Circuit Theory and Electronics Fundamentals

Department of Electrical and Computer Engineering, Técnico, University of Lisbon

Laboratory One Report

Francisco Batista, 95792.
Miguel Jesus, 95833.
Pedro Monteiro, 96465.

March 22, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study the circuit that can be seen if Figure 1.

The circuit is composed by eleven components. This include two voltage sources (one of which is independent and the other current-controlled), two current sources (one independent and another voltage-controlled) and seven resistors. Furthermore, the circuit contains thirteen meshes (of which four of them are essential) and eight 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.

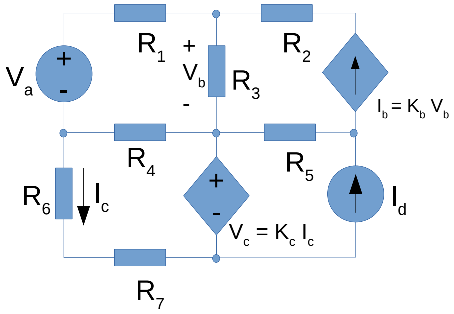

Figure 1: Studied circuit.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, in terms of its nodes' voltages and meshes' equations. In order to do so, we first had to number and identify all the nodes of the given circuit. In Figure 2 we can see the distribution of such nodes.

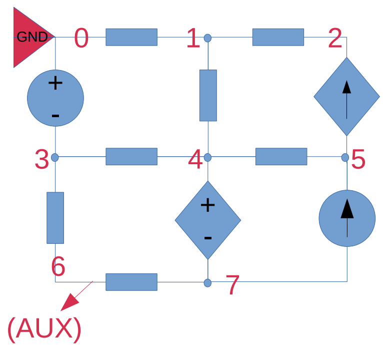

Figure 2: Circuit Node Identification.

This analysis was performed using the Mesh Method and the Node Method. In the Mesh Analysis we used the Kirchoff Voltage Law (KVL) and an additional equation to solve for the currents in the essential meshes. These currents' directions are represented in Figure 3.

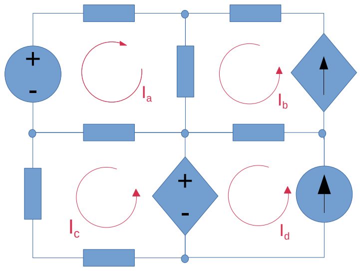

Figure 3: Essential meshes' current direction.

On the other hand, the Node Analysis was done with the Kirchoff Current Law (KCL) applied to the nodes that weren't connected to voltage sources. A supernode was also considered on this analysis in order to simplify the calculation. A supernode was defined by combining the nodes 4 and 7 as these are connected to a voltage source. The supernode equation is found on (5). The system of equations obtained in each method are presented in the following expressions.

$$V1(-\frac{1}{R1}-\frac{1}{R2}-\frac{1}{R3})+V2\frac{1}{R2}+V4\frac{1}{R2}=0 \tag{1}$$

$$V1(Kb+\frac{1}{R2})-V2\frac{1}{R2}-V4Kb=0 \tag{2}$$

$$V3=-Va \tag{3}$$

$$V1\frac{1}{R3}+V3\frac{1}{R4}+V4(-\frac{1}{R3}-\frac{1}{R4}-\frac{1}{R5})+V5\frac{1}{R5}+V6\frac{1}{R7}-V7\frac{1}{R7}=Id \tag{4}$$

$$V1Kb-V1(\frac{1}{R5}-Kb)+V5\frac{1}{R5}=Id \tag{5}$$

$$V3\frac{1}{R6}+V5(-\frac{1}{R6}-\frac{1}{R7})+V7\frac{1}{R7}=0 \tag{6}$$

$$V3\frac{Kc}{R6}-V4-V6\frac{Kc}{R6}+V7=0 \tag{7}$$

$$Ia(R1+R2+R3)+IbR3+IcR4=Va \tag{8}$$

$$IaR4+Ic(R4+R6+R7-Kc)=0 \tag{9}$$

$$IaR3+Ib(-\frac{1}{Kb}+R3)=0 \tag{10}$$

After obtaining the system of equations, the Octave tool was used to run all the necessary calculation so as to obtain the theoretical results that are displayed in Table 1 and Table 2.

Nodes	Value [V]
v1	-0.202617
v2	-0.619394
v3	-5.201027
v4	-0.174256
v5	3.485906
v6	-7.293091
v7	-8.322033

Table 1: Theoretical voltage values for each node, expressed in Volt.

Meshes	Value [A]
Ia	0.000212
Ib	-0.000222
Ic	0.001098

Table 2: Theoretical current values for each node, expressed in Ampere.

3 Simulation Analysis

3.1 Operating Point Analysis

Following our theoretical analysis, the circuit was simulated in the Ngspice software, which returned the values of the voltage for each node. These values, that are shown in Table 3, allowed us to easily calculate the electrical currents. The inputs given to the Ngspice were the values of the resistors, as well as the nodes to which they are connected (the positive being followed by the negative), and their values. Moreover, the independent current and voltage source, as well as the values for Kb and Kc, which are fundamental to the controlled sources, are also introduced. In order for the current that commands the controlled voltage to flow from the positive to the negative node, the creation of a control, ficticious node was necessary to sense this current. When compared, the results of the theoretical analysis and the simulated ones are approximately the same, with tiny differences that might be owed to factors such as Octave rounding errors in the matrix equations.

Name	Value [A or V]
@Id[i]	1.0311e-3
@gi[i]	-2.03901e-04
@i[current]	1.031098e-03
@r1[i]	-1.94985e-04
@r2[i]	-2.03901e-04
@r3[i]	-8.91660e-06
@r4[i]	-1.20321e-03
@r5[i]	-1.23500e-03
@r6[i]	1.008221e-03
@r7[i]	1.008221e-03
v(1)	-2.04563e-01
v(2)	-6.23996e-01
v(3)	-5.13416e+00
v(4)	-1.76513e-01
v(5)	3.643090e+00
v(6)	-7.22987e+00
v(7)	-8.28569e+00
aux	-7.22987e+00

Table 3: Operating point. Simulated Values for voltage (V) and current (A) using Ngspice.

4 Conclusion

To sum up, in this assignment, the main goal was successfully achieved. This objective was to analyze circuits with linear components such as resistors and controlled sources using two different methods.

The proximity verified between the theoretical and simulated values in this laboratory project can be explained by the simplicity and linearity of the system, which rendered the theoretical calculation a linear set of equations that were quickly solved by the Octave software.

Nevertheless, the small discrepancies between both sets of values are due to approximations of the given data (resistance, voltage values) by the Octave and the Ngspice, as these software have a different level of digit precision.