Circuit Theory and Electronics Fundamentals

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

Example Laboratory Report

February 27, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit that contains two current sources, two voltage sources and seven resistors. The circuit has a dependent voltage source V_c and an independent one V_a. One of the current sources is dependent, I_b, and an independent one, I_d.

A representation of the circuit made with *LibreOffice Draw* can be seen in figure 1.
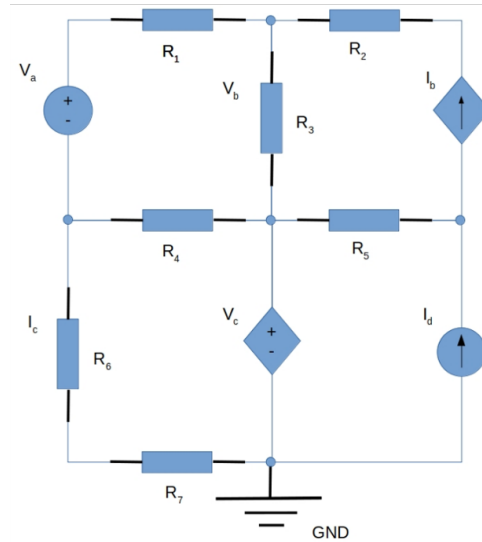
Figure 1: Studied Circuit

In Section 2, a theoretical analysis 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.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, using the mesh method and the nodes method. The known values can be seen in the following table.

R_1	1.02362892933kOhm
R_2	2.08640382129kOhm
R_3	3.09996108706kOhm
R_4	4.08183334334kOhm
R_5	3.0416957579kOhm
R_6	2.04156679366kOhm
R_7	1.04156790057kOhm
V_a	5.17885996884V
I_d	1.04908336809mA
K_b	7.29571922963mS
K_c	8.16840649221kOhm

Table 1: Known Data

2.1 Mesh Method

To fully understand this particular method, the concept of mesh needs to be clarified. A mesh is a loop that contains no other loops. Then, it is possible to determine, via Kirchhoff Voltage Law (KVL), the current in each mesh. With that information, using Ohm's Law, it is possible to determine every voltage in the circuit, knowing the resistances in the mesh. This method requires an attentive observer that can identify meshes, therefore, is not often used in automation processes.

Firstly, four meshes were indentified in the circuit. The equations were written and the flow of the current was stipulated, as can be seen in figure 2.
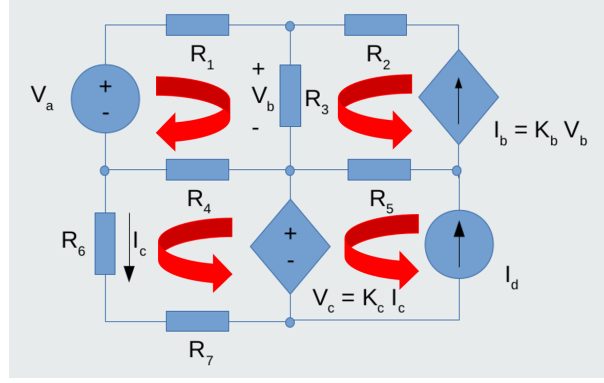
Figure 2: Mesh Method applied to the circuit

Simplifying the mesh method, the following system of equations 2.1 was writen.

$$
\begin{cases}
V_a = R_1 I_a + R_3(I_a + I_b) + R_4(I_a + I_c) \\
V_b = \frac{I_b}{K_b} = R_3(I_a + I_b) \\
V_c = K_c I_c = R_4(I_a + I_c) + R_6 I_c + R_7 I_c
\end{cases}
$$

It was transformed into the following system 1 and solved using the software *Octave*.

$$
\begin{bmatrix}
R_1 + R_3 + R_4 & R_3 & R_4 & 0 \\
R_4 & 0 & -K_c + R_4 + R_6 + R_7 & 0 \\
R_3 & R_3 - \frac{1}{K_b} & 0 & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
I_a \\ I_b \\ I_c \\ I_d
\end{bmatrix}
=
\begin{bmatrix}
V_a \\ 0 \\ 0 \\ 0
\end{bmatrix}
\tag{1}
$$

Therefore, the values for the voltage and current sources were determined and can be seen in the following table 2, being the unit for current mA and the unit for voltage V.

I_a	2.401364132117077e-01
I_b	2.512453816512571e-01
I_c	9.768380052260230e-01
I_d	1.049083368090000e+00
V_a	5.178859968840000
V_b	3.443736987998083e-02
V_c	7.979209903725712

Table 2: Table of results for the mesh method

2.2 Node Method

Nodes are any regions between any two elements of the circuit. With Kirchhoff Current Law (KCL) it is possible to determine any nodal voltage. The KCL cannot be used in nodes that are connected to a voltage source. This means those nodes must have relations with the source, making it possible to solve the system. In opposition to the mesh method, this method is easy to automate due to its methodical approach, being possible to extract values in a simulation very easily. The first approach is to name every node so it is possible to start the analysis.

