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 of voltage and current sources alonside various resistors. To do so we've obtained the current in the various meshes and the voltage in all the nodes.

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.

The following table displays the info that was already given to us:

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

2 Theoretical Analysis

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

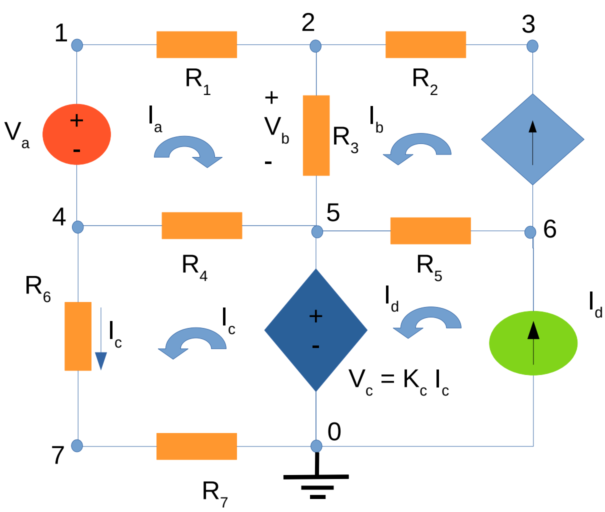

Figure 1: Circuit

2.1 Nodal analysis

The nodal analysis or the branch current method is a method of determining the voltage (potential difference) between ”nodes” (points where elements or branches connect) in an electrical circuit in terms of the branch currents. Nodal analysis writes an equation at each electrical node, requiring that the branch currents incident at a node must sum to zero. Since there are 8 nodes in total in this circuit we must have 8 equations in order to find all the 8 voltages.

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

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 circuits for the currents at any place in the electrical circuit.
This analysis makes use of Kirchhoff’s voltage law to arrive at a set of equations guaranteed to be solvable if the circuit has a solution.
In this case, we use four equations in order to find the four circulation currents, since we have four elementar meshes in this circuit.

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

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 7). This Vaux was intended to allow the measurement of the current Ic which voltage source Vc depends on, because ngspice doesn't measure the current between two nodes, only in resistors and independent voltage sources. Vaux's voltage is equal to 0 V, as expected by its name, 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).

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

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 4: 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. Based on this, 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.