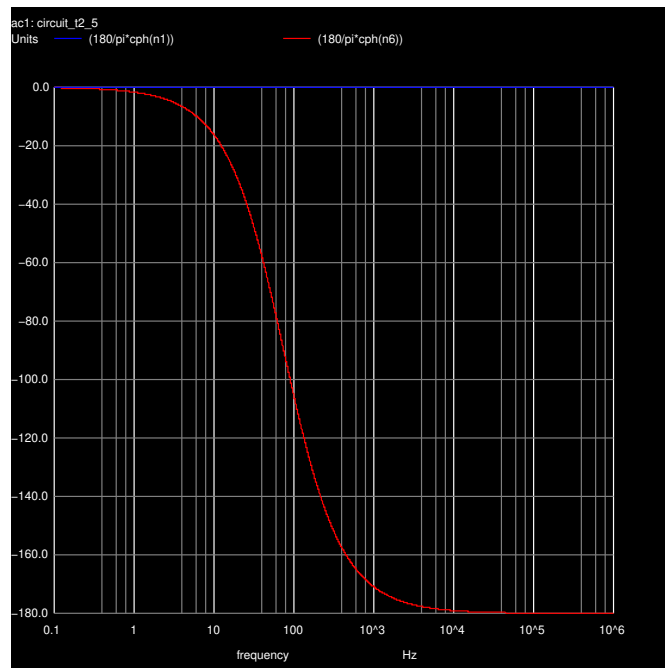


Figure 16: Frequency response - 2