# Circuit Theory and Electronics Fundamentals

Integrated Masters in Aerospace Engennering, Técnico, University of Lisbon

Laboratory Report 1- Group 28

Beatriz Pedroso 95773, Teresa Gonçalves 95826, Tiago Escalda 95851

March 24th, 2021

# Contents

# 1 Introduction

# 2 Introduction

The aim of this laboratory assignment is to analyse a RC circuit, which contains a sinusoidal voltage source $v_s$ and a capacitor $C$. The other components present in this four mesh circuit are 7 resistors (from $R_1$ to $R_7$) and a linearly dependent current $I_b$ and voltage $V_d$ sources.

The voltage controlled current source depends on the constant $K_d$ and the current controlled voltage source has a linear dependence on the constant $K_b$.

The voltage source varies in time as it follows:

$$v_s(t) = V_s u(-t) + sin(2\pi ft)u(t) \tag{1}$$

where

$$u(t) = e \begin{cases} 0 & t<0 \\ 1 & t \geq 0 \end{cases} \tag{2}$$

The data generated automatically by the Python script is given in the table below.

| Name | Value |
|------|-------|
| R1 | 1.041113e+03 |
| R2 | 2.099452e+03 |
| R3 | 3.131091e+03 |
| R4 | 4.119470e+03 |
| R5 | 3.115588e+03 |
| R6 | 2.047994e+03 |
| R7 | 1.027544e+03 |
| Vs | 5.068716e+00 |
| C | 1.041275e-06 |
| Kb | 7.287471e-03 |
| Kd | 8.115684e+03 |

Table 1: Initial data generated by Python script. All variables are expressed in Ohms, V, F, S or A.

The nodes (from $V_1$ to $V_8$) are displayed as it shows in Figure **??**. The fourth node is considered to be the ground one.

In Section 1, for $t < 0$, the voltage and the currents in all branches were determined with the node method. The circuit is analysed by simulations using the program Ngspice. In Section 2, both the equivalent resistor $R_eq$, seen from the capacitor terminals, and the nodes voltage were computed. An operating poin analysis is used to analyse the circuit when $t$ ¡ 0 and also to determine the time constant.The natural solution is determined in the interval [0, 20]ms in Section 3, followed in the next section with the computation of the forced solution.Both natural and forced solutions are computed with a transiet analysis. These solutions were superimposed. Phasors were converted to real time functions. In Section 6, the voltage ($v_c$, $V_s$ and $V_6$) responses due to variation of frequencies were studied. Theoretical analysis are made in all of the sections. Frequency analysis are also made. All the Ngspices simulations are compared with the theoretical results. The conclusions of this study are outlined in the final section.
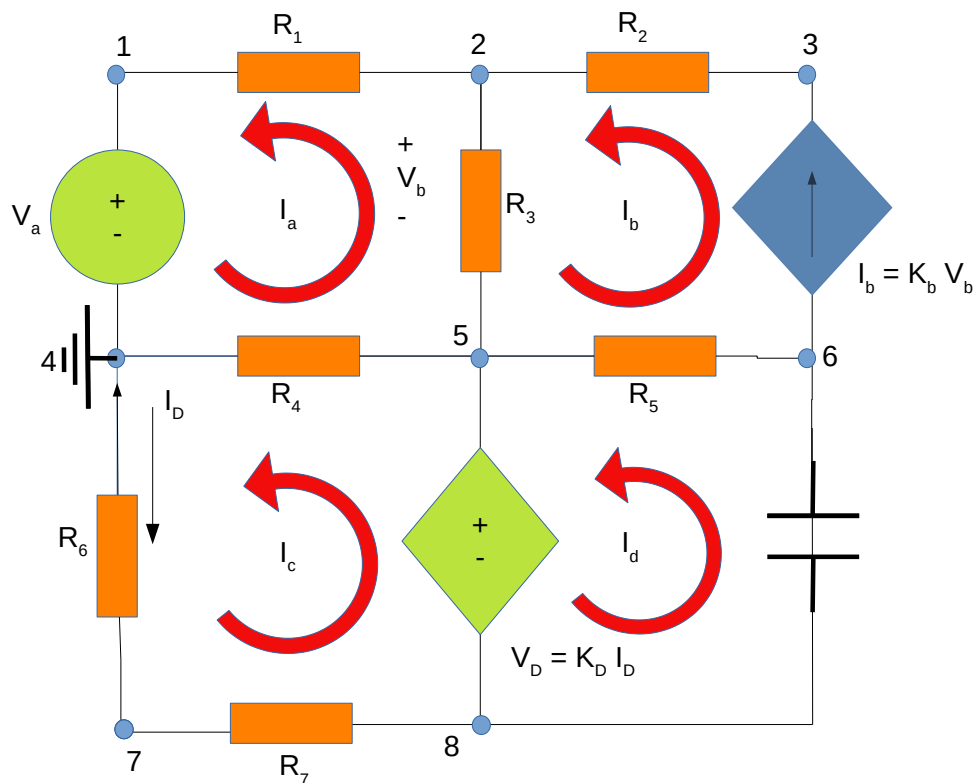
3

Figure 1: Circuit analysed.

