

Circuit Theory and Electronics Fundamentals - T1

João D. Álvares (96541)¹, João M. Teixeira (96544)¹, and Rui M. Martins (96565)¹

¹*Instituto Superior Técnico, Av. Rovisco Pais 1, 1049-001 Lisboa*

24th March 2021

Abstract

EN: In this report, we show a concise analysis of a 4 single mesh circuit through mesh and nodal analysis. Hitherto Ngspice has been considered a good circuit analyser and thus chosen it was to run the same circuit for comparison with the aforementioned theoretical analysis.

PT: Neste relatório, mostramos uma análise concisa de um circuito composto por 4 malhas através das análises nodal e de malhas. Até ao dia presente, o Ngspice é considerado um bom simulador de circuitos e, por esse mesmo motivo, foi escolhido de forma a poder-se comparar com a supramencionada análise teórica.

Contents

1	Introduction	1
2	Theoretical Analysis	1
2.1	Mesh Analysis	1
2.2	Nodal Analysis	3
3	Simulation Analysis	4
4	Conclusion	6

1 Introduction

The main goal of this laboratory assignment is to explore different circuit analysis methods and to compare the results obtained with the results given by the *Ngspice* simulation. For the theoretical analysis, the circuit will be analysed using mesh and nodal methods (both result from the Kirchoff's Laws) - and the equations resulting from these methods will be solved using *Octave*. On the other hand, the circuit simulation will be made, as said above, with *Ngspice*. The analysed circuit can be seen in Fig. 1.

As a starting point to solve this circuit, some values of the symbolic variables presented in Fig. 1 were given and are presented in Table 1:

Resistors [$k\Omega$]		Currents [mA]	
R_1	1.03994439216	I_d	1.01674167773
R_2	2.07923431764	Voltages [V]	
R_3	3.06168544529	V_a	5.03847501972
R_4	4.09516986362	Dependent sources constants	
R_5	3.00136467001	K_b	7.01505323139 mS
R_6	2.03324628446	K_c	8.37372457746 $k\Omega$
R_7	1.02216788331		

Table 1: Summary table with all the known values at the starting point

2 Theoretical Analysis

2.1 Mesh Analysis

This circuit analysis method results from the application of Kirchhoff's Voltage Law (KVL) and it is based on loop currents flowing around meshes. The analysis is performed with a certain sequence

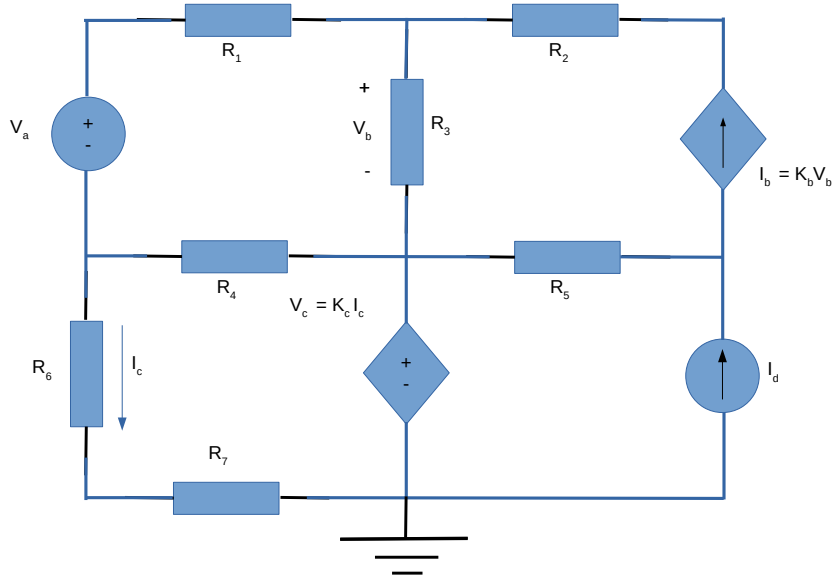


Figure 1: *Circuit analysed. It consists, essentially, in a circuit with 4 essential meshes, 8 nodes and 11 branches (in which we count 7 resistors, 2 voltage sources - one of them controlled by the current I_C - and 2 current sources - one independent and one dependent, controlled by the voltage V_b as it can be seen in the figure. The labels used in this figure will be used throughout this report.*

