Circuit Theory and Electronics Fundamentals

T1 Laboratory Report

Group 34

José Bento, Nº 95815
Thomas Childs, Nº 95847
Luís Pacheco, Nº 96425

Aerospace Engineering (MEAer), Técnico, University of Lisbon

March 22, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing independent and linearly dependent voltage and current sources. The circuit also contains 7 resistors, totaling 11 components.

The circuit has 8 nodes and 4 meshes. The nodes of the circuit were numbered arbitrarily (from V_0 to V_7), and it was considered that *node 0* was the ground node. The voltage-controlled current source I_b has a linear dependence on Voltage V_b, of constant K_b. The voltage V_b is the voltage drop at the ends of resistor R_3. The current-controlled voltage source V_c has a linear dependece on current I_c, of constant K_c. The control current I_c is the current that passes through the resistor R_6. The circuit can be seen in **Figure 1**.

The values for the resistors, the independent sources and the constants for the dependent sources are presented in **Table 1**. These values were obtained using the Python script provided by the Professor responsable for the laboratory assigment and using the number 95815 as the seed.

In Section 2, a theoretical analysis of the circuit is presented using two methods: the mesh analysis and the nodal analysis. In Section 3, the circuit is analysed by simulation using the program Ngspice, and the results are compared to the theoretical results obtained in Section 2. The conclusions of this study are outlined in Section 4.
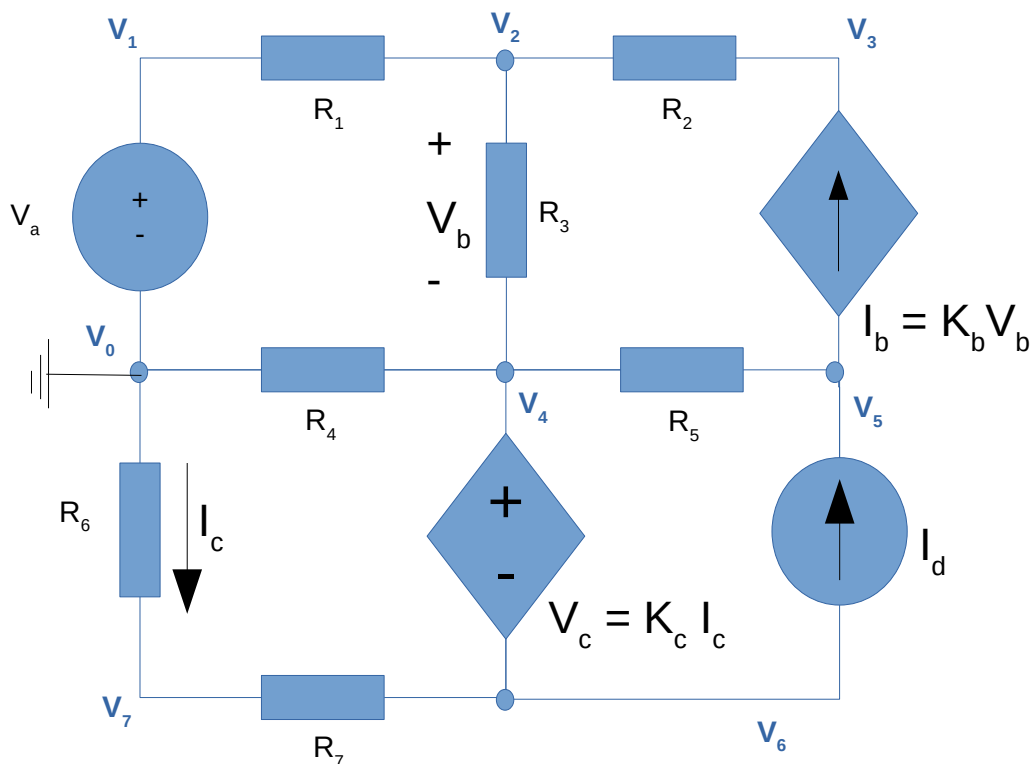