# 3 Question 1- Nodal Analysis

## 3.1 Theoretical Analysis

## 3.2 Operating Point Analysis

First of all, to contextualize the values obtained using the tools in ngspice, it is necessary to state that, as node 0 is connected to ground, its nodal voltage does not appear on the table of results. Furthermore, to be able to describe the voltage flowing in the dependent source, it is necessary to know the current in resistor 6. However, ngspice is not able to compute this value when the depedent source is described. So, in order to do that, an extra dependent voltage source (whose voltage drop is equal to 0 V) was created, and put in series after the resistor 6. This led to the appeerence of node 8, that has the same voltage drop as node 6. So, by doing that, ngspice is able to determine the current in this auxiliar independent source, which is exactly the value needed. The circuit with these changes is shown in the drawing below.

## 3.3 Comparison

After running the simulation, the results were put in the table below. Then, a careful analysis of the aforementioned table was conducted. It shows the simulated operating point results for the circuit that is being studied, allowing the group to obtain the current flowing in every risistor, the voltage in the dependent voltage source and even the current flowing in the dependent currrent source.

A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.
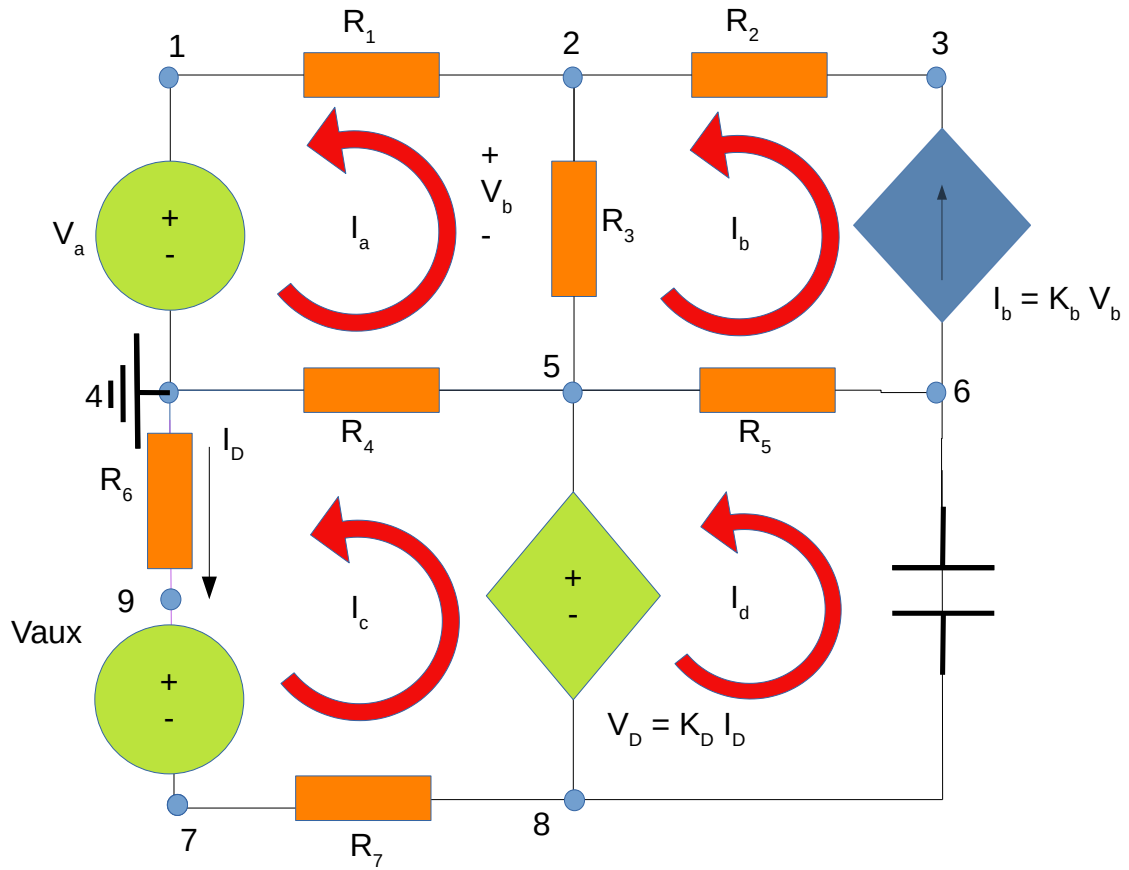
Figure 2: Circuit analysed in ngspice.

