Circuit Theory and Electronics Fundamentals

Department of Electrical and Computer Engineering, Técnico, University of Lisbon

T2's Laboratory Report

Group 5

Carlos Ribeiro (96364), João Diniz (96416), Paulo Clemente (96462)

April 5, 2021

Contents

1 Introduction **1**

2 Theoretical Analysis **2**

2.1 Question 1: Node Analysis 2

2.2 Question 2: Equivalent Resistance (R_{eq}) 3

2.3 Question 3: Natural solution $v_{6t}(t)$ 4

2.4 Question 4: Forced solution $v_{6t}(t)$ (f=1Khz) 5

2.5 Question 5: Final solution $v_{6t}(t)$ 6

3 Simulation analysis **6**

3.1 Question 1: 6

3.2 Question 2: 7

3.3 Question 3: 8

3.4 Question 4: 8

3.5 Question 5: 10

4 Conclusion **11**

1 Introduction

The objective of this laboratory assignment is to study a circuit with resistances, sinusoidal voltage source and a capacitor (Fig.1). The main objective is to simulate the circuit using NGspice and compare results with the theoretical analysis.

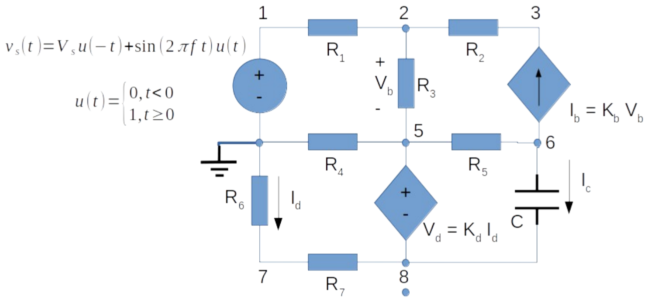

Figure 1: T2's given circuit.

Firstly, in the theoretical analysis we will determine the voltages in all nodes and the currents in all branches using the nodal method, like in the previous laboratory assignment. Then, we calculate the equivalent resistance (R_{eq}) as seen from the capacitor terminals and determined the natural solution for $V_{6n}(t)$ and plot the result from 0ms to 20ms. The forced solution was also obtainned in the same interval (for f=1KHz) and the total solution was plotted for -5ms to 20ms. At last, the frequency response was calculated and the results were analised.

At the same time, the circuit was simulated using NGspice in order to obtain the same results and, considering that some of them could be lightly different, they were compared and analised.

The data used was the following:

Name	**Value [F, V, Ω or S]**
R_1	1.04765357286e3
R_2	2.06140068334e3
R_3	3.03459085363e3
R_4	4.00398818216e3
R_5	3.11499853456e3
R_6	2.04588646991e3
R_7	1.04390152967e3
V_s	5.02522591213
C	1.00982536324e-6
K_b	7.28209304852e-3
K_c	8.36641247715e3

2 Theoretical Analysis

2.1 Question 1: Node Analysis

This preliminary analysis of the circuit is the basis of the rest of the work and to do this we used the nodal method in the same way as used in T1. We have the equations: (the nomenclature for all nodes and branches is present in figure 1).

Node 0 (ground):

$$\frac{1}{R_1}(V_2 - V_1) + \frac{1}{R_4}(V_5 - 0) - I_d = 0 \tag{1}$$

Node 2:

$$\frac{1}{R_1}(V_1 - V_2) + \frac{1}{R_2}(V_3 - V_2) + \frac{1}{R_3}(V_5 - V_2) = 0 \tag{2}$$

Node 3:
$$\frac{1}{R_2}(V_2 - V_3) + I_b = 0 \tag{3}$$
Node 5:
$$\frac{1}{R_3}(V_2 - V_5) + \frac{1}{R_5}(V_6 - V_5) + \frac{1}{R_4}(0 - V_5) - I_d = 0 \tag{4}$$
Node 6:
$$\frac{1}{R_5}(V_5 - V_6) - I_b - I_c = 0 \tag{5}$$
Node 7:
$$\frac{1}{R_6}(V_7 - 0) + I_d = 0 \tag{6}$$
Node 8:
$$\frac{1}{R_7}(V_7 - V_8) - I_d + I_c = 0 \tag{7}$$
Additional equation:
$$V_5 - V_8 - K_d I_d = 0 \tag{8}$$
Additional equation:
$$V_2 - V_5 - \frac{I_b}{K_b} = 0 \tag{9}$$

