Circuit Theory and Electronics Fundamentals

Integrated Masters in Aerospace Engennering, Técnico, University of Lisbon

Laboratory Report 1- Group 28

Beatriz Pedroso 95773, Teresa Gonçalves 95826, Tiago Escalda 95851

March 24th, 2021

Contents

1 Introduction

The aim of this laboratory work regarding the topics studied in the first three weeks of the course was to analyse a circuit constituted by an independent voltage source, an independent current source, a voltage controlled dependent current source, a current controlled dependent voltage source and seven resistors, as shown in the Figure 1 below. For this, a theorical analysis was made using both node and mesh methods, whose results will be discussed in Section 2. To validate these results, a simulation was conducted, as will appear in Section 3.

The results were then compared and the conclusions of the group summarized in Section 4.

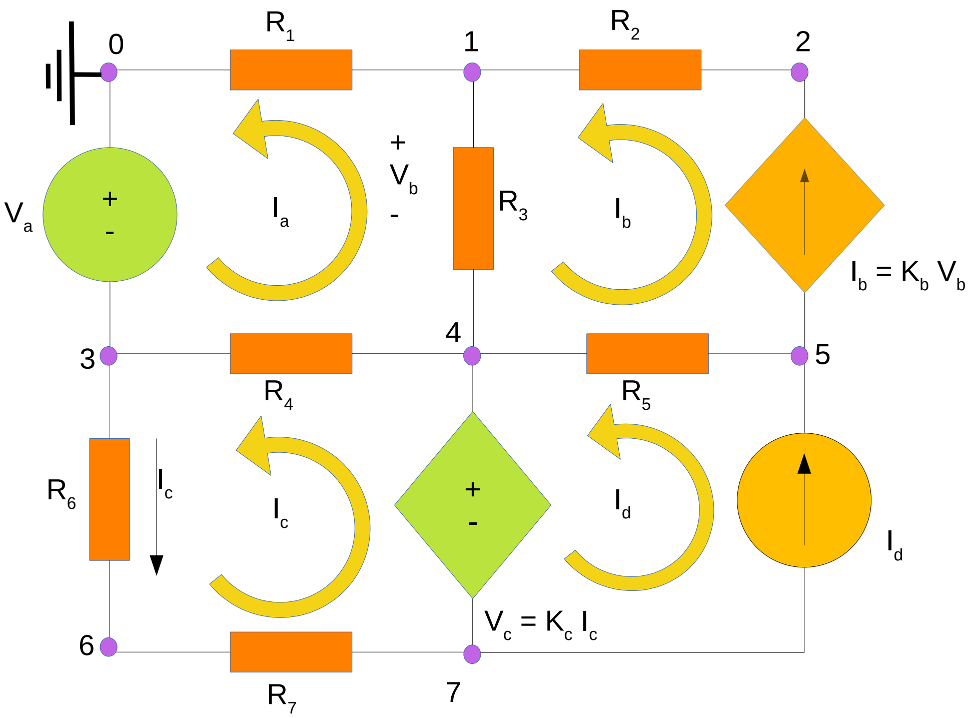

Figure 1: Circuit analysed.

The initial data was generated randomly by datagen.py as it is presented in the table below.

Generated Data										
Resistors			Voltages			Currents			Constants	
R1	1.04111259479		Va	5.06871572779		Id	1.04127523824		Kb	7.28747116393
R2	2.09945227782								Kc	8.11568444746
R3	3.13109125645									
R4	4.11947040212									
R5	3.1155879392									
R6	2.04799381798									
R7	1.02754401839									

Table 1: Units for the values: V, mA, KOhm and mS

2 Theoretical Analysis

In this section, a theoretical analysis of the circuit was conducted. Two approaches were chosen: the mesh and the node methods.

2.1 Mesh Method

The currents IA and IC were determined by examining the loop formed by R1, R3, R4 and Va and the loop formed by R4, R6, R7 and Vc, respectively. The third independent equation was obtained by matching IB to Kb*Vb (Vb=R3*(IB-IA)). These were then rearrenged in a matrix form as shown below. Octave math tools were used to solve the system.

$$\begin{bmatrix} R1+R3+R4 & -R3 & -R4 \\ -R4 & 0 & -Kc+R4+R6+R7 \\ -Kb*R3 & Kb*R3-1 & 0 \end{bmatrix} \begin{bmatrix} IA \\ IB \\ IC \end{bmatrix} = \begin{bmatrix} -Va \\ 0 \\ 0 \end{bmatrix}$$

Name	Value [A]
Ia	-2.161572e-04
Ib	-2.260646e-04
Ic	9.671728e-04

Table 2: Octave Mesh Method Results. All variables are of type *current* and expressed in Ampere.

