Circuit Theory and Electronics Fundamentals

Mestrado em Engenharia Física Tecnológica, Técnico, University of Lisbon

Ana Martins (96506), Miguel Moreira (96556), Sara Oliveira (96566)

T2

April 5, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing two independent sources, V_a and I_d, one voltage controlled source, I_b, one current controlled source, V_c, connected to seven resistors. The circuit can be seen in Figure **??**.

t1-1.pdf

Figure 1: This is circuit that was analysed. The nodes were numbered, as were the currents in each mesh.

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

2 Theoretical Analysis

2.1 Node Analysis ($t < 0$)

In this particular section, the circuit is analysed in $t < 0$, therefore, we can apply the usual node analysis, with no sinusoidal tensions in the voltage source Vs: for $t < 0$ $v_s = Vs$.

Labels were assigned to identify nodes from zero to eight (no node four given in instructions), and directions to currents as seen in figure (**??**), to then proceed with the node analysis. We can, therefore, derive direct equations in terms of the voltage at these nodes.

For this procedure, one applies KCL (Kirchoff's Current Law), which states that the sum of currents leaving a node must be the same as the sum of currents entering a node. Because this kind of approach is only possible for the nodes which aren't directly connected to a terminal of a voltage source, it's crucial to find other equations to cover all the unknown variables of the system. In this case, it's important to consider factor like the voltage gain between two nodes separated by a voltage source: for example, from nodes 0 and 1, the terminals of v_s, we can take the equation: $V_1 = V_0 + V_s$. As V_0 is identified as the ground ($V_0 = 0V$) we're left with: $V_1 = V_s$.

Moreover, we can relate the currents associated with the current or voltage controlled sources with the nodes voltages, giving us two more equations, the nineth and thenth.

The equations are as follows:

$$\begin{cases} Node\,0 : V_0 = 0 \\ Node\,1 : V_1 = V_s \\ Node\,2 : \frac{V_1-V_2}{R_1} + \frac{V_3-V_2}{R_2} - \frac{V_2-V_5}{R_3} = 0 \\ Node\,3 : -\frac{V_3-V_2}{R_2} + I_b = 0 \\ Node\,5 : V_5 - V_8 = V_d \\ Node\,6 : \frac{V_5-V_6}{R_5} - I_b = 0 \\ Node\,7 : \frac{V_0-V_7}{R_6} - \frac{V_7-V_8}{R_7} = 0 \\ Supernode\,5-8 : \frac{V_2-V_5}{R_3} - \frac{V_5-V_6}{R_5} + \frac{V_7-V_8}{R_7} - \frac{V_5-V_0}{R_4} = 0 \\ I_d = \frac{V_0-V_7}{R_6} \\ I_b = K_b\,(V_2 - V_5) \end{cases} \tag{1}$$

Note that as there's no sinusoidal excitation ($v_s = constant$), the capacitor behaves like an open circuit and so the current I_c is null and not included in the equations, because:

$$I_c = C\frac{dv}{dt} \tag{2}$$

Using *GNUOctave*, we computed the values of the node voltages by solving the system KCL equations above with matrixes. The currents in each branch were solved by aplication of Ohm's law ($I = \frac{V}{R}$) in each resistor, for example:

$$I_1 = \frac{V_1 - V_2}{R_1} \tag{3}$$

The value obtained are listed in the next tables:

Name	Voltages [V]

Table 1: T2 1) Node Analysis Computation Results: Voltages computed using KCL equations

Name	Currents [A]

Table 2: T2 1) Node Analysis Computation Results: Currents (A) computed using ohms law

2.2 Determining the Equivalent Resistor as Seen From Capacitor Terminals

TABELAS DO 2 DO OCTAVE

Name	Voltages [V]
V0	-4.9144604e-32
hline V2	-1.3014247e-15
hline V3	-3.8742839e-15
hline V5	-1.1245383e-15
hline V6	8.6836959e+00
hline V7	6.1145673e-16
hline V8	1.3292118e-15
hline Vb	-1.7688637e-16
Vd	-2.4537501e-15

Table 3: T2 1) Node Analysis Computation Results: Currents (A) computed using ohms law

2.3 Determining the Natural Solution for $V_6(t)$

After reducing the rather complex circuit to a circuit with one voltage source, one resistor, R_{eq}, (between nodes with voltage V_s and V_6, with this direction of voltage drop) and one capacitor, we can easily determine the natural solution on the system. More specifically, we can observe the behaviour of the capacitor, dissipating voltage through the resistors without any external excitation or driving force.

To study this, we've worked out the voltage natural solution in node six (V_{6n}), in the interval [0, 20]ms.

For a simple RC series circuit (with a voltage source) we can derive the differential equation with describes the system. As is a system formed only by components in series the current (I_c) is the same in the resistor and in the capacitor:

$$I_c = I_c \Leftrightarrow \frac{V_s - V_6(t)}{R_{eq}} = C\frac{dV_6(t)}{dt} \tag{4}$$

$$\Leftrightarrow V_s = V_6(t) + R_{eq}C\frac{dV_6(t)}{dt} \tag{5}$$

As said before, in this step we want to determine the natural solution and, for that, we need to turn off external excitations, particularly, the voltage source v_s: $V_s = 0$. Which yields:

$$V_6(t) + R_{eq}C\frac{dV_6(t)}{dt} = 0 \tag{6}$$

Name	Currents [A]
Ix	2.8670509e-03
hline Iy	-2.8670509e-03
hline I1	1.2662274e-18
hline I2	-1.2386447e-18
hline I3	-5.6774006e-20
hline I4	-2.7353981e-19
hline I5	-2.8670509e-03
hline I7	-7.1450436e-19
hline Ib	-1.2386447e-18
hline Id	-3.0123519e-19
hline	

Table 4: T2 1) Node Analysis Computation Results: Currents (A) computed using ohms law

This is, recognizably, a first order homogeneous differential equation, which have very well known solutions in the form of:

$$V_6(t) = Kexp(-\frac{t}{RC}) \tag{7}$$

,where K is an integration constant.