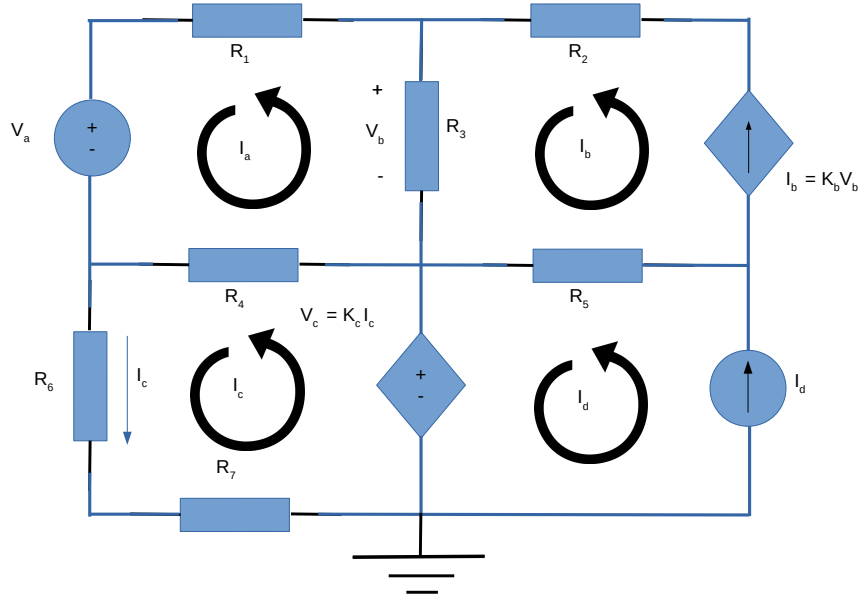


Figure 2: *Mesh analysis scheme for this circuit that shows the adopted current direction for each mesh*

of steps. First, we start by identifying the meshes and assigning a current to each of those, as it can be seen in Fig. 2.

After that, the next step is to write linearly independent Kirchhoff's Voltage Law equations for each mesh that allows finding the unknown current values. For this circuit, as I_d is known, and following the current directions represented in Fig. 2, we obtained the following system of equations:

$$\begin{cases} (R_1 + R_4)I_a - (\frac{1}{K_b})I_b - R_4I_c = -V_a & \text{(mesh A)} \\ -R_4I_a + (R_4 + R_6 + R_7 - K_c)I_c = 0 & \text{(mesh B)} \\ R_3I_a + (\frac{1}{K_b} - R_3)I_b = 0 & \text{(restriction involving meshes A and B)} \end{cases} \quad (1)$$

Note that the restriction equation used as the third equation of the system of equation urges as a result of the resistor R_3 terminals voltage. One can write the voltage of the resistor R_3 as being defined by

$$V_b = R_3(I_b - I_a) \quad (2)$$

Furthermore, it is known from the equation associated with the dependent current source in mesh B, that the voltage V_b can be defined as

$$V_b = \frac{I_b}{K_b} \quad (3)$$

In fact, combining equations (2) and (3) one can obtain:

$$R_3(I_b - I_a) = \frac{I_b}{K_b} \iff R_3 I_a + \left(\frac{1}{K_b} - R_3 \right) I_b = 0 \quad (4)$$

Converting this system of equations in its matricial form and using *Octave* to solve it, we get that:

$$\begin{bmatrix} R_1 + R_4 & -\frac{1}{K_b} & -R_4 \\ -R_4 & 0 & R_4 + R_6 + R_7 - K_c \\ R_3 & \frac{1}{K_b} - R_3 & 0 \end{bmatrix} \begin{bmatrix} I_a \\ I_b \\ I_c \end{bmatrix} = \begin{bmatrix} -V_a \\ 0 \\ 0 \end{bmatrix} \iff \begin{bmatrix} I_a \\ I_b \\ I_c \end{bmatrix} = \begin{bmatrix} -0.26949 \\ -0.28265 \\ 0.90226 \end{bmatrix} mA \quad (5)$$

