Circuit Theory and Electronics Fundamentals

T1 Laboratory Report

Group 34

José Bento, Nº 95815
Thomas Childs, Nº 95847
Luís Pacheco, Nº 96425

Aerospace Engineering (MEAer), Técnico, University of Lisbon

March 22, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing independent and linearly dependent voltage and current sources. The circuit also contains 7 resistors, totaling 11 components.

The circuit has 8 nodes and 4 meshes. The nodes of the circuit were numbered arbitrarily (from V_0 to V_7), and it was considered that *node 0* was the ground node. The voltage-controlled current source I_b has a linear dependence on Voltage V_b, of constant K_b. The voltage V_b is the voltage drop at the ends of resistor R_3. The current-controlled voltage source V_c has a linear dependece on current I_c, of constant K_c. The control current I_c is the current that passes through the resistor R_6. The circuit can be seen in **Figure 1**.

The values for the resistors, the independent sources and the constants for the dependent sources are presented in **Table 1**. These values were obtained using the Python script provided by the Professor responsable for the laboratory assigment and using the number 95815 as the seed.

In Section 2, a theoretical analysis of the circuit is presented using two methods: the mesh analysis and the nodal analysis. In Section 3, the circuit is analysed by simulation using the program Ngspice, and the results are compared to the theoretical results obtained in Section 2. The conclusions of this study are outlined in Section 4.

Figure 1: Circuit in study

Name	Python-generated values
R1	1.04606282456
R2	2.00732621328
R3	3.06060705885
R4	4.07055531265
R5	3.1225213804
R6	2.06927045958
R7	1.01531018068
Va	5.24359648479
Id	1.01891541651
Kb	7.0473187437
Kc	8.3479788681

Table 1: The variables that start with an *R* are the values of the resistors and are expressed in kiloohm (kOhm); the variable *Va* is a *voltage* and is expressed in Volt (V) and the variable *Id* is a *current* and expressed in miliAmpere (mA). The constants *Kc* and *Kb* are expressed in kiloOhm (kOhm) and miliSiemens (mS), respectively.

2 Theoretical Analysis

In this section, the circuit shown in **Figure ??** is analysed theoretically using two methods: the mesh analysis and the nodal analysis.

2.1 Nodal Analysis

For the nodal analysis we apply the Kirchhoff Current Law (KCL) to the nodes that are not conected to voltage sources (equations 3 to 7). We obtain 2 more equations from the potencial difference at the terminals of the voltage sources (equations 8 and 9). As was stated before, *node 0* is the ground node and therefore the voltage at this node is 0V (equation 1). For the simulation analysis it was necessesary to add another "fictional" voltage source that provides 0 volts to the circuit, between node 7 and resistor 6. This "fictional" voltage source was also considered in the Theoretical Analysis, as it has no real effect on the circuit. It was needed to define in the ngspice code the current on which the current-controlled voltage source depends on. This yields equation number 2. Now we have 9 equations for 9 unkown variables - voltage V_0 to V_8. The schematics correponding to this approach can be seen in **Figure 1** Using Octave, we determine the values of every node voltage of the circuit.

$$V_0 = 0 \tag{1}$$

$$V_8 = V_7 \tag{2}$$

$$\frac{V_1 - V_2}{R_1} + \frac{V_0 - V_8}{R_6} + \frac{V_0 - V_4}{R_4} = 0 \tag{3}$$

$$\frac{V_2 - V_4}{R_3} + \frac{V_2 - V_3}{R_2} + \frac{V_2 - V_1}{R_1} = 0 \tag{4}$$

$$-K_b(V_2 - V_4) + \frac{V_3 - V_2}{R_2} = 0 \tag{5}$$

$$\frac{V_5 - V_4}{R_5} + K_b(V_2 - V_4) - Id = 0 \tag{6}$$

$$\frac{V_7 - V_6}{R_7} + \frac{V_7 - V_0}{R_6} = 0 \tag{7}$$

Figure 2: Diagram of the circuit considered for the computations

$$V_1 - V_0 = V_a \tag{8}$$

$$V_4 - V_6 = K_c \frac{V_0 - V_7}{R_6} \tag{9}$$

In **Table 2** the values for the branch currents and the node voltages obtained from the Octave script for both methods are presented. Here, the node voltages in the mesh method were computed from the respective currents, which were determined as described in the previous subsection.