In order to validate the results obtained in NGSPICE, relative errors between the theoretical values, obtained in octave and the ones obtained in ngspice, were calculated. These were put in the table below.

| Name | Value [A or V] |
|---|---|
| @c1[i] | 0.000000e+00 |
| @gb[i] | -2.26065e-04 |
| @r1[i] | 2.161572e-04 |
| @r2[i] | -2.26065e-04 |
| @r3[i] | -9.90741e-06 |
| @r4[i] | 1.183330e-03 |
| @r5[i] | -2.26065e-04 |
| @r6[i] | -9.67173e-04 |
| @r7[i] | 9.671730e-04 |
| v(1) | 5.068716e+00 |
| v(2) | 4.843672e+00 |
| v(3) | 4.369060e+00 |
| v(5) | 4.874693e+00 |
| v(6) | 5.579017e+00 |
| v(7) | -1.98076e+00 |
| v(8) | -2.97458e+00 |
| v(9) | -1.98076e+00 |

| Name | Value [A or V] |
|---|---|
| V1 | 5.068716e+00 |
| V2 | 4.843672e+00 |
| V3 | 4.369060e+00 |
| V4 | 0.000000e+00 |
| V5 | 4.874693e+00 |
| V6 | 5.579017e+00 |
| V7 | -1.980764e+00 |
| V8 | -2.974577e+00 |

Table 3: Simulation and Calculus of Req (NgSpice)

Table 2: Simulation and Calculus of Req (NgSpice)

# 4   Question 2- Nodal Analysis, and Req

The purpose of this task was to compute the Req (equivalant resistance) of the circuit.

## 4.1   Theoretical Analysis

It is known that:

$$R_{eq} = \frac{V_x}{I_x} \tag{3}$$

IN order to determine the variable aforementioned, first it was sugested to calculate it seen from the capacitors terminals. Then, using the Thevenin and Norton theorem, we put the independent voltage source to 0V. $V_x$ is equivalent to Thevenin's Voltage, and $I_x$ to Norton's Current. This is necessary because the depedent voltage short cannot be put equal to 0V(short-circuit) and the independent current source cannot be erased from the circuit.

An ilustration of the circuit analysed is showed in figure **??**

Then, the values of current of every branch and the nodal values are obtained using node method. The matrix used in octave is the one that follows.

$$
\begin{bmatrix}
1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
-G1 & G1+G2+G3 & -G2 & 0 & -G3 & 0 & 0 & 0 & 0 \\
0 & -G2-Kb & G2 & 0 & Kb & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\
0 & -G1 & 0 & 0 & -G4 & 0 & -G6 & 0 & 0 \\
0 & Kb & 0 & 0 & -G5-Kb & G5 & 0 & 0 & 1 \\
0 & 0 & 0 & 0 & 0 & 0 & G6+G7 & -G7 & 0 \\
0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 & 0 \\
0 & 0 & 0 & KdG6 & -1 & 0 & -Kd*G6 & 1 & 0
\end{bmatrix}
\begin{bmatrix}
V1 \\ V2 \\ V3 \\ V4 \\ V5 \\ V6 \\ V7 \\ I_x
\end{bmatrix}
=
\begin{bmatrix}
0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ V_x \\ 0
\end{bmatrix}
$$

