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

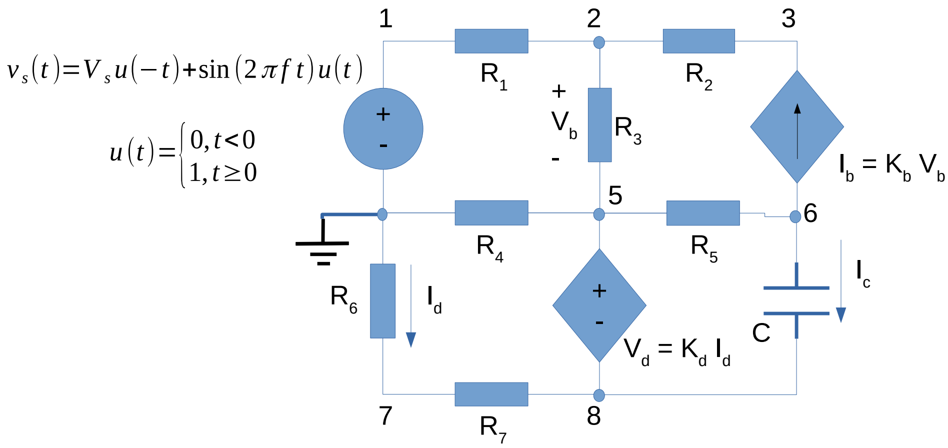
Contents

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 3.

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.

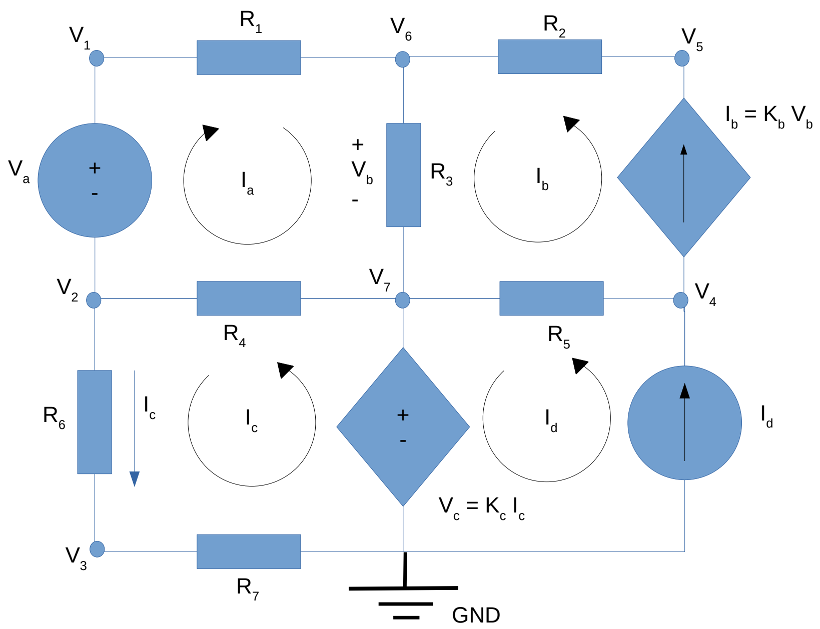


Figure 2 shows the mesh currents considered for the circuit analysis, with the current I_a flowing clockwise and the rest of the currents (I_b, I_c and I_d) flowing counter-clockwise. In the meshes containing I_b and I_d, the currents were considered to be the same as the current sources in said meshes.

From this circuit, there can then be extracted 3 equations to figure out the value of the components necessary for the circuit analysis.

The first one, Equation 1, was obtained by using Ohm's Law, assuming it is known the value of the voltage and the resistance in resistor 3 and that the current flowing through it is (I_a + I_b).

$$I_b = K_b(I_a + I_b)R_3, \tag{1}$$

Equation 2 was figured out by analysing the top left mesh, using Kirchoff's Voltage Law and Ohm's Law for the resistors. Since the current I_a is flowing clockwise, the voltage in V_a is negative and the currents in resistors 3 and 4 are, correspondingly, (I_a + I_b) and (I_a + I_c), as these pairs of currents are flowing the same way in said resistors.

$$-V_a + I_aR_1 + (I_a + I_b)R_3 + (I_a + I_c)R_4 = 0, \tag{2}$$

Finally, from the bottom left mesh, there is Equation 3, in which was also used Kirchoff's Voltage Law and Ohm's Law. The voltage in V_c is negative due to the current flow.

