Circuit Theory and Electronics Fundamentals

MEAer (Integrated Master In Aerospace Engineering), Técnico, University of Lisbon

Laboratory 2: RC Circuit analysis

Group 3

Diogo Faustino, nº95782
Henry Machado, nº95795
Rúben Novais, nº95843

April 5, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study the behaviour of a circuit containing 4 independent meshes and a total of 11 branches: an independent voltage source V_A, an independent current source I_D, a dependent current source I_B, and a dependent voltage source V_C, all connected to 7 resistances (from R_1 through to R_7). In order to determine the electrical variables (current and voltage) of each of it components, we will use different methods to determine these variables and we will compare the theoretical with experimental data. The circuit can be seen in Figure 1.

In Section 2, a theoretical analysis of the circuit using both the mesh method and the node method is presented. In Section 3, the circuit is analysed by simulation, and the results in operating point are compared to the theoretical results obtained in Section 2. The conclusions of this study are laid out in Section 4.

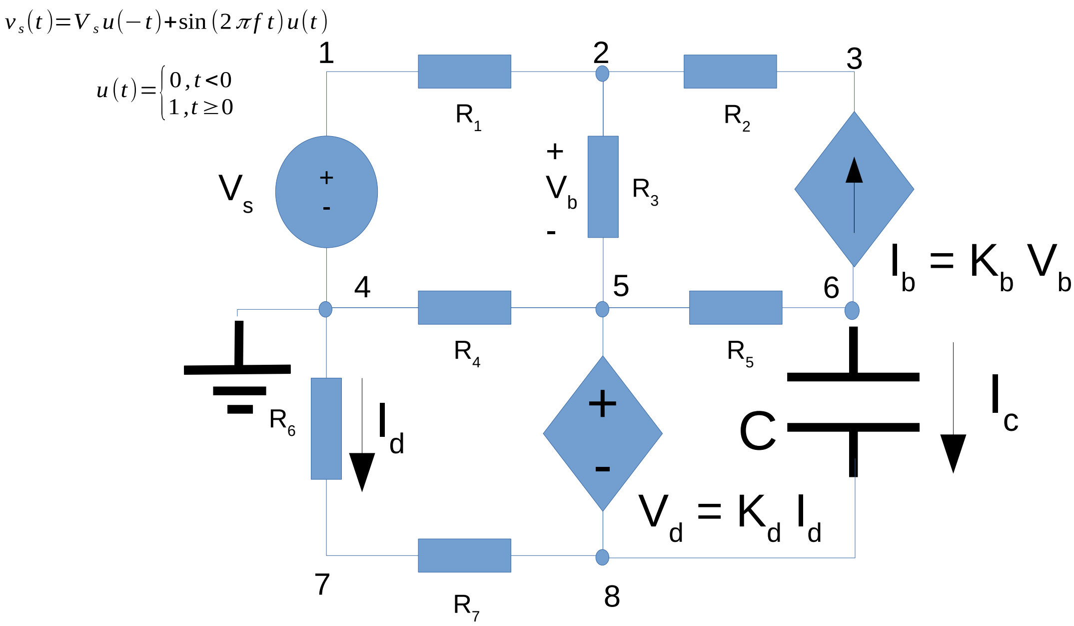

Figure 1: Circuit with linear components.

2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, using both mesh and node analysis. Here, we lay out the background information that was essential for the execution of this assignment. Since the circuit at hand is a simple one we only needed the following information (this is all referenced in Horowitz, P; Hill,W(2015). The Art of Electronics. Cambridge University Press(3rd ed, pp.2-4)): The sum of the currents into a point in a circuit equals the sum of the currents out. Each of these points is referred to as a node. This is also known as Kirchhoff's current law (KCL). The sum of the voltage drops around any closed circuit is zero.This is Kirchhoff's voltage law (KVL). The power consumed by a circuit device is

$$P = VI; \tag{1}$$

When voltages and currents are in the same direction they produce a positive power (the component is consuming energy). Voltages and currents in opposite direction produce negative power (the component is giving forth energy to the circuit) A resistor is characterized by its resistance:

$$R = \frac{V}{I}; \tag{2}$$

This is known as Ohm's law. Independent V/I sources impose voltage/current regardless of current/voltage. Linearly dependent V/I sources impose voltage/current that is linearly dependent on a specific variable of other circuit component.