Theoretically speaking, once the voltage source is equal to 0V, $V_4$ and $V_1$ are also to be 0.
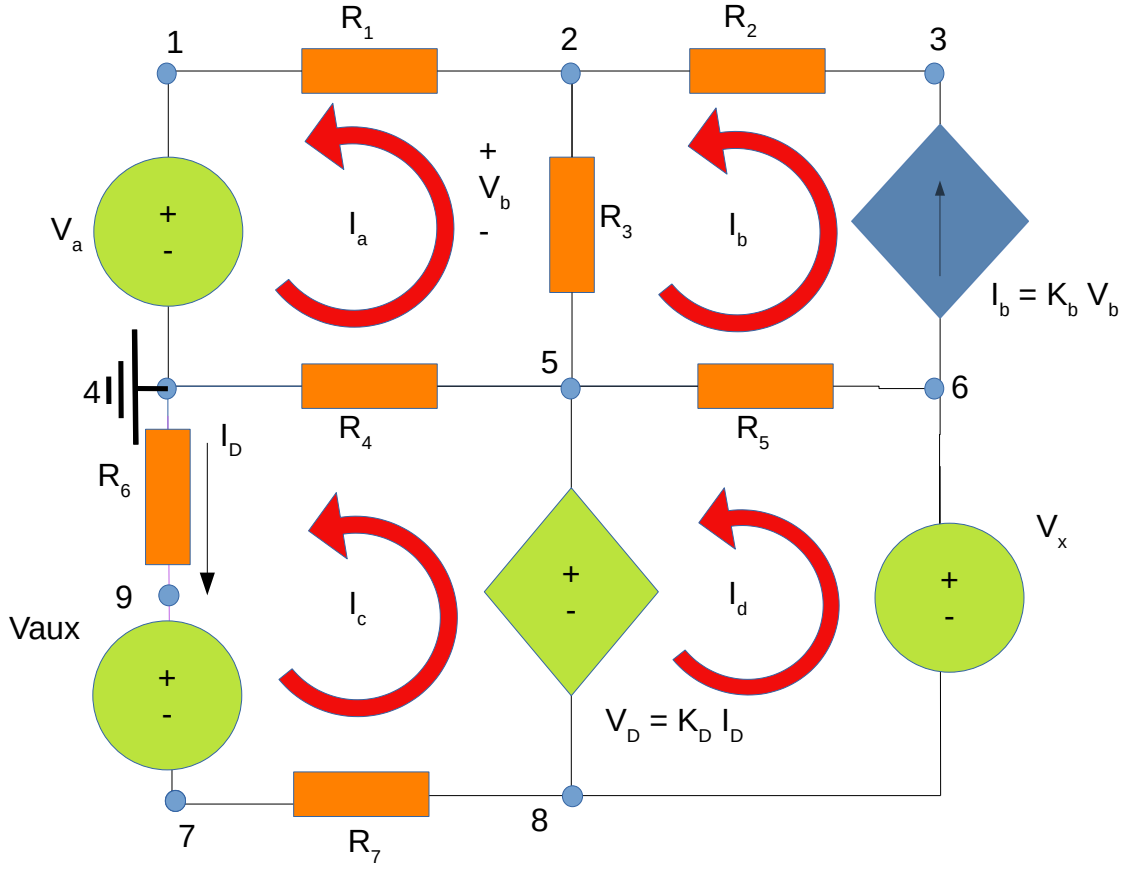
Figure 3: RC Circuit analysed in section 2.

## 4.2 Operating Point Analysis

As requested, an operating point analysis was conducted. The capacitor was replaced by the independent voltage source $V_x$ (**??**). The values of currents and nodal voltages were then put in a table, as well as the $R_{eq}$, hence:

$$R_{eq} = \frac{v(6) - v(8)}{vxbranch} \tag{4}$$

## 4.3 Comparison

After running the node analysis in octave and the simulation in ngspice, the results were put in the tables below. Then, a careful analysis of the aforementioned tables was conducted.

A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt. $R_{eq}$ is presented as in **??** in Ohm.

In order to validate the results obtained in NGSPICE, relative errors between the theoretical values, obtained in octave and the ones obtained in ngspice, were calculated. These were put in the table below.

After the careful evaluation of the results, several observations need to be made. Firstly, in ngspice, there are some node voltage results different from 0V, with values in the order of magnitude of $10^{-15}$, $10^{-16}$ V. After questioning the professor, these will be considered 0V.

On the other hand, octave results have equally those discrepancies. That said, the procedure was the same and those values were considered 0. We believe the main reason for this situation

| Name | Value [A or V] |
|------|----------------|
| @gb[i] | -4.13942e-18 |
| @r1[i] | -3.95801e-18 |
| @r2[i] | -4.13942e-18 |
| @r3[i] | 1.814124e-19 |
| @r4[i] | 8.624201e-19 |
| @r5[i] | -2.74542e-03 |
| @r6[i] | -8.67362e-19 |
| @r7[i] | -1.72875e-18 |
| v(1) | 0.000000e+00 |
| v(2) | -4.12073e-15 |
| v(3) | -1.28112e-14 |
| v(5) | -3.55271e-15 |
| v(6) | 8.553593e+00 |
| v(7) | 1.776352e-15 |
| v(8) | 3.552714e-15 |
| v(9) | 1.776352e-15 |

Table 4: Simulation and Calculus of Req (NgSpice)

| Name | Value [A or V] |
|------|----------------|
| V1 | 0.000000e+00 |
| V2 | 0.000000e+00 |
| V3 | 9.496396e-16 |
| V4 | 0.000000e+00 |
| V5 | 5.935248e-17 |
| V6 | 8.553593e+00 |
| V7 | -2.967624e-17 |
| V8 | 0.000000e+00 |

Table 5: Calculus of $R_{eq}$- Octave

is that, when the data.txt file is read from the datagen.py in octave, a rounding happens, making it impossible for the nodal voltages to be 0.

