Laboratory Report 2:RC Circuit Analysis

Circuit Theory and Electronics Fundamentals

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

5, 2021

Work by:

Beatriz Contente 95772

Francisco Fonseca 95789

Manuel Carvalho 95823

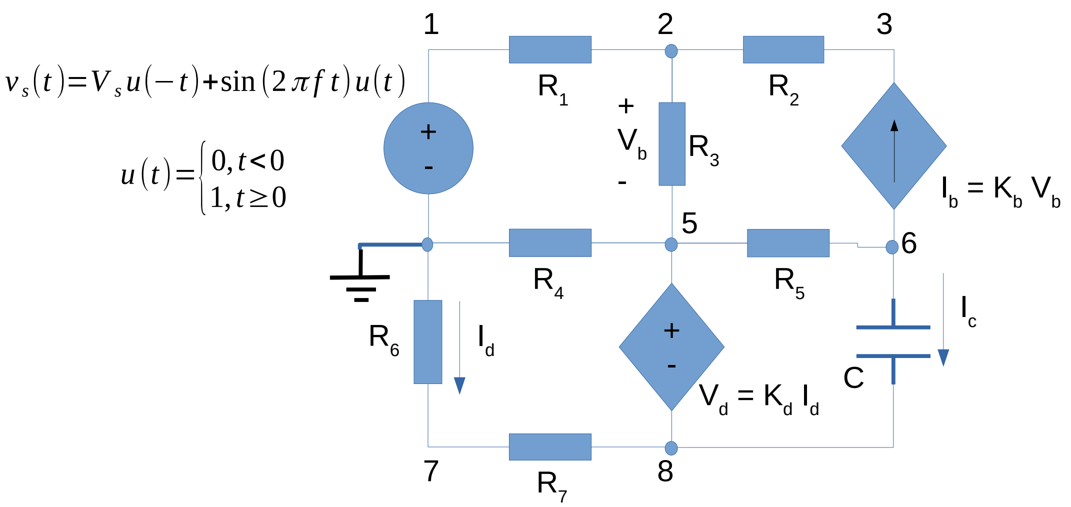
Contents

1 Introduction

The objective of this laboratory assignment is to analyze a RC circuit to find the natural and forced response as well as doing a frequency analysis. Furthermore, is it asked to run a simulation using NgSpice to detect small diferences between the different approaches and understand why said differences happen. The circuit can be seen in Figure 1.

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: RC Circuit with alternate voltage source (V_s), linear dependent sources (V_d-linear current controlled voltage source and I_b-linear voltage controlled current source) and capacitor C



The values given for this report can be found in table 1.

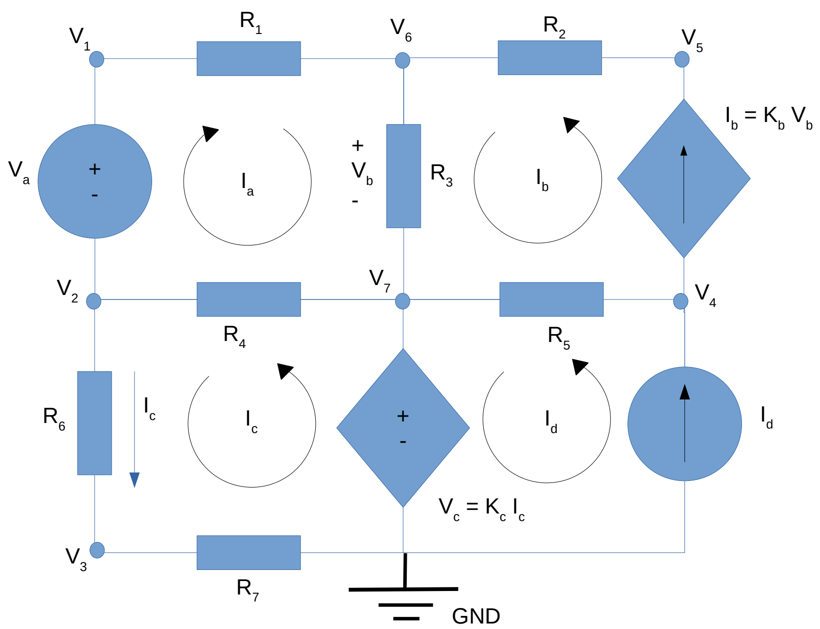
Name	Values
R1	1.01949191994 KΩ
R2	2.05054429461 KΩ
R3	3.09286027724 KΩ
R4	4.12838973576 KΩ
R5	3.06635427647 KΩ
R6	2.01254230153 KΩ
R7	1.00502981701 KΩ
Va	5.24204797361 V
C	1.01905568201 uF
Kb	7.23185131759 mS
Kd	8.12820254987 KΩ

