Circuit Theory and Electronics Fundamentals

Mestrado em Engenharia Física Tecnológica, Técnico, University of Lisbon

Ana Martins (96506), Miguel Moreira (96556), Sara Oliveira (96566)

Analysis of the Circuit Given for Laboratory T1

March 22, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing two independent sources, V_a and I_d, one voltage-controlled source, I_b, one current-controlled source, V_c, connected to seven resistors. The circuit can be seen in Figure **??**.

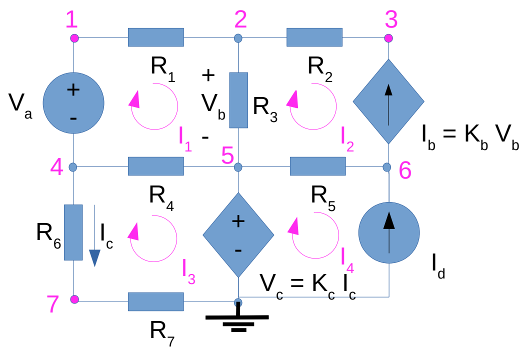

Figure 1: This is circuit that was analysed. The nodes were numbered, as were the currents in each mesh.

In Section **??**, a theoretical analysis of the circuit is presented. In Section **??**, the circuit is analysed by simulation, and the results are compared to the theoretical results obtained in Section **??**. The conclusions of this study are outlined in Section **??**.

2 Theoretical Analysis

In this section, the circuit shown in Figure **??** is analysed via node analysis and mesh analysis.

2.1 Node analysis

Labels were assigned to identify nodes from zero to seven, to then proceed with the mesh and node analysis (the 0th of which being the ground). This is helpful because we can, therefore, derive direct equations in terms of the voltage at these nodes.

The voltage V_i refers to the node i. For this procedure, one applies KCL (Kirchoff's Current Law), which states that the sum of currents leaving a node must be the same as the sum of currents entering a node. It's relevant to notice that this kind of approach is only possible for the nodes which aren't directly connected to a terminal of a voltage source, therefore, we can only derive 4 of these equations: in that case, it's crucial to find other equations to cover all the unknown variables of the system. The approach is then to consider the voltage gain between to nodes that have a voltage source between them: for example, the nodes 5 and 0 are connected to V_c, which means $V_5 = V_c + V_0$, which wields us the 7^{th} equation in **??**. Moreover, we can relate the currents associated with the controlled sources with the nodes we numerated: for example, in I_c there is a voltage drop, which gives us the 6^{th} equation. The current flow direction is considered, whenever possible, the same as indicated by each I_i, with i ranging from 1 to 4, as shown in **??**. The cases where this is not possible are the currents going through nodes 2 to 5 and 5 to 6; for these, we considered, respectively, the direction of V_b and of I_d.

The equations are as follows

$$\begin{cases} Node\,2: \frac{V_1-V_2}{R_1} + \frac{V_2-V_5}{R_3} + \frac{V_2-V_3}{R_2} = 0 \\ Node\,3: \frac{V_2-V_3}{R_2} + I_b = 0 \\ Node\,6: \frac{V_6-V_5}{R_5} - I_b = \ -I_d \\ Node\,7: \frac{V_7-V_4}{R_6} - \frac{V_7}{R_7} = 0 \\ \frac{V_2-V_5}{R_3} - I_b = 0 \\ \frac{V_4-V_7}{R_6} - I_c = 0 \\ V_5 - V_c = 0 \end{cases} \tag{1}$$

2.2 Mesh analysis

In the mesh analysis, every mesh is given an arbitrary current (I_1, I_2, I_3, I_4), represented in Figure **??** with round arrows. This is helpful because we can, therefore, derive direct equations in terms of the current passing through these meshes.

The current I_i refers to the mesh i. For this method, we apply KVL (Kirchoff's Voltage Law), which states that the sum of all the voltages around any closed loop in a circuit is equal to zero, and relate the fictional currents we created to currents given in the circuit (for example, I_d).

This can't be done for every mesh as the Law would specifically entail, but we can reach conclusions about every one of them by inspection of said mesh: for example, the 2^{nd} mesh is connected to a current source, which automatically wields the 2^{nd} equation.