However, and despite the small erros above explaned, the values of $R_{eq}$, $V_6$ and $I_x$ match perfectly, which led us to fully validate the theorical procedure and the results obtained.

# 5 Question 3-Natural Response

It was proposed to study and determine the natural response of the circuit over time, in node 6. The natural response of the circuit is what the circuit does including the initial conditions (initial voltage of the capacitor) but with the imput supressed.

## 5.1 Theoretical Analysis

## 5.2 Simulation Analysis

In this section, a transient analysis was made in order to evaluate the natural response of the circuit, which means, the variation over time. The description of the circuit included the initial values of v(6) and v(8), calculated in question 1. As the ngspice and octave results matched, as observed in **??**, the theorectical values were imported from a .cir file created by octave. The time interval considered was [0,20]ms.

# 6  Question 4- Theoretical Forced Response

The forced response of a circuit is calculated with the sources turned on, but with the initial conditions (internal stored energy) set to zero.What is forced response? The forced response is where the output (the voltage on the capacitor) is going to end up in the long run after all stored energy eventually dissipates. The forced response does this by ignoring the presence of energy storage elements (in this case, it ignores the capacitor and its initial voltage). However, the forced response can't tell us what happens at t=0, or during the transition to the final state, because it ignores the stored energy.

Like so, a phasor was used, with $V_s = 1$. The capacitor was replaced by its impiedence $Z$. Then, the nodal method was run again, with this new variable.

$$w = 2 * pi * f \tag{5}$$

$$Zc = 1/(j * w * C); \tag{6}$$

The matrix used was the following:

$$
\begin{bmatrix}
1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\
-G1 & G1+G2+G3 & -G2 & 0 & -G3 & 0 & 0 & 0 \\
0 & -G2-Kb & G2 & 0 & Kb & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\
0 & G3 & 0 & G4 & -G3-G5-G4 & G5+1/Zc & G7 & -G7-1/Zc \\
0 & Kb & 0 & 0 & -G5-Kb & G5+1/Z_c & 0 & -1/Z_c \\
0 & 0 & 0 & 0 & 0 & 0 & G6+G7 & -G7 \\
0 & 0 & 0 & KdG6 & -1 & 0 & -Kd*G6 & 1
\end{bmatrix}
\begin{bmatrix}
V1 \\ V2 \\ V3 \\ V4 \\ V5 \\ V6 \\ V7
\end{bmatrix}
$$

$$
=
\begin{bmatrix}
1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0
\end{bmatrix}
$$

Then, the complex amplitudes with the knowlegde that the amplitude of the forced response is the absolute value of the complex $V_6$, and the phase is the argument, the forced solution is then given by:

$$V6_f = A * sin(w * t + Ph) \tag{7}$$

The table below presents the values of the complex amplitudes of every nodal voltage:

| Name | Value [V] |
|------|-----------|
| V1 | 1.000000e+00 |
| V2 | 9.556014e-01 |
| V3 | 8.619659e-01 |
| V4 | 2.801649e-17 |
| V5 | 9.617215e-01 |
| V6 | 5.886212e-01 |
| V7 | 3.907822e-01 |
| V8 | 5.868501e-01 |

Table 6: Amplitudes of Nodal Voltages

# 7 Question 5 (Theorectial) and 4 (Simulation)-Natural and Forced Response

The total response of a circuit can be teased apart into a forced response plus a natural response. These responses can be combined using the principle of superposition. This principle pressupose the addition of the natural response and the forced response, both calculated in question 3 and 4.

## 7.1 Theoretical Analysis

## 7.2 Simulation Analysis

Once again, a transient analysis was made in order to meet the goal above. The main difference between the analysis in point 3 and this one is that Vs was considered a sinusoidal voltage source. This way, the plot obtained is the sum of both responses.

## 7.3 Comparison

# 8 Question 6 (Theoretial) and 5 (Simulation)-Frequency Response

In this section, both octave and ngspice were used in order to obtain plots of the phase response and of the magnitude response, using logscale. This approach is very useful hence it provides a much better plot fit and, therefore it provides great visualization for users. The magnitude in debicels is of interest for analysis of sound waves, and the analysis of the phase or angle delay is a very interesting way of study another parameter to compare signal. Frequency range in both analysis was from 0.1 Hz to 1MHz. The plots made were v6(f), vs(f) and vc=(v6(f)-v8(f)).

