



TÉCNICO
LISBOA

Circuit Theory and Electronics Fundamentals

Masters of Aerospace Engineer, Técnico, University of Lisbon

Laboratory Report

Group 37

Afonso Magalhães, nº95765
Fábio Monteiro, nº95786
Leonardo Encarnação, nº95816

March 25, 2021

Contents

1	Introduction	3
2	Theoretical Analysis	4
3	Simulation Analysis	4
4	Conclusion	4

1 Introduction

In this laboratory assignment we study a circuit containing various elements, amongst them a capacitor, resistances and dependent and independent sources of voltage and current in order to accomplish various objectives. At first we analyse the circuit when $t \geq 0$, using the nodal method to determine the voltages in all nodes and currents in all branches. Then, in order to find the R_{eq} , we change the circuit into what is displayed in Figure 2, and with this new circuit, we'll find the total solution of the voltage V_s . To finalize, we are going to determine the frequency responses of the voltage in the capacitor and in the node 6.

In Section 2, a theoretical analysis of the circuit is presented. In Section 3, the circuit is analysed by simulation, and the results are compared to the theoretical results obtained in Section 2. The conclusions of this study are outlined in Section 4.

2 Theoretical Analysis

3 Simulation Analysis

In order to run the simulation, we wrote the ngspice code according to the image below. It is important to note that an extra voltage source, V_{aux} , was added and therefore, another node was also added (node 8). This V_{aux} was intended to allow the measurement of the current I_c which voltage source V_c depends on, since ngspice doesn't allow us to introduce Resistor R_6 's current in the computation. V_{aux} 's voltage is equal to 0 V, since it is only an auxiliary component that doesn't interfere with the circuit (node's 8 voltage is equal to node's 7 voltage) and allowed us to obtain the current through it.

4 Conclusion

In this laboratory assignment the objective of analysing the circuit specified in the introduction has been achieved. All analyses have been performed both theoretically using the Octave maths tool and by circuit simulation using the Ngspice tool.