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

April 8, 2021

Contents

1 Introduction

In this laboratory assignment we study a circuit (Fig. 1) containing various elements, to be more specific, 1 capacitor, 7 resistances, 1 independent voltage source V_s, 1 dependent voltage source and 1 dependent current source.

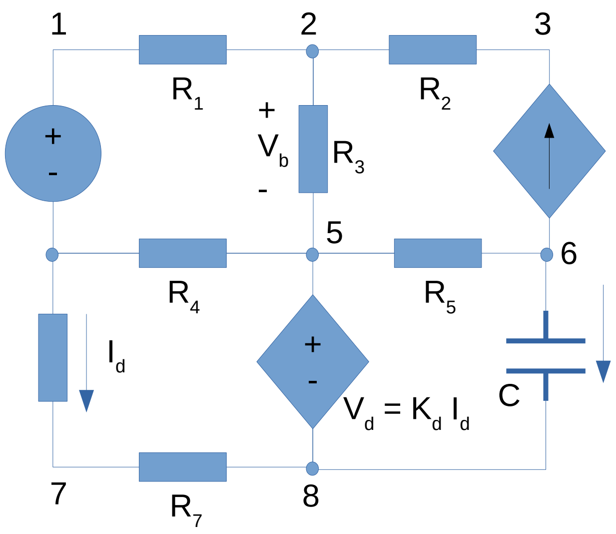


Figure 1: The RC Circuit

The independent voltage source V_s varies in time exactly as it follows:

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

Where

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

The next table displays the data generated automatically by the Python Script:

Octave - Voltages (V)	
R1	1.013609e+03 Ohm
R2	2.016578e+03 Ohm
R3	3.006816e+03 Ohm
R4	4.049229e+03 Ohm
R5	3.053925e+03 Ohm
R6	2.092502e+03 Ohm
R7	1.022320e+03 Ohm
C	1.029587e-06 F
Kb	7.213324e-03 A/V
Kd	8.321035e+03 V/A

Table 1: Initial data

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

In this section we will explain with some detail how do we analyse the circuit theoretically. To do so, and because there were several things to be analyse, the following subsections are going to detail the analysis of each of the six powerpoint points related to the theoretical analysis. Just as a final note, the results obtained theoretically will only be shown in the symulation section, to compare the both.

2.1 Point 1: Determine the initial state of the circuit

Initially, the circuits's independent voltage source inputs in the circuit a constant voltage. Because of this, and considering that the circuit is working for a really long time (started working for t = $-\infty$), we can assume that the capacitor is already fully charged and, consequentaly, it behaves like an open circuit. So to analyse the circuit's initial state we just need to do a node analysis just like we did in *laboratory nº1*.

After performing the node analysis we've obtained the following matrix:

$$\begin{pmatrix} 1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\ -G_1 & G_1+G_2+G_3 & -G_2 & 0 & -G_3 & 0 & 0 & 0 \\ 0 & -G_2-K_b & G_2 & 0 & K_b & 0 & 0 & 0 \\ G_1 & -G_1 & 0 & G_4+G_6 & -G_4 & 0 & -G_6 & 0 \\ 0 & 0 & 0 & -K_d*G_6 & 1 & 0 & K_d*G_6 & 0 \\ 0 & K_b & 0 & 0 & -G_5-K_b & G_5 & 0 & 0 \\ 0 & 0 & 0 & -G_6 & 0 & 0 & G_6+G_7 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \end{pmatrix} \begin{pmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix} = \begin{pmatrix} V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

After resolving the matrix, we've obtained the theoretical results for the voltages in all nodes.

2.2 Point 2: Determine the equivalent resistance seen by the capacitor

To determine the equivalent resistance, we've defined a voltage source Vx between the nodes 6 and 8 and we've powered off the independent voltage source Vs. After that, we use the node analysis to determine the volatge in all nodes, which resulted in the following matrix:

$$\begin{pmatrix} 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 \\ -G_1 & G_1+G_2+G_3 & -G_2 & 0 & -G_3 & 0 & 0 & 0 \\ 0 & -G_2-K_b & G_2 & 0 & K_b & 0 & 0 & 0 \\ G_1 & -G_1 & 0 & G_4+G_6 & -G_4 & 0 & -G_6 & 0 \\ 0 & 0 & 0 & -K_d*G_6 & 1 & 0 & K_d*G_6 & 0 \\ 1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -G_6 & 0 & 0 & G_6+G_7 & -G_7 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \end{pmatrix} \begin{pmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix} = \begin{pmatrix} V_x \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

After knowing these voltages, it's easy to determine the currents in all branches. To determine the equivalent resistance we need to determine the current in the branch 6-8, Ix, just because $R_{eq} = V_x/I_x$. We've obtained Ix by analysing the node 8 and we've obtained the following expression for Ix:

$$I_x = -Kb*(V_2 - V_5) + G_5*(V_5 - V_6) \tag{3}$$

After knowing Ix, we now can obtain Req, using the previous shown formula.