## 8.1 Theoretical Analysis

To examine the frequency responses, the system of equations in Section 4 is solved in a loop cycle, what allows us to calculate the $V_6$, $V_c$ and $V_s$ for each frequence. For each result of these complex vectors, the values were saved.

### 8.1.1 Frequency Response- Magnitude

To represent the magnitude in dB, the absolute values were converted ($X_{dB}$=20$log_{10}$(X)). The frequencies were put in a logarithmic scale.

The magnitude of $V_s$ does not suffer any alteration with the variation of the frequency of the signal. Since its amplitude is 1, as one can observe in the graphics shown at the end of the section, the plot shows a constant horizontal line, with the value zero (0=$log_{10}$(1)).

On the contrary, as the frequency is increasing, the magnitudes of $V_6$ and $V_c$ decrease. The value of $V_c$ changes as it is expected in a RC circuit. This is due to the impedance of the capacitor (Z=-j/wC).

### 8.1.2 Frequency Response- Phase

The angles of the values saved were transformed from radians to degrees. The phase in $V_s$ is 0, therefore we do not have to do any further calculations. The phase of each voltage corresponds to the exact angles. In the plot shown below, the frequencies were also put in the logarithmic scale. As the frequency increases the phase tend to negative values, which varies as it is expected.

## 8.2 Simulation Analysis

In this part of the assignment, an AC (Small Signal Analysis was conducted, in order to match the goal aforementioned. This type of analysis allows to study the frequency response of the circuit. In other words, there is no frequency variation over time, the so called steady-state analysis.

## 8.3 Comparison

After comparing the graphics showed below, it is clear to admit that the results in ngspice and octave match. Any minor difference may be explained by aproximation errors.

### 8.3.1 Frequency Response- Magnitude

### 8.3.2 Frequency Response- Phase

# 9 Conclusion

It was agreed by the members of the group that the main goal of the task proposed was achieved. As presented, both theoretical and simulation results(obtained using Octave tools and ngpsice simulator, respectively) matched, reaching total accuracy. Despite the initial belief that the considerable number of components of the circuit could cause some disparity in the results, such did not happened. This proves not only the efficency of both mesh and node methods to analyse the circuit, as well as the simulator used.

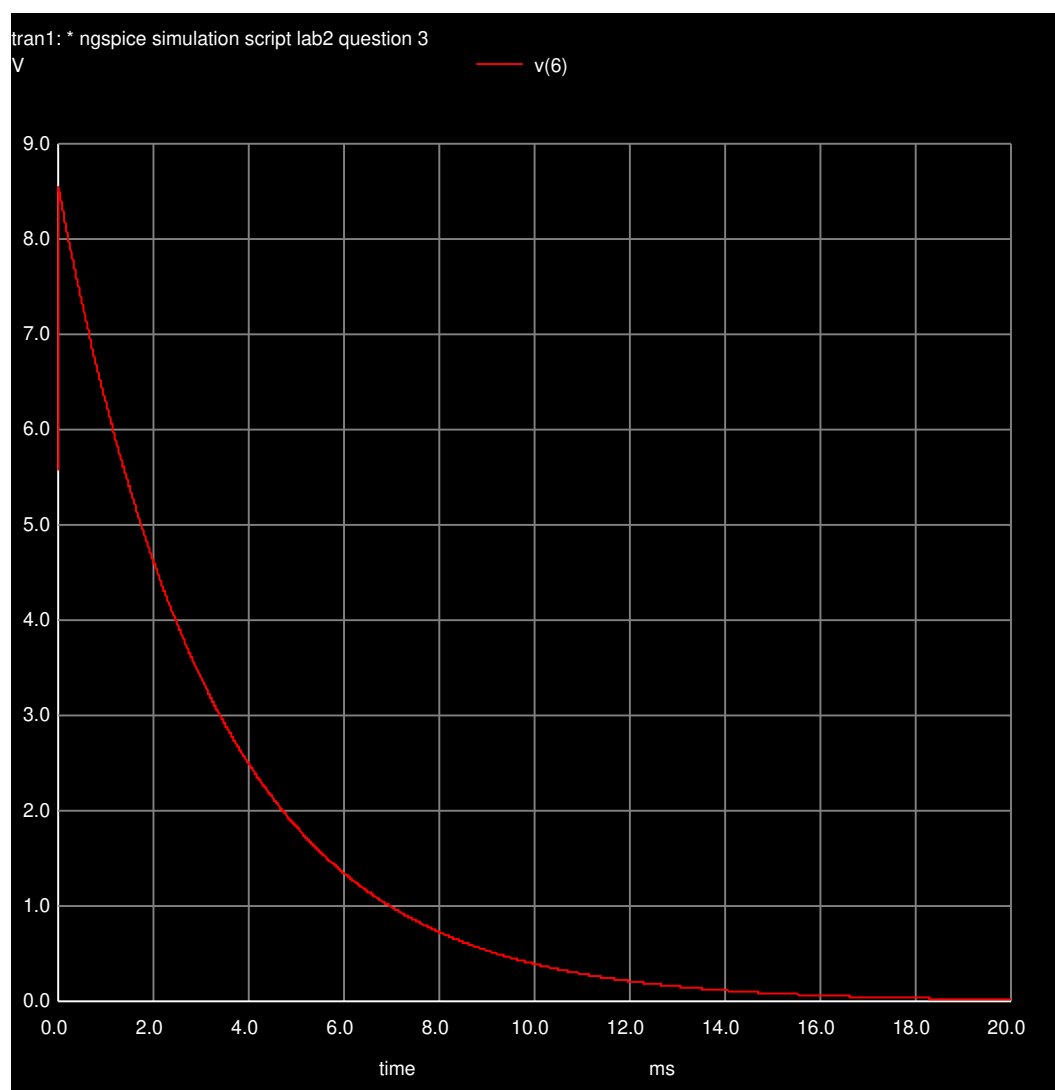tran1: * ngspice simulation script lab2 question 3