First, the nodes were represented in the circuit as can be seen in figure 3. The software used to represent the circuit and to provide another simulation to confirm the one explained on 3 was *LTSpice*.
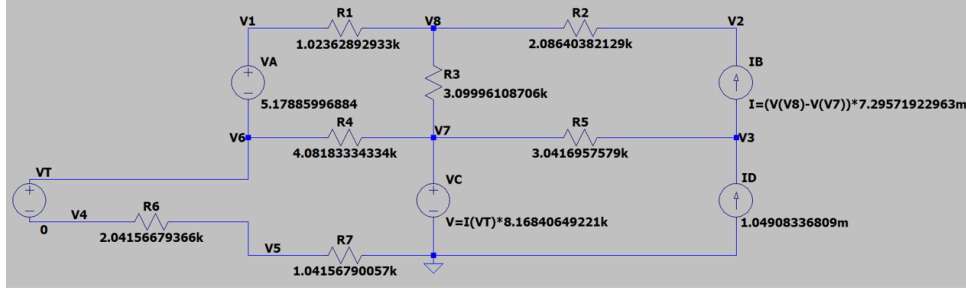
Figure 3: Circuit represented through *LTSpice*

Then, the equations for the nodes method were writen according to the numeration visible in the circuit above. The voltage source V_T was created to help in the simulation analysis, the reason behind it will be explained later.

$$
\begin{cases}
V_1 - V_6 = V_a \\
\frac{V_2 - V_8}{R_2} - K_b * (V_8 - V_7) = 0 \\
-I_d + \frac{V_3 - V_7}{R_5} + K_b (V_8 - V_7) = 0 \\
\frac{V_5 - V_6}{R_6} + \frac{V_5}{R_7} = 0 \\
\frac{V_8 - V_1}{R_1} + \frac{V_8 - V_2}{R_2} - \frac{V_8 - V_7}{R_3} = 0 \\
\frac{V_6 - V_7}{R_4} + \frac{V_6 - V_5}{R_6} - \frac{V_1 - V_8}{R_1} = 0 \\
V_7 = -K_c \frac{V_5 - V_6}{R_6}
\end{cases}
$$

The equations above were transformed into the following system 2 so that *Octave* could solve it.

$$
\begin{bmatrix}
1 & 0 & 0 & 0 & -1 & 0 & 0 \\
0 & \frac{1}{R_2} & 0 & 0 & 0 & K_b & \frac{-1}{R_2} - K_b \\
0 & 0 & \frac{1}{R_5} & 0 & 0 & \frac{-1}{R_5} - K_b & K_b \\
0 & 0 & 0 & \frac{1}{R_6} + \frac{1}{R_7} & \frac{-1}{R_6} & 0 & 0 \\
\frac{-1}{R_1} & \frac{-1}{R_2} & 0 & 0 & 0 & \frac{-1}{R_3} & \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \\
\frac{1}{R_1} & 0 & 0 & \frac{-1}{R_6} & \frac{1}{R_4} + \frac{1}{R_6} & \frac{-1}{R_4} & \frac{-1}{R_1} \\
0 & 0 & 0 & \frac{K_c}{R_6} & \frac{-K_c}{R_6} & 1 & 0]
\end{bmatrix}
\begin{bmatrix}
V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8
\end{bmatrix}
=
\begin{bmatrix}
V_a \\ 0 \\ I_d \\ 0 \\ 0 \\ 0 \\ 0
\end{bmatrix}
\quad (2)
$$

Hence, the values for the current sources and the voltage in each point V_1 to V_8 were calculated and can be seen in the table below, again being the unit for current mA and the unit for voltage V. V_4 is only in the circuit for simulation purposes, with voltage equal to V_6, V_4 is supressed in the analysis.

V_1	8.190583113394776e+00
V_2	7.420573209487090e+00
V_3	1.193441434568911e+01
V_5	1.017443110300255e+00
V_6	3.011723144554777e+00
V_7	7.979209903725710e+00
V_8	7.944772533845730e+00
I_a	2.401364132117075e-01
I_b	2.512453816512545e-01
I_c	9.768380052260227e-01
I_d	1.049083368090000e+00

Table 3: Table of results for the node method

Using

$$V_b = \frac{I_b}{K_b} \tag{3}$$

and

$$V_c = K_c I_c \tag{4}$$

one can get the following results:

V_a	5.178859968840000
V_b	3.443736987998047e-02
V_c	7.979209903725709

Table 4: Values of V_a, V_b and V_c

It is evident that the two methods return values with notorious resemblance.