And the results are shown in the following table:

Name	**Value [A or V]**
V_1	5.025226e+00
V_2	4.724476e+00
V_3	4.104661e+00
V_5	4.765766e+00
V_6	5.702373e+00
V_7	-1.847813e+00
V_8	-2.790649e+00
I_1	-2.870701e-04
I_2	-3.006765e-04
I_3	1.360640e-05
I_4	1.190255e-03
I_5	3.006765e-04
I_6	9.031846e-04
I_7	9.031846e-04
I_s	-2.870701e-04
I_d	9.031846e-04
I_b	-3.006765e-04
I_c	-2.168404e-19

All the variables preceded by I are currents and are expressed in Ampere, the other variables, preceeded by V are voltages and are expressed in Volt.

2.2 Question 2: Equivalent Resistance (R_{eq})

The resolution of this question was based on the suggestion presented. Firstly, we set $V_s = 0$ and replaced the capacitor by a voltage source $V_x = V_6 - V_8$ and ran the nodal analysis in order to obtain the current I_x witch is the current supplied by V_x.

Then, with the equation:

$$R_{eq} = \frac{V_x}{I_x} \tag{10}$$

We obtain the Equivalent resistance.

The time constant τ is calculated by doing $\tau = R_{eq}C$

Since the capacitor was replaced by a Voltage source witch terminals have the same difference of potential as $V_6 - V_8$ in Question 1, this is a known variable that corresponds to V_x (V_{eq}). I_x can also be obtainned by running the nodal method with $V_s = 0$. We needed to do this because there is no faster way to calculate the Equivalent Resistance in a circuit where there are resistances in paralell, in series and a capacitor. Basically, the procedure was based on *Thévenin* theorem to obtain V_{eq}. By doing this, we have $V_{eq} = R_{eq}I_x$, where R_{eq} is the only unknown variable.

The results:

Name	**Value [A or V]**
V_x	8.493021e+00
V_1	0.000000e+00
V_2	0.000000e+00
V_3	-0.000000e+00
V_5	0.000000e+00
V_6	8.493021e+00
V_7	0.000000e+00
V_8	0.000000e+00
I_1	0.000000e+00
I_2	0.000000e+00
I_3	0.000000e+00
I_4	0.000000e+00
I_5	2.726493e-03
I_6	-0.000000e+00
I_7	-0.000000e+00
I_s	0.000000e+00
I_d	-0.000000e+00
I_b	0.000000e+00
I_x	2.726493e-03
$Req(kOhm)$	3.114999e+00
$tau(ms)$	3.145605e+00

2.3 Question 3: Natural solution $v_{6t}(t)$

The natural solution $v_{6t}(t)$ in the interval [0,20]ms calculating:

$$v_{6n} = V_x - V_8 e^{\frac{-t}{\tau}} \tag{11}$$

Where V_x is the same as in question 1 ($t < 0$). By doing this, the plot obtained is the one shown below:

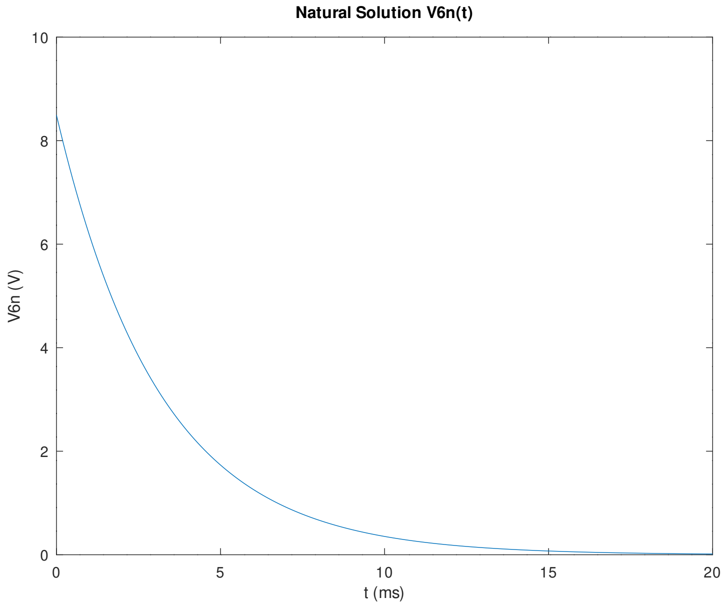

Figure 2: Plot alinea 3.