Table 1: Values received by the Python program that can be found in folder $python$.

2 Theoretical Analysis

2.1 Exercise 1

Figure 2: Representation of mesh currents in the circuit.



The first thing which must be noticed is that there is an independent voltage source (V_s) and a linear current controlled voltage source (V_d) in this circuit. Knowing that a nodal analysis can't include the analysis of nodes that are connected to voltage sources, it becomes clear that is useless to analyze nodes 1 and 4 (connected to V_s) and also nodes 5 and 8 (connected to V_d) using this method.

From figure **??**, we can easily conclude that there are 11 unknown variables: V_b, I_b, V_d, I_d, V_1, V_2, V_3, V_5, V_6, V_7 and V_8. In this exercise, V_s is constant and the capacitor is assumed to be also constant and fully charged, meaning that the current I_c is 0 (open circuit behavior).

For the node analysis, it is necessary to consider 7 linearly independent equations to reach all the values corresponding to the voltages in each node and consequently solve the circuit. As referred above, it is possible to use the nodal method to analyze nodes 2, 3, 6 and 7.

We begin by establishing the following equations, being that equations 1 and 2 were given by the professor. Equation 3 was obtained by relating the voltage difference between nodes 2 and 5 to the voltage V_b. Finally, by using Ohm's Law for resistor R_6 and remembering that V_4 is null, we get the last equation (4) for I_d.

$$I_b = K_b V_b, \tag{1}$$

$$V_d = K_d I_d, \tag{2}$$

$$V_b = V_2 - V_5, \tag{3}$$

$$I_d = -V_7 G_6, \tag{4}$$

The equation written below (5) is a direct consequence of our choice of connecting node 4 to the ground (GND), because this choice makes evident that the value of V_4 is 0 and then:

$$V_1 - V_s = 0, \tag{5}$$

By analysing node 2 using Kirchoff's Current Law and Ohm's Law for the resistors R_1, R_2 and R_3, we get the following equation:

$$(V_3 - V_1)G_1 + (V_2 - V_3)G_2 + (V_2 - V_5)G_3 = 0, \tag{6}$$

The following equation (7), in which was also used Kirchoff's Current Law and Ohm's Law for resistor R_2, refers to node 3. Here we consider the given equation 1 and the equation 3 to substitute the current I_b.

$$(V_3 - V_2)G_2 - (V_2 - V_5)K_b = 0, \tag{7}$$

For node 6, the ensuing equation (8) was figured out by resorting to Ohm's Law for the resistor R_5 and Kirchoff's Current Law. Equations 1 and 3 were once again used to avoid using I_b. Remember that, for this exercise, I_c is null.

$$(V_6 - V_5)G_5 + (V_2 - V_5)K_b = 0, \tag{8}$$

Finally, for node 7, Kirchoff's Circuit Law and Ohm's Law (for resistors R_6 and R_7) were used to establish the following mathematical relation:

$$V_7G_6 + (V_7 - V_8)G_7 = 0, \tag{9}$$

Since there are 7 unkown varibles, we need two more equations. The first one (10) is obtained by relating V_d to the voltage difference in nodes 5 and 8 and replacing V_d for the equations 2 and 4.

$$-V_7G_6K_d - (V_5 - V_8) = 0, \tag{10}$$

Ultimately, to discover the last equation, there are some theoretical concepts that must be considered. Kirchoff's Current Law implies that there is no current stuck at any node. It is also known that neither voltage sources nor resistors retain current. Then, any branch that only contains one of the said elements does not retain current as well. Merging the two branches placed on the left side of circuit (branch containing V_s with branch containing R_6) and calling Supernode to the result of this merger, it is still true that no current is retained in the Supernode. Then, considering that all unkown currents are diverging from the nodes, the resultant equation (11) is the one written right below.