Name	Node method
V0	0
V1	5.054818641360000
V2	4.793704691824189
V3	4.258197784082119
V4	-1.934225550007436
V5	4.831047093349573
V6	5.668298372306746
V7	-1.934225550007436
V8	-2.905231322705667
Ir1	-2.536487650531725e-01
Ir2	-2.658975828505401e-01
Ir3	1.224881779736725e-02
Ir4	1.205310280142027
Ir5	2.658975828505403e-01
Ir6	-9.516615150888543e-01
Ir7	-9.516615150888547e-01
Gb	2.658975828505403e-01

Table 2: A variable preceded by @ is of type *current* and expressed in milliampere (mA); other variables are of type *voltage* and expressed in Volt (V).

As can be seen, the values obtained via the mesh and the node analysis are the same, which suggests that for this simple circuit, the methods are equally precise, as expected.

2.2   Equeivalent resistor form capacitor's prespective

In **Table 2** the values for the branch currents and the node voltages obtained from the Octave script for both methods are presented. Here, the node voltages in the mesh method were computed from the respective currents, which were determined as described in the previous subsection.

Name	Values for aux circuit
V0	0
V1	0
V2	0
V3	0
V4	0
V5	0
V6	8.573529695010000
V7	0
V8	0
Req	-3.148773561540000
τ	-3.239212e-03

Table 3: A variable preceded by @ is of type *current* and expressed in milliampere (mA); other variables are of type *voltage* and expressed in Volt (V).

Name	Complex amplitude voltage
V0	0
V1	1
V2	9.483435572941628e-01
V3	8.424036718627852e-01
V4	3.826498411201222e-01
V5	9.557310432110339e-01
V6	1.121194149101466
V7	3.826498411201222e-01
V8	5.747449174406016e-01

Table 4: The table for the complex amplitudes in point 4:

3 Simulation Analysis

In this section we will describe the steps needed to simulate this circuit using the software Ngspice.

3.1 Operating Point Analysis

Table 5 shows the simulated operating point results for the circuit under analysis. The current flows considered in the theoretical section were coherent with the polarity implicitly declared when defining the circuit to be simulated in the Ngspice script.

As mentioned previously, in Section 2, we had to create a "fictional" voltage source, between node 7 and resistor 6 (providing 0V to the circuit in order not to alter the behaviour of the rest of the circuit) so as to be able to define the dependecy for the current-controlled voltage source V_c. This has no specific reason to be, other than the particularities of the Ngspice software. The consequences of adding this additional voltage have already been described (namely the creation of a new node) and dealt with in the previous section - see, for example, equation 2.

The following steps in the simulations are to be conducted:

- for t¡0 (operating point only, in order to obtain the voltages in all nodes and the currents in all branches);

- operating point for $V_s(0) = 0$, replacing the capacitor with a a voltage source Vx = V(6)-V(8), where V(6) and V(8) are the voltages in nodes 6 and 8 as obtained in the previous step (this step is necessary given that we must the compute the boundary conditions that guarantee continuity in the capacitor's discharge - such may imply that the boundary conditions differ from those computed for t¡0);

- for the natural response of the circuit (using the boundary conditions V(6) and V(8) as obtained previously);

- repeating the third step, for the natural and forced response on node 6 with V_s (as given in **Figure 7**) and f = 1kHz.

- for the frequency response in node 6 given these conditions

Name	Value [mA or V]
@c[i]	0.000000e+00
@gb[i]	-2.65897e-04
@r1[i]	2.536486e-04
@r2[i]	2.658975e-04
@r3[i]	-1.22488e-05
@r4[i]	-1.20531e-03
@r5[i]	-2.65897e-04
@r6[i]	9.516617e-04
@r7[i]	9.516617e-04
v(1)	5.054819e+00
v(2)	4.793705e+00
v(3)	4.258199e+00
v(4)	-1.93423e+00
v(5)	4.831048e+00
v(6)	5.668299e+00
v(7)	-1.93423e+00
v(8)	-2.90523e+00

Table 5: Step 1: Operating point for t¡0. A variable preceded by @ is of type *current* and expressed in miliAmpere; other variables are of type *voltage* and expressed in Volt.

4 Conclusion

In this laboratory assignment the objective of analysing a circuit containing independent, linearly dependent voltage and current sources and other linear components (resistors) has been achieved. Static analyses have been performed both theoretically, using the Octave maths tool, and by circuit simulation, using the Ngspice tool. Not only did the theoretical results obtained using different methods agree perfectly, they also matched the simulation results precisely.

The reason for this perfect match is the fact that this is a straightforward circuit containing only linear components - this implies, as pointed out in the theory classes, that the theoretical and simulation models cannot differ. For more complex components and circuits, the theoretical and simulation models could differ (given the greater complexity of the models implemented by the simulator Ngspice, when compared to those studied in the theoretical classes); however, that is not the case for this particular circuit.

