



Circuit Theory and Electronics Fundamentals

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

March 24, 2021

Laboratory Assignment - T2

Group nº59

José Miguel Goulão - 95814

Lourenço Pacheco - 95817

André Gomes - 96352

Contents

1	Introduction	1
2	Simulation Analysis	2
2.1	Simulated results	3
2.2	Comparison	3
3	Conclusion	3

1 Introduction

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

- seven resistors (R_1 - R_7)
- one voltage source (V_s)
- one capacitor (C)
- one voltage-controlled current source (I_b)
- one current-controlled voltage source (V_d)

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 referred to as G_1 and V_d as H_1 (explanation can be found in Subsection 2.1).

In Section ??, a theoretical analysis (using two distinct methods) of the circuit is presented. In Section 2, 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 3.

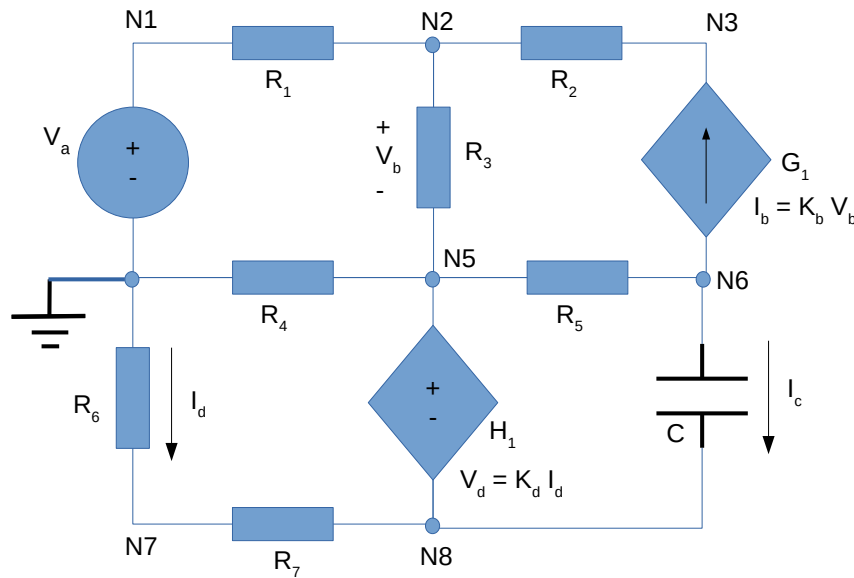


Figure 1: Circuit T2

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

2 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 achieved. Afterwards, a comparison is done between those values and the ones attained in Section ??.

Name	Value
R_1	1003.59
R_2	2042.99
R_3	3025.03
R_4	4056.48
R_5	3077.81
R_6	2012.77
R_7	1019.93
V_s	5.11403
C	1.03896e-06
K_b	0.00723768
K_d	8335.26

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

2.1 Simulated results

Ngspice is a simulator for electronic 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 additional voltage source (V_{aux}) was added so that the current in R_6 (I_c) is known. This source (not displayed in Figure ??) has a voltage of 0V and it was implemented between R_6 and R_7 . Therefore an additional node had to be added (node N_7).

As previously stated, I_b is referred 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 2 shows the simulated operating point results for Circuit T1.

2.2 Comparison

By comparing both Tables, we confirm that all the absolute values displayed in Table 2 are identical to the ones shown in Section ??, 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 2 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).

3 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

Name	Value [A or V]
i(vaux)	9.187367e-04
i(h1)	-9.18737e-04
@c[i]	0.000000e+00
@g1[i]	-2.95727e-04
@r1[i]	-2.82220e-04
@r2[i]	-2.95727e-04
@r3[i]	1.350709e-05
@r4[i]	-1.20096e-03
@r5[i]	-2.95727e-04
@r6[i]	9.187367e-04
@r7[i]	-9.18737e-04
n1	5.114030e+00
n2	4.830797e+00
n3	4.226630e+00
n5	4.871656e+00
n6	5.781848e+00
n7	-1.84921e+00
n7.	-1.84921e+00
n8	-2.78625e+00
v(n5,n2)	4.085936e-02
v(n5,n8)	7.657909e+00
v(n6,n8)	8.568100e+00

Table 2: 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).

matched the theoretical results very precisely. So in that order it is safe to say that our objective was achieved successfully.