The simulation, explained on the next section, will be compared with the two theoretichal analysis in order to identify any incompability with the theory, allowing further discussion.

3   Simulation Analysis

For this simulation analysis, it is convenient to represent graphically, with *LTSpice*, the studied circuit, that can be seen in the following figure.
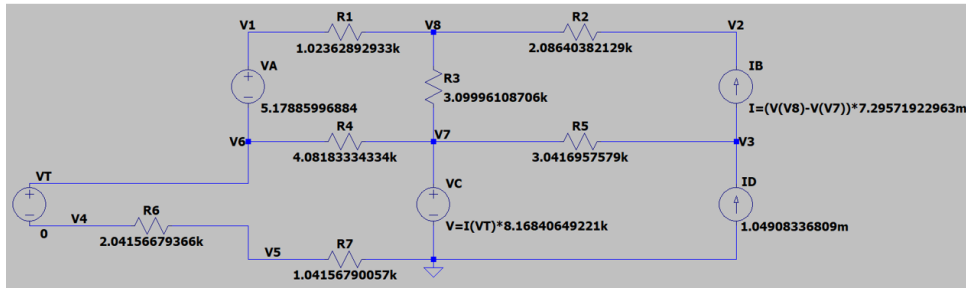

Figure 4: Circuit represented through *LTSpice*

V_T is set to 0 meaning it will function as an ammeter. This is strictly for the purpose of the simulation. In the *Spice* language, in particular in the *NGSpice* software, for a current-controled voltage source there is an input that requires the current, which is controling the voltage source. We can measure that current by inserting an independent voltage source set to 0, making it possible to determine the current that is flowing in that branch without disturbing the system. The V_T source will be connected to 2 nodes that have the same voltage (V_6 and V_4), so, although both nodes are used in the simulation, for theoretical analysis one of them was supressed. For coherence purposes it was decided that the same numeration would be used in all models, so we decided to supress V_4, therefore not showing it in any analysis.

Running the simulation with *NGSpice* we obtained the following results shown in Table 5.

5

Name	Value [A or V]
@g1[i]	-2.51245e-04
@i1[current]	1.049083e-03
@r1[i]	-2.40136e-04
@r2[i]	-2.51245e-04
@r3[i]	-1.11090e-05
@r4[i]	1.216974e-03
@r5[i]	1.300329e-03
@r6[i]	-9.76838e-04
@r7[i]	-9.76838e-04
v(1	8.190583e+00
v(2	7.420573e+00
v(3	1.193441e+01
v(4	3.011723e+00
v(5	1.017443e+00
v(6	3.011723e+00
v(7	7.979210e+00
v(8	7.944773e+00

Table 5: Operating point. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

The data provided by the simulation is enough to solve all the circuit. That being said, some values need some work so they can be compared. For instance, V_7 can be associated to V_c, because V_c imposes the voltage between the ground and V_7 implying that V_7 equals to V_c. V_a is known data, therefore no calculations are executed. Also:

$$V_b = V_8 - V_7 \tag{5}$$

For the calculations of the currents we can assume that I_{R_1} and I_{R_6} are equal to I_a and I_c, respectively. The current I_b is the current in "g1", the voltage-controlled current source. Once again I_d is a known data.

Due to linearity of the resistors, all voltages and currents can be determined knowing the resistance of the resistors, which is given.

Finally we can say the circuit has been solved and the results are the following:

I_a	2.40136e-04
I_b	2.51245e-04
I_c	9.76838e-04
V_b	3.4437e-02
V_c	7.979210e+00

Table 6: Table of results for the simulation in A and V

4 Conclusion

The circuit studied is linear because it only has linear components. The Kirchhoff Laws only work because of this linearity, as do the two methods that we used, which are the circuit mesh and nodal analysis methods. The all-linear relationships have as consequence the validation of the Superposition Theorem. The superposition means that makes sense having multiple mesh currents circulating in only one element of the circuit, and multiple currents enables the use of the mesh currents as our independent variables, using all this to solve the circuit.

As for the second, the nodal analysis method, it's based on the KCL and this method is included in the *NGSpice* simulator that we used. This one is based on the thought of tension in the nodes, referring to the potential differences between 2 nodes of a circuit.

As we can see, the reason why both the methods and the *NGSpice* results accomplished are the same is because they are all based on the Kirchhoff's circuit Laws and the linearity of the circuit that allows us to use for example the Superposition Theorem.

In conculsion, the simulation and the theoretical analysis have similar results due to the linearity of the circuit.