This lab assignment has been useful to learn and give us the chance to put into practice our knowledge of the invaluable software tools and automation procedures required for this and other types of work, from Git and Makefile to Octave and the syntax of Ngspice. In more theoretical terms, we have become more at ease with the analysis of linear circuits, and, knowing what we now know, we would have made differente choices regarding the analysis even of this circuit (for example, we probably would have chosen a different node to assign to the Ground potential).

Name	Value [mA or V]
@gb[i]	6.332294e-18
@r1[i]	-6.04059e-18
@r2[i]	-6.33229e-18
@r3[i]	2.917029e-19
@r4[i]	-1.32956e-18
@r5[i]	-2.72282e-03
@r6[i]	8.673617e-19
@r7[i]	1.320008e-20
v(1)	0.000000e+00
v(2)	6.218372e-15
v(3)	1.897135e-14
v(4)	-1.76289e-15
v(5)	5.329071e-15
v(6)	8.573530e+00
v(7)	-1.76289e-15
v(8)	-1.77636e-15
Ix	-2.72282e-03
Vx	8.573530e+00
Req	-3.14877e+03

Table 6: Step 2: Operating point for $v_s(0) = 0$, with the capacitor replaced by a voltage source $V_x = V(6) - V(8)$ with these as obtained in the last step. A variable preceded by @ is of type *current* and expressed in miliAmpere; other variables are of type *voltage* and expressed in Volt.
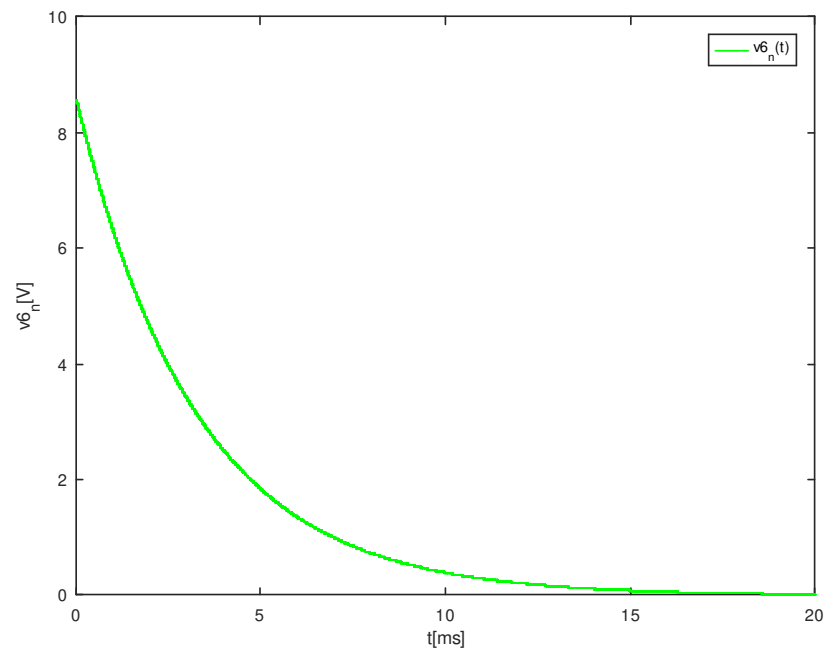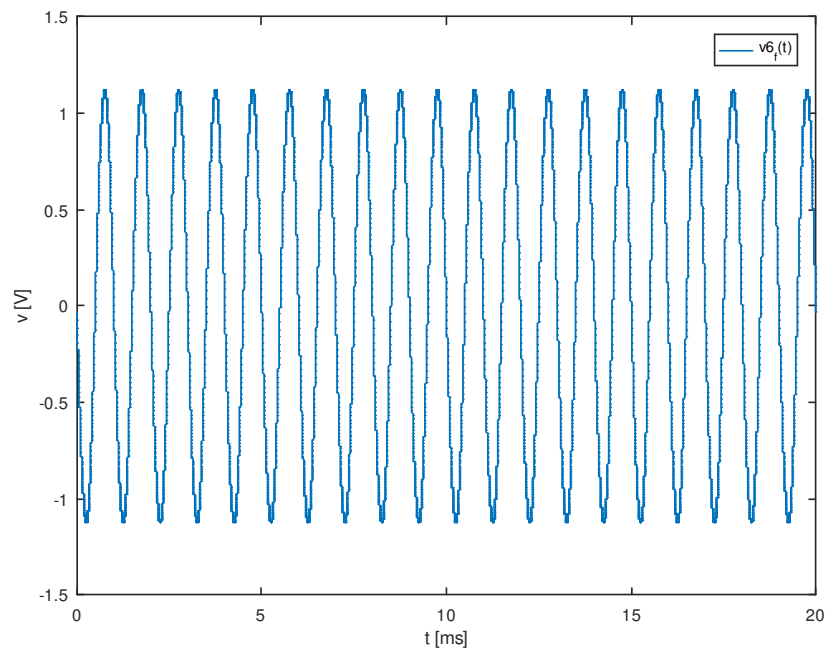
Figure 3: Graph for natural response point 3