The circuit consists of four independent meshes, and 11 branches where different currents circulate. These will be our variables in the mesh analysis. The current flow is depicted in Figure 2

The components we are working with have a linear relationship between the voltage and current, therefore we will need to solve a system of linear equations to determine the variables of each component given some initial values. We will use two different methods that are based on KCL and KVL. These methods are the Mesh and Node methods. Each one of them will produce its set of equations that when solved, will give us either the currents flowing through each mesh (Mesh Method) or the voltage on each node (Node Method). We've written a program on GNU Octave to help us determine the solution for the linear system. The simulation of the circuit was carried out by the Ngspice tool. We've written a simple code describing the circuit.

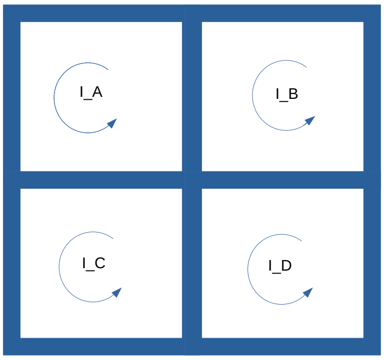

Figure 2: Direction of each mesh current.

2.1 Mesh Analysis

Applying the Kirchhoff Voltage Law (KVL) in the different loops, we get four different equations, which we can then solve as a matrix:

$$I_D = I; \tag{3}$$

$$(R_1 + R_3 + R_4)I_A - R_3 I_B - R_4 I_C = -V_A; \tag{4}$$

$$(R_4 + R_6 + R_7 - K_C)I_C - R_4 I_A = 0; \tag{5}$$

$$(R_3 K_B - 1)I_B - R_3 K_B I_A = 0. \tag{6}$$

$$\begin{pmatrix} 0 & 0 & 0 & 1 \\ R1+R3+R4 & -R3 & -R4 & 0 \\ -R4 & 0 & R4+R6+R7-Kc & 0 \\ -R3Kb & R3Kb-1 & 0 & 0 \end{pmatrix} \begin{pmatrix} Ia \\ Ib \\ Ic \\ Id \end{pmatrix} = \begin{pmatrix} I \\ -V \\ 0 \\ 0 \end{pmatrix} \tag{7}$$

Name	**Value [A]**
$@I_a$	0.000834
$@I_b$	0.000875
$@I_c$	-0.000330
$@I_d$	0.000000

Table 1: Theoretical analysis results from Octave. (A variable preceded by @ is of type *current*)

2.2 Node Analysis

As in the previous section, we extract a set of equations from the circuit, this time using the node method. This yields 8 equations, one for each node. Then, we can solve those 8 equations in the form of the matrix presented here.

$$\begin{pmatrix} G7 & 0 & 0 & 0 & 0 & 0 & G6 & 0 \\ 1 & -1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & 0 & -G2 & G1+G2+G3 & -G1 & 0 & -G3 \\ 0 & 0 & G5 & 0 & G3 & 0 & G4+G6 & -G3-G4-G5 \\ 1 & 0 & 0 & 0 & 0 & 0 & Kc*G6 & -1 \\ 0 & 0 & 0 & 0 & G1 & -G1 & -G4-G6 & G4 \\ 0 & 0 & 0 & -G2 & Kb+G2 & 0 & 0 & -Kb \end{pmatrix} \begin{pmatrix} V1 \\ V2 \\ V3 \\ V4 \\ V5 \\ V6 \\ V7 \\ V8 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ V \\ 0 \\ I \\ 0 \\ 0 \\ 0 \end{pmatrix} \tag{8}$$

Name	Value [V]
V_1	5.113399
V_2	4.260917
V_3	2.507106
V_4	-0.000000
V_5	4.382714
V_6	7.045254
V_7	-0.667990
V_8	-1.000947

Table 2: Theoretical analysis results from Octave.