Figure 4: Circuit analysed.

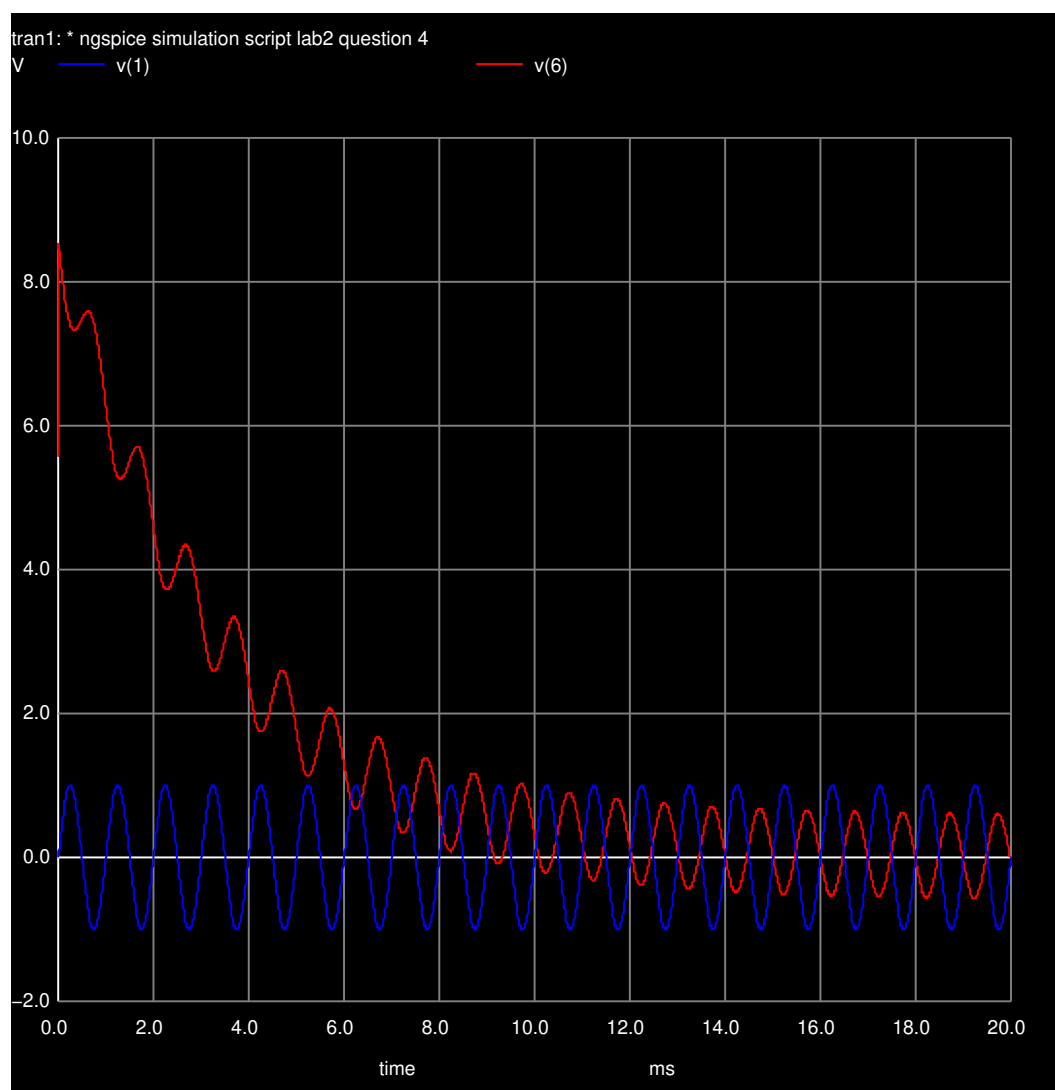part4.pdf

Figure 5: Circuit analysed.