$$-K_cI_c + I_cR_6 + I_cR_7 + (I_a + I_c)R_4 = 0, \tag{3}$$

By developing these 3 equations, the matrix below (4) is achieved as to simplify the calculations. This matrix was solved in Octave, getting the values of the currents I_a, I_b and I_c that can be found in table **??**. It was not necessary to solve for the value of the current in the bottom right mesh since it is already known (equivalent to I_d).

$$\begin{bmatrix} -K_bR_3 & 1-K_bR_3 & 0 \\ R_1+R_3+R_4 & R_3 & R_4 \\ R_4 & 0 & R_6+R_7-K_c+R_4 \end{bmatrix} \times \begin{bmatrix} I_a \\ I_b \\ I_c \end{bmatrix} = \begin{bmatrix} 0 \\ V_a \\ 0 \end{bmatrix} \tag{4}$$

With these currents, it is possible to discover the values of the voltages in each node, using the equations 5 through 11 down below and knowing that I_b = K_bV_b and V_c=K_cI_c.

$$V_7 = V_c, \tag{5}$$

$$V_6 = V_7 + V_b, \tag{6}$$

$$V_5 = V_6 + R_2I_b, \tag{7}$$

$$V_4 = V_7 + R_5(I_d - I_b), \tag{8}$$

$$V_3 = V_0 + R_7I_c, \tag{9}$$

$$V_2 = V_3 + R_6I_c, \tag{10}$$

$$V_1 = V_2 + V_a, \tag{11}$$

Table 2 shows the nodes' voltages discovered by replacing the known variables in equations 5 to 11. The branch currents were obtained with the following equations 12 to 17 (resorting to figure 2).

$$R_1[i] = I_a, \tag{12}$$

$$R_2[i] = I_b, \tag{13}$$

$$R_3[i] = I_a + I_b, \tag{14}$$

$$R_4[i] = I_a + I_c, \tag{15}$$

$$R_5[i] = Id - Ib, \tag{16}$$

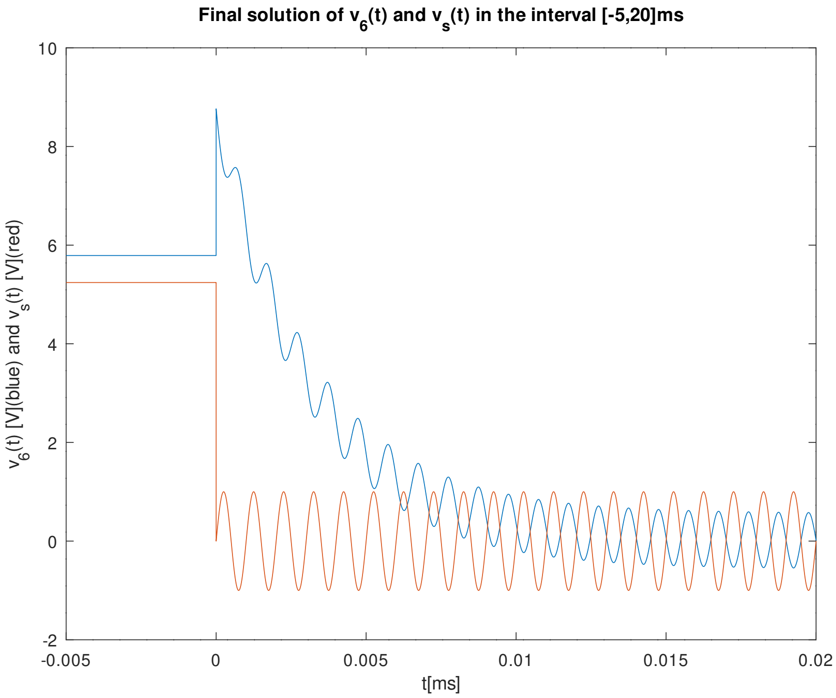
$$R_6[i] = R_7[i] = I_c, \tag{17}$$

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

Branch	Current[A]
I_b	-2.45466679e-04
I_c	0.00000000e+00
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_6	9.85578501e-04
R_7	9.85578501e-04

Table 2: Voltage and Current values(Exercise 1)

Figure 3: Final solution of $v_6(t)$ and $v_s(t)$ in the interval $[-5, 20]ms$



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 programm. In figure 4 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.