$$(V_1 - V_2)G_1 - V_5G_7 - V_7G_6 = 0, \tag{11}$$

A system with the 7 linearly independent equations and 7 variables (regarding the voltage in each node) is, of course, possible to solve but not easy (and certainly not pratical) to deal with. The following matrix equation (12) summarizes the 7 referred equations so it is easier to read and to instantaneously solve (by using Octave).

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G_1 & G_1+G_2+G_3 & -G_2 & -G_3 & 0 & 0 & 0 \\ 0 & -G_2-K_b & G_2 & K_b & 0 & 0 & 0 \\ 0 & K_b & 0 & -K_b-G_5 & G_5 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & G_6+G_7 & -G_7 \\ G_1 & -G_1 & 0 & -G_4 & 0 & -G_6 & 0 \\ 0 & 0 & 0 & -1 & 0 & -K_dG_6 & 1 \end{bmatrix} \times \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix} = \begin{bmatrix} V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{12}$$

To find all the required branches' currents we can use Ohm's Law for each resistor, which can be seen in equations 13 to 19: Now we no longer consider the currents to be diverging from the nodes, but assume the directions shown in figure **??**.

$$R_1[i] = \frac{V_1 - V_2}{R_1}, \tag{13}$$

$$R_2[i] = \frac{V_2 - V_3}{R_2}, \tag{14}$$

$$R_3[i] = \frac{V_5 - V_2}{R_3}, \tag{15}$$

$$R_4[i] = \frac{V_5}{R_4}, \tag{16}$$

$$R_5[i] = \frac{V_6 - V_5}{R_5}, \tag{17}$$

$$R_6[i] = -\frac{V_7}{R_6}, \tag{18}$$

$$R_7[i] = \frac{V_7 - V_8}{R_7}, \tag{19}$$

Node	**Voltage[V]**
V_b	-3.39424400e-02
V_d	8.01098169e+00
V_1	5.24204797e+00
V_2	5.00298504e+00
V_3	4.49964474e+00
V_5	5.03692748e+00
V_6	5.78961528e+00
V_6	-1.98351843e+00
V_8	-2.97405421e+00

Table 2: Voltage and Current values(Exercise 1)

Branch	**Current[A]**
I_b	-2.45466679e-04
I_c	0.00000000e+00
$R_6 = I_d$	9.85578501e-04
R_1	2.34492229e-04
R_2	2.45466679e-04
R_3	1.09744498e-05
R_4	1.22007073e-03
R_5	2.45466679e-04
R_7	9.85578501e-04

3 Simulation Analysis

3.1 Exercise 1

In this section we proceed to do the anlysis of the circuit through the use of the Ngspice simulation program. In figure 3 we have the circuit that was inputed into Ngspice (and also the considered current flows and nodes). The file can be found at the sim folder inside the $T2$ folder.

