Circuit Theory and Electronics Fundamentals

T1 Laboratory Report

Aerospace Engineering, Técnico, University of Lisbon

March 25, 2021

Group 19

João Silva, n° 95802
Miguel Cunha, n° 95832
Frederico Baptista, n° 97196

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing two independent and two linearly dependent sources. V_a is the independent voltage source and I_d is the independent current source.

The voltage controlled current source I_b is determined by $K_b \cdot V_b$, and the current controlled voltage source V_c is calculated with $K_c \cdot I_c$. The circuit can be seen in Figure 1, where it is found that along with these sources, the circuit is composed of seven more resistors, named R_1, R_2, ..., R_7.

The resistor values were obtained from a *Python* script, given by the professor and using the student number 95802 to generate the data. These values are shown in the table below, where names starting with R represent resistors, and the units are *kOhm* (kiloohm). K_b is given in *mS* (millisiemens), whereas K_c is expressed in *kOhm*. V_a and I_c are expressed in *V* (volt) and *mA* (milliampere), respectively.

Name	**Value**
R_1	1.01658203395
R_2	2.08071598482
R_3	3.12527703197
R_4	4.13615246449
R_5	3.04804224053
R_6	2.06609096892
R_7	1.01589064139
K_b	7.30340439475
K_c	8.13276803722
V_a	5.08211987776
I_c	1.02439571082

In Section 2, two theoretical analysis of the circuit are presented, using the mesh method and the node method. 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.

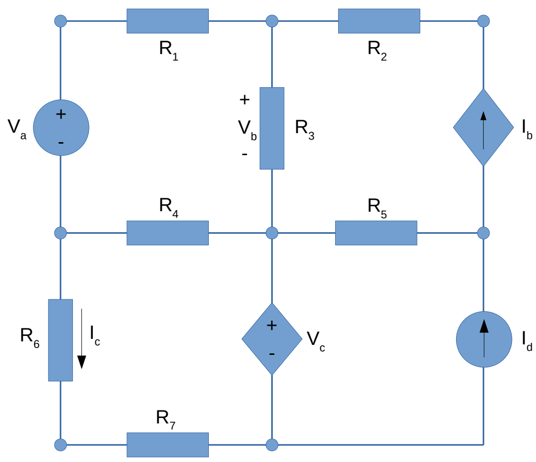

Figure 1: T1 circuit.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically using both the Mesh Method and Nodal Method.

2.1 Mesh Method

For the Mesh Method, circular currents are introduced in the different single meshes of the circuit as shown in 2 and then the circuit is evaluated considering those new currents.

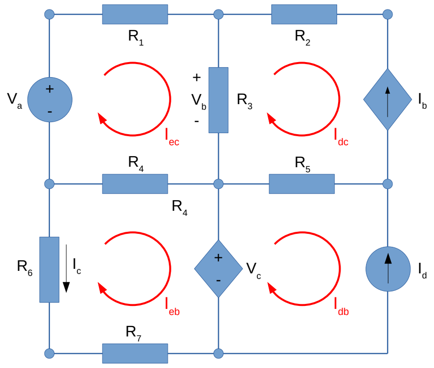

Figure 2: Mesh Analysis

After identifying all the mesh currents, considering the above, it is now necessary to use Kirchhoff Voltage Law (KVL) in those meshes that don't contain current sources (Mesh EC (**1**) and EB (**4**)) and relate the mesh currents to those imposed by the sources (Mesh DC (**2**) and DB (**3**).

$$-V_a + R_1 I_{ec} + V_b + R_4(I_{ec} - I_{eb}) = 0 \tag{1}$$

$$I_{dc} = -I_b \tag{2}$$

$$I_{db} = -I_d \tag{3}$$

$$V_c + R_7 I_{eb} + R_6 I_{eb} + R_4(I_{eb} - I_{ec}) = 0 \tag{4}$$

Analysing the equations it is noticeable that we have 8 variables and that we'll need four more equations to solve the circuit. It is, also, important to notice that two of them are already given:

$$V_c = K_c I_c \tag{5}$$

$$I_b = K_b V_b \tag{6}$$

The other two are trivial equations found by the careful examination of the circuit with Ohm's Law:

$$I_c = -I_{eb} \tag{7}$$

$$V_b = R_3(I_{ec} - I_{dc}) \tag{8}$$

In the following table, both current and voltage for each component are presented:

Name	Current (A)	Tension (V)
I_{EC}	2.143086e-04	-
I_{DC}	2.241280e-04	-
I_{DB}	-1.024396e-03	-
I_{EB}	-9.691453e-04	-
V_b	-	-3.068815e-02
V_c	-	7.881834e+00
I_b	-2.241280e-04	-
I_c	9.691453e-04	-

2.2 Nodal Metod

To determine the values of the current and voltage using the Nodal Method is first necessary to find all the knots in the circuit, as shown in 3.