2.4 Question 4: Forced solution $v_{6t}(t)$ (f=1Khz)

In this question, we have used a phasor voltage source $V_s = 1$, so:

$$v_s = V_s e^{-i\frac{\pi}{2}} \tag{12}$$

Then, replacing C with impedance Z_c and running the nodal analysis to determine the phasor voltages in all nodes we have the following results:

Name	Value [A or V]
V_1	6.123234e-17
V_2	5.756770e-17
V_3	5.001526e-17
V_5	5.807082e-17
V_6	8.493021e+00
V_7	-2.251559e-17
V_8	-3.400403e-17

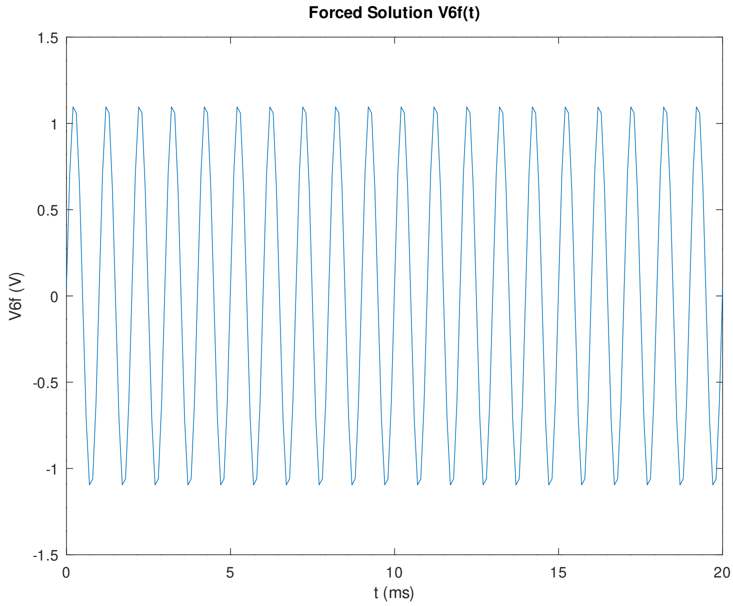

Figure 3: Plot alinea 4.

We have calculated the module of V_6 with "abs" Octave's function. Then, the phase is calculated in the expression:

$$phase = 0 + arctan(2\pi R_{eq} C) \tag{13}$$

Where R_{eq} and C are the equivalent resistance and the Capacitor's capacitance, respectively.
And, finally, the forced solution is given by the expression:

$$V_{6f} = V_6 cos(t2\pi - phase) \tag{14}$$

where V_6 is the module value, calculated in the previous expression.

2.5 Question 5: Final solution $v_{6t}(t)$

3 Simulation analysis

In this section we will be describing the simulations that we made on NGSpice where we made 4 scripts describing the given circuit (figure 1), as it is requested in T2's lab questions. After running the scripts, we will be able to compare the results we got from the theoretical analysis.
This simulation analysis includes 5 questions, from which the last two are answered using the same script (the circuit is the same, we only want different results).

3.1 Question 1:

For the first question of this simulation analysis we begun by adding the values for all the components used in the circuit, thus describing the circuit in order to simulate it.
All the simulation was done using the same node nomenclature as in figure 1.
After describing the circuit in a NGSpice script we simulated the operating point for $t < 0$, for which $v_s = V_s$, in order to obtain the volatges in all nodes and the currents in all branches.
The results we got are in the table below:

Name	Value [A or V]
@c0[i]	0.000000e+00
@g0[i]	-3.00677e-04
@r1[i]	2.870701e-04
@r2[i]	-3.00677e-04
@r3[i]	-1.36064e-05
@r4[i]	1.190255e-03
@r5[i]	-3.00677e-04
@r6[i]	9.031846e-04
@r7[i]	9.031846e-04
n1	5.025226e+00
n2	4.724476e+00
n3	4.104661e+00
n5	4.765766e+00
n6	5.702373e+00
n7	-1.84781e+00
n8	-2.79065e+00
n9	-2.79065e+00

3.2 Question 2:

After completing the first step of the simulation analysis, we replaced the capacitor with a voltage source V_x whose voltage is equal to $v_6 - v_8$ as obtained in question 1, and did the same operating point simulation in order to obtain the voltages and currents in the circuit.

