Circuit Theory and Electronics Fundamentals

Integrated Master in Aerospace Engineering, Técnico, University of Lisbon

Lab1: Circuit analysis methods

March 22, 2021

João Maria Mendes, Maria Magalhães, Mariana Afonso

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit with four elementary meshes and eight nodes. This circuit contains an independent voltage source V_a, an independent current source I_d, a voltage source dependent from a current V_c and a current source dependent from a voltage I_b. Besides this, it has seven resistors, R_1, R_2, R_3, R_4, R_5, R_6 and R_7. The circuit can be seen in Figure 1.
In Section 2, a theoretical analysis of the circuit is presented, using the mesh method and the node method, in Octave. In Section 5, the circuit is analysed by simulation using ngspice, and the results are compared to the theoretical results obtained in Section 2. The conclusions of this study are outlined in Section 6.

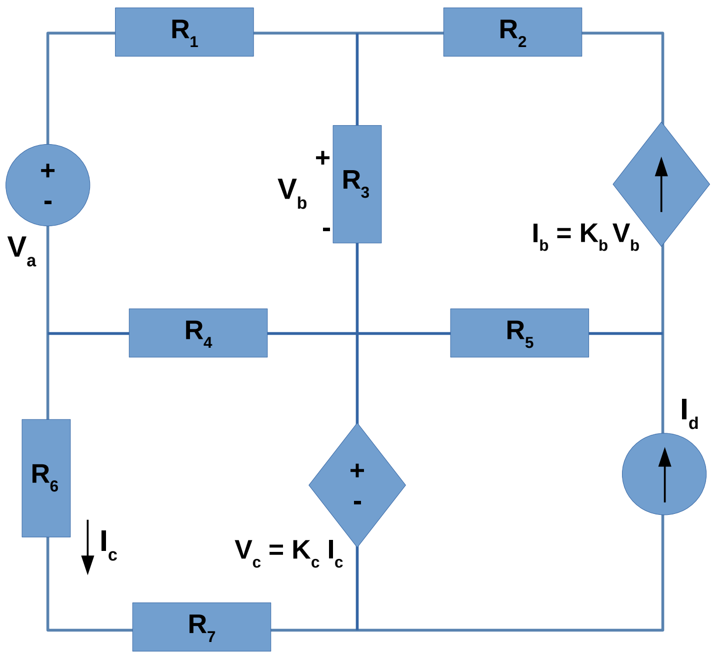

Figure 1: Circuit containing voltage sources, current sources and resistors.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, using the mesh method and the node method.
In the first one, we define a current for each elementary mesh, which is a loop containing no other loops, with an arbitrary direction. These currents will be our unknown values and the objetive is to compute them, using the Ohm's Law (V = RI) and Kirchhoff Voltage Law. Therefore, in this circuit we wil have four currents to compute, so we have to find four equations to solve this problem.
In the second one, each circuit node have a defined voltage. We also choose a node to have a null voltage. Then, we use Ohm's Law and Kirchhoff Current Law in nodes that are not connected to a voltage source. In the end, we write additional equations for the nodes related by voltage sources. Consequently, we will have an equal number of nodes and equations. To calculate the unkown voltages and currents, we use the following values (Resistances in kOhm, voltages in Volts, currents in mA, K_b in S and K_c in Ohms):

Table 1: Given values.

R_1	1,00332071212
R_2	2,04460853047
R_3	3,08291730437
R_4	4,16061678649
R_5	3,04022345043
R_6	2,06711403452
R_7	1,03302701196
V_a	5,13988034104
I_d	1,02475824097
K_b	7,0544535009 x 10^{-3}
K_c	8,16113797582 x 10^{3}