From this result, it is possible to infer that, according to the current directions assigned and represented in Fig.2, I_a and I_b flow in the clockwise direction while I_c and I_d (with positive signs) flow in the counterclockwise direction. To finish the mesh analysis of this circuit, we just need to use the current values obtained in (5) to calculate what is still left unknown in the circuit:

$$V_c = K_c I_c \iff V_c = 7.5553V \quad (6)$$

$$V_b = (I_b - I_a)R_3 = -0.040292V \quad (7)$$

2.2 Nodal Analysis

Let $\vec{b} = [V_1, V_2, V_3, V_4, V_5, V_6, V_7, I_a, I_c]$, such that $[V_i] = V$, $[I_i] = A$. For the nodes, we used the ones given in Fig. 3, as well as the currents there presented.

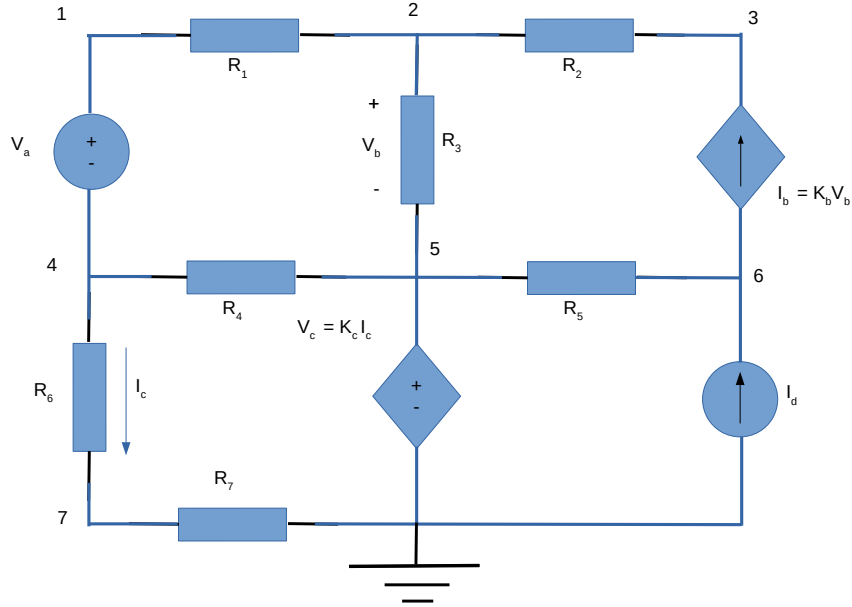


Figure 3: Nodal analysis scheme for this circuit that shows the labels given to each of the circuit's nodes

Thus, for each node, we get the following linearly independent equations (as it was supposed, we obtained 7 linearly independent equations for a circuit with 8 nodes),

$$\begin{cases} G_1(V_1 - V_2) + I_a = 0 & \text{(node 1)} \\ G_1(V_2 - V_1) + G_2(V_2 - V_3) + G_3(V_2 - V_5) = 0 & \text{(node 2)} \\ G_2(V_3 - V_2) - K_b(V_2 - V_5) = 0 & \text{(node 3)} \\ G_4(V_4 - V_5) + G_6(V_4 - V_7) - I_a = 0 & \text{(node 4)} \\ G_3(V_5 - V_2) + G_4(V_5 - V_4) + G_4(V_5 - V_6) - I_c = -I_d & \text{(node 5)} \\ G_5(V_6 - V_5) + K_b(V_2 - V_5) = I_d & \text{(node 6)} \\ G_6(V_7 - V_6) + G_7(V_7 - V_8) = 0 & \text{(node 7)} \end{cases} \quad (8)$$

Note that node 8 will have null voltage, $V_8 = 0$, because ground (GND) was assigned to it. Furthermore, by inspection (and it is pretty clear from the scheme presented in Fig.3), one can also conclude that:

$$\begin{cases} V_1 - V_4 = V_a \\ V_5 - K_c I_c = 0 \end{cases} \quad (9)$$

Thus, having as many equations as variables, one can now solve this system (10), using *Octave*.