2.3 Point 3: Obtaining the natural solution for the capacitor

Using the results in point 2, we can determine the natural solution of the circuit by making a simple RC circuit, with the only components being the capacitor and the equivalent resistance, just as shown in the figure:

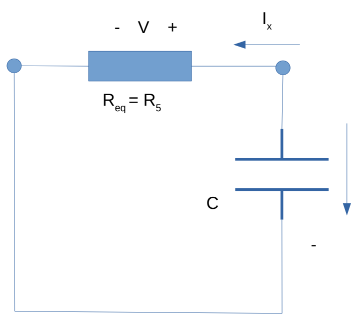


Figure 2: Circuit3

Using the Kirchhoff's voltage law for this circuit, we've obtain the folling diferential equation:

$$\frac{dV}{dt} + \frac{V}{C*R_{eq}} = 0 \tag{4}$$

Knowing that the $V_{capacitor_i} = V_6 - V_8$ and using the values for these voltages obtained in point 1, the natural solution for the capacitor is given by:

$$V_{6n} = V_{capacitor_i} * e^{-\frac{t}{C*R_{eq}}} \tag{5}$$

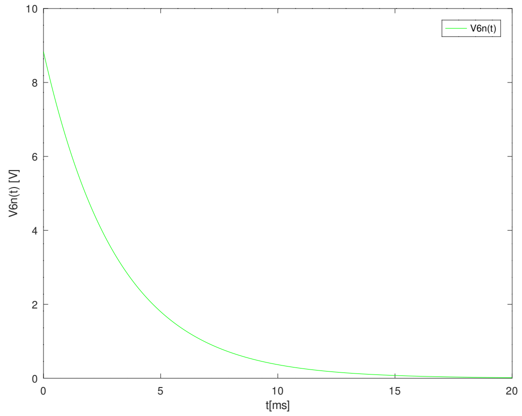


Figure 3: Natural Response

2.4 Point 4: Determine the forced solution for the circuit

We already know that in a forced circuit the voltages in all nodes will gain the same frequency as the source. Knowing this we know that the solution for the node i will be $V_i = V_{i_{max}} * cos(w * t + \phi_i)$. So, in order to determine the voltages for all nodes, we just need to determine the voltages amplitudes and phases, or in other words, their complex amplitude. This brings an interesting point: if we consider the complex world the solutions will be in the form of $V_i = V_{i_{max}} * e^{w*t} * e^{\phi_i}$, which makes easy to obtain both the amplitudes and the phases. Alongside this, we just need to solve the node analysis for all the circuit, which will result in the following matrix:

$$\begin{pmatrix} 1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\ -G_1 & G_1+G_2+G_3 & -G_2 & 0 & -G_3 & 0 & 0 & 0 \\ 0 & -G_2-K_b & G_2 & 0 & K_b & 0 & 0 & 0 \\ G_1 & -G_1 & 0 & G_4+G_6 & -G_4 & 0 & -G_6 & 0 \\ 0 & 0 & 0 & -K_d*G_6 & 1 & 0 & K_d*G_6 & -1 \\ 0 & K_b & 0 & 0 & -G_5-K_b & G_5+\frac{1}{Z_c} & 0 & -\frac{1}{Z_c} \\ 0 & 0 & 0 & -G_6 & 0 & 0 & G_6+G_7 & -G_7 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \end{pmatrix} \begin{pmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{pmatrix} = \begin{pmatrix} V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

As explained, all nodes will have the same frequency, so by using the complex amplitudes we can use the matrix obtained in the nodal analysis just to determine these amplitudes. To explain thsi statement let's take a look in what happens in the node 6:

$$K_b V_2 + (-K_b - G_5)V_5 + (G_5 + \frac{1}{Z_c})V_6 - \frac{1}{Z_c}V_8 \tag{6}$$

And this is what we see in the matrix

Solving the matrix using the given values, the complex amplitudes generated are the following:

Octave - Voltages (V)	
V1	1.000000e+00 + i(-1.549812e-33) V
V2	9.469010e-01 + i(-1.466518e-17) V
V3	8.361543e-01 + i(5.184518e-16) V
V4	0.000000e+00 + i(-1.549812e-33) V
V5	9.545144e-01 + i(-5.131501e-17) V
V6	-5.667484e-01 + i(-8.549146e-02) V
V7	-3.836423e-01 + i(2.062473e-17) V
V8	-5.710758e-01 + i(3.070122e-17) V