3 Mesh method

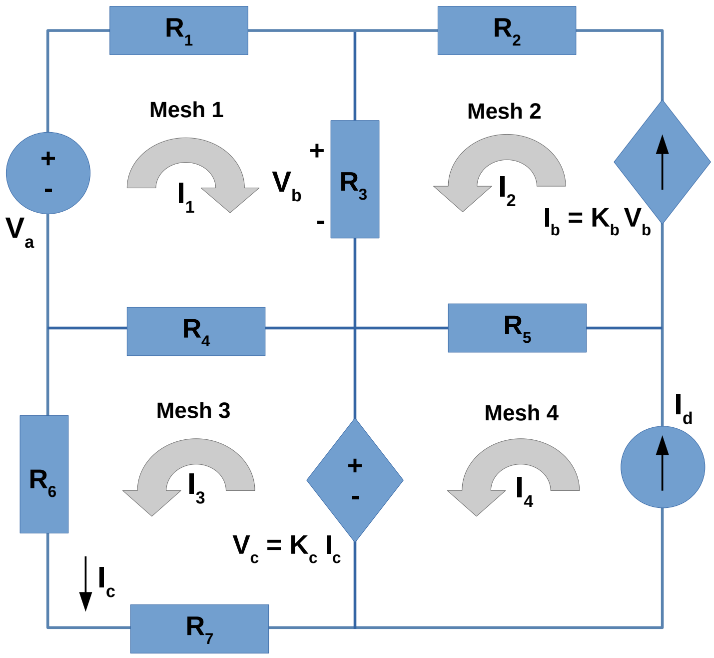

Figure 2: Circuit with current defined in each mesh.

In Figure 2, we defined a current for each circuit essencial loop. As said before, we will have four equations, because we need to compute four currents (I_1, I_2, I_3 and I_4). In this analysis, we assumed that the current in mesh one is flowing clockwise and the three other currents are flowing in the other direction. Once we calculate the values to each current with the KVL method, we know that the negative ones are not flowing in the direction we assumed.
Starting with mesh 3, we can see by inspection that $I_c = I_3$ and that R_3 depends on I_1 and I_3, which have the same direction. The other 3 components depend only on I_3. In the voltage source, the current flows to the positive side to the negative, so it is not aligned with the direction we selected. Hence, we must consider it negative. So we have this equation:

$$(I_3 + I_1)R_4 + I_3R_7 + I_3R_6 - K_cI_3 = 0 \tag{1}$$

In circuit analysis, we always consider the current flowing from the negative side to the positive, so in mesh 1, the voltage source is not aligned with the selected direction. For that reason, we consider V_a negative. The current in R_1 depends only in I_1, which we can see by inspection, and R_3 and R_4 depend on two different currents with the same direction, which is positive because we assumed the clockwise direction. So we have:

$$(I_1 + I_3)R_4 - V_a + R_3(I_2 + I_1) + R_1I_1 = 0 \tag{2}$$

We already have two equations, but we are going to need two additional ones, since we can not use this method in meshes with current sources. We can get these two equations by inspection of the two current sources. Starting with the dependent source I_b, we already know the relation between this source and V_b, so we only have to write this term as a function of resistances and currents. We already wrote V_b as a function of R_3, I_2 and I_1 at the third term of the previous equation for mesh 1. So we write it again:

$$I_2 = K_bR_3(I_2 + I_1) \tag{3}$$

Finishing with the simplest equation, we can look at the fourth mesh to verify that:

$$I_4 = I_d \tag{4}$$

All these equations can be transformed in a matricial system as it is shown here:

$$\begin{bmatrix} R_4 & 0 & R_6 + R_4 + R_7 - K_c & 0 \\ R_4 + R_3 + R_1 & R_3 & R_4 & 0 \\ K_b R_3 & -1 + K_b R_3 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ I_3 \\ I_4 \end{bmatrix} = \begin{bmatrix} 0 \\ V_a \\ 0 \\ I_d \end{bmatrix}$$

Solving this system using Octave and the values given, we have obtained this values:

Table 2: Meshes table.

Meshes currents	Values obtained (Ampers)
I_1	2,00
I_2	1,50
I_3	1,50
I_4	3,00

With this currents, we can discover node voltages and currents passing through each resistance, with the following equations, and knowing that V_c = $K_c I_3$:

$$I_b = I_2 \tag{5}$$

$$I_d = I_4 \tag{6}$$

$$R_1[i] = I_1 \tag{7}$$

$$R_2[i] = I_2 \tag{8}$$

$$R_3[i] = I_1 + I_2 \tag{9}$$

$$R_4[i] = I_1 + I_3 \tag{10}$$

$$R_5[i] = I_4 - I_2 \tag{11}$$

$$R_6[i] = I_3 \tag{12}$$

$$R_7[i] = I_3 \tag{13}$$

$$V_b = \frac{I_b}{K_b} \tag{14}$$

$$V_7 = V_c \tag{15}$$

$$V_6 = V_b + V_7 \tag{16}$$

$$V_5 = V_6 + R_2 I_2 \tag{17}$$

$$V_4 = V_7 + R_5(I_4 - I_2) \tag{18}$$

$$V_3 = R_7 I_3 \tag{19}$$

$$V_2 = V_3 + R_6 I_2 \tag{20}$$

$$V_1 = V_2 + V_a \tag{21}$$

4 Node method

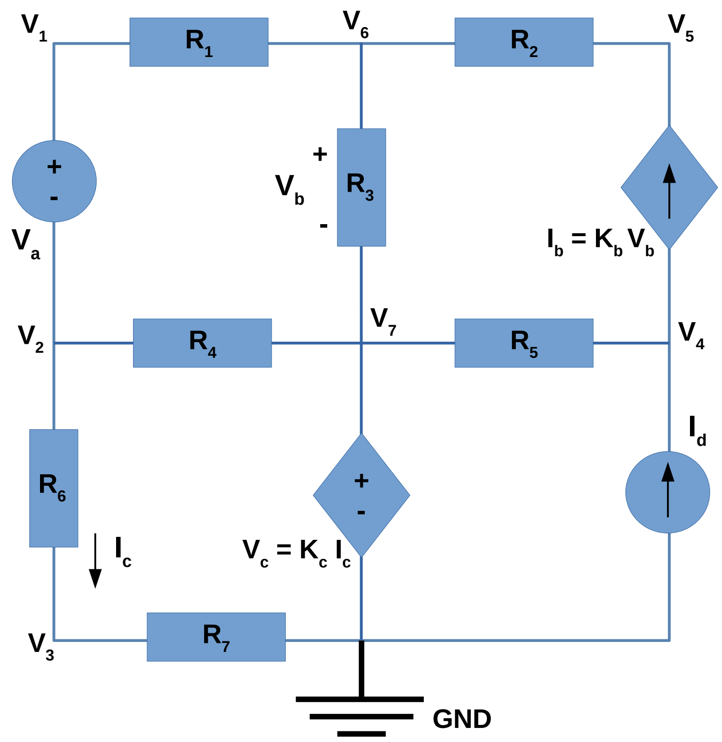

Figure 3: Circuit with voltages defined in each node.

In Figure 3, we defined a voltage for each circuit node and a node with null voltage. As said before, we will have seven equations, because we need to compute seven voltages (V_1, V_2, V_3, V_4, V_5, V_6 and V_7). In this analysis, we consider currents diverging from the node as positive values and currents converging as negative values, using KCL.
Starting with node 3, for this node, we considered all currents diverging, so we have this equation:

$$\frac{V_3 - V_2}{R_6} + \frac{V_3}{R_7} = 0 \tag{22}$$

In node 4, only I_d is converging. V_B is equal to V_6 - V_7, due to the direction represented in the circuit for R_3 voltage:

$$\frac{V_4 - V_7}{R_5} - I_d + K_b(V_6 - V_7) = 0 \tag{23}$$

Moving on to node 5, in which only I_b is converging, having this equation:

$$\frac{V_5 - V_6}{R_2} - K_b(V_6 - V_7) = 0 \tag{24}$$

On the other side, in node 6, we considered all currents diverging:

$$\frac{V_6 - V_7}{R_3} + \frac{V_6 - V_5}{R_2} + \frac{V_6 - V_1}{R_1} = 0 \tag{25}$$

The next equation establish the relation between nodes 1 and 2 voltages:

$$V_1 - V_2 = V_a \tag{26}$$

The relation between GND and node 7 is showed in the next equation:

$$V_7 = \frac{K_c(V_2 - V_3)}{R_6} \tag{27}$$

Since there is still an equation missing, we considered a super node containing V_a and R_6 and all currents diverging:

$$\frac{V_1 - V_6}{R_1} + \frac{V_2 - V_7}{R_4} + \frac{V_3}{R_7} = 0 \tag{28}$$

All this equations can be transformed in a matricial system as it is showed here:

$$\begin{bmatrix} 0 & -\frac{1}{R_6} & \frac{1}{R_6} + \frac{1}{R_7} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{R_5} & 0 & K_b & -K_b - \frac{1}{R_5} \\ 0 & 0 & 0 & 0 & \frac{1}{R_2} & -K_b - \frac{1}{R_2} & K_b \\ -\frac{1}{R_1} & 0 & 0 & 0 & -\frac{1}{R_2} & \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} & -\frac{1}{R_3} \\ 1 & -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{K_c}{R_6} & -\frac{K_c}{R_6} & 0 & 0 & 0 & -1 \\ \frac{1}{R_1} & \frac{1}{R_4} & \frac{1}{R_7} & 0 & 0 & -\frac{1}{R_1} & -\frac{1}{R_4} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \end{bmatrix} = \begin{bmatrix} 0 \\ I_d \\ 0 \\ 0 \\ V_a \\ 0 \\ 0 \end{bmatrix}$$

Solving this system using Ocatve and the values given, we have obtained this values:

Table 3: Nodes table.

Nodes Voltages	Values obtained (Volts)
V_1	8.17732927e+00
V_2	3.03744893e+00
V_3 1.01213679e+00	
V_4	1.17872793e+01
V_5	7.51018511e+00
V_6	7.96459596e+00
V_7	7.99610065e+00

Having discovered nodes voltages, we need now to obtain the currents flowing in each resistance and I_b, with the following equations:

$$I_b = K_b(V_6 - V_7) \tag{29}$$

$$R_1[i] = \frac{(V_6 - V_1)}{R_1} \tag{30}$$

$$R_2[i] = \frac{(V_5 - V_6)}{R_2} \tag{31}$$

$$R_3[i] = \frac{(V_6 - V_7)}{R_3} \tag{32}$$

$$R_4[i] = \frac{(V_2 - V_7)}{R_4} \tag{33}$$

$$R_5[i] = \frac{(V_4 - V_7)}{R_5} \tag{34}$$

$$R_6[i] = \frac{(V_2 - V_3)}{R_6} \tag{35}$$

$$R_7[i] = \frac{V_3}{R_7} \tag{36}$$

5 Simulation Analysis

5.1 Currents and Voltages Analysis

This section covers the circuit simulation in ngspice. However, for this simulation, a new node 8 and an auxiliary independent voltage source - V_8 - with a voltage of 0V had to be created. Node 8 is located between the resistance R6 and node 3, while the terminals of V_8 are connected to nodes 3 and 8. This happens because, to create a current dependent voltage source, ngspice needs the current value of a voltage source. As the current Ic doesn't flow through any voltage source, one had to be created.
Table 4shows the simulated operating point results for the circuit under analysis. A variable preceded by @ is of the type current and expressed in Ampere; other variables are of type voltage and expressed in Volt. The values obtained, by ngspice, for currents flowing in each resistance (Ampers) and nodes voltages (Volts) are showed in the following table:

Name	Value [A or V]
@gb[i]	-2.22248e-04
@id[current]	1.024758e-03
@r1[i]	-2.12029e-04
@r2[i]	-2.22248e-04
@r3[i]	-1.02191e-05
@r4[i]	-1.19181e-03
@r5[i]	1.247007e-03
@r6[i]	9.797777e-04
@r7[i]	9.797777e-04
v(1)	8.177329e+00
v(2)	3.037449e+00
v(3)	1.012137e+00
v(4)	1.178728e+01
v(5)	7.510185e+00
v(6)	7.964596e+00
v(7)	7.996101e+00
v(8)	1.012137e+00

Table 4: Operating point. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt. (g in "gib" refers to the Ngspice notation of a current controlled by a voltage source).

6 Conclusion

Comparing the theoretical analysis and the simulation, it can be concluded that both the values obtained in the nodal and mesh methods are similar to those obtained in the simulation.
Regarding the nodal analysis, the data obtained was the same as the one in the simulation. Such might happen because this is a systematic method preferred for circuit simulation programs such as ngspice. As for the mesh method, the values are also very similar to the ones generated in the simulation.
This similarity between theoretical analysis and simulation by ngspice can also be related to the low complexity of the considered circuit and to the fact that it only contains linear components. Taking all of this into account, the theoretical analysis showed coherent values relative to the simulation, proving Node and Mesh Methods to be reliable for the analysis of this circuit.