2.2 Node Method

The aim of using this method is to determine every node voltage. To do so, a reference node(with voltage =0V) was chosen. Then, seven independent equations were written in orther to find the remaining unknown node voltage values. The equations were then put in the form of the matrix shown below. Octave math tools were used to solve the system.

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ G1 & G1-G2-G3 & G2 & 0 & G3 & 0 & 0 & 0 \\ 0 & G2+Kb & -G2 & 0 & -Kb & 0 & 0 & 0 \\ 1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\ 0 & G1 & 0 & -G4-G6 & G4 & 0 & G6 & 0 \\ 0 & -Kb & 0 & 0 & G5+Kb & -G5 & 0 & 0 \\ 0 & 0 & 0 & G6 & 0 & 0 & -G6-G7 & G7 \\ 0 & 0 & 0 & -KcG6 & -1 & 0 & Kc*G6 & -1 \end{bmatrix} \begin{bmatrix} V0 \\ V1 \\ V2 \\ V3 \\ V4 \\ V5 \\ V6 \\ V7 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ Va \\ 0 \\ -Id \\ 0 \\ 0 \end{bmatrix}$$

Name	Value [V]
V0	0.000000e+00
V1	-2.250439e-01
V2	-6.996557e-01
V3	-5.068716e+00
V4	-1.940229e-01
V5	3.754486e+00
V6	-7.049480e+00
V7	-8.043292e+00

Table 3: Octave Node Method Results. All variables are of type *voltage* and expressed in Volt.

With these results, we are able to compare both methods. Calculating $(V1 - V0)/R1$ with the voltages of the node method we get Ia. Repeating the same process for V2, V1 and R2 we obtain Ib and for V3, V6 and R6 we get Ic. The calculations lead us to the following table:

Name	Value [A]
Ia	-2.161572e-04
Ib	-2.260646e-04
Ic	9.671728e-04

Table 4: Current Results . All variables are of type *current* and expressed in Ampere.

As expected, the results of node method and mesh method are equal. We can infer both analysis are correct.

3 Simulation Analysis

3.1 Operating Point Analysis

First of all, to contextualize the values obtained using the tools in ngspice, it is necessary to state that, as node 0 is connected to ground, its nodal voltage does not appear on the table of results. Furthermore, to be able to describe the voltage flowing in the dependent source, it is necessary to know the current in resistor 6. However, ngspice is not able to compute this value when the depedent source is described. So, in order to do that, an extra dependent voltage source (whose voltage drop is equal to 0 V) was created, and put in series after the resistor 6. This led to the appearence of node 8, that has the same voltage drop as node 6. So, by doing that, ngspice is able to determine the current in this auxiliar independent source, which is exactly the value needed. The circuit with these changes is shown in the drawing below.

