

Circuit Theory and Electronics Fundamentals

Integrated Masters in Aerospace Engineering, 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 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 ?? below. . For this, a theoretical 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 appear 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 (Section ??), and the conclusions of the group summarized in Section ??.

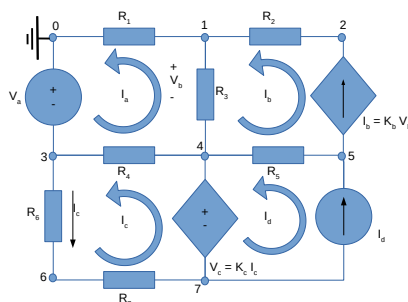


Figure 1: Voltage driven serial circuit.

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 order 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 seven equations.

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

$$C = \begin{bmatrix} 0 \\ 0 \\ 0 \\ Va \\ 0 \\ -Id \\ 0 \\ 0 \end{bmatrix}$$

3 Simulation Analysis

3.1 Operating Point Analysis

After careful evaluation of the Table ?? below, which shows the simulated operating point results for the circuit that is being studied. This simulation allowed the group to obtain the current flowing in every resistor, the voltage in the dependent voltage source and even the current flowing in the dependent current source.

After comparing the results obtained in ngspice with the obtained using the octave math tools, we conclude that every result 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 ngspice 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 happen. This proves not only the efficiency of both mesh and node methods to analyse the circuit, as well as the simulator used.

Name	Value [A or V]
@gb[i]	-1.99401e-03
@id[current]	1.041275e-03
@r1[i]	-1.90662e-03
@r2[i]	-1.99401e-03
@r3[i]	8.738847e-05
@r4[i]	-8.14991e-04
@r5[i]	-3.03528e-03
@r6[i]	-1.09163e-03
@r7[i]	-1.09163e-03
v(1)	-1.98500e+00
v(2)	-6.17133e+00
v(3)	-5.06872e+00
v(4)	-1.71138e+00
v(5)	7.745306e+00
v(6)	-2.83307e+00
v(7)	-1.71137e+00
v(8)	-2.83307e+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.