TÉCNICO LISBOA

Circuit Theory and Electronics Fundamentals

Masters of Aeroespace Engineer, Técnico, University of Lisbon

Laboratory Report

Group 37

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

March 25, 2021

Contents

1 Introduction

In this laboratory assignment we accomplish various objectives, In first place we analyse the circuit when t¡0, using the nodal method to determine the voltages in all nodes and currents in all branches.(fig1) Then, in order to find the R_eq, we change the circuit into what in the fig2 is displaying and work with that circuit to find the solution to , and to finalise The objective of this laboratory assignment is to study a circuit containing dependent and independent sources of both voltage and current alonside various resistors (2 dependent sources, 2 independent souces and 7 resistors to be more precise). To do so we've obtained the current in the various meshes and the voltage in all the nodes, using two diferent methods alongside a simlulation to corroborate the results. The circuit that will be analysed is represented in the image below and the values of each constant are especifed in the table below.

The objective of this laboratory assignment is to study a circuit when t¡0 containing dependent and independent sources of both voltage and current alongside various resistors and 1 capacitor(2 dependent sources, 2 independent souces and 7 resistors to be more precise). To do so we've obtained the current in the various meshes and the voltage in all the nodes, using two diferent methods alongside a simlulation to corroborate the results.

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, Vaux, was added and therefore, another node was also added (node 8). This Vaux was intended to allow the measurement of the current Ic which voltage source Vc depends on, since ngspice doesn't allow us to introduce Resistor R6's current in the computation. Vaux'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 especified 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. 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.