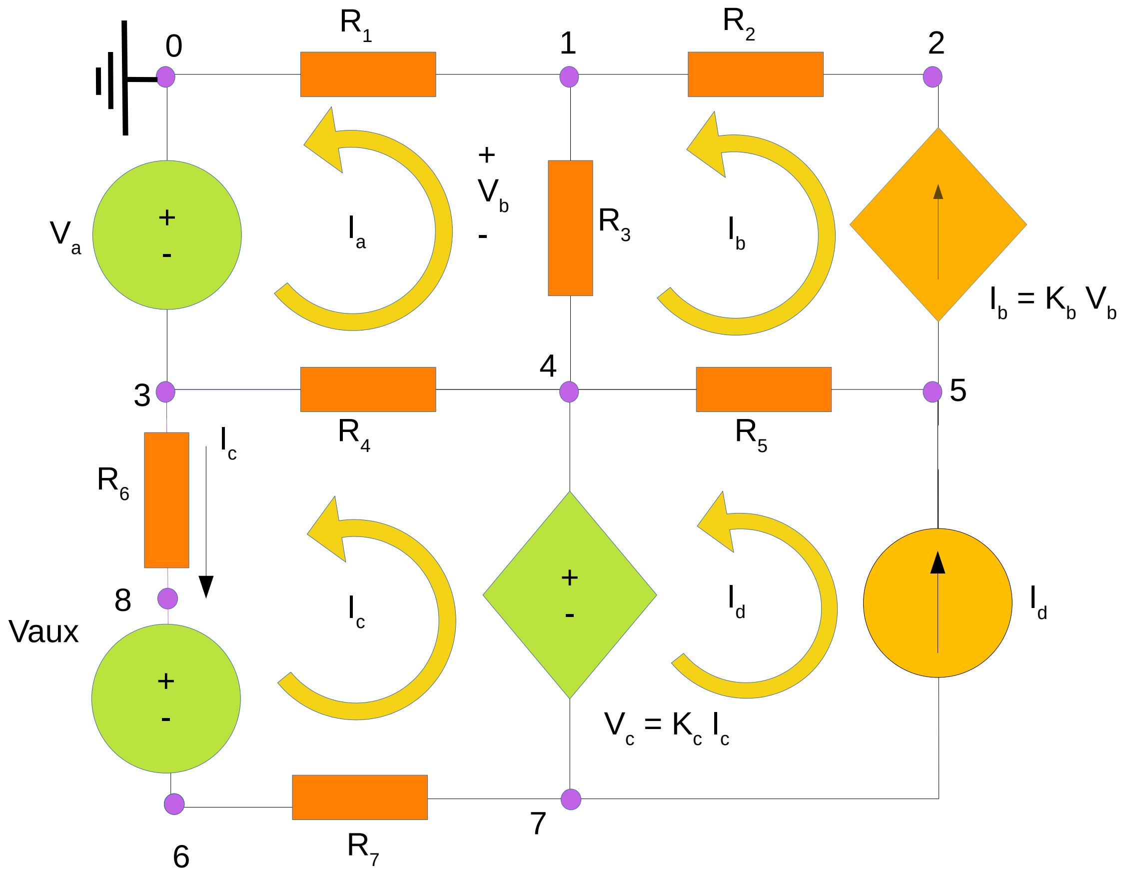

Figure 2: Circuit analysed in ngspice.

After running the simulation, the results were put in the table below. Then, a careful analysis of the aforementioned table was conducted.It shows the simulated operating point results for the circuit that is being studied, allowing the group to obtain the current flowing in every risistor, the voltage in the dependent voltage source and even the current flowing in the dependent currrent source.

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

In order to validate the results obtained in NGSPICE, relative errors between the theoretical values, obtained in octave and the ones obtained in ngspice, were calculated. These were put in the table below.

Name	Value [A or V]
@gb[i]	-2.26065e-04
@id[current]	1.041275e-03
@r1[i]	-2.16157e-04
@r2[i]	-2.26065e-04
@r3[i]	9.907405e-06
@r4[i]	-1.18333e-03
@r5[i]	-1.26734e-03
@r6[i]	9.671728e-04
@r7[i]	9.671728e-04
v(1)	-2.25044e-01
v(2)	-6.99656e-01
v(3)	-5.06872e+00
v(4)	-1.94023e-01
v(5)	3.754486e+00
v(6)	-7.04948e+00
v(7)	-8.04329e+00
v(8)	-7.04948e+00

Table 5: NgSpice Results

Relative Errors (%)	
V1	4.444e-05
V2	4.288e-05
V3	7.892e-05
V4	5.154e-05
V5	0
V6	0
V7	2.487e-05
IA	9.253e-05
IB	1.769e-05
IC	0

Table 6: Relative Errors between Octave and NgSpice results

After the anaylysis of these errors, we conclude that the accuracy is extremely high supported by the fact that the relative maximum error is 9.253e-05%. Nevertheless, we assume that these errors are due to dissipated power in the resistors. Even so, the simulation results can be validated.

4 Conclusion

It was agreed by the members of the group that the main goal of the task proposed was achieved. As presented, both theoretical and simulation results(obtained using Octave tools and ngpsice simulator, respectively) matched, reaching total accuracy. Despite the initial belief that the considerable number of components of the circuit could cause some disparity in the results, such did not happened. This proves not only the efficency of both mesh and node methods to analyse the circuit, as well as the simulator used.