The reason we have to do this step is in all ways similar to the reason in question 2 of the theoretical analysis and is described in section 2.2.

The results we got from the simulation are in the table below:

Name	Value [A or V]
@g0[i]	0.000000e+00
@r1[i]	0.000000e+00
@r2[i]	0.000000e+00
@r3[i]	0.000000e+00
@r4[i]	0.000000e+00
@r5[i]	2.726492e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
n1	0.000000e+00
n2	0.000000e+00
n3	0.000000e+00
n5	0.000000e+00
n6	-8.49302e+00
n7	0.000000e+00
n8	0.000000e+00
n9	0.000000e+00

3.3 Question 3:

After getting the results from question 2 we then proceeded to make another NGSpice script simulating the natural response of the circuit (with no voltage sources connected), using the boundary conditions v_6 and v_8 as obtained in question 2.

In this question we made a transient analysis in order to get v_6 during the [0,20] ms time window. This is, the variation of the voltage in node 6 as the capacitr discharges.

The result we got is in the plot below:

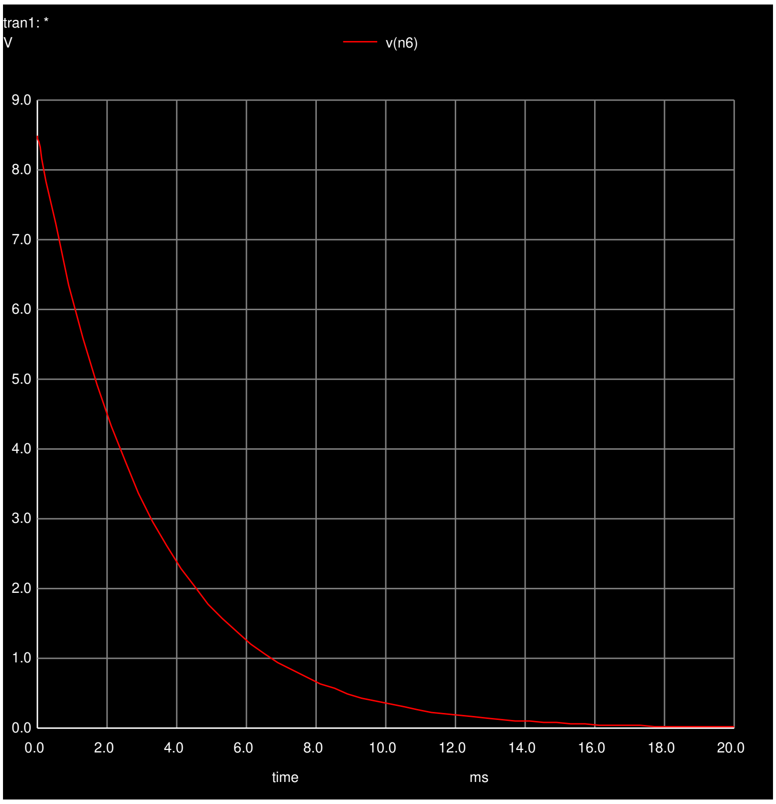

Figure 4: Transient Analysis of Question 3.

3.4 Question 4:

This question and the next were answered using the same script as it features the same exact circuit.

In these two questions, we are simulating the circuit with $v_s(t)$ as given in figure 1.

In the present question 4, we will be simulating the total response (forced and natural together) on node 6 using a frequency of 1kHz on the voltage source v_s, and the same time interval as in question 3.

