Circuit Theory and Electronics Fundamentals

Integrated Masters in Aerospace Engineering, Instituto Superior Técnico, University of Lisbon

Hugo Tavares dos Santos, 86639

Ricardo Esteves Rodrigues, 95841

Víctor Negrini Liotti, 94774

March 24, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit composed of eleven branches and eight nodes, arranged in four independant meshes, as detailed bellow. This circuit has seven resistences, numbered from R_1 trough R_7, dependant current and volatage sources, V_C and I_B respectively, and independant current and voltage sources, V_A and I_D respectively. The study of the circuit will be subdivided in two major steps. In Section 2 we will analyse the circuit using the Mesh and Node Analysis. In Section 3 the circuit is going to be simulated with Ngspice, and we will compare the results of that simulation with the ones obtained in Section 2. Finally, the Conclusions of this assignment are detailed in Section 4.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed using the Mesh and Node Methods, after which both results are compared. Those methods are based upon the Kirchoff Laws, and the behaviour of the elements of the circuit, which are detailed bellow.
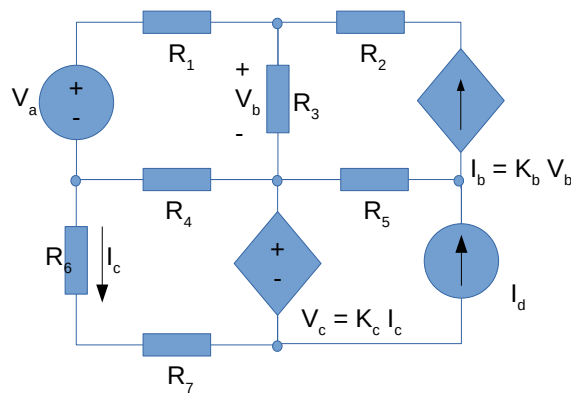
Kirchoff Laws

Figure 1: Circuit to be analyzed in the laboratory assignment.

The analysis of this circuit is based on the Kirchoff's Circuit Laws, more specifically, the Kirchoff's Current and Voltage laws (KCL and KVL). The first law states that for a given node, the sum of the electrical currents flowing out of the node is equal to the sum of currents flowing into that node. It can be translated into the following equation:

$$\sum_{k=1}^{n} I_k = 0. \tag{1}$$

The second law states that the sum of all the voltages in mesh must be zero, and can be translated into the following equation:

$$\sum_{k=1}^{n} V_k = 0. \tag{2}$$

Resistor

A resistor is an electrical component that poses resistance (as the name implies) to current flow, reducing it. The functioning of a resistor can be described by the following equation:

$$V = R * I \tag{3}$$

Voltage Source

- Imposes a voltage V regardless of current I

- Zero internal resistance

Current Source

- Imposes a current I regardless of voltage V

- Infinite internal resistance

For the simulation and analysis we will use the following values with the numerical values given:

$R_1 = 1.00781211614 \, k\Omega$ $R_2 = 2.00311223204 \, k\Omega$ $R_3 = 3.04503555589 \, k\Omega$ $R_4 = 4.17896607062$ $k\Omega$ $R_5 = 3.10615699135 \, k\Omega$ $R_6 = 2.06090154363 \, k\Omega$ $R_7 = 1.00634569025 \, k\Omega$ $V_a = 5.04864033546$ V $I_d = 1.02502620056$ mA $K_b = 7.05958243797$ mS $K_c = 8.03913881798 \, m\Omega$

In the simulation we used the following simplified circuit, which is equivalent of the one given:
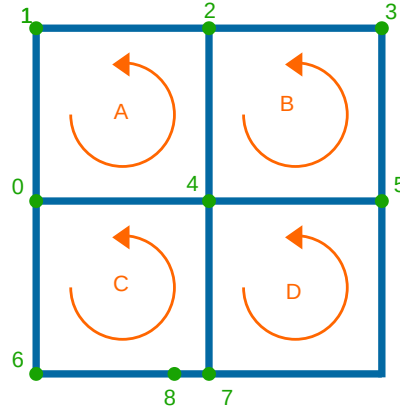
Figure 2: In this image you can see the number of each node as used for the simulation as well as the direction used for each mesh.

2.1 Mesh Method Analysis

After the aplication of the Mesh Method to the circuit, we obtainded the following equations:

$$I_D = I_d \tag{4}$$

$$(R_1 + R_3 + R_4)I_A - R_3 I_B - R_4 I_C = -V_A \tag{5}$$

$$(R_4 + R_6 + R_7 - K_C)I_C - R_4 I_A = 0 \tag{6}$$

$$(R_3 K_B - 1)I_B - R_3 K_B I_A = 0 \tag{7}$$

From which we can obtain the following matrix equation:

$$
\begin{bmatrix}
0 & 0 & 0 & 1 \\
R_1 + R_3 + R_4 & -R_3 & -R_4 & 0 \\
-R_4 & 0 & R_4 + R_6 + R_7 - K_C & 0 \\
-R_3 * K_B & R_3 * K_B - 1 & 0 & 0
\end{bmatrix}
\times
\begin{bmatrix}
I_A \\
I_B \\
I_C \\
I_D
\end{bmatrix}
=
\begin{bmatrix}
I_d \\
-V_A \\
0 \\
0
\end{bmatrix}
\tag{8}
$$

Finally, solving the problem with Octave we obtain:

Name	Value [mA]
I_a	-0.186554
I_b	-0.195655
I_c	0.983196
I_d	1.025026

Table 1: Mesh currents expressed in mA

3

2.2 Nodal Method Analysis

After the aplication of the Nodal Method to the circuit, we obtainded the following matrix, which in turn was obtained from the equivalent set of equations.

$$
\begin{bmatrix}
1 & 0 & 0 & 0 & 0 & 0 & 0 \\
-G_1 & (G_1 + G_2 + G_3) & -G_2 & -G_3 & 0 & 0 & 0 \\
0 & (G_1 + K_B) & -G_2 & -K_B & 0 & 0 & 0 \\
0 & K_B & 0 & (-G_5 - K_B) & G_5 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & (G_6 + G_7) & -G_7 \\
0 & -G_3 & 0 & (G_3 + G_4 + G_5) & -G_5 & -G_7 & G_7 \\
0 & 0 & 0 & 1 & 0 & (K_C * G_6) & -1
\end{bmatrix}
\times
\begin{bmatrix}
V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7
\end{bmatrix}
=
\begin{bmatrix}
V_A \\ 0 \\ 0 \\ I_D \\ 0 \\ -I_D \\ 0
\end{bmatrix}
\tag{9}
$$

Once again, solving the matrix equation with Octave we obtain:

Name	Value [V]
V_0	0.000000
V_1	5.048640
V_2	4.860629
V_3	4.468710
V_4	4.888344
V_5	8.679973
V_6	-2.026270
V_7	-3.015705

Table 2: Node voltages expressed in V

3 Simulation Analysis

For each resistor, the associated negative voltage node $< n- >$ was always defined as a number greater than the positive one $< n+ >$ based on Figure 2.

Table 3 shows the simulated operating point results for the circuit under analysis. Compared to the theoretical analysis results, no difference is perceived, which was to be expected, since for simple circuits of linear equations ngspice uses the same algorithm/method as we do for the analysis.

Name	Value [A or V]
@gb[i]	-1.95655e-04
@id[current]	1.025026e-03
@r1[i]	1.865537e-04
@r2[i]	1.956553e-04
@r3[i]	-9.10165e-06
@r4[i]	-1.16975e-03
@r5[i]	-1.22068e-03
@r6[i]	9.831960e-04
@r7[i]	9.831960e-04
v(1)	5.048640e+00
v(2)	4.860629e+00
v(3)	4.468710e+00
v(4)	4.888344e+00
v(5)	8.679973e+00
v(6)	-2.02627e+00
v(7)	-3.01571e+00
v(8)	-3.01571e+00

Table 3: Operating point. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt. Obs: 8 is the same node as 7 and it was created to place a fictitious 0V voltage source to aid the analysis

4 Conclusion

In this laboratory assignment we analysed the circuit indicated in Figure 1 successfully. This analysis was done both theoretically, making use of the Kirchoff's Laws and the Octave math tool to perform the calculations, and by circuit simulation, using the Ngspice tool. Unlike in theory, in real life we could expect several experimental errors, due to the measurement instruments, the way the measurement is performed, internal resistance of the components of the circuit, temperature, etc.

However, using the computer simulation, we came to the conclusion that the results we obtained matched the ones we calculated theoretically precisely. This is because this is a relatively simple circuit with few components. This corroborates with what was taught in theory classes, that theoretical and simulation models cannot differ; unless we are talking about more complex circuits, which is clearly not the case.

Finally, this lab assignment was useful to put to test our knowledge of circuits and the "laws" that they obey to and familiarize ourselves with sophisticated softwares, like Git, Makefile, Octave and Ngspice.