TÉCNICO LISBOA

Circuit Theory and Electronics Fundamentals

Masters of Aeroespace Engineer, Técnico, University of Lisbon

Laboratory Report

Group 37

Afonso Magalhães, nº95765
Fábio Monteiro, nº95786
Leonardo Encarnação, nº95816

March 25, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing dependent and independent sources of both voltage and current alonside various resistors (2 dependent sources, 2 independent souces and 7 resistors to be more precise). To do so we've obtained the current in the various meshes and the voltage in all the nodes, using two diferent methods alongside a simlulation to corroborate the results.

The circuit that will be analysed is represented in the image below and the values of each constant are especifed in the table below.

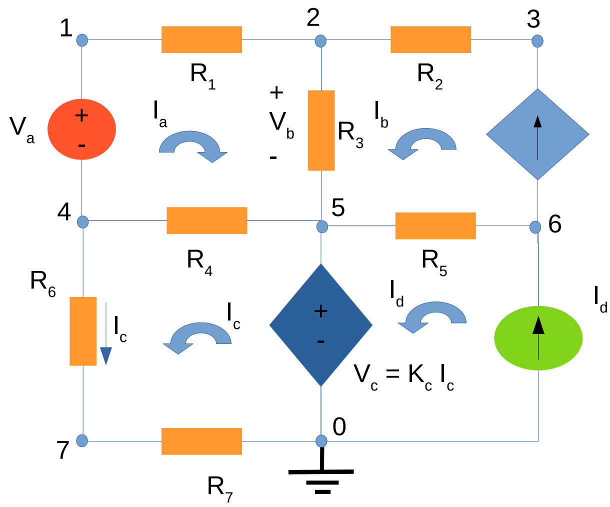

Figure 1: Circuit

R1	1.013609e+00 kOhm
R2	2.016578e+00 kOhm
R3	3.006816e+00 kOhm
R4	4.049229e+00 kOhm
R5	3.053925e+00 kOhm
R6	2.092502e+00 kOhm
R7	1.022320e+00 kOhm
Id	1.029587e+00 mA
Kb	7.213324e+00 mA/V
Kc	8.321035e+00 mA/V

Table 1: Initial info

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.

2 Theoretical Analysis

In this section, the circuit shown in the introduction is analysed theoretically, in terms of its current and voltage. Analyzing a circuit using Kirchhoff's circuit laws, one can either use the nodal analysis method, a method derivated of Kirchhoff's current law (KCL) or the mesh analysis method, a method derivated of Kirchhoff's voltage law (KVL). In the subsections bellow we will explain how the two methods are used in order to solve the circuit.

2.1 Nodal analysis

The nodal analysis or the branch current method is a method of determining the voltage in each node (points where elements or branches connect) in an electrical circuit in terms of the branch currents, using one of the nodes as reference (in this case the node 0). Nodal analysis writes an equation at each electrical node, requiring that the branch currents incident at a node must sum to zero. For example, in the node 2 the current from the branch 1-2, from the branch 2-3, and the branch 2-5 must sum zero. however the current in each branch is the potencial difference of the two nodes over the resistence. considering this and simplify this equation, we've obtained a equation for the voltages in the nodes 1, 2, 3 and 5. To make this easier, instead of using the resistences , we use their inverse, the condutivities. Since there are 8 nodes in total in this circuit we must have 8 equations in order to find all the 8 voltages. After finding all these 8 equations we obtain the following matrix:

$$\begin{pmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & -1 & 0 & 0 & 0 \\ 0 & -G1 & G1+G2+G3 & -G2 & 0 & -G3 & 0 & 0 \\ 0 & 0 & -G2-Kb & G2 & 0 & Kb & 0 & 0 \\ 0 & G1 & -G1 & 0 & G4+G6 & -G4 & 0 & -G6 \\ 0 & 0 & 0 & 0 & -Kc*G6 & 1 & 0 & Kc*G6 \\ -1 & 0 & Kb & 0 & 0 & -G5-Kb & G5 & 0 \\ -G7 & 0 & 0 & 0 & -G6 & 0 & 0 & G6+G7 \end{pmatrix} \begin{pmatrix} V0 \\ V1 \\ V2 \\ V3 \\ V4 \\ V5 \\ V6 \\ V7 \end{pmatrix} = \begin{pmatrix} 0 \\ Va \\ 0 \\ 0 \\ Id \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

The following table displays the various solutions to the various voltages :

Voltage (V)	
V1	8.194795e+00
V2	7.917828e+00
V3	7.340169e+00
V4	2.978754e+00
V5	7.957540e+00
V7	9.776608e-01
V8	0.000000e+00

Table 2: Octave nodal analysis results

2.2 Mesh analysis

Mesh analysis is a method that is used to solve the currents at any place in the electrical circuit. This analysis makes use of Kirchhoff's voltage law to ensure that the sum of the potencial difference in every circuit loop is zero. To do this we find the loops that don't contain other loops, or meshes, and within these meshes we define a single current for each one. After these we just need to apply the Kirchhoff's voltage law for every mesh and using these currents (circulation currents).