The results we got are in the plots below, with the first one being the stimulus and the second one the response:

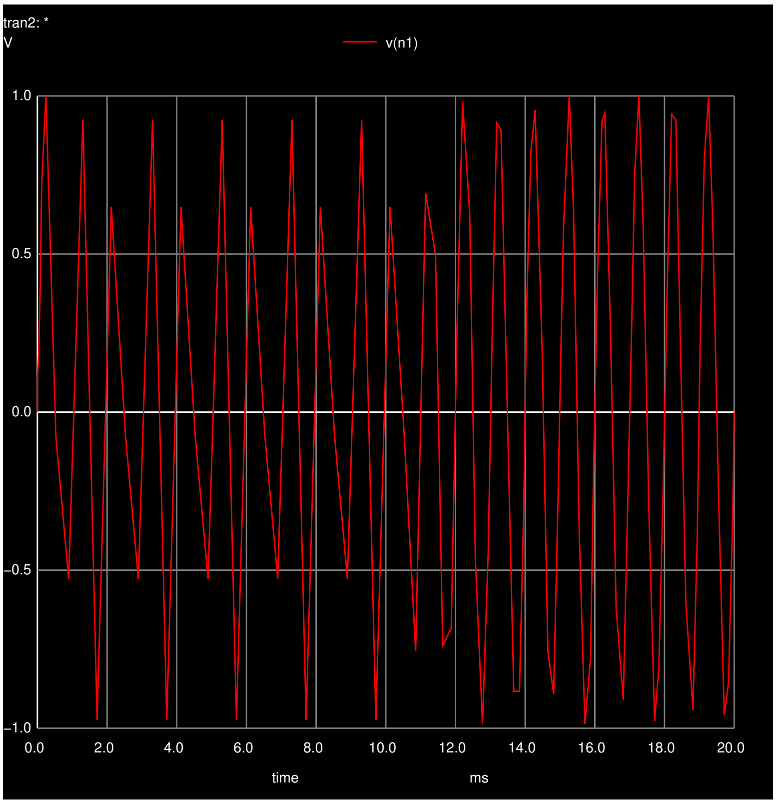

Figure 5: Transient Analysis of the Stimulus on Question 4.

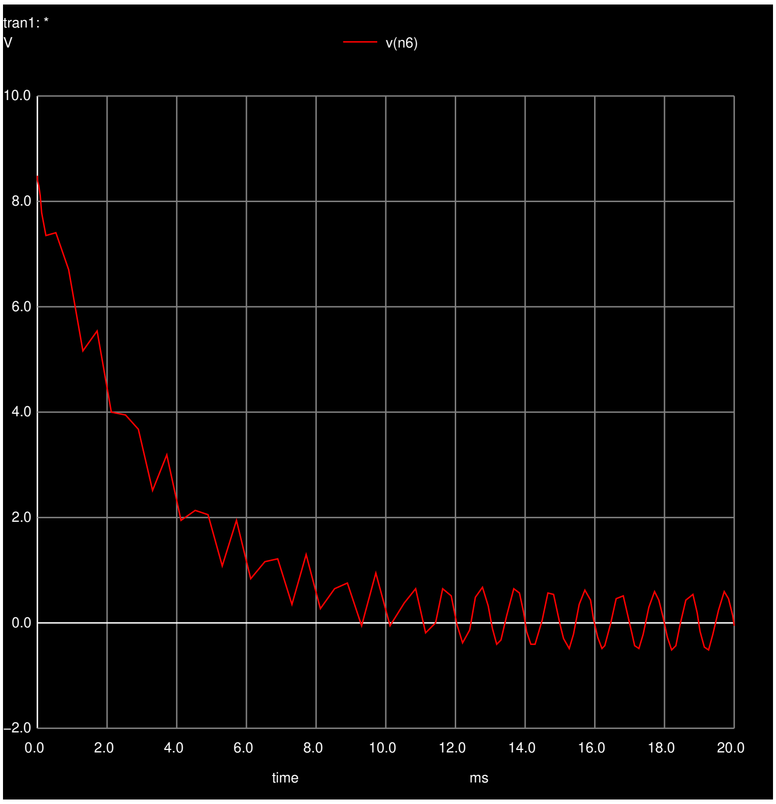

Figure 6: Transient Analysis of the Response on Question 4.

3.5 Question 5:

As said before, the circuit in this question is the exact same as the one in question 4.