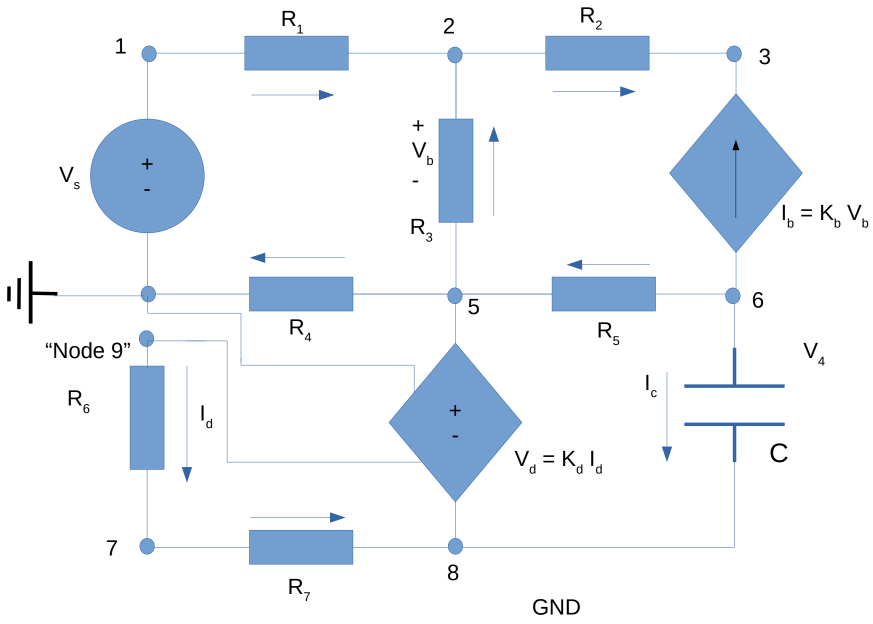


Figure 3: Considered circuit for Ngspice simulation

V_9 refers to an extra ficticious node created specifically for the Ngspice simulation, and it is below $0(GND)$ and above resistor R6 as it can be seen above in figure 3.The reason this node is necessary is because when creating a current controlled voltage source, Ngspice gets the current value by refering to a voltage source from where the current goes through. Since I_d does not go through any voltage source in the circuit (does not go through V_s) we used this extra node to create a voltage source of $0V$ (Which can be confirmed since V_9 =GND$=0$) from which we are certain I_d is passing by.

Table 3 shows the simulated operating point results for the circuit under analysis given the values found on Table 1, considering t<0, which means $v_S(t)$ = V_s as seen in Figure 1. The variables representation and format are automatically determined by Ngspice.

We can get all the missing values given the voltage different of the nodes where they are defined.

$$V_b = v(v2, v5) = -3.39424e - 02V \tag{20}$$

$$V_d = v(v5, v8) = 8.010982e + 00V \tag{21}$$

3.2 Exercise 2

In this section, we simulate the operating point for $v_s(0) = 0$, replacing the capacitor with a voltage source $V_x = V_6 - V_8$ using the values of the respective nodes as obtained in section 3.1.

Table 3: Operating point for $t < 0$. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt. (the g in "gib" refers to the Ngspice notation of a voltage controlled current source)

Name	**Value [A or V]**
@c1[i]	0.000000e+00
@gib[i]	-2.45467e-04
@r1[i]	2.344922e-04
@r2[i]	2.454667e-04
@r3[i]	1.097445e-05
@r4[i]	1.220071e-03
@r5[i]	2.454667e-04
@r6[i]	9.855785e-04
@r7[i]	9.855785e-04
v1	5.242048e+00
v2	5.002985e+00
v3	4.499645e+00
v5	5.036927e+00
v6	5.789615e+00
v7	-1.98352e+00
v8	-2.97405e+00
v9	0.000000e+00

It is required to replace the capacitor since we can assume that an infinite time as passed until t=0 and it is fully charged, meaning it starts with the voltage given before we start the time.

Table 4 shows the simulated operating point results for the circuit under analysis given the values found on Table 1, considering t=0, and the above mentioned considerations.

Table 4: Operating point for $t = 0$, $v_s(0) = 0$ and capacitor replaced with $V_x = V_6 - V_8$. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt. (the g in "gib" refers to the Ngspice notation of a voltage controlled current source)