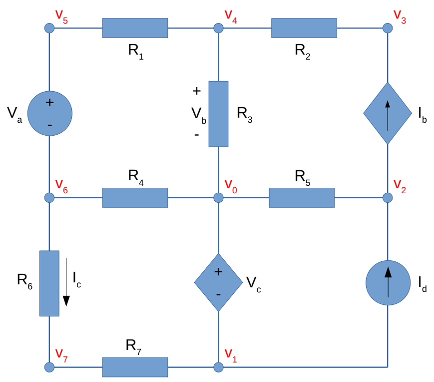

Figure 3: Node Analysis

It is then used the Kirchhoff Current Law (KCL) in the nodes that are not connected to voltage sources (from 9 to 12)

$$V_5G_1 - V_4(G_1 + G_2) + V_3G_2 - V_bG_3 = 0 \tag{9}$$

$$-V_3G_2 + V_4G_2 + I_b = 0 \tag{10}$$

$$V_2G_5 + V_0G_5 + I_d = I_b \tag{11}$$

$$G_6(V_6 - V_7) - I_c = 0 \tag{12}$$

In 13 and 14, knot's voltage relate to the voltage sources that are connected to them:

$$V_5 - V_6 = V_a \tag{13}$$

$$V_c = V_1 - V_0 \tag{14}$$

By analysing the circuit we get:

$$V_c = K_c I_c \tag{15}$$

$$I_b = K_b V_b \tag{16}$$

$$V_b = V_4 - V_0 \tag{17}$$

$$V_0 = 0 \tag{18}$$

Using Ohm's Law:

$$I_c = V_6 G_6 - V_7 G_6 \tag{19}$$

The continuity of current in voltage sources allows to write this equation:

$$V_5 G_1 - V_4 G_1 + I_c + V_0 G_4 - V_6 G_4 = 0 \tag{20}$$

In the following table, both current and voltage for each component are presented:

Name	**Current (A)**	**Tension (V)**
V_0	-	0.000000e+00
V_1	-	-7.881834e+00
V_2	-	3.805553e+00
V_3	-	-4.970348e-01
V_4	-	-3.068815e-02
V_5	-	1.871742e-01
V_6	-	-4.894946e+00
V_7	-	-6.897288e+00
V_b	-	-3.068815e-02
V_c	-	7.881834e+00
I_b	-2.241280e-04	-
I_c	9.691453e-04	-

3 Simulation Analysis

3.1 Operating Point Analysis

Table 1 shows the simulated operating point results for the circuit under analysis. It is important to note the creation of an auxiliary node, N8, as well as a fictional voltage source, V3 (providing 0V in order to not affect the rest of the circuit). This creation is due to no theoretical reason whatsoever and it's only necessary because of Ngspice software requirements for defining a current controlled voltage source.

Name	**Value [A or V]**
Id	1.02439571082e-3
@gb[i]	-2.24128e-04
@r1[i]	-2.14309e-04
@r2[i]	-2.24128e-04
@r3[i]	-9.81934e-06
@r4[i]	1.183454e-03
@r5[i]	1.248524e-03
@r6[i]	-9.69145e-04
@r7[i]	-9.69145e-04
n0	4.894946e+00
n1	-2.98689e+00
n2	8.700499e+00
n3	4.397911e+00
n4	4.864258e+00
n5	5.082120e+00
n7	-2.00234e+00
n8	-2.00234e+00

Table 1: Operating point

Note: A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

As it can be seen, the simulation results coincide up to the last digit with the theoretical results.

4 Conclusion

In this laboratory assignment the objective of analysing this circuit has been achieved. The analysis of the linearly dependent sources (one current controlled voltage source and one voltage controlled current source) have been performed both theoretically using the Octave maths tool and by circuit simulation using the Ngspice tool. The simulation results matched the theoretical results precisely. The reason for this perfect match is the fact that this is a straightforward circuit containing only linear components, so the theoretical and simulation models cannot differ. For more complex components, the theoretical and simulation models could differ but this is not the case in this work. This fact can easily be justified due to the nature of this simulation, since Ngspice uses the exact same methods as those that were used in the theoretical calculations to solve the circuit, therefore it was expected to output the same results as those calculated before.

Moreover, since Ngspice provides a simulation, all the resistors are perfect, and so are the branches, nodes and 'wires', with no electrical resistance and no energy wasted on heating. Furthermore, since there are no real measurements, it is found that no errors emerge from inaccurate readings, and hence the theoretical results being precisely the same as the simulated ones.