Circuit Theory and Electronics Fundamentals

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

March 24, 2021

Laboratory Assignment - T1

Group nº59

José Miguel Goulão - 95814
Lourenço Pacheco - 95817
André Gomes - 96352

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing:

- seven resistors (R_1-R_7)
- one voltage source (V_a)
- one current source (I_d)
- one voltage-controlled current source (I_b)
- one current-controlled voltage source (V_c)

Circuit T1 is presented in Figure 1. All components, including nodes ($N1$-$N8$) and ground(GND or 0), are identified with their respective names. Note that I_b is also refered to as G_1 and V_c as H_1 (explanation can be found in Subsection 3.1). Furthermore, the existence of $N6$ and $N7$ is also explained in Subsection 3.1, however, consider the voltages in both nodes to be equal.

In Section 2, a theoretical analysis (using two distinct methods) 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.

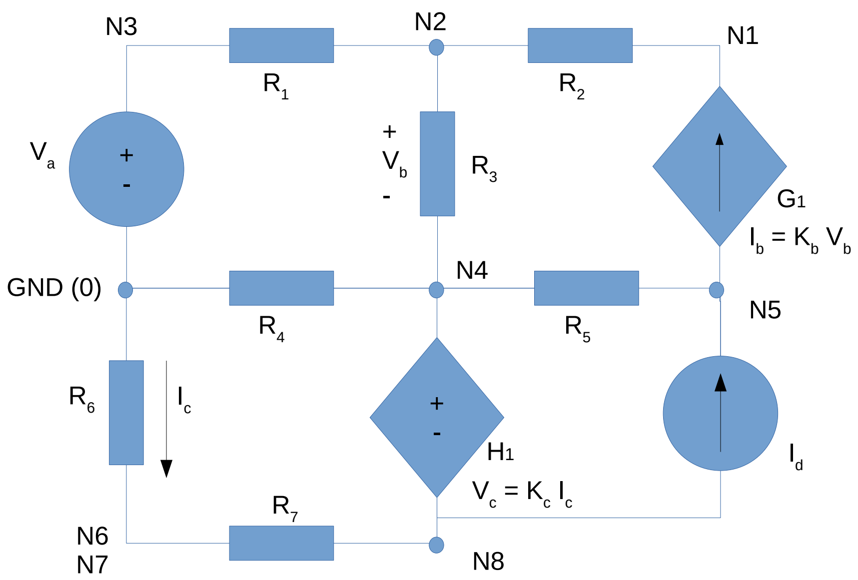

Figure 1: Circuit T1

For this laboratory assignment, the values considered for all the varibles can be found on Table 1. They were obtained through a Python script that generates random values.

Name	Value
$R1$	1.00359089673
$R2$	2.04298963569
$R3$	3.02503141993
$R4$	4.05647775356
$R5$	3.07781188185
$R6$	2.01277040929
$R7$	1.01993304256
V_a	5.11402517827
I_d	1.03896393154
K_b	7.23768458527
K_c	8.33526265782

Table 1: Values provided by the Python sript. Units for the values: kOhm, V, mA and mS

2 Theoretical Analysis

In this section, the circuit in Figure 1 is analysed theoretically.

Two methods were used and both will be explained and presented. In Subsection 2.1 the aplication of the mesh method and its results are shown. In Subsection 2.2 the same is done with the node method.

In both of these methods, three important equations were used: both Kirchhoff's laws (Kirchhof's current law (KCL) - eq.(1) and Kirchhoff's voltage law (KVL) - eq.(2)); Ohm's law (eq.(3)).

The algebraic sum of all the currents in any given node is zero:

$$\sum I_i = 0 \tag{1}$$

The algebraic sum of all the voltages in any given closed-loop circuit (mesh) is zero:

$$\sum V_i = 0 \tag{2}$$

The potential difference between the two nodes connected to a resistor is equal to the current that passes through the resistor multiplied by its resistance.

$$V_i = R_i I_i \tag{3}$$

2.1 Mesh method

To correclty use the mesh method, firstly, four currents must be considered, one for each simple mesh. They were identified as follows: I_1 - associated with the top left mesh; I_2 - associated with the top right mesh; I_3 - associated with the bottom left mesh; I_4 - associated with the bottom right mesh. Each one of these currents is assumed to run counterclockwise.

Afterwards, eq.(2) needs to be applied in the meshes not containing any type of current sources (eq.(4) and eq.(5)) . Moreover, it is essential to relate the remaining mesh current with those created by the current sources (eq.(6) and eq.(7)). Likewise, eq.(8) is also obtained.

$$V_a = I_1(R_1 + R_3 + R_4) - I_2(R_3) - I_3(R_4) \tag{4}$$

$$-V_c = I_3(R_4 + R_6 + R_7) - I_1(R_4) \tag{5}$$

$$I_2 = -I_b \tag{6}$$

$$I_4 = -I_d \tag{7}$$

$$I_3 = -I_c \tag{8}$$

Further relations need to be composed in order to solve the circuit. We must not forget the two equations from the linear dependent sources (eq.(9) and eq.(10)). In adition, by making use of eq.(3), one more equation is acquired, eq.(11).

$$I_b = K_b V_b \tag{9}$$

$$V_c = K_c I_b \tag{10}$$

$$I_1(R_3) - I_2(R_3) = V_b \tag{11}$$

With eight equations and eight unknown variables, the system can be solved. The results were computed by Octave and organized in Table 2

Name	**Value [A or V]**
$@I_1$	2.822201e-04
$@I_2$	2.957272e-04
$@I_3$	9.187358e-04
$@I_4$	-1.038964e-03
V_b	-4.085937e-02
V_c	-7.657904e+00
$@I_b$	-2.957272e-04
$@I_c$	-9.187358e-04

Table 2: Values computed by Octave. Variables identified with a '@' have a corresponding value in Ampere (A). The others are expressed in Volts (V).