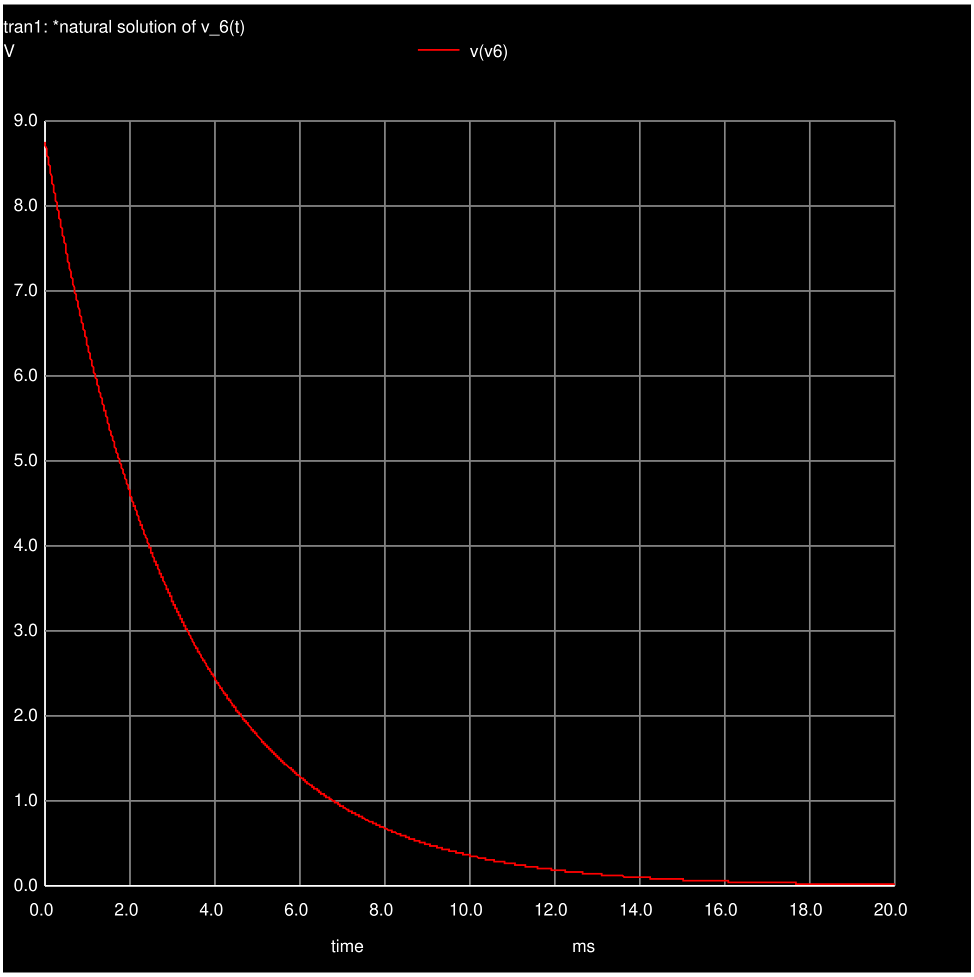
Name	**Value [A or V]**
@gib[i]	0.000000e+00
@r1[i]	0.000000e+00
@r2[i]	0.000000e+00
@r3[i]	0.000000e+00
@r4[i]	0.000000e+00
@r5[i]	2.858009e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
v1	0.000000e+00
v2	0.000000e+00
v3	0.000000e+00
v5	0.000000e+00
v6	8.763669e+00
v7	0.000000e+00
v8	0.000000e+00
v9	0.000000e+00

3.3 Exercise 3

In this section we used the results obtained in section 3.2 to determine the natural solution $v_{6n}(t)$ in the interval $[0, 20]ms$ using the boundary conditions V_6 and V_8.

Figure 4 shows the plot of the required result that was obtained using the transient analysis.