The equations are as follows

$$\begin{cases} Mesh\,1: R_1\,I_1 + R_3\,(I_1 - I_2) + R_4(I_1 - I_3) = V_a \\ Mesh\,2: I_2 + I_b = 0 \\ Mesh\,3: R_4(I_3 - I_1) + V_c + R_7\,I_3 + R_6\,I_3 = 0 \\ Mesh\,4: I_4 = -I_d \\ R_3\,(I_1 - I_2) - V_b = 0 \\ I_3 + I_c = 0 \end{cases} \tag{2}$$

2.3 Circuit Solution

To make sense out of the equations that were presented already, we also have to add

$$\begin{cases} K_b\,V_b - I_b = 0 \\ K_c\,I_c - V_c = 0 \end{cases} \tag{3}$$

We then use *GNU Octave*, a $software$ that can solve this system of equations, to obtain the values of all the unknowns. Knowing all the voltages allows us to know every current as well, which means the circuit is solved. The results of these computations are compiled in this table.

height Name	Value [A and V]
V1	7.1581440e+00
V2	7.1581440e+00
V3	6.6424218e+00
V4	9.9835499e-01
V5	7.9316995e+00
V6	4.0555787e+00
V7	-9.7816698e-01
Vb	-3.5456366e-02
Vc	7.9316995e+00
I1	2.3690260e-04
I2	2.4828279e-04
I3	-9.7373690e-04
I4	-1.0314759e-03
Ib	-2.4828279e-04
Ic	9.7373690e-04

Table 1: Node and Mesh Analysis Computation Results. A variable starting with I is of type *current* and expressed in Ampere; a variable starting with V is of type *voltage* and expressed in Volt.

3 Simulation Analysis

3.1 Operating Point Analysis

Table **??** shows the simulated operating point results for the circuit under analysis.

height Name	Value [A or V]
@gb[i]	-2.48283e-04
@id[current]	1.031476e-03
@r1[i]	2.369026e-04
@r2[i]	-2.48283e-04
@r3[i]	-1.13802e-05
@r4[i]	1.210639e-03
@r5[i]	1.279759e-03
@r6[i]	9.737369e-04
@r7[i]	9.737369e-04
v(1)	8.139731e+00
v(2)	7.896243e+00
v(3)	7.380521e+00
v(4)	2.954689e+00
v(5)	7.931699e+00
v(6)	1.180782e+01
v(7)	9.781670e-01
v(8)	2.954689e+00

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

These results were produced using the *Ngspice software*. In order for the *software* to be able to recognise the Current-Controlled Voltage Source, defined in Figure **??** as H_c, we had to add a new Independent Voltage Source with a voltage of 0V, which is also represented in **??**.

To compare the results between the theoretical calculations and the simulation, it's important to keep in mind that the current values and directions represented in **??** by round arrows: I_1, I_2, I_3 and I_4 correspond,

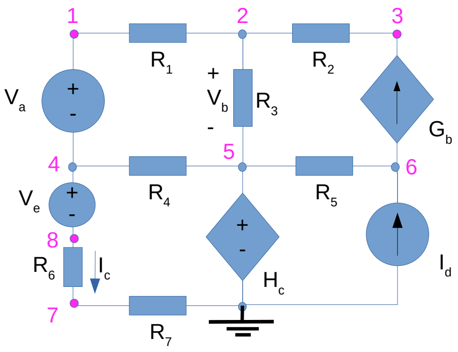

Figure 2: The original circuit with an added voltage source of value 0V.

respectively, to the following values in the simulation data table (**??**): @r1[i], -@r2[i], -@id[current], -@r6[i] (or @r7[i], they're equivalent).

On a general basis, the values obtained by the simulation greatly resemble the ones obtained using the theoretical models and the application *GNUOctave* to compute them.

In fact, by inspection, it's possible to conclude that practically all the values fit in the same magnitude as its correspondent.

As it's possible to check in the following table, the relative uncertainties ($|theoretical - simulated|/|theoretical|$) are considerably slim. Although we have a rather significant uncertainty for the value of V7 (theoretical value: -9.7816698e-01 V , simulated value: 8.139731e+00 V, uncertainty: 2.0000000e+00); the smallest uncertainty, which null, reports to the value of I_1 (theoretical value: 2.3690260e-04 A , simulated value: 2.369026e-04 A). It's relevant to note that this number isn't actually zero, but instead, the program doesn't have enough precision to compute a detectable error.

height **Name**	**Relative Uncertainty [A or V]**
V1	1.3712870e-01
V2	1.0311318e-01
V3	1.1111899e-01
V4	1.9595575e+00
V5	6.3038192e-08
V6	1.9115006e+00
V7	2.0000000e+00
I1	0.0000000e+00
I2	8.4580973e-07
I3	5.9296407e-02
I4	5.5977071e-02

Table 3: Relative Uncertainty of the simulation results relative to the theorical values.

4 Conclusion

In this laboratory assignment, the objective of analysing a circuit with several current and voltage sources (one linearly dependent and one independent of each) in parallel and in series with resistors has been achieved.

Ideally, the current and voltage theoretical analysis (computed using *GNUOctave*) should precisely resemble the circuit simulated by the *Ngspice*.

As seen in the previous section, it's possible to conclude that the simulations follow very closely the theoretical model used in the analysis: node and mesh analysis. In fact, for a circuit containing only linear components, the simulation values should not differ from the theoretically obtained ones.

References

[1] Phyllis R. Nelson, *Introduction to SPICE Source Files* Slides

[2] *GNU Octave* Documentation Files

[3] José Teixeira de Sousa, *Circuit Theory and and Eletronic Fundamentals* Class Slides