

## Circuit Theory and Electronics Fundamentals

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

Laboratory 1: Circuit analysis methods

Group 3

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

March 22, 2021

### Contents

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>Theoretical Analysis</b>	<b>2</b>
<b>3</b>	<b>Mesh Analysis</b>	<b>2</b>
<b>4</b>	<b>Simulation Analysis</b>	<b>3</b>
4.1	Operating Point Analysis . . . . .	4
<b>5</b>	<b>Conclusion</b>	<b>5</b>

### 1 Introduction

The objective of this laboratory assignment is to study 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  $R_7$ ). 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 4, 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 5.

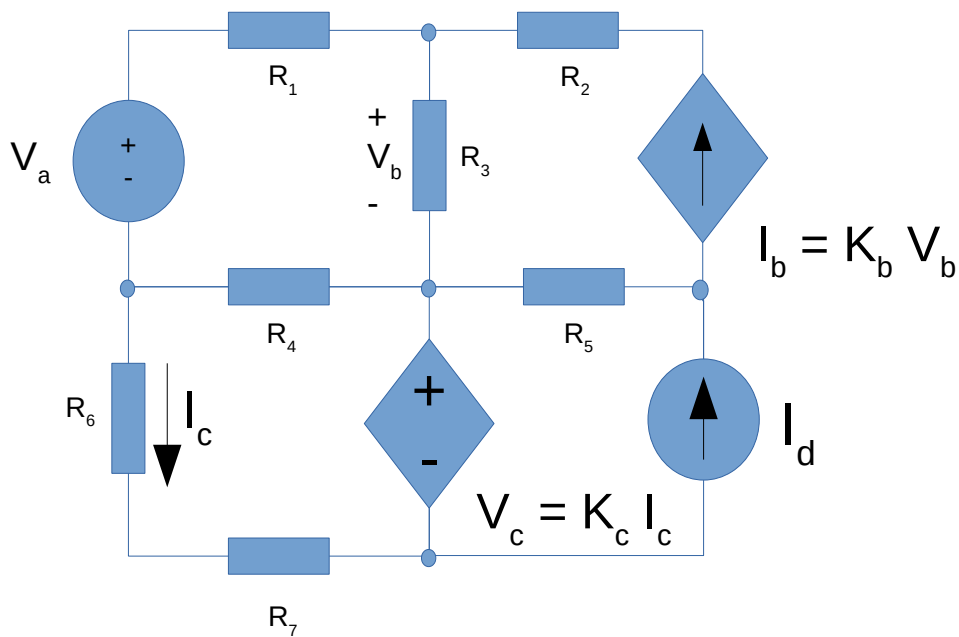


Figure 1: Circuit with linear components.

## 2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically, using both mesh and node analysis.

## 3 Mesh Analysis

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

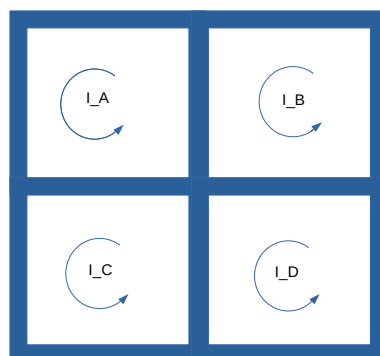


Figure 2: Direction of each mesh current.

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

$$I_D = I; \quad (1)$$

$$(R_1 + R_3 + R_4)I_A - R_3I_B - R_4I_C = -V_A; \quad (2)$$

$$(R_4 + R_6 + R_7 - K_C)I_C - R_4I_A = 0; \quad (3)$$

$$(R_3K_B - 1)I_B - R_3K_BI_A = 0. \quad (4)$$

$$v_O(t) = v_{On}(t) + v_{Of}(t). \quad (5)$$

As learned in the theory classes the natural solution is of the form

$$v_{On}(t) = Ae^{-\frac{t}{RC}}, \quad (6)$$

where  $A$  is an integration constant.

$$V_{Of}(t) = |\bar{V}_{Of}| \cos(\omega t + \angle \bar{V}_{Of}), \quad (7)$$

Name	Value [A]
@ $I_a$	-0.000219
@ $I_b$	-0.000230
@ $I_c$	0.000966
@ $I_d$	0.001005

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

Name	Value [V]
$V_1$	-0.973914
$V_2$	-0.973914
$V_3$	10.634329
$V_4$	6.382611
$V_5$	6.843347
$V_6$	7.067298
$V_7$	1.953900
$V_8$	6.875344

Table 2: Theoretical analysis results from Octave.

## 4 Simulation Analysis

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

## 4.1 Operating Point Analysis

For this circuit's simulation on Ngspice, there was a need to introduce a nule 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 shows the simulated operating point results for the circuit under analysis. Compared to the theoretical analysis results, one notices the following differences: describe and explain the differences.

Using the results of the theoretical analysis for mere guidance, we know the voltage values for the nodes, that should result 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!).

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.

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 algharisms) can be slightly different than the precision we used in GNU Octave (a maximum of 2 significant algharisms of difference). Different algharisms only start to appear by the 6<sup>th</sup> decimal case, which is a disposable calculation error (Shrinking the theoretical results to Ngspice precision, we get equal theoretical and simulation results).

That being said, the theorectical 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.

## 5 Conclusion

In this laboratory assignment the objective of analysing an RC circuit has been achieved. Static, time and frequency analyses have been performed both theoretically using the Octave maths tool and by circuit simulation using the Ngspice tool. The simulation results matched the theoretical results precisely. The reason for this perfect match is the fact that this is a straightforward circuit containing only linear components, so the theoretical and simulation models cannot differ. For more complex components, the theoretical and simulation models could differ but this is not the case in this work.

Lorem ipsum dolor sit amet, consectetur adipiscing elit. Ut purus elit, vestibulum ut, placerat ac, adipiscing vitae, felis. Curabitur dictum gravida mauris. Nam arcu libero, nonummy eget, consectetur id, vulputate a, magna. Donec vehicula augue eu neque. Pellentesque habitant morbi tristique senectus et netus et malesuada fames ac turpis egestas. Mauris ut leo. Cras viverra metus rhoncus sem. Nulla et lectus vestibulum urna fringilla ultrices. Phasellus eu tellus sit amet tortor gravida placerat. Integer sapien est, iaculis in, pretium quis, viverra ac, nunc. Praesent eget sem vel leo ultrices bibendum. Aenean faucibus. Morbi dolor nulla, malesuada eu, pulvinar at, mollis ac, nulla. Curabitur auctor semper nulla. Donec varius orci eget risus. Duis nibh mi, congue eu, accumsan eleifend, sagittis quis, diam. Duis eget orci sit amet orci dignissim rutrum.