Figure 4: Natural solution, $v_{6n}(t)$ in V in the interval $[0, 20]ms$

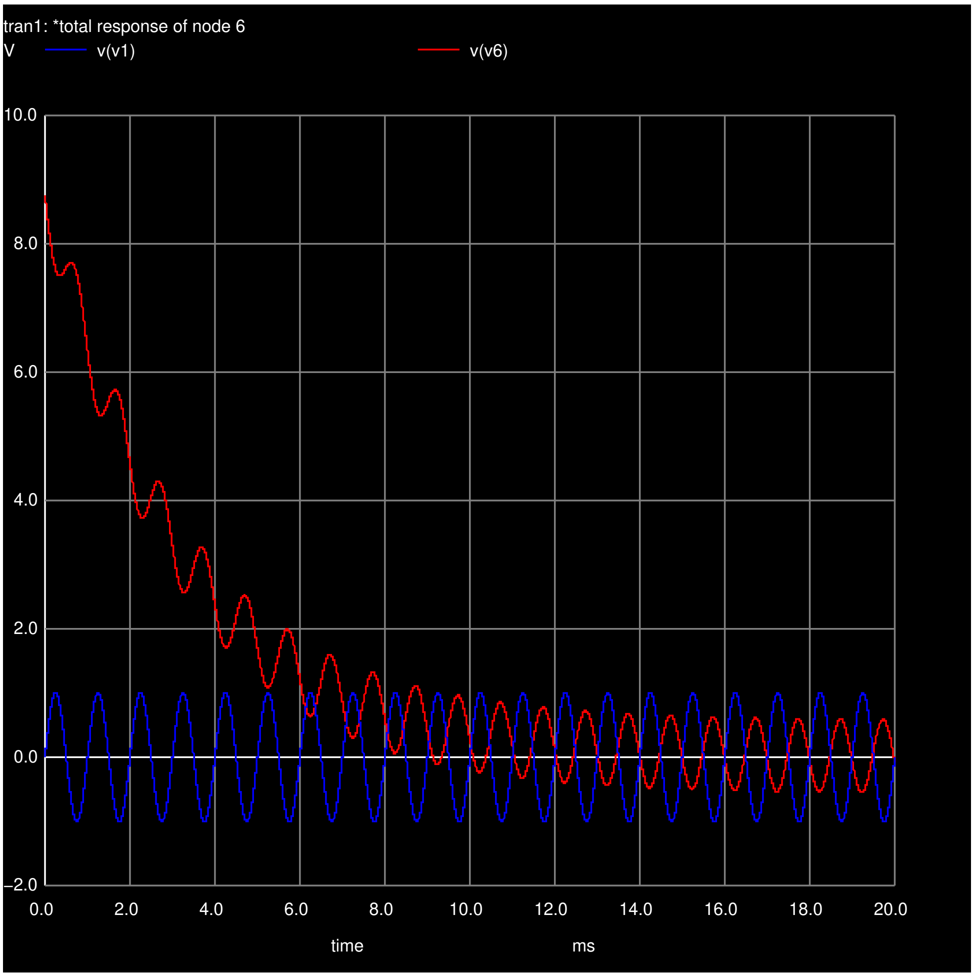


3.4 Exercise 4

In this section we are required to simulate the total response in node 6 $v_6(t)$ by repeating the procedure of 3.3 with the initially given $v_s(t)$ (shown in 1, with a frequency of $1000Hz$.

Figure 5 shows the plot of the required result that was obtained using the transient analysis as well as the stimulus (shown as $v(V_1)$ since $v_1(t) = v_s(t)$

Figure 5: Total response, $v_6(t)$ in V, and stimulus $v_s(t)$, in V in the interval $[0, 20]ms$