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 4.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.

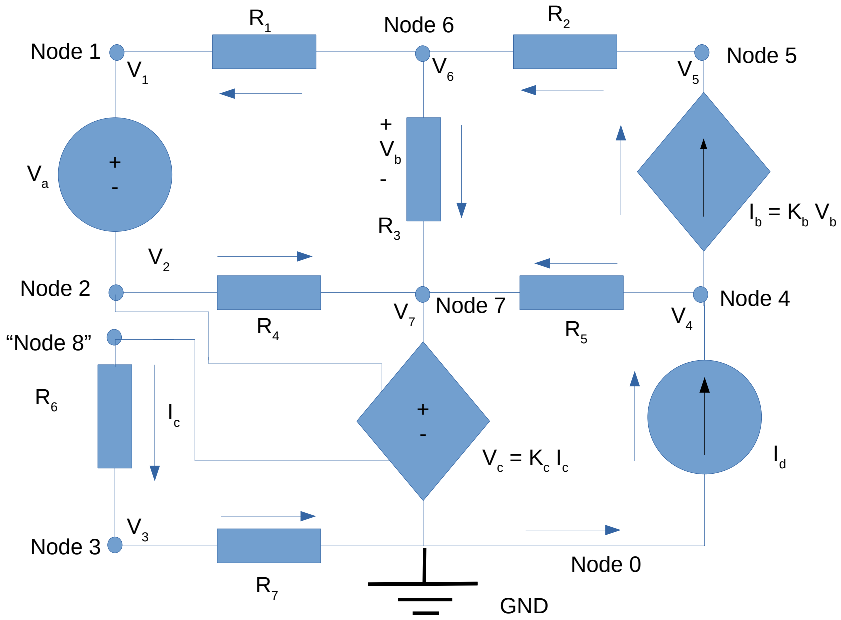


Figure 4: Considered circuit for Ngspice simulation

Table 3 shows the simulated operating point results for the circuit under analysis given the values found on Table 3. The variables representation and format are automatically determined by Ngspice.

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

Table 3: Operating point. 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)

We can get all the missing values given the equations showed in section 2.

$$V_c = V_7 \tag{18}$$

$$V_b = V_6 - V_7 => V_b = -3.3943 * 10^{-2} V \tag{19}$$

From Table 3 we can directly get the value of I_d:

$$I_d = @r6[i] \tag{20}$$

With this we have finalized the circuit anlysis through simulation with Ngspice.

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 5).

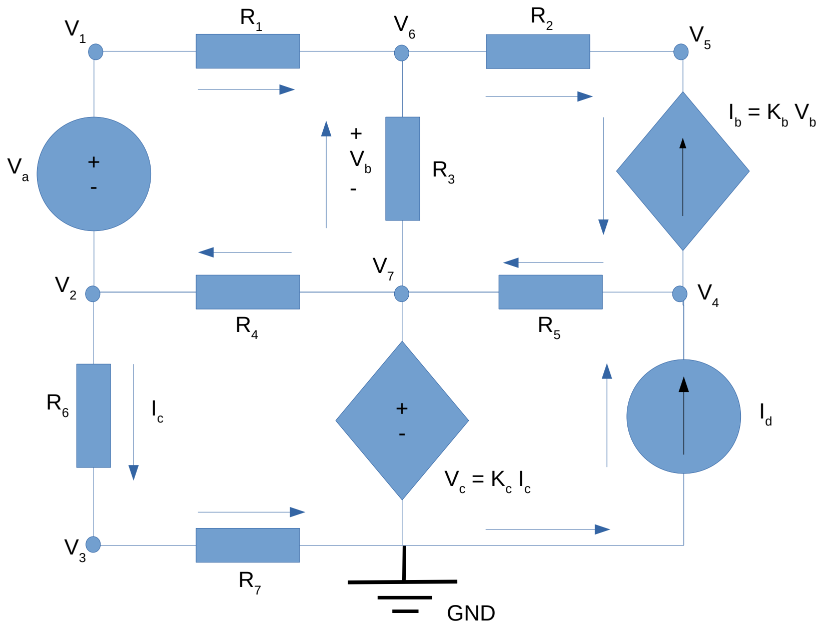


Figure 5: Representation of the final circuit with all the correct directions for the currents.

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).