$$\begin{bmatrix} G1 & -G1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\ -G1 & G1 + G2 + G3 & -G2 & 0 & -G3 & 0 & 0 & 0 & 0 \\ 0 & -G2 - Kb & G2 & 0 & Kb & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & G4 + G6 & -G4 & 0 & -G6 & -1 & 0 \\ 0 & -G3 & 0 & -G4 & G3 + G4 + G5 & -G5 & 0 & 0 & -1 \\ 0 & Kb & 0 & 0 & -G5 - Kb & G5 & 0 & 0 & 0 \\ 0 & 0 & 0 & -G6 & 0 & 0 & G6 + G7 & 0 & 0 \\ 1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & -Kc \end{bmatrix} \vec{b} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ -I_d \\ I_d \\ 0 \\ V_a \\ 0 \end{bmatrix} \quad (10)$$

The matrix system 10 is composed of a very sparse matrix, thus we also advise using correspondingly adapted sparse matrix solvers if possible, but this is already out of the scope of this discipline and laboratory. Getting back to the point, the solution to the matrix system is,

$$\vec{b} = \begin{bmatrix} 7.79526 \\ 7.51501 \\ 6.92732 \\ 2.75679 \\ 7.5553 \\ 11.4552 \\ 0.922264 \\ -0.000269487 \\ 0.000902263 \end{bmatrix} \quad (11)$$

which produces, $V_c = 7.5553V$, $V_b = -0.0402915V$, as expected.

3 Simulation Analysis

To be sure of the values obtained in 2, we made use of Ngspice, a more stable version of the Berkeley SPICE, which is an "open source spice simulator for electric and electronic circuits", as stated in [1]. The code used was the following,

```
TCFELab1
.options savecurrents

Vin 1 4 5.03847501972
Id 0 6 1.01674167773m
Vdumb CFP1 7 0

R1 2 1 1.03994439216k
R2 3 2 2.07923431764k
R3 2 5 3.06168544529k
R4 4 5 4.09516986362k
R5 5 6 3.00136467001k
R6 4 CFP1 2.03324628446k
R7 7 0 1.02216788331k

Gb 6 3 2 5 7.01505323139m
Hc 5 0 Vdumb 8.37372457746k

.op
.control
run
print all
.endc
```

.end

The first line of this code is the title of the simulation itself, which is followed by the option to save currents, a command that allows the program to, when printing all the variables, also print the currents flowing in every component of the circuit. Then, V_{in} and I_d are the V_a , I_d corresponding to what is shown in Fig. 3. V_{dumb} is a dumb voltage source that was introduced for a specific reason that will be later explained. In this simulation, we chose node 8 to be GND, which, by Ngspice standards, has to be named node 0.

After this, all the resistances were declared in the form $R<name> <n+> <n-> <value>$, where n+ and n- mean the positive and negative nodes of the electronic component, according to the schematics shown in Fig.4.

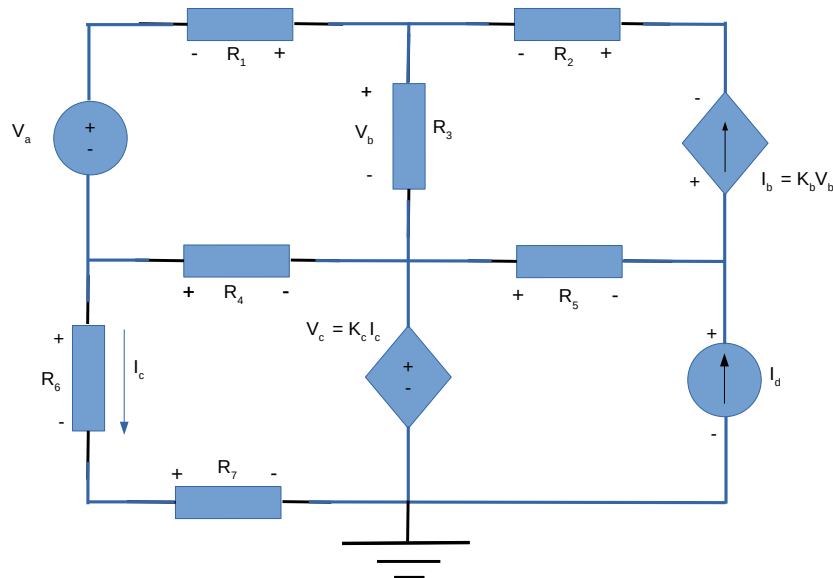


Figure 4: Representation of the convention used to the Ngspice simulation. In the figure above, one can see the conventioned positive and negative terminal for each branch

