TÉCNICO LISBOA

Circuit Theory and Electronics Fundamentals

Laboratory 1: Resistive Circuit

Integrated Masters in Aerospace and Technological Physics Engineering, Técnico, University of Lisbon

João Peixoto, 95807; Pedro Farinha, 95838; Patrícia Curado, 96559

March 25, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing resistors connected both in parallel and in series to other active components. These resistors are connected to independent voltage and current sources and dependent voltage-controlled current and current-controlled voltage sources.

In Section 2, a theoretical analysis using both the nodal and the mesh method is presented. The results are obtained using *Octave* [1]. In Section 3, the circuit is analysed by simulation using *NgSpice* [2]. The results are then compared to the theoretical results obtained in Section 2. The conclusions of this study are outlined in Section 4. Below, Figure 1 shows the circuit that was analysed and the respective node numbering that the group chose.

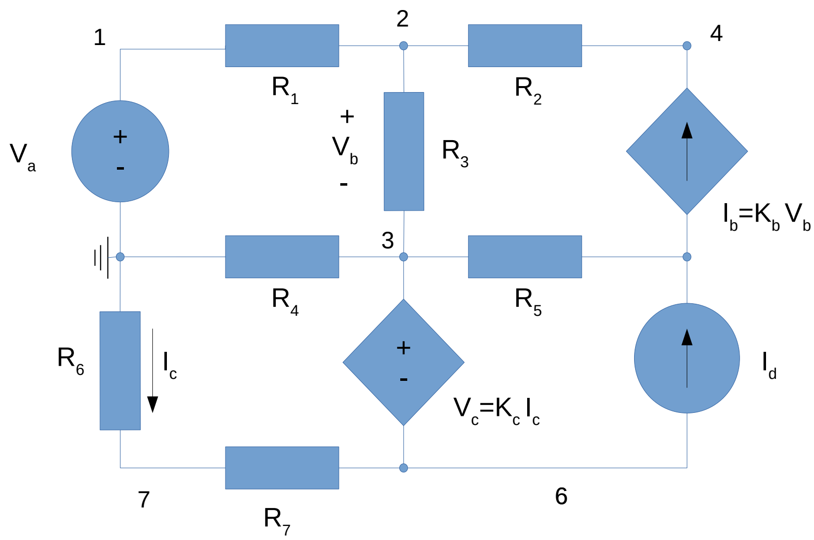

Figure 1: Circuit T1 and respective nodes

In order to start analysing the circuit, some data was needed. This data, such as the values of the resistances, the voltage of V_a, the current of I_d and K_b and K_c values, were all generated by the *Python* script, previously handed to us. The following data, using the lowest student number in our group, 95807, was obtained:

Name	**Value [Ω, V, A or S]**
R_1	1.04921233729$\times10^3$
R_2	2.01121557182$\times10^3$
R_3	3.04491334831$\times10^3$
R_4	4.07370420497$\times10^3$
R_5	3.04829678473$\times10^3$
R_6	2.08879558009$\times10^3$
R_7	1.01335761883$\times10^3$
V_a	5.22566789497
I_d	1.04037874222$\times10^{-3}$
K_b	7.00318569445$\times10^{-3}$
K_c	8.05999483103$\times10^3$

Table 1: Values for the constants used in the circuit analysis

2 Theoretical Analysis

The circuit presented in Figure 1 has eight nodes, eleven branches, four elementary meshes and thirteen loops. It is going to be attributed zero potential to one of the nodes, which we will call *reference* or *ground node*. Consequently, we will need seven node equations to get the voltages in the other nodes and four mesh equations to get the mesh currents.

2.1 Analysis using the mesh method

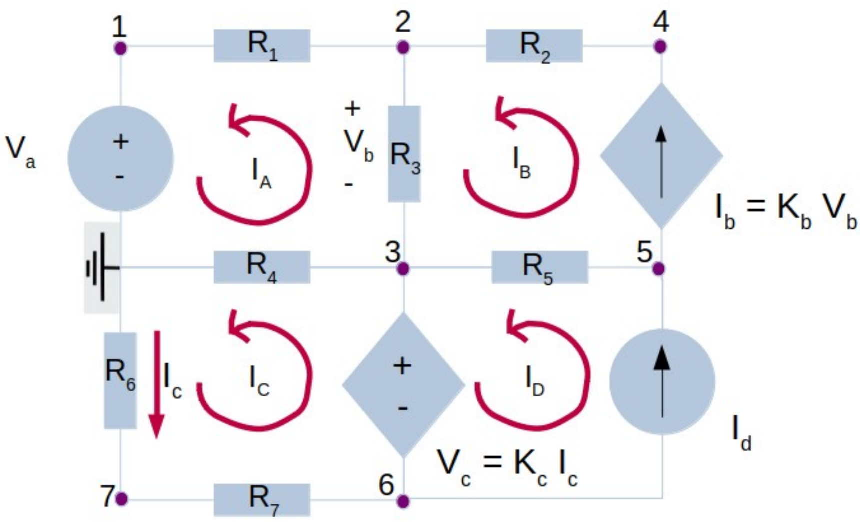

Figure 2: Circuit from the mesh method point of view

Mesh analysis is a useful method for determining the currents flowing through each resistor. We define the mesh currents (I_A, I_B, I_C and I_D as shown in Figure 2), one for each mesh. Then we apply KVL and Ohm's Law to the two meshes without current sources to get