In this question we simulated the frequency response in node 6 for the frequency range $[0.1, 1]MHz$. The figure below shows the results we got for not only the node 6 but also v_s, we did this for comparation purposes. The frequency is in logscale, the magnitude in dB and phase in degrees.

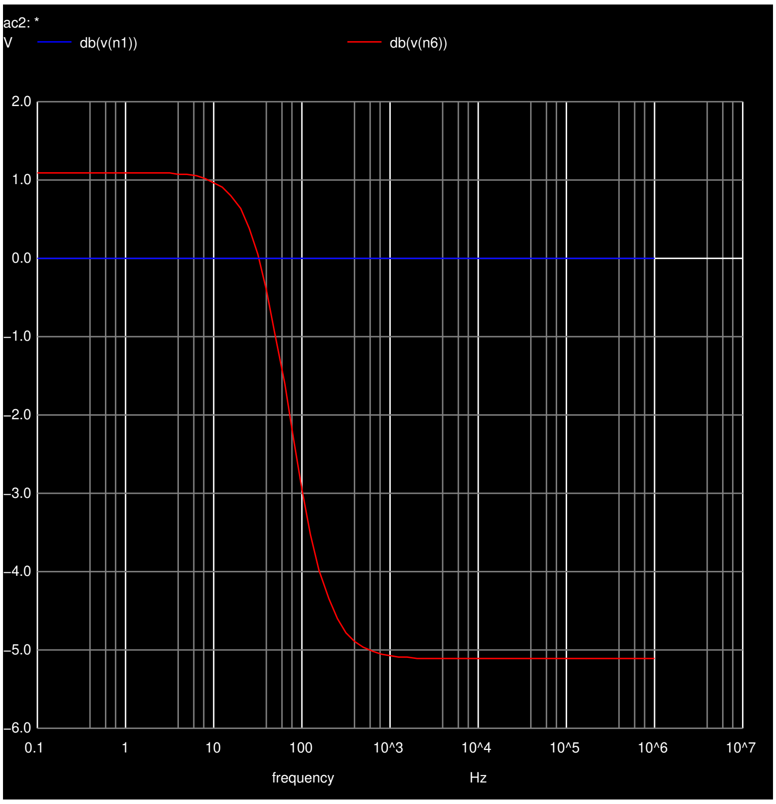

Figure 7: Frequency Analysis of Question 5.

EXPLICAÇÃO

4 Conclusion

Summing up, in this laboratory assignment (T2) we have made both theoretical analysis as well as simulations of several different but similar and correlated circuits.

The results obtained in both analysis were either put in a table or plotted in order to make comparations between the theoretical and simulation analysis. The reasoning behind these results is that we studied a very simple circuit containing only linear components. If the components used were more complex, the case would not be the same and we could detect real differences between the values calculated and simulated.

That being said we would like to end this report by presenting both the theoretical and simulation results side by side in order to easily compare the results.

Name	Value [A or V]
V_1	5.025226e+00
V_2	4.724476e+00
V_3	4.104661e+00
V_5	4.765766e+00
V_6	5.702373e+00
V_7	-1.847813e+00
V_8	-2.790649e+00
I_1	-2.870701e-04
I_2	-3.006765e-04
I_3	1.360640e-05
I_4	1.190255e-03
I_5	3.006765e-04
I_6	9.031846e-04
I_7	9.031846e-04
I_s	-2.870701e-04
I_d	9.031846e-04
I_b	-3.006765e-04
I_c	-2.168404e-19

Figure 8: Theoretical Question 1

Name	Value [A or V]
@c0[i]	0.000000e+00
@g0[i]	-3.00677e-04
@r1[i]	2.870701e-04
@r2[i]	-3.00677e-04
@r3[i]	-1.36064e-05
@r4[i]	1.190255e-03
@r5[i]	-3.00677e-04
@r6[i]	9.031846e-04
@r7[i]	9.031846e-04
n1	5.025226e+00
n2	4.724476e+00
n3	4.104661e+00
n5	4.765766e+00
n6	5.702373e+00
n7	-1.84781e+00
n8	-2.79065e+00
n9	-2.79065e+00

