

# Circuit Theory and Electronics Fundamentals

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

## Laboratory Report

March 24, 2021

## Contents

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>Theoretical Analysis</b>	<b>2</b>
2.1	Mesh Method Analysis . . . . .	3
2.2	Nodal Method Analysis . . . . .	3
<b>3</b>	<b>Simulation Analysis</b>	<b>4</b>
3.1	Operating Point Analysis . . . . .	4
<b>4</b>	<b>Conclusion</b>	<b>4</b>

## 1 Introduction

The objective of this laboratory assignment is to study a circuit composed of eleven branches and eight nodes, arranged in four independent meshes, as detailed below. This circuit has seven resistances, numbered from  $R_1$  through  $R_7$ , dependent current and voltage sources,  $V_C$  and  $I_B$  respectively, and independent 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.

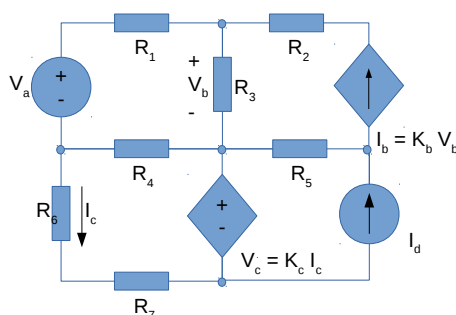


Figure 1: Circuit to be analyzed in the laboratory assignment.

## 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 Kirchhoff Laws, and the behaviour of the elements of the circuit, which are detailed below.

### Kirchoff Laws

The analysis of this circuit is based on the Kirchhoff's Circuit Laws, more specifically, the Kirchhoff'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:

$$\sum_{k=1}^n I_k = 0. \quad (1)$$

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

$$\sum_{k=1}^n V_k = 0. \quad (2)$$

[INSERIR FONTES]

### 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:

$$V = R * I \quad (3)$$

### 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

[DESCREVER PROGRAMA DO GNU OCTAVE]

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

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

## 2.1 Mesh Method Analysis

After the application of the Mesh Method to the circuit, we obtained the following equations:

$$I_D = I_d \quad (4)$$

$$(R_1 + R_3 + R_4)I_A - R_3I_B - R_4I_C = -V_A \quad (5)$$

$$(R_4 + R_6 + R_7 - K_C)I_C - R_4I_A = 0 \quad (6)$$

$$(R_3K_B - 1)I_B - R_3K_BI_A = 0 \quad (7)$$

From which we can obtain the following matrix equation:

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

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

## 2.2 Nodal Method Analysis

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

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

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

### 3 Simulation Analysis

#### 3.1 Operating Point Analysis

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]
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.