$$V_a + R_1 I_A + R_4(I_A - I_C) = R_3(I_B - I_A)$$
$$R_6 I_C + R_7 I_C = V_C + R_4(I_A + I_C)$$

In order to complete the four equations needed, we observe

$$I_D = I_d$$
$$I_B = I_b$$

Substituting I_b for its corresponding formula $K_b V_b$ (due to it being a dependent current source), the final system can be derived and thus the mesh currents can be determined:

$$\begin{cases} I_D = I_d \\ I_B = K_b R_3(I_B - I_A) \\ V_a + R_1 I_A + R_4(I_A - I_C) = R_3(I_B - I_A) \\ R_6 I_C + R_7 I_C = V_C + R_4(I_A + I_C) \end{cases}$$

Name	Value [A or V]
I_b	-0.000231
I_d	0.001040
I_1	0.000220
I_2	0.000231
I_3	-0.000011
I_4	0.001234
I_5	-0.001271
I_6	0.001014
I_7	0.001014
V_1	5.225668
V_2	4.994742
V_3	5.027716
V_4	4.530306
V_5	8.903022
V_6	-3.145874
V_7	-2.118235

Table 2: Mesh analysis results. All currents *I* expressed in Ampère; all voltages *V* expressed in Volt.

2.2 Analysis using the node method

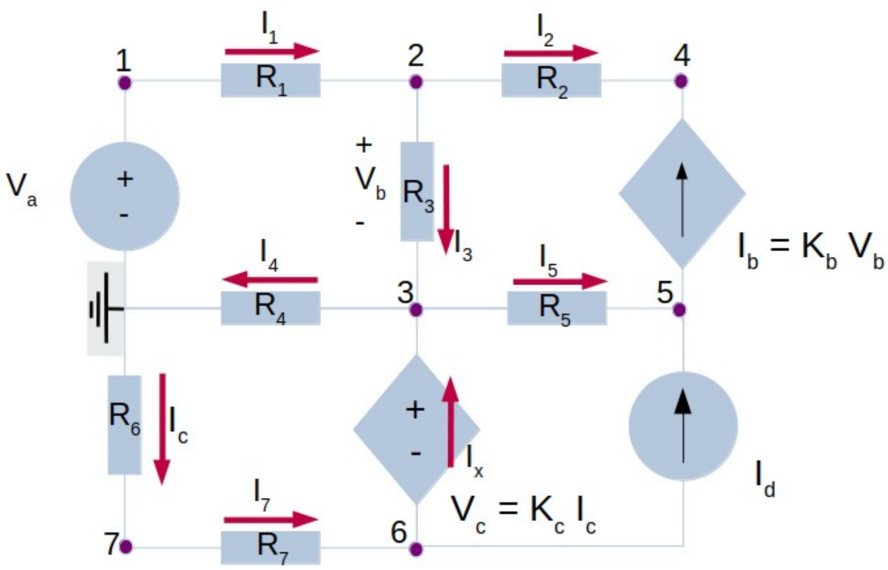

Figure 3: Circuit from the node method point of view

The main use of the node analysis is identifying the voltage in every node of the circuit. The first step to do so is to define the ground, the one node that has zero potential ($V_0 = 0$). The second step is to number each other node, so that when we reference the voltage in the node i we use the abbreviation V_i. We also use the notation I_i to refer to the current flowing through the resistor R_i, with the exceptions of I_c flowing through R_6 and I_x flowing through the dependent voltage source. Applying KCL to every non-reference node, we are able to deduce the five equations below.

$$\begin{aligned} I_7 &= I_C \\ I_4 &= I_1 + I_C \\ I_3 + I_x &= I_4 + I_5 \\ I_b &= I_5 + I_d \\ I_1 &= I_2 + I_3 \end{aligned}$$

By direct observation, we are also able to equate

$$\begin{aligned} V_1 &= V_a \\ V_c &= V_3 - V_6 \end{aligned}$$

Note that, since V_c is the voltage of the dependent voltage source, its value can be calculated by the subtraction of the voltages nodes 3 and 6, as stated in the second equation of the last system. To solve the seven equations and later on compute them into a matrix, we have to rewrite them using only V_i as variables with $i = 1, 2, 3, 4, 5, 6, 7$. It's important to point out that the current always flows from a higher potential to a lower one and, therefore, we can define the currents I_c, I_b and I_x as

$$\begin{aligned} I_c &= \frac{V_0 - V_7}{R_6} = -\frac{V_7}{R_6} \\ I_b &= K_b V_b = K_b(V_2 - V_3) \\ I_x &= I_7 - I_d = \frac{V_7 - V_6}{R_7} - I_d \end{aligned}$$