Figure 9: Simulation Question 1

Name	Value [A or V]
V_x	8.493021e+00
V_1	0.000000e+00
V_2	0.000000e+00
V_3	-0.000000e+00
V_5	0.000000e+00
V_6	8.493021e+00
V_7	0.000000e+00
V_8	0.000000e+00
I_1	0.000000e+00
I_2	0.000000e+00
I_3	0.000000e+00
I_4	0.000000e+00
I_5	2.726493e-03
I_6	-0.000000e+00
I_7	-0.000000e+00
I_s	0.000000e+00
I_d	-0.000000e+00
I_b	0.000000e+00
I_x	2.726493e-03
$Req(kOhm)$	3.114999e+00
$tau(ms)$	3.145605e+00

Figure 10: Theoretical Question 2

Name	Value [A or V]
@g0[i]	0.000000e+00
@r1[i]	0.000000e+00
@r2[i]	0.000000e+00
@r3[i]	0.000000e+00
@r4[i]	0.000000e+00
@r5[i]	2.726492e-03
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
n1	0.000000e+00
n2	0.000000e+00
n3	0.000000e+00
n5	0.000000e+00
n6	-8.49302e+00
n7	0.000000e+00
n8	0.000000e+00
n9	0.000000e+00

Figure 11: Simulation Question 2

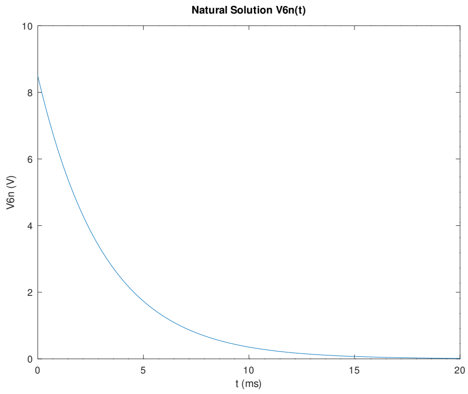

Figure 12: Theoretical Question 3

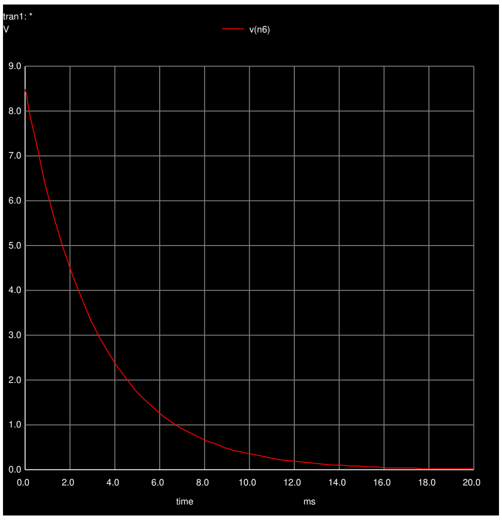

Figure 13: Simulation Question 3

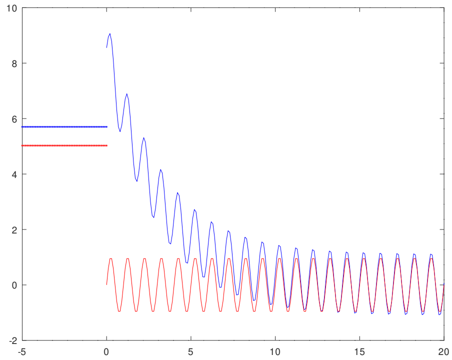

Figure 14: Theoretical Question 5

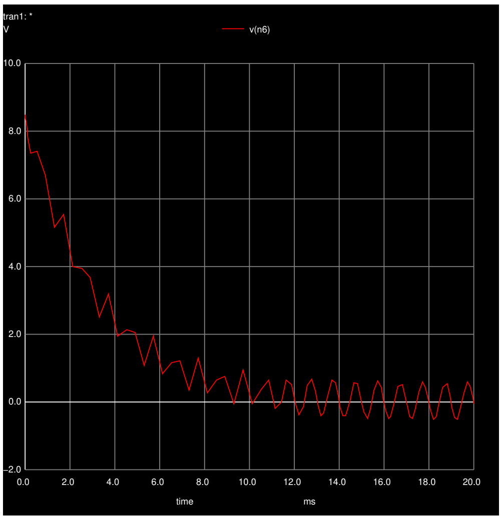

Figure 15: Simulation Question 4