Figure 6: Circuit analysed.
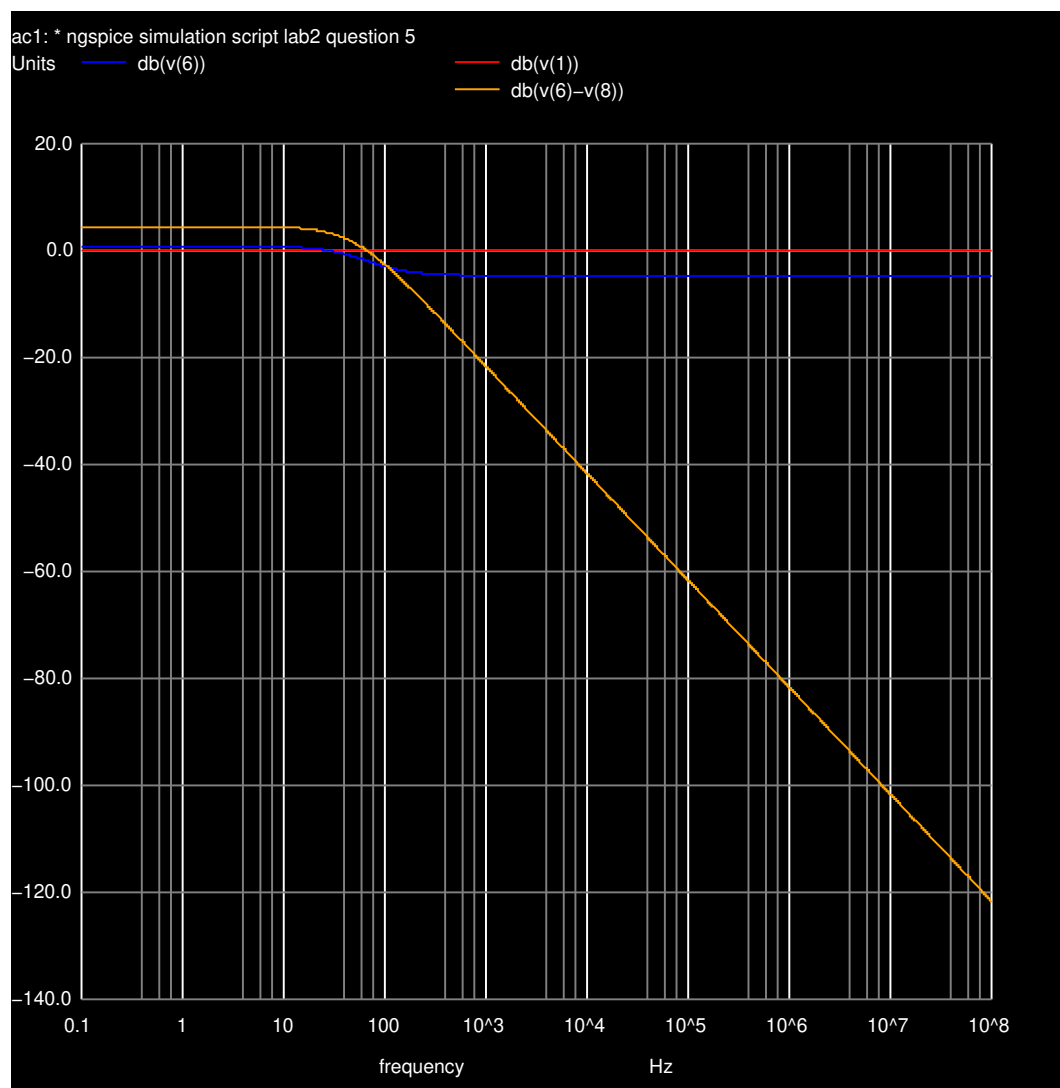
part6_amp.pdf

Figure 7: Circuit analysed.

Figure 8: Magnitude Response (in decibels)

part6_ang.pdf

Figure 9: Circuit analysed.

sim5_ph.pdf

Figure 10: Phase Response (in degrees)