To determine K, we need an initial condition, such as $V_6(t=0) = V_x$, determined in the last section. The final expression for $V_6(t)$ is then:

$$V_{6n}(t) = V_x exp(-\frac{t}{RC}) \tag{8}$$

We used *GNUOctave* to compute and plot V_{6n} against time ([0, 20] ms), using a 1×10^{-6} s step, obtaining the following graphic:

FALTA METER O GRAFICO I STILL NEED TO FIGURE THAT OUT

2.4 Determining the Forced Solution for $V_6(t)$

2.5 Determining the Final Total Solution for $V_6(t)$

2.6 NEM PERCEBO O QUE RAIO ELES QUEREM AQUI

3 Simulation Analysis

3.1 Operating Point Analysis

Table **??** shows the simulated operating point results for the circuit under analysis.

Name	Value [A or V]
@gb[i]	-2.48284e-04
@r1[i]	2.369027e-04
@r2[i]	-2.48284e-04
@r3[i]	-1.13810e-05
@r4[i]	1.210640e-03
@r5[i]	-2.48284e-04
@r6[i]	9.737374e-04
@r7[i]	9.737374e-04
v(1)	5.185042e+00
v(2)	4.941554e+00
v(3)	4.425830e+00
v(5)	4.977013e+00
v(6)	5.729012e+00
v(7)	-1.97652e+00
v(8)	-2.95469e+00
v(9)	0.000000e+00

Table 5: Operating point results. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

These results were produced using the *Ngspice software*. In order for the *software* to be able to recognise the Current-Controlled Voltage Source, defined in Figure **??** as H_c, we had to add a new Independent Voltage Source with a voltage of 0V, which is also represented in **??**.

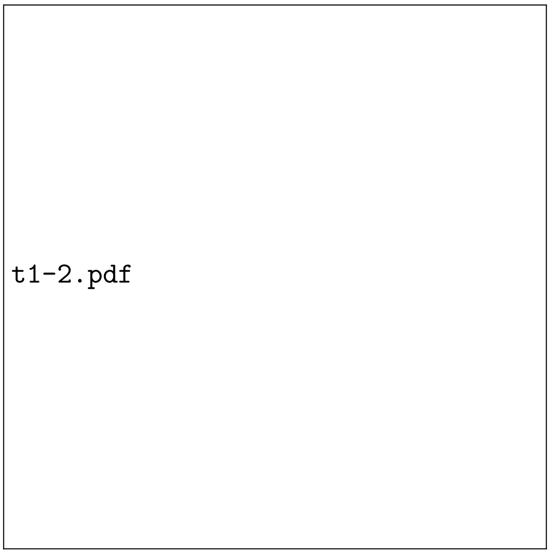

Figure 2: The original circuit with an added voltage source of value 0V.

To compare the results between the theoretical calculations and the simulation, it's important to keep in mind that the current values and directions represented in **??** by round arrows: I_1, I_2, I_3 and I_4 correspond, respectively, to the following values in the simulation data table (**??**): @r1[i], -@r2[i], -@id[current], -@r6[i] (or @r7[i], they're equivalent).

On a general basis, the values obtained by the simulation greatly resemble the ones obtained using the theoretical models and the application *GNUOctave* to compute them.

In fact, by inspection it's possible to conclude that pratically all the values fit in the same magnitude as its correspondent.

As it's possible to check in the following table, the relative uncertainties ($|theoretical - simulated|/|theoretical|$) are considerably slim. Although we have a rather significant uncertainty for the value of V7 (theoretical value: -9.7816698e-01 V , simulated value: 8.139731e+00 V, uncertainty: 2.0000000e+00); the smallest uncertainty, which null, reports to the value of I_1 (theoretical value: 2.3690260e-04 A , simulated value: 2.369026e-04 A). It's relevant to note that this number isn't actually zero, but instead, the program doesn't have enough precison to compute a detectable error.

Name	**Relative Uncertainty [A or V]**

Table 6: Relative Uncertainty of the simulation results relative to the theorical values.

4 Conclusion

In this laboratory assignment the objective of analysing a circuit with several current and voltage sources (one linearly dependent and one independent of each) in parallel and in series with resistors has been achieved.

Ideally, the current and voltage theoretical analysis (computed using *GNUOctave*) should precisely resemble the circuit simulated by the *Ngspice*.

As seen in the previous section, it's possible to conclude that the simulations follows very closely the theorical model used in the analysis: node and mesh analysis. In fact, for a circuit containing only linear components, the simulation values should not differ from the theoreticallt obtained ones.

References

[1] Phyllis R. Nelson, *Introduction to SPICE Source Files* Slides

[2] *GNU Octave* Documentation Files

[3] José Teixeira de Sousa, *Circuit Theory and and Eletronic Fundamentals* Class Slides