Logical substitutions lead us to the system of equations that is present below.

$$\begin{cases} V_1 = V_a \\ V_3 - V_6 = -\frac{K_c}{R_6} V_7 \\ \frac{V_7 - V_6}{R_7} = -\frac{V_7}{R_6} \\ \frac{V_3}{R_4} = \frac{V_1 - V_2}{R_1} - \frac{V_7}{R_6} \\ \frac{V_2 - V_3}{R_3} + \frac{V_7 - V_6}{R_7} - I_d = \frac{V_3}{R_4} + \frac{V_3 - V_5}{R_5} \\ K_b(V_2 - V_3) = \frac{V_3 - V_5}{R_5} + I_d \\ \frac{V_1 - V_2}{R_1} = \frac{V_2 - V_3}{R_3} + \frac{V_2 - V_4}{R_2} \end{cases}$$

Name	Value [A or V]
I_b	-0.000231
I_d	0.001040
I_1	0.000220
I_2	0.000231
I_3	-0.000011
I_4	0.001234
I_5	-0.001271
I_6	0.001014
I_7	0.001014
V_1	5.225668
V_2	4.994742
V_3	5.027716
V_4	4.530306
V_5	8.903022
V_6	-3.145874
V_7	-2.118235

Table 3: Nodal analysis results. All currents *I* expressed in Ampère; all voltages *V* expressed in Volt.

3 Simulation Analysis Using *NgSpice*

Table 4 shows the simulated operating point results for the circuit under analysis. It is possible to see that table 4 shows the current that goes through all the resistors as well as the voltage in each node.

However, as we can see there is an additional node when compared to the number of nodes that the circuit truly has. Since the circuit has a current-controlled voltage source, in order to be able to define this component in the *NgSpice* netlist, a current value that goes through another voltage source is required. However, in this circuit, this current value must be the same that goes through the resistor R_6. Therefore, an additional voltage source, V_{aux}, was added to the circuit between R_6 and R_7 with voltage equal to 0V, DC. Thus, this additional voltage source will have the same current that goes through R_6. A new node 8 that connects R_6 to V_{aux}, is created. Accordingly, node 7 will connect V_{aux} to R_7.

Name	Value [A or V]
@gb[i]	-2.30923e-04
@id[current]	1.040379e-03
@r1[i]	2.200941e-04
@r2[i]	2.309233e-04
@r3[i]	-1.08292e-05
@r4[i]	1.234188e-03
@r5[i]	-1.27130e-03
@r6[i]	1.014094e-03
@r7[i]	1.014094e-03
v(1)	5.225668e+00
v(2)	4.994742e+00
v(3)	5.027716e+00
v(4)	4.530306e+00
v(5)	8.903022e+00
v(6)	-3.14587e+00
v(7)	-2.11823e+00
v(8)	-2.11823e+00

Table 4: Operating point. Values obtained from the *NgSpice* script. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

The *Ngspice* netlist is done according to the following ”positive” and ”negative” nodes associated to the passive elements:

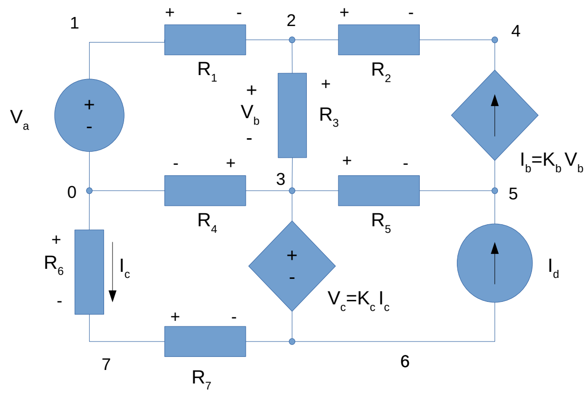

Figure 4: T1 Circuit with ”positive” and ”negative” nodes associated to the passive elements

4 Conclusion

Looking at the results shown in table 4 and comparing them to the theoretical analysis present in tables 2 and 3, it is clear that the *NgSpice* simulation results were equal to the values predicted by the methods applied. As the ”experimental” results are merely a simulation based on the same theoretical models used in this report, it is not surprising that this is the case. The circuit is comprised of only linear components (current and voltage sources and resistors), as such no symbolic calculations were required in a mathematical software like *Octave*. This contributes as well to the exact correspondence between simulation and theoretical results.

References

[1] GNU Octave, version 6.2.0 (February 20 2021)

[2] NgSpice, *open-source spice simulator*, version 31