Circuit Theory and Electronics Fundamentals

BSc Aerospace Engineering, Técnico, University of Lisbon

Lab 1: Circuit analysis methods

March 25, 2021

Group 48
Dinis Papinha, 84379
Filipa Gonçalves, 89662
Carlos de Vasconcelos, 90227

Contents

1 Introduction

The objective of this laboratory assignment is to study a resistive circuit containing linear components, such as resistors (R_i), independent (circle shaped) and dependent (rhombus shaped) voltage (V)

2 Theoretical Analysis

2.1 Mesh Method

Name	Amplitude
V1	1.00000000000
V2	0.95451213476
V3	0.85940656896
V5	0.96104661213
V6	0.60656522123
V7	0.40122803814
V8	0.60479191683

Table 1: Values for the components using the node method.
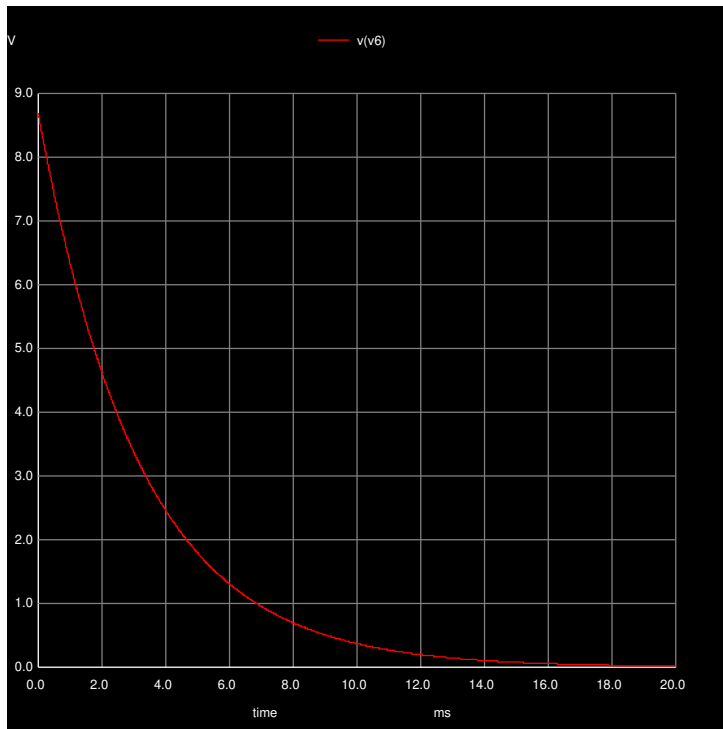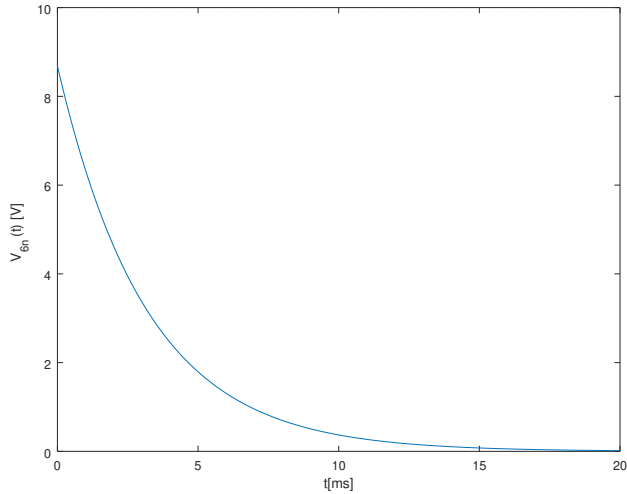
Figure 1: Teste

Figure 2: Teste

3 Simulation Analysis

To accurately simulate the circuit in ngspice, precaution is needed with the positive and negative nodes associated with each component, since an error in these parameters would cause currents and voltage sources operating in the wrong direction.

It's important to add that due to a peculiarity of the program, we couldn't get the current in the resistance R_5 as a reference for the current dependent voltage source. To solve that, we created a

Name	Amplitude
V1	-0.00000000000
V2	-0.00000000000
V3	0.00000000000
V5	-0.00000000000
V6	-2.99994331014
V7	3.14159265359
V8	3.14159265359

Table 2: Values for the components using the node method.

4 Conclusion

In this laboratory assignment, the objective of analysing a resistive circuit has been achieved. Analyses of current intensity in branches and nodal voltages 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 reasons for the match may be related to the fact that this is a simple linear circuit, in which the currents and voltages do not vary with time, and the components are considered to be ideal in both models. Furthermore, the mesh and nodal methods are systematic and structured. The theoretical and the Ngspice simulation models do not differ.