Figure 1: Circuit in study

Name	Python-generated values
R1	1.04606282456
R2	2.00732621328
R3	3.06060705885
R4	4.07055531265
R5	3.1225213804
R6	2.06927045958
R7	1.01531018068
Va	5.24359648479
Id	1.01891541651
Kb	7.0473187437
Kc	8.3479788681

Table 1: The variables that start with an *R* are the values of the resistors and are expressed in kiloohm (kOhm); the variable *Va* is a *voltage* and is expressed in Volt (V) and the variable *Id* is a *current* and expressed in miliAmpere (mA). The constants *Kc* and *Kb* are expressed in kiloOhm (kOhm) and miliSiemens (mS), respectively.

2 Theoretical Analysis

In this section, the circuit shown in **Figure 1** is analysed theoretically using two methods: the mesh analysis and the nodal analysis.

2.1 Mesh Analysis

For the mesh analysis 4 variables were created - the 4 mesh currents. I_{SE} is the mesh current of the mesh on the top left corner of our circuit and it flows clockwise. I_{SD} is the mesh current of the mesh on the top right corner of our circuit and it flows counter clockwise. I_{IE} is the mesh current of the mesh on the lower left corner of our circuit and it flows counter clockwise. I_{ID} is the mesh current of the mesh on the lower right corner of our circuit and it flows counter clockwise. Knowing the mesh currents, and using Ohm's law all node voltages and branch currents can be determined. To determine the 4 unknown mesh currents the following 4 equations were used:

$$R_1 I_{SE} + R_3(I_{SE} + I_{SD}) + R_4(I_{SE} + I_{IE}) = Va \tag{1}$$

$$R_6 I_{IE} + R_7 I_{IE} + R_4(I_{IE} + I_{SE}) = K_c I_{IE} \tag{2}$$

$$I_{SD} = K_b R_3(I_{SE} + I_{SD}) \tag{3}$$

$$I_{ID} = Id \tag{4}$$

With the help of Octave, these equations yielded the following values for the mesh currents:

Name	Current (mA)
I_{SE}	0.27805
I_{SD}	-0.29157
I_{IE}	0.94884
I_{ID}	1.0189

2.2 Nodal Analysis

For the nodal analysis we apply the Kirchhoff Current Law (KCL) to the nodes that are not conected to voltage sources (equations 7 to 11). We obtain 2 more equations from the potencial difference at the terminals of the voltage sources (equations 12 and 13). As was stated before, *node 0* is the ground node and therefore the voltage at this node is 0V (equation 5). For the simulation analysis it was necessesary to add another "fictional" voltage source that provides 0 volts to the circuit, between node 7 and resistor 6. This "fictional" voltage source was also considered in the Theoretical Analysis, as it has no real effect on the circuit. It was needed to define in the ngspice code the current on which the current-controlled voltage source depends on. This yields equation number 6. Now we have 9 equations for 9 unkown variables - voltage V_0 to V_8. Using Octave, we determine the values of every node voltage of the circuit.

$$V_0 = 0 \tag{5}$$

$$V_8 = V_7 \tag{6}$$

$$\frac{V_1 - V_2}{R_1} + \frac{V_0 - V_8}{R_6} + \frac{V_0 - V_4}{R_4} = 0 \tag{7}$$

$$\frac{V_2 - V_4}{R_3} + \frac{V_2 - V_3}{R_2} + \frac{V_2 - V_1}{R_1} = 0 \tag{8}$$

$$-K_b(V_2 - V_4) + \frac{V_3 - V_2}{R_2} = 0 \tag{9}$$

$$\frac{V_5 - V_4}{R_5} + K_b(V_2 - V_4) - Id = 0 \tag{10}$$

$$\frac{V_7 - V_6}{R_7} + \frac{V_7 - V_0}{R_6} = 0 \tag{11}$$

$$V_1 - V_0 = V_a \tag{12}$$

$$V_4 - V_6 = K_c\frac{V_0 - V_7}{R_6} \tag{13}$$

In **Table 2** the values for the branch currents and the node voltages obtained from the Octave script for both methods are presented. Here, the node voltages in the mesh method were computed from the respective currents, which were determined as described in the previous subsection.