Table 2: Complex Amplitudes - Octave Results

2.5 Point 5: Determine the solution for the voltage in node 6

Using the data obtained in the points 3 and 4 is relatively easy to find the solution for the voltage in the node 6. We just need to find the forced solution using the amplitude and phase for the node 6 obtained in point 4 e aplying it in this formula: $V_{6f} = V_{6_{max}} * cos(w * t + \phi_6)$. After that it's just add it to the natural solution and we've obtained the following:

$$V_6(t) = \begin{cases} V_{6i} & \text{para t} \in [-5, 0] \\ V_{6n}(t) + V_{6f}(t) = V_{6i}e^{\frac{t}{C*R_5}} + V_{6_{max}}cos(\omega t - \phi_6) \end{cases} \tag{7}$$

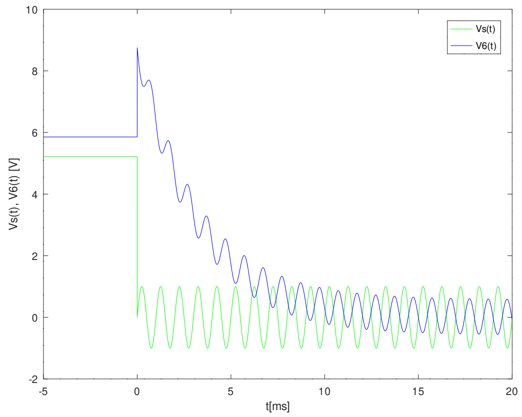


Figure 4: Natural and forced response

Note: In this plot we can also see the $V_s(t)$, which is detailed in the introduction.

2.6 Point 6: Determine the frequency response for Vc(f), V6(f) and Vs(f)

In this final subsection we want to determine the frequency response for Vc(f), V6(f) and Vs(f) for f belonging to [0.1;1M](Hz). To do so we've repeated the point 4 of this analysis, but varying the frequencys, which are logaritmic spaced from one another just to plot the grapth related to the $log_{10}(f)$. The results can be seen below alongside their description.

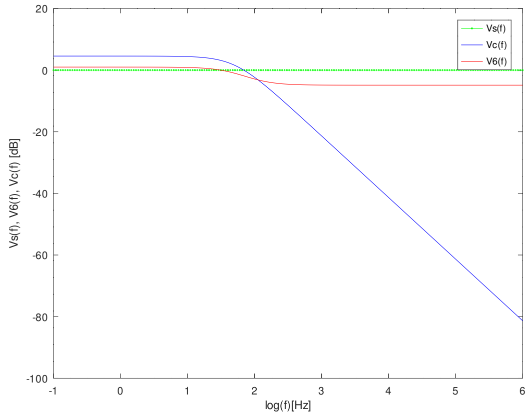


Figure 5: Amplitude

In the graphic shown above we can see the amplitude of the 3 voltages that we are studing. As predicted, Vs(f) is constant because is the user that inputs this value and just looking at the formula in the introduction we see that Vs = 1V during all the time studied. The value of Vs is 0 in the graph due to Vs = 1V = 20*log(1)[dB] = 0 dB. Relatively to V6(f), we see that it decreases slightly with the frequency's increase, but it tends to estabalize. As for Vc, it decreases a lot with the frequency increse, which is according to the expected, since the impedance of the capacitor is $Z_c = 1/(iwC)$.

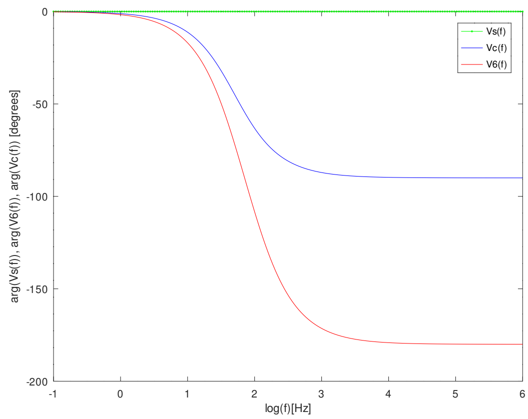


Figure 6: Arguments

The above graphic now details the phases of the 3 studied voltages. The phase of Vs is once more constant and equal to 0 since itś provided by input. The phases of the othe voltages tend to decrease as the frequency increses, but estabalizing near the -90 degrees for Vc and near -180 degrees for V6.

3 Simulation Analysis

3.1 Operating point analysis for t<0

In this section, an operating point analysis of the circuit (referência ao circuito) was conducted in order to calculate the voltage in all nodes and the current through the resistors for a t $<$ 0. 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. It is important to note that an extra voltage source, Vaux, was added and therefore, another node was also added (node 9). This Vaux was intended to allow the measurement of the current Id which voltage source Vd 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 7 voltage is equal to node's 9 voltage) and