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 **3**

2 Theoretical Analysis **4**
2.1 Node Method 4
2.2 Mesh Method 4

3 Simulation Analysis **5**
3.1 Operating Point Analysis 5

4 Conclusion **7**

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 of an independent voltage source, an independent voltage source, a voltage controlled dependent current source, a current controlled dependent voltage source and seven resistors, as shown in the Figure t1draw below. . For this, a theorical analysis was made using both node and mesh methods, whose results will be discussed in section one. To validate these results, a simulation was conducted, as will apeear in section 2.

The forementioned analysis was divided into a theoretical one, presented in section .In order to be able to validate the results obtained, a simulation was also conducted, as shown in Section . The results were then compared , and the conclusions of the group summarized in Section

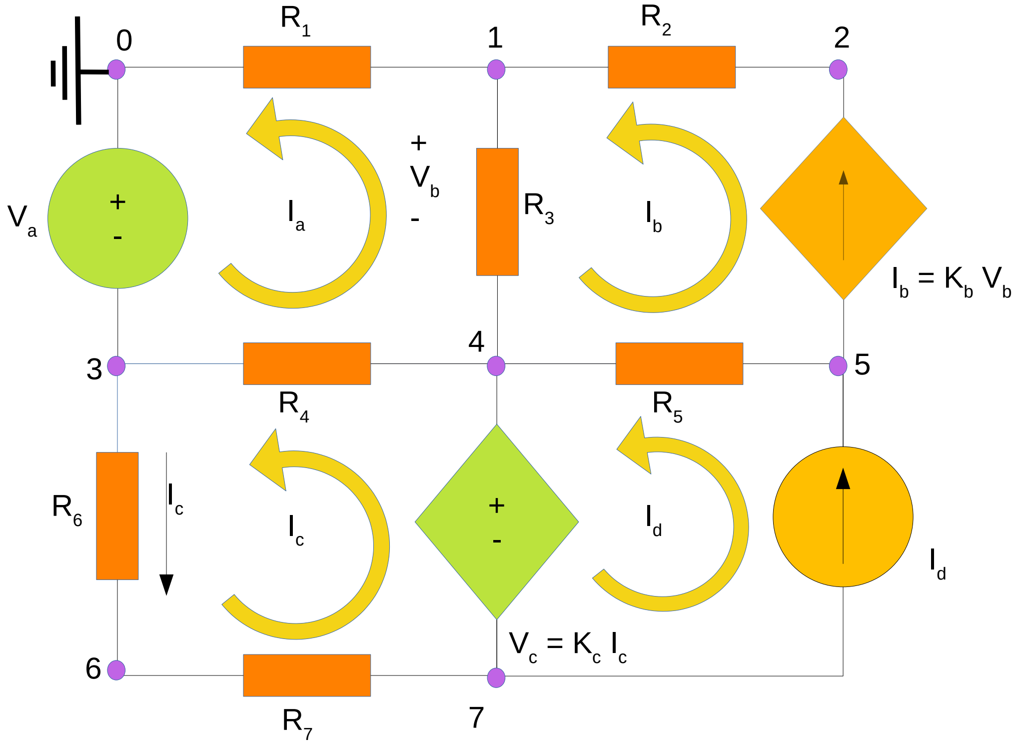

Figure 1: Circuit analysed.

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 Node Method

The aim of using this method to analyse the circuit 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 matriz shown below. Octave math tools were used to solve the seven equations.

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

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

2.2 Mesh Method

The mesh analysis was the chosen method to determine the currents IA, IB and IC. This was achieved by examining the loop formed by R1, R3, R4 and Va and the loop constituted by R4, R6, R7 and Vc, in which the circulating currents are IA and IC, respectively. The third independent equation was obtained equaling IB to Kb*Vb. The equations were then rearrenged in a matrix form as shown below. Octave math tools were used to solve the three equations.

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

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

Table 2: NgSpice Results . A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

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, an extra voltage source (whose voltage is equal to 0 V) was created, and put after the resistor 6 and node 6. This source led to the appearence of node 8, that has the same voltage of node 6. This was necessary because ngspice does not measure the current between two nodes, only in resistors and in independent voltage sources. The circuit with these changes is shown in the drawing below.

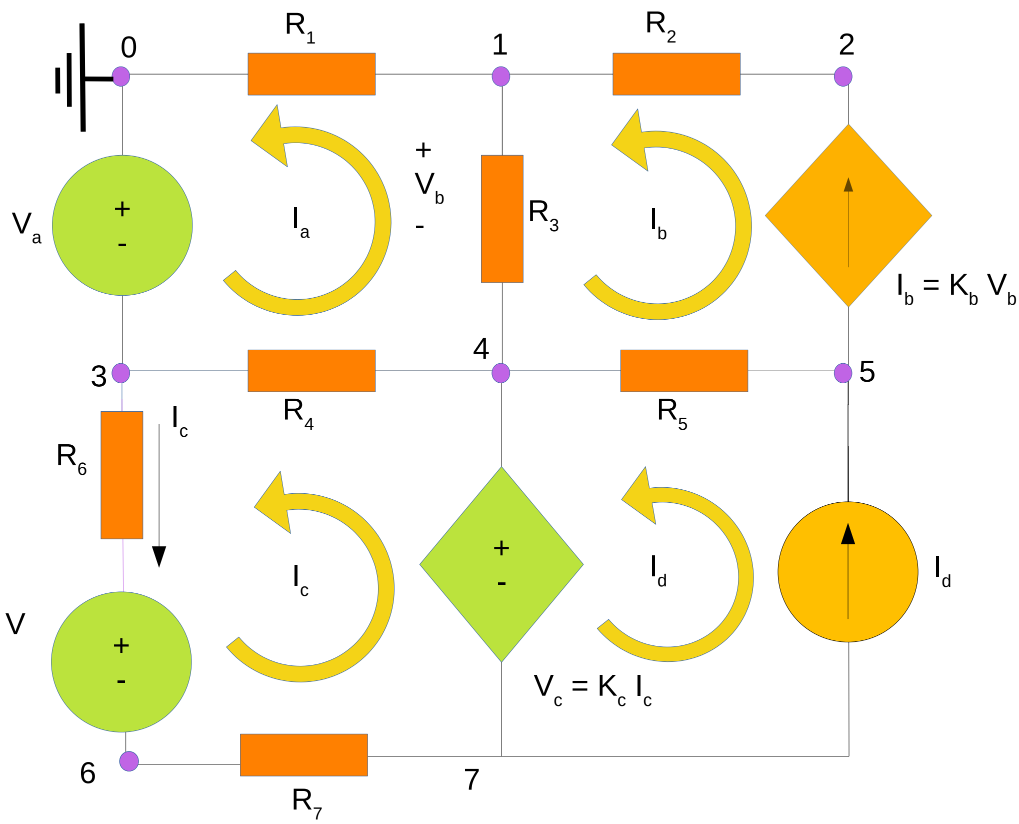

Figure 2: Circuit analysed in ngspice.

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.

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 3: NgSpice Results

After comparing the results obtained in ngspice with the obtained using the octave math tools, analysed in Section 2, we conclude that every result, Ia,Ib and Ic match.

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, reatching 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.