In this case, we use four equations in order to find the four circulation currents, since we have four meshes in this circuit. After finding the four equations we find the following matrix:

$$\begin{pmatrix} R1+R3+R4 & R3 & R4 & 0 \\ Kb*R3 & Kb*R3-1 & 0 & 0 \\ R4 & 0 & R4+R6+R7-Kc & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix} \begin{pmatrix} Ia \\ Ib \\ Ic \\ Id \end{pmatrix} = \begin{pmatrix} Va \\ 0 \\ 0 \\ Id \end{pmatrix}$$

The following table displays the various solutions to the various currents:

Current (mA)	
Ia	2.732478e-01
Ib	-2.864551e-01
Ic	9.563162e-01
Id	1.029587e+00

Table 3: Octave mesh analysis results

3 Simulation Analysis

In order to run the simulation, we wrote the ngspice code according to the image below. It is important to note that an extra voltage source, Vaux, was added and therefore, another node was also added (node 8). This Vaux was intended to allow the measurement of the current Ic which voltage source Vc depends on, since ngspice doesn't allow us to introduce Resistor R6's current in the computation. Vaux's voltage is equal to 0 V, since it is only an auxiliary component that doesn't interfere with the circuit (node's 8 voltage is equal to node's 7 voltage) and allowed us to obtain the current through it.

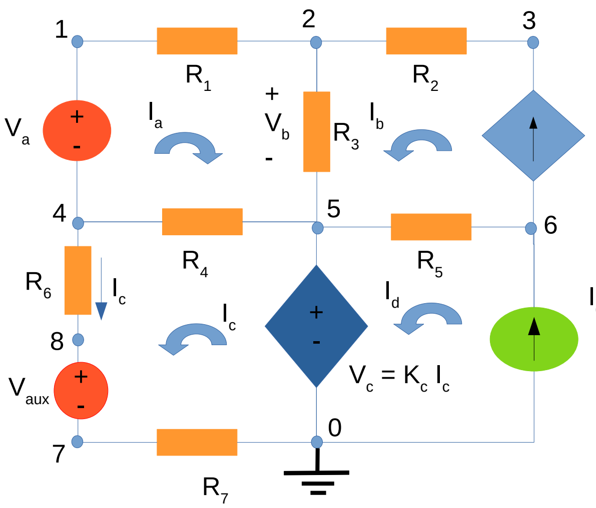

Figure 2: Circuit

The results obtained from the simulation are shown in the table below.

Name	Value [A or V]	Name	Value [A or V]
@Ib	-2.86455e-04	V1	8.194795
@Id	1.029587e-03	V2	7.917828
@R1[i]	2.732478e-04	V3	7.340169
@R2[i]	-2.86455e-04	V4	2.978754
@R3[i]	-1.32073e-05	V5	7.957540
@R4[i]	1.229564e-03	V6	1.197664e+01
@R5[i]	1.316042e-03	V7	9.776608e-01
@R6[i]	9.563162e-04	V8	9.776608e-01
@R7[i]	9.563162e-04	Vc	7.327105e-05
Vaux	9.563162e-04	Va	-2.73248e-04

Table 4: NgSpice simulation results

Then, an analysis of the values obtained was conducted, in order to ascertain the compatibility with the expected values from the theorethical analysis. This result validation was achieved by calculating the relative errors between the theoretical values obtained in octave and the experimental values obtained in ngspice.

Relative Errors (%)	
V1	0
V2	0
V3	0
V4	0
V5	0
V6	0
V7	0
IA	0
IB	3.49095e-05
IC	0

Table 5: Relative Errors between Octave and NgSpice results

After the analysis of these errors, it is possible to infer that the accuracy is extremely high. The maximum relative error is 3.49095e-05, which is extremely low. ”This error is associated to the dissipated power in the resistors”.This error is most likely due to rounding up of the ngspice, since the decimal digits obtained using ngspice and in octave are different. Based on this, it is possible to conclude that the simulation results are validated.

4 Conclusion

In this laboratory assignment the objective of analysing the circuit especified in the introduction has been achieved. All analyses 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.