Afterwards, G_b and H_c stand for the voltage-controlled current source I_b and current-controlled voltage source V_c respectively. The first two parameters received by G_b are its positive and negative nodes as expected and then the two nodes where it should calculate the corresponding potential. For H_c , the third parameter is V_{dumb} which, as promised, is a voltage source that was introduced for the single purpose that current-controlled voltage sources in *Ngspice* take as a third parameter only voltage sources through which the current in question is flowing through. Thus, because the purpose is to retrieve I_c , a dumb voltage source was added after R_6 , creating a new node that was called CFTP1, which is where R_6 then connects. To not alter the circuit at all, the actual induced voltage by V_{dumb} is 0V, thus CFTP1 and node 7 are in short circuit, i.e., the circuit does not "see" V_{dumb} . At last, inside *control*, the simulation is ran and everything is printed out, giving the following output:

Name	Current/Voltage [A/V]
@gb[i]	-2.82647e-04
@id[current]	1.016742e-03
@r1[i]	-2.69487e-04
@r2[i]	-2.82647e-04
@r3[i]	-1.31599e-05
@r4[i]	-1.17175e-03
@r5[i]	-1.29939e-03
@r6[i]	9.022631e-04
@r7[i]	9.022631e-04
v(1)	7.795262e+00
v(2)	7.515011e+00
v(3)	6.927322e+00
v(4)	2.756787e+00
v(5)	7.555303e+00
v(6)	1.145524e+01
v(7)	9.222643e-01
cfp1	9.222643e-01

Table 2: Values for several variables declared in the .net file.

The values obtained were expected in some way given to the results obtained in the theoretical analysis and that were presented in the last section.

However, due to *machine epsilons* and related things, some decimal places might be a bit different. The main cause of this is that Ngspice also uses, if possible, nodal analysis. Thus, for solving matrix 10, one has to be aware that inverting a 9x9 or 7x7 matrix will introduce a significant margin of error, which explains the registered difference in the results obtained.

4 Conclusion

The goal of this assignment was to analyse the circuit using different methods. Theoretically, they were used nodal and mesh methods, that result from the application of Kirchhoff's Laws (current and voltage, respectively). It was also used *Ngspice*, a program that allowed us to simulate the circuit and to obtain the different currents and voltages in it, although this program also uses the theoretical methods used and Kirchhoff's laws in its bases.

The mesh analysis provides directly the fictional current, being easy to infer any current on the circuit. On the other hand, nodal analysis allows to determine the voltage in every circuit node and one can confirm that the methods are equivalent by comparing the voltage values V_c and V_b from each one and the value of V_c . Doing that, one can prove that the values were coincident in all the provided digits and V_b only diverges from one another in the fifth non-zero digit (0.0025%). Furthermore, another pair of values obtained in both methods is I_a and I_c which one can verify that match at least 5 decimal places, as well.

Being both theoretical methods used very consistent with each other, we already had expected values for each current and voltage that one would obtain from Ngspice's output. Unsurprisingly, those values matched with the obtained values from Ngspice with almost every shown digit being the same as in the first two methods. As said before, the small differences registered in these values may be justified with errors of numeric nature such as in matrix solving algorithms (that for this problem in question could involve inverting big matrices, which can introduce some margin of error) or just the *machine epsilon* of the program(s).

That being said, it was nothing more than expected since the voltages and currents in this circuit are all constant and the relationship between every component is linear so there was no reason for the simulation to not match the theoretical results. Therefore, and concluding, it can be stated that the goals for this laboratory were successfully achieved.

References

- [1] Ngspice official website, <http://ngspice.sourceforge.net/>