

Circuit Theory and Electronics Fundamentals

Integrated Master's in Aerospace Engineering, Técnico, University of Lisbon

Laboratory 1 Report

Francisco Alves 95787, Tomás Nunes 95855, Rodrigo Sequeira 96480

March 22nd, 2021

Contents

1	Introduction	1
2	Theoretical Analysis	2
2.1	Nodal Analysis	2
2.2	Mesh Analysis	3
3	Simulation Analysis	4
3.1	Ngspice Analysis	4
4	Conclusion	5

1 Introduction

The objective of this laboratory assignment is to study a circuit containing an independent voltage source V_a , an independent current source I_d , seven resistors R_1 to R_7 , a linearly dependent current source I_b and a linearly dependent voltage source V_c . The circuit can be seen in Figure 1. The study of the circuit is done using both mesh and nodal analysis methods.

In Section 2, a theoretical analysis of the circuit using Octave is presented. In Section 3, the circuit is analysed by simulation with Ngspice software and the results are compared to the theoretical results obtained in Section 2. Conclusions of this study can be found in Section 4.

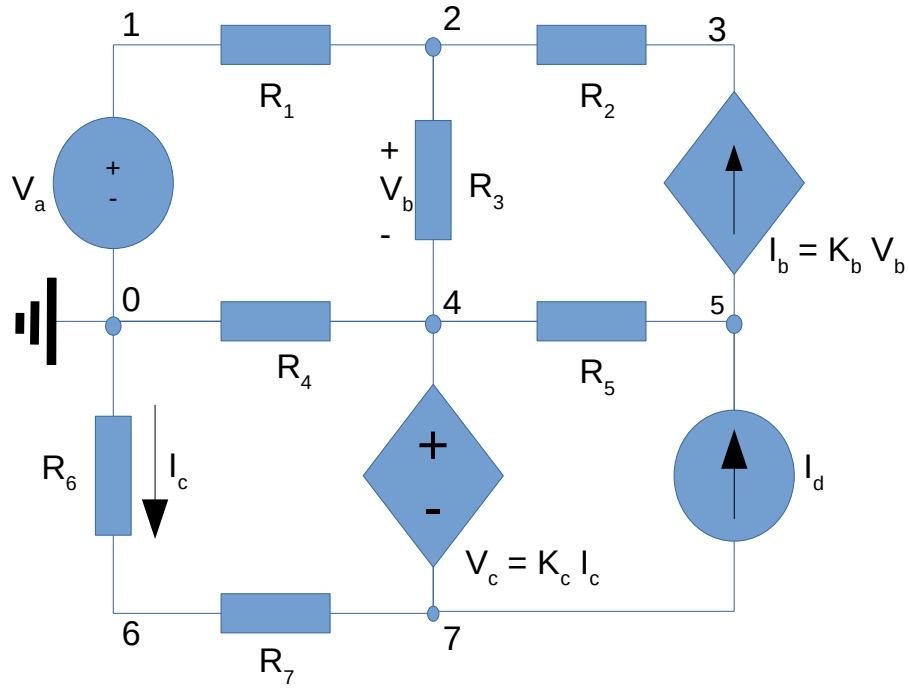


Figure 1: Circuit in study.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, in terms of each branch's current and voltage. The current flows considered in the analysis are the following: R_1 from node 1 to 2; R_2 from node 2 to 3; R_3 from node 2 to 4; R_4 from node 4 to 0; R_5 from node 4 to 5; R_6 from node 0 to 6; R_7 from node 6 to 7. Also, the potential in Node 0 was considered to be 0V (ground).

2.1 Nodal Analysis

This method is based on Kirchhoff's Current Law (KCL), and firstly consists in deriving equations for the current flow in nodes not connected to voltage sources, followed by writing additional equations in nodes related to voltage sources. The node identification can be seen on Figure 1. Moreover, the equations relative to nodes 0, 2, 5 and 0, respectively, are

$$(V_0 - V_4)G_4 + (V_0 - V_6)G_6 + (V_1 - V_2)G_1 = 0 \quad (1)$$

$$(V_2 - V_1)G_1 + (V_2 - V_4)G_3 + (V_2 - V_3)G_2 = 0 \quad (2)$$

$$(V_5 - V_4)G_5 + I_b - I_d = 0, I_b = K_b(V_2 - V_4) \quad (3)$$

$$(V_6 - V_7)G_7 + (V_0 - V_4)G_4 + (V_1 - V_2)G_1 = 0 \quad (4)$$

The additional equations for the method are

$$V_0 = 0 \quad (5)$$

$$V_1 = V_a \quad (6)$$

$$V_4 - V_7 = K_c(V_0 - V_6)G_6 \quad (7)$$

$$I_b = (V_3 - V_2)G_2, I_b = K_b(V_2 - V_4) \quad (8)$$

Name	Value [mA or V]
I_b	-0.265898
I_d	1.028722
I_{R1}	0.253649
I_{R2}	0.265898
I_{R3}	-0.012249
I_{R4}	1.205310
I_{R5}	-1.294619
I_{R6}	0.951662
I_{R7}	0.951662
V_1	5.054819
V_2	4.793705
V_3	4.258198
V_4	4.831047
V_5	8.907510
V_6	-1.934226
V_7	-2.905231
V_8	-1.934226

Table 1: Nodal Analysis' variable values, where I_j is expressed in milliampere and V_j is expressed in Volt.

2.2 Mesh Analysis

This method is based on Kirchoff's Voltage Law (KVL), and consists on analysing the circuit's meshes after creating arbitrary currents with arbitrary flows for each mesh in the circuit. The top left mesh was named mesh A with clockwise current flow I_A ; the top right mesh was named mesh B with counterclockwise current flow I_B ; the bottom left mesh was named mesh C with counterclockwise current flow I_C ; the bottom right mesh was named mesh D with counterclockwise current flow I_D . Furthermore, the equations associated with these meshes and currents are

$$R_1 I_A + R_3(I_A + I_B) + R_4(I_A + I_C) - V_a = 0 \quad (9)$$

$$R_6 I_C + R_7 I_C - K_c I_C + R_4(I_A + I_C) = 0 \quad (10)$$

$$I_D = I_d \quad (11)$$

$$I_B = K_b R_3(I_A + I_B) \quad (12)$$

Name	Value [mA or V]
I_b	-0.265898
I_d	1.028722
I_{R1}	0.253649
I_{R2}	0.265898
I_{R3}	-0.012249
I_{R4}	1.205310
I_{R5}	-1.294619
I_{R6}	0.951662
I_{R7}	0.951662
V_1	5.054819
V_2	4.793705
V_3	4.258198
V_4	4.831047
V_5	8.907510
V_6	-1.934226
V_7	-2.905231
V_8	-1.934226

Table 2: Mesh Analysis' variable values, where I_j is expressed in milliampere and V_j is expressed in Volt.

3 Simulation Analysis

3.1 Ngspice Analysis

Table 3 shows the simulated operating point results for the circuit under analysis. Once again, current flows are the ones referred in 2 and node 0 is considered to have 0V potential.

A new voltage source V_{aux} with voltage 0V (so it doesn't affect the circuit) was added to the circuit between components R_6 and R_7 so that Ngspice could simulate and calculate the current value in that branch. This current is needed since the voltage source V_c depends on its value and as expected, this current's value is the same as the one that flows through R_6 and R_7 . Adding a new voltage source led to the creation of node 8 between R_6 and V_{aux} . Also, has predicted, voltage V_8 is the same as voltage V_6 since the voltage in V_{aux} is 0V.

Name	Value [A or V]
gb[i]	-2.65898e-04
id[current]	1.028722e-03
r1[i]	2.536488e-04
r2[i]	2.658976e-04
r3[i]	-1.22488e-05
r4[i]	1.205310e-03
r5[i]	-1.29462e-03
r6[i]	9.516615e-04
r7[i]	9.516615e-04
v(1)	5.054819e+00
v(2)	4.793705e+00
v(3)	4.258198e+00
v(4)	4.831047e+00
v(5)	8.907510e+00
v(6)	-1.93423e+00
v(7)	-2.90523e+00
v(8)	-1.93423e+00

Table 3: Operating point. A variable proceeded by [i] is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

4 Conclusion

Comparing the results given by Nodal and Mesh analysis, it can be observed that both methods achieved the same values. Also, the results given by the Ngspice simulation are the same as those of the methods stated above. All in all, the analysis of the given circuit using the stated methods was achieved successfully. Nodal and mesh methods were performed both theoretically using the Octave maths tool and by circuit simulation using the Ngspice software. Given the fact that the circuit only has linear components and it is a simple circuit to analyse, Ngspice probably used the same models as we did in the theoretical analysis, so it makes sense that no discrepancies were detected. However, if we had been given a circuit with more complex components, the theoretical and simulation models could not match as precisely as they did in this laboratory assignment.