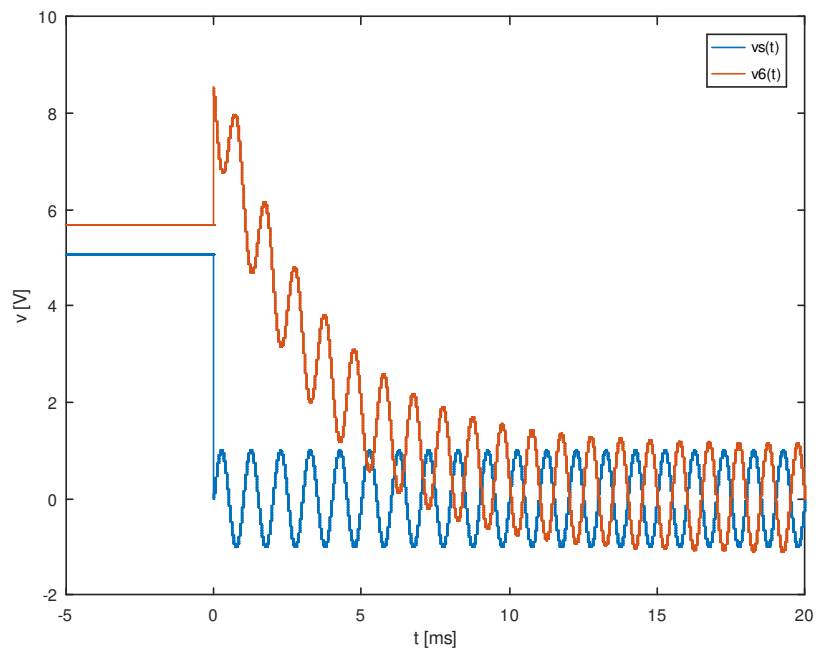Figure 4: Graph for forced response point 4

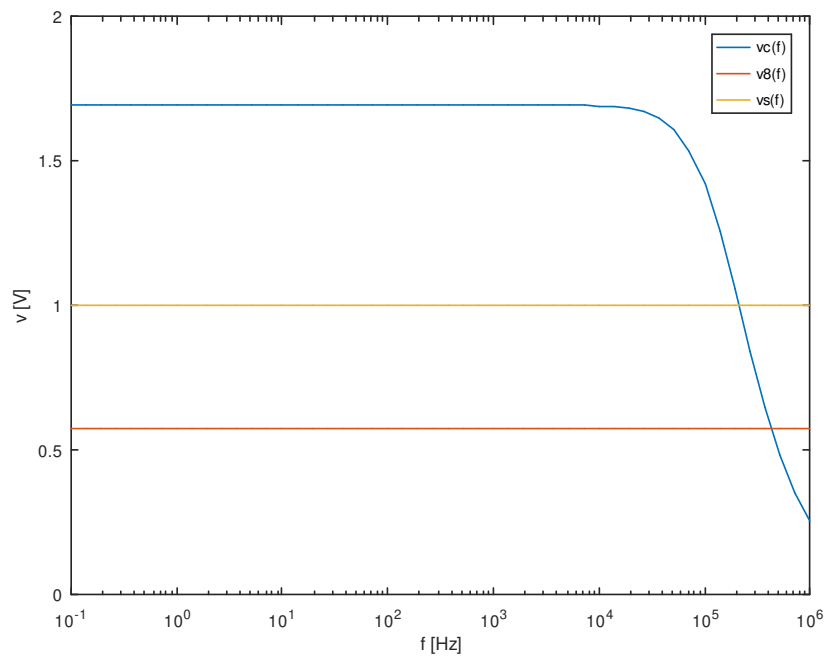Figure 5: point 5: Graph for time interval from -5 to 20 ms

Figure 6: point 6: Graph for frequency response of Vc, V8 and Vs for frequencies ranging from 0.1Hz to 1MHz

$$v_s(t) = V_s u(-t) + \sin(2\pi f t) u(t)$$

$$u(t) = \begin{cases} 0, t < 0 \\ 1, t \geq 0 \end{cases}$$

Figure 7: Time step conditions