2.2 Node method

Similarly to the previous subsection, for the node method, ground was considered to be where it is identified in Figure 1. In adition, assume V_{Ni} to be the voltage in node Ni (every node position can also be found in Figure 1).

The node method uses KCL in conjunction with Ohm's law to define equations that when solved give the voltage value of each nove in relation to ground (Node 0, $V_0 = 0$). In this circuit we defined six equations that equate the currents entering a particular node with the currents leaving said node.

In order to have equations that solve for the node's voltage, a relation between curent and voltage is made using Ohm's law (given a resistance between two nodes, the current that passes the resistance can be written as $I = \frac{V_2 - V_1}{R_1}$)

To simplify the equations it is useful to use the conductance G_n which is the inverse of the resistance R_n ($G_n = \frac{1}{R_n}$)

$$G_2(V_1 - V_2) + G_1(V_3 - V_2) - G3(V_2 - V_4) = 0 \tag{12}$$

$$G_2(V_1 - V_2) + K_b(V_4 - V_2) = 0 \tag{13}$$

$$G3(V_2 - V_4) + G_5(V_5 - V_4) - G_4V_4) + I_c = 0 \tag{14}$$

$$K_b(V_2 - V_4) - G_5(V_5 - V_4) = -I_d \tag{15}$$

$$G_7(V_6 - V_7) - I_c = I_d \tag{16}$$

$$G_6(-V_6) - G_7(V_6 - V_7) = 0 \tag{17}$$

In order to solve the circuit two more equations are used to relate the voltage difference between the nodes that are conected to the voltage sources.

$$V_3 = V_a \tag{18}$$

$$K_c\frac{(-V_6)}{G_6} - (V_4 - V_7) = 0 \tag{19}$$

With these 8 equations it is possible to solve the system using Octave. The results were organized in Table 3

Name	Value [A or V]
V_{N1}	4.226624e+00
V_{N2}	4.830792e+00
V_{N3}	5.114025e+00
V_{N4}	4.871651e+00
V_{N5}	8.979579e+00
V_{N6}	-1.849204e+00
V_{N8}	-2.786253e+00
V_b	4.085937e-02
V_c	7.657904e+00
$@I_b$	2.957272e-04
$@I_c$	9.187358e-04
$@I_{H1}$	-1.202281e-04

Table 3: Values computed by Octave. Variables identified with a '@' have a corresponding value in Ampere (A). The others are expressed in Volts (V).

3 Simulation Analysis

In this section, Circuit T1 is reproduced with the help of Ngspice.

Firstly, the outcome of the simulation is shown, as well as a brief explanation on how it was achived. Afterwards, a comparison is done between those values and the ones attained in Section 2.

3.1 Simulated results

Ngspice is a simulator for eletronic circuits that can output a variety of results. This emulator computes the voltages in every node, as well as the potential difference between two given nodes. Apart from that, the group made use of the command *.options savecurrents* which also enables the output of the currents that pass through all branches.

With the limitation that Ngspice only provides the current in the components and not through the nodes, an aditional voltage source ($Vaux$) was added so that the current in R_6 (I_c) is known. This source (not displayed in Figure 1) has a voltage of 0V and it was implemented between R_6 and R_7. Therefore an aditional node had to be added (node $N7$).

As previously stated, I_b is refered to as G_1. This is because, in Ngspice, a voltage-controlled current source is identified with capital 'g' (G). In the case of V_c, all current-controlled voltage source are identified with H.

Table 4 shows the simulated operating point results for Circuit T1.

Name	Value [A or V]
i(vaux)	9.187358e-04
i(h1)	1.202281e-04
@g1[i]	-2.95727e-04
@id[current]	1.038964e-03
@r1[i]	-2.82220e-04
@r2[i]	-2.95727e-04
@r3[i]	1.350709e-05
@r4[i]	-1.20096e-03
@r5[i]	-1.33469e-03
@r6[i]	9.187358e-04
@r7[i]	-9.18736e-04
n1	4.226624e+00
n2	4.830792e+00
n3	5.114025e+00
n4	4.871651e+00
n5	8.979579e+00
n6	-1.84920e+00
n7	-1.84920e+00
n8	-2.78625e+00
v(n4,n2)	4.085937e-02
v(n4,n8)	7.657904e+00

Table 4: Values from Ngspice. Variables identified with a '@' or are of the type $i(...)$ have a corresponding value in Ampere (A). The others are expressed in Volts (V).

3.2 Comparison

By comparing both Tables, we confirm that all the absolute values displayed in Table 4 are identical to the ones shown in Section 2, including all decimal digits.

All the voltages in every node match with high precision. Moreover, V_b and V_c are equal to the simulated values, which are presented in Table 4 as $v(n2, n4)$ and $v(n4, n8)$, respectively. Finally, theoretical I_d is also the same as the one obtained via Ngspice ('$@g1[i]$').

It is also worth noting that all theoretical calculations consider every element of the circuit to be ideal (without energy loss nor self-inductance nor any other phenomena that could alter the results). Similarly, Ngspice also considers all components to be ideal. Therefore every source of discrepancies between theoretical and simulated results are removed (apart from the small limitations concerning calculations and the rounding of values).

4 Conclusion

For this laboratory assignment, we were given a circuit composed by resistors, dependent and independent current and voltage sources and had the objective of analyzing and simulating it and then compare the results obtained.

Static analyses were performed theoretically, through mesh and node analysis and by circuit simulation, using the Octave math tool and Ngspice tool, respectively. The simulation results matched the theoretical results very precisely. So in that order it is safe to say that our objective was achieved successfully.