3 Simulation Analysis

Since this circuit is a steady one, the voltage or current values of the various components doesn't vary in time. Therefore, only the Operating Point Analysis is necessary in this circuit simulation

3.1 Operating Point Analysis

For this circuit's simulation on Ngspice, there was a need to introduce a null voltage source between the ground node and the R7 resistor. The voltage related to node V2 will not appear in following table 3 because it has the same voltage as node V1 (its omission is necessary for the simulation to run correctly).

Table 3 below shows the simulated operating point results for the circuit under analysis.

Using the results of the theoretical analysis for mere guidance, we know the voltage values for the nodes: that should follow from our simulation. We know then in which direction the voltage drops happen and we use this order for all branches except the current sources. In the current sources the order follows the current flow of such sources, as it is norm in Ngspice. Following the order we stated for the other branches means that the current that flows through every resistor must have a positive value in table 3, because in resistors the voltage drop and current flow have the same direction (resistors always consume energy).

Analysing table 3, we notice that the current flowing through every resistor has a positive value, as it should be. I_a is the current flowing through R1, I_b is the current flowing through R2, I_c is the current flowing through R6 (and R7), and, finally, I_d is equivalent to Idd. Comparing the simulation results for the voltage and current values with the theoretical analysis values, we notice that the values for voltage in nodes V_1 to V_9 and the current for all four mesh currents are exactly equal, if we exclude all roundings carried out by NgSpice, since its precision (up to 7 significant figures) can be slightly different than the precision we used in GNU Octave (a maximum of 2 significant figures of difference). Different digits only start to appear by the 6^{th} decimal case, which is a disposable calculation error (Shrinking the theoretical results to Ngspice precision, we get equal theoretical and simulation results).

With that being said, the theoretical results are equal to the simulation results (the complete accuracy is fruit of the staticness of the circuit), which confirms our theoretical analysis. The fact that the voltage and current results are equal obviously results in equal power values for each branch.

Name	Value [A or V]
@gb[i]	-2.29771e-04
@idd[current]	1.005042e-03
@r1[i]	2.191669e-04
@r2[i]	2.297712e-04
@r3[i]	1.060424e-05
@r4[i]	1.185502e-03
@r5[i]	1.234813e-03
@r6[i]	9.663347e-04
@r7[i]	9.663347e-04
v(1)	-9.73914e-01
v(3)	1.063433e+01
v(4)	6.382611e+00
v(5)	6.843347e+00
v(6)	7.067298e+00
v(7)	1.953900e+00
v(8)	6.875344e+00
v(9)	0.000000e+00

Table 3: Operating point analysis. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

4 Conclusion

In this laboratory assignment the objective of studying the presented circuit has been achieved. All the calculations for the Mesh and Node methods have been carried out using the Octave Maths tool and the circuit simulation has been done using the Ngspice tool. In a real, presential lab class, all sorts of experimental errors would be present: the resistance of the wiring, internal resistance of the sources, the temperature, external noise, etc. Besides that, reading and assembly errors would be expected. In this case, the simulation results matched the theoretical results: the reason for this match is the fact that this is a straightforward circuit containing only linear components, so if done correctly, the theoretical and simulation models cannot have noticeable differences. For more complex components, the theoretical and simulation models could differ greatly but this is not the case in this work. Despite the proximity of the values, there is still some difference between them (around the 6th decimal place). This can be considered negligible for the experiment in case, which doesn't require a higher degree of accuracy since the whole purpose is just to study a simple circuit (it could be a problem if this belonged to a bigger functional structure where every small discrepancy between the expected and real data mattered). Nevertheless, these differences may have been caused by the employment of different numerical methods for solving the linear system and for different rounding criteria employed by the two different tools.