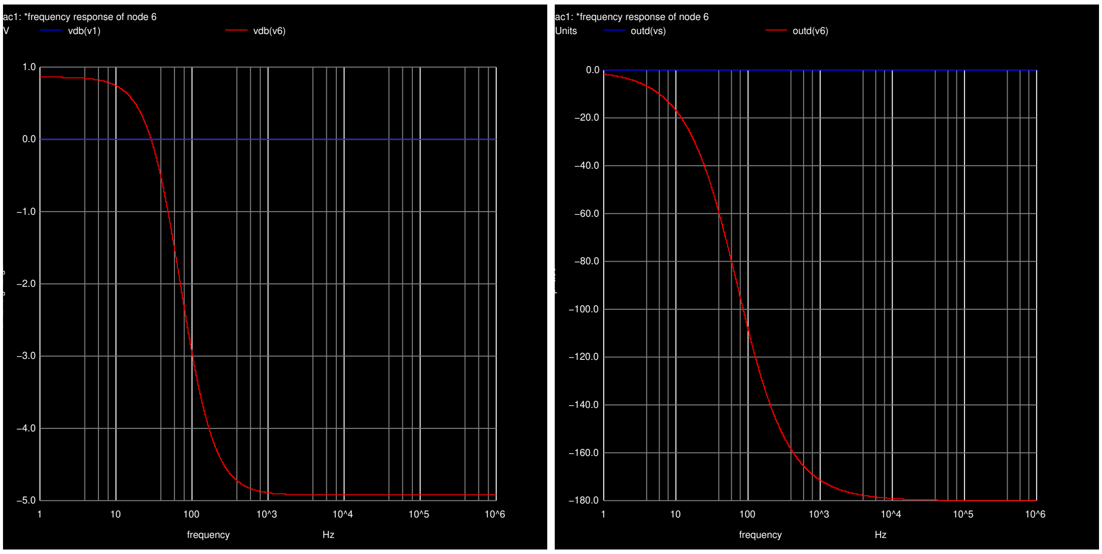
3.5 Exercise 5

In this section we are required to simulate the frequency response in node 6 $v_6(t)$ with the frequency in logscale, magnitude in dB and phase in degrees for a range of $0.1Hz$ to $1MHz$.

Figure 6 shows the plot of the required result, $v_6(f)$ as well as $v_s(f)$.

Since the source of of frequency change is v_s itself, it is expected to not to show a frequency response since it changes alongside frequency (and in $log(1) = 0$), where as v_6 is dependant of the value that v_s takes.

Figure 6: Frequency response, $v_6(f)$ in V, and $v_s(f)$, in V. Phase is in degrees and Magnitude in dB



4 Conclusion

By analysing the circuit theoretically, with both the mesh and the node methods, and then simulating the circuit using Ngspice, we can verify that the values of the unknown components match almost perfectly and all approaches agree on the final currents' directions across the circuit's branches (which can be seen below in figure 7).

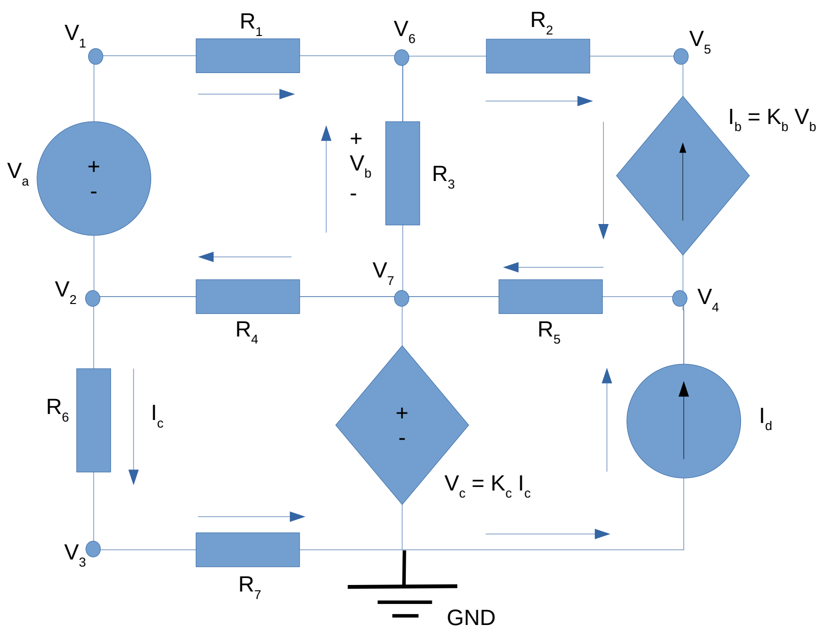


Figure 7: Representation of the final circuit with all the correct directions for the currents.

Some small discrepancies like the value of V_b obtained in section 3 differing from the ones found on sections **??** and **??** are due to the small number of decimal places considered by

Ngspice leading to slight inaccuracies. However, considering that the circuit complexity is still not considered, the differences are negligible.

In any case, the node analysis uses the Kirchoff Current Law while the Mesh Analysis uses the Kirchoff Voltage Law. This means that if a circuit contains more voltage sources than current sources, the mesh method is going to be more exact, as it is the other way around.

In this circuit, both types of sources have the same number of components; a fact that might help justify the accuracy of the results. Even more, it only includes linear components, making it so that the theoretical values are expected to be almost the same as the ones obtained in the simulation. Furthermore, as already mentioned, the low complexity of the circuit makes it that both methods are still extremely reliable.

All of this leads to the conclusion that the making of this laboratory assignment was coherent and that the main goal was attained: to achieve the circuit analysis through 3 different methods (mesh, nodal and simulated analysis).