Name	Mesh method	Node method
@Gb	-0.291567	-0.291567
@id	1.018915	1.018915
@r1	-0.278049	-0.278049
@r2	-0.291567	-0.291567
@r3	-0.013518	-0.013518
@r4	1.226887	1.226887
@r5	1.310482	1.310482
@r6	-0.948838	-0.948838
@r7	-0.948838	-0.948838
V1	5.243596	5.243596
V2	4.952740	4.952740
V3	4.367469	4.367469
V4	4.994112	4.994112
V5	9.086122	9.086122
V6	-2.926767	-2.926767
V7	-1.963402	-1.963402
V8	-1.963402	-1.963402

Table 2: A variable preceded by @ is of type *current* and expressed in milliampere (mA); other variables are of type *voltage* and expressed in Volt (V).

As can be seen, the values obtained via the mesh and the node analysis are the same, which suggests that for this simple circuit, the methods are equally precise, as expected.

3 Simulation Analysis

In this section we will describe the steps needed to simulate this circuit using the software Ngspice.

3.1 Operating Point Analysis

Table 3 shows the simulated operating point results for the circuit under analysis. The current flows considered in the theoretical section were coherent with the polarity implicitly declared when defining the circuit to be simulated in the Ngspice script.

As mentioned previously, in Section 2, we had to create a "fictional" voltage source, between node 7 and resistor 6 (providing 0V to the circuit in order not to alter the behaviour of the rest of the circuit) so as to be able to define the dependecy for the current-controlled voltage source V_c. This has no specific reason to be, other than the particularities of the Ngspice software. The consequences of adding this additional voltage have already been described (namely the creation of a new node) and dealt with in the previous section - see, for example, equation 6.

As can already be observed, the results of the simulation coincide up to the last digit with the results of the theoretical analysis.

Name	Value [mA or V]
@gb[i]	-2.91567e-01
@id[current]	1.018915e+00
@r1[i]	-2.78049e-01
@r2[i]	-2.91567e-01
@r3[i]	-1.35178e-02
@r4[i]	1.226887e+00
@r5[i]	1.310482e+00
@r6[i]	-9.48838e-01
@r7[i]	-9.48838e-01
v(1)	5.243596e+00
v(2)	4.952740e+00
v(3)	4.367469e+00
v(4)	4.994112e+00
v(5)	9.086122e+00
v(6)	-2.92677e+00
v(7)	-1.96340e+00
v(8)	-1.96340e+00

Table 3: Operating point. A variable preceded by @ is of type *current* and expressed in mil-iAmpere; other variables are of type *voltage* and expressed in Volt.

4 Conclusion

In this laboratory assignment the objective of analysing a circuit containing independent, linearly dependent voltage and current sources and other linear components (resistors) has been achieved. Static analyses have been performed both theoretically, using the Octave maths tool, and by circuit simulation, using the Ngspice tool. Not only did the theoretical results obtained using different methods agree perfectly, they also matched the simulation results precisely.

The reason for this perfect match is the fact that this is a straightforward circuit containing only linear components - this implies, as pointed out in the theory classes, that the theoretical and simulation models cannot differ. For more complex components and circuits, the theoretical and simulation models could differ (given the greater complexity of the models implemented by the simulator Ngspice, when compared to those studied in the theoretical classes); however, that is not the case for this particular circuit.

This lab assignment has been useful to learn and give us the chance to put into practice our knowledge of the invaluable software tools and automation procedures required for this and other types of work, from Git and Makefile to Octave and the syntax of Ngspice. In more theoretical terms, we have become more at ease with the analysis of linear circuits, and, knowing what we now know, we would have made differente choices regarding the analysis even of this circuit (for example, we probably would have chosen a different node to assign to the Ground potential).