

## **Circuit Theory and Electronics Fundamentals**

### **Lab 1 - Circuit Analysis Methods**

#### **Aerospace Engineering**

Laboratory Report

March 24, 2021

Eva Claro, 95785

Miguel Isidoro, 95834

Pedro Braz, 95837

# Contents

<b>1</b>	<b>Introduction</b>	<b>3</b>
<b>2</b>	<b>Theoretical Analysis</b>	<b>5</b>
2.1	Mesh Analysis Method . . . . .	5
2.2	Nodal Analysis Method . . . . .	5
<b>3</b>	<b>Simulation Analysis</b>	<b>7</b>
3.1	Operating Point Analysis . . . . .	7
<b>4</b>	<b>Conclusion</b>	<b>11</b>

# 1 Introduction

This report is being made for the subject of Circuit Theory and Electronics Fundamentals and is related to the 1<sup>st</sup> laboratory being its objective to study a circuit containing seven resistors, one voltage source, one current source, one current controlled voltage source and one voltage controlled current source. The four elementary meshes are named after the current to which they are attributed.

The current controlled voltage source  $V_c$  is calculated by multiplying  $K_c$  with the current  $I_c$ , whereas the voltage controlled current source  $I_b$  can be determined by multiplying  $K_b$  with the voltage source  $V_b$ .

The display of this circuit can be seen in Figure 1.

In Section 2 it will be analysed theoretically the circuit by both Nodal and Mesh Method ending with the presentation of the results obtained by Octave.

Secondly, in Section 3 it will be simulated the circuit using ngspice, the results obtained will be presented and compared with the ones gathered from Section 2.

The conclusions of this study are outlined in Section 4.

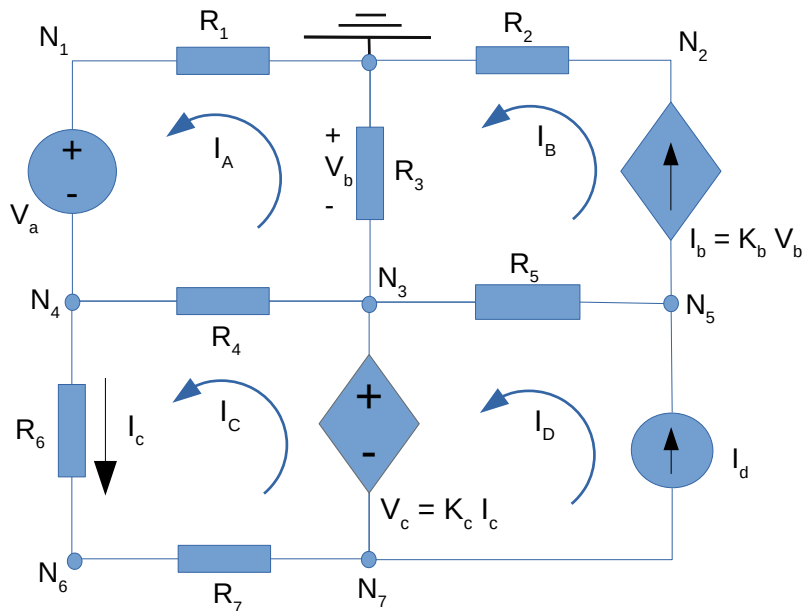


Figure 1: Circuit in analysis

Where:

$$R_1 = 1.03431507833$$

$$R_2 = 2.02853090731$$

$$R_3 = 3.1462050633$$

$$R_4 = 4.03438547455$$

$$R_5 = 3.12170042214$$

$$R_6 = 2.07116379646$$

$$R_7 = 1.01597753093$$

$$V_a = 5.156959346$$

$$I_d = 1.01455683569$$

$$K_b = 7.1497941196$$

$$K_c = 8.12593642585$$

The units of the elements whose name starts with R (the resistors) are  $k\Omega$  (kiloohm), the ones that start with I are expressed in  $mA$  (milliampere) and the ones starting with V are expressed in  $V$  (volts). While Kb is given in  $MS$  (milisiemens), Kc is also given in  $k\Omega$ .

These values were obtained using the Python script using the lowest student number on our group - 95785.

## 2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, in terms of its node voltages and mesh currents.

### 2.1 Mesh Analysis Method

For this mesh method, circular currents are defined in the counter-clockwise direction and then the circuit is evaluated considering those new currents.

Starting by number the nodes arbitrarily, assigning current names and directions to all branches also arbitrarily and defining one node as ground (GND).

Being mesh A the one with the resistors  $R_1$ ,  $R_3$  and  $R_4$ , and the voltage source  $V_a$ , the mesh B with the resistors  $R_2$ ,  $R_3$  and  $R_5$  and the voltage controlled current source  $I_b$ , the mesh C with the resistors  $R_4$ ,  $R_6$  and  $R_7$ , and the current controlled voltage source  $V_c$ , and, at last, the mesh D with the resistor  $R_5$ , the current source  $I_d$  and the current controlled voltage source  $V_c$ .

A system of equations obtained applying the Kirchhoff Current Law (KCL) to each mesh can be written as

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

It was used 3 equations (4 meshes - 1 = 3 linearly independent equations): Mesh A, Mesh C and an additional equation which is

$$I_b = K_b V_b, \text{ where } V_b = (I_B - I_A) R_3 \quad (2)$$

It's important to notice that D loop is independent of the remaining ones so it isn't don't need to determine the current  $I_D$  as it is given on the initial data.

### 2.2 Nodal Analysis Method

The Nodal Analysis Method is another general procedure for analysing circuits using node voltages as the circuit variables.

To find the nodal voltages we chose 7 equations (8 nodes - 1 = 7 linearly independent equations) that comprise:

- KCL in nodes not connected to voltage sources;
- Additional equations for nodes related by voltage sources.

It was used the equations regarding the nodes 0, 2, 5, 6 therefore it was necessary three additional equations.

We chose to put the ground zero between three branches corresponding to the ones with  $R_1$ ,  $R_2$  and  $R_3$  because it will facilitate the system of equations.

The next equation was used for node 1 because node 1 and node 4 are connected to an independent voltage source.

$$V_1 - V_4 = V_a \quad (3)$$

Secondly knowing that  $V_c = K_c I_c$  and  $V_c = V_3 - V_7$  it was concluded that for node 7 the equation obtained was

$$V_3 - V_7 = K_c (V_3 - V_6) G_6 \quad (4)$$

To find the third equation (Equation 5) it was used the continuity of the current to create a "super-knot" (nodes 3 and 7)

$$(V_4 - V_3)G_4 + (V_0 - V_3)G_3 + (V_5 - V_3)G_5 - I_d + (V_6 - V_7)G_7 = 0 \quad (5)$$

The system of equations that will be solved is:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G_1 - G_2 - G_3 & G_1 & G_2 & G_3 & 0 & 0 & 0 & 0 \\ K_b + G_2 & 0 & -G_2 & -K_b & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & -1 & 0 & 0 & 0 \\ -K_b & 0 & 0 & K_b + G_5 & 0 & -G_5 & 0 & 0 \\ 0 & 0 & 0 & 0 & G_6 & 0 & -G_6 - G_7 & G_7 \\ 0 & 0 & 0 & 1 & -K_c G_6 & 0 & K_c G_6 & -1 \\ G_3 & 0 & 0 & -G_4 - G_3 - G_5 & G_4 & G_5 & G_7 & -G_7 \end{bmatrix} \begin{bmatrix} V_0 \\ V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ V_a \\ -I_d \\ 0 \\ 0 \\ I_d \end{bmatrix} \quad (6)$$

### 3 Simulation Analysis

#### 3.1 Operating Point Analysis

The Table 2 shows the simulated operating point results for the circuit described in Figure 1.

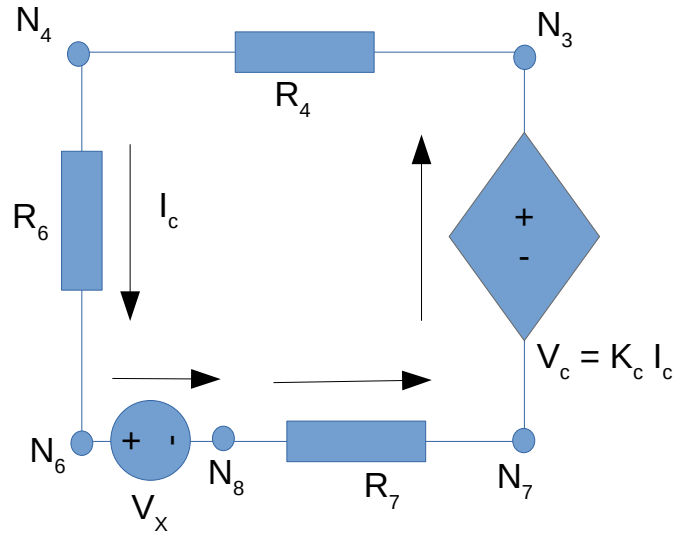


Figure 2: C Mesh with an adicional voltage source

In this simulation is important to explain the creation of an auxiliary voltage  $V_b$  (with a voltage equal to  $0V$ ) that was put between N6 and R7 as shown in Figure 4. Consequently, this led to the appearance of a node that we designated by N8 that has the same voltage as N6.

This was necessary because of Ngspice software requirements.

By observing the Table 2, Table ?? and Table ?? values it's possible to conclude that the simulated results are the same as the theoretical results.

However, was also calculated the relative errors made in order to understand the accuracy of the results.

Related to that calculations, it was noticed that in an experimental procedure, the calculation of relative errors is made by comparing experimental values and theoretical values, meaning that the decimal places used in the theoretical value to be considered will be in accordance with the experimentally obtained places.

Considering that, in our case, the experimental values are obtained through NGSice, we must use theoretical values with the same number of decimal places returned by the simulation for calculating the errors, as it has a number of decimal places lower than that of the octave. By doing this calculating, all error values absolute and relative are equal to zero.

Therefore, we can see that the order of magnitude of the errors will be residual.

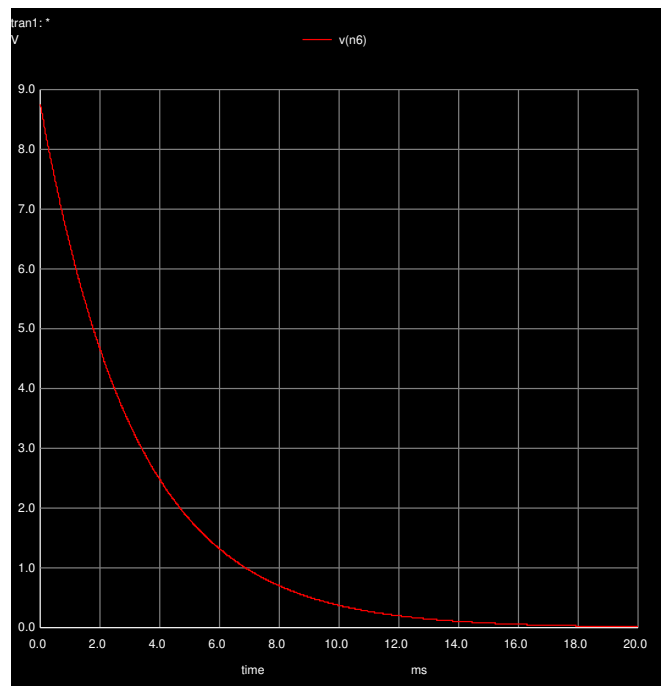


Figure 3: C Mesh with an adicional voltage source



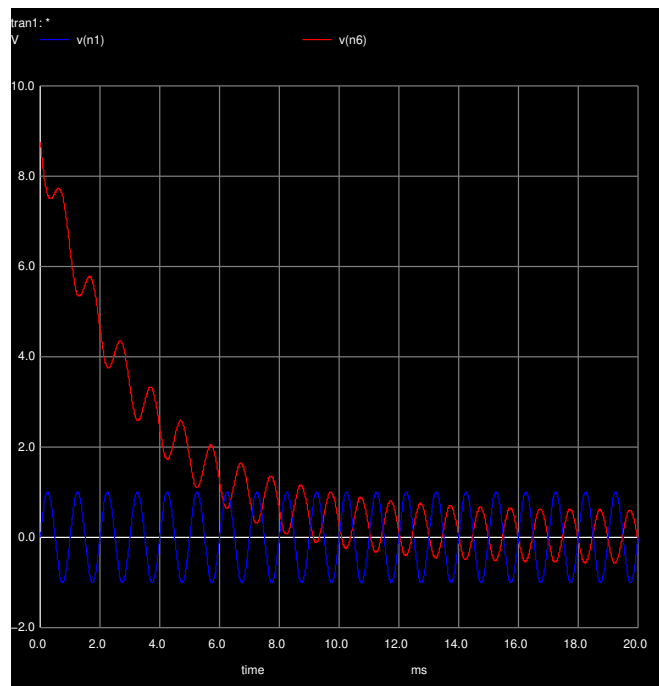


Figure 4: C Mesh with an adicional voltage source

Name	Value [A or V]
@vaux[i]	9.804367e-04
@hd[i]	-9.80437e-04
@vs[i]	-2.44092e-04
@c[i]	0.000000e+00
@gb[i]	-2.55448e-04
@r1[i]	-2.44092e-04
@r2[i]	-2.55448e-04
@r3[i]	-1.13559e-05
@r4[i]	-1.22453e-03
@r5[i]	-2.55448e-04
@r6[i]	9.804367e-04
@r7[i]	9.804367e-04
n1	5.156959e+00
n2	4.904492e+00
n3	4.386308e+00
n5	4.940220e+00
n6	5.737650e+00
n7	-2.03064e+00
n8	-3.02675e+00
n9	-2.03064e+00

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.

Name	Value [A or V]
@vaux[i]	4.336809e-19
@hd[i]	2.807571e-03
@vs[i]	4.001056e-18
@gb[i]	4.187197e-18
@r1[i]	4.001056e-18
@r2[i]	4.187197e-18
@r3[i]	1.861414e-19
@r4[i]	-8.80608e-19
@r5[i]	-2.80757e-03
@r6[i]	4.336809e-19
@r7[i]	-8.84098e-19
n1	0.000000e+00
n2	4.138353e-15
n3	1.263221e-14
n5	3.552714e-15
n6	8.764397e+00
n7	-8.98224e-16
n8	0.000000e+00
n9	-8.98224e-16

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.

## 4 Conclusion

The objective of this laboratory assignment is to analyse the circuit and solve it. After discussing with all members of the group we can conclude that this goal was achieved.

As presented the results obtained by the Octave math tool and Ngspice simulation tool are the same. This perfect match was achieved because the circuit is not very complex, being only composed of linear components so both models (Ngspice and Octave) used the same methods to solve the circuit and therefore the results can not differ.

Also, all the components used in this circuit (resistors, branches, nodes,...) are perfect this means they don't dissipate energy by heating. This is one of the advantages of simulating rather than doing it on the laboratory, the other one being the elimination of "human error". It's known that this type of error can influence the experimental results causing considerable relative errors, which in our case weren't made.

Finally, this similarity proves the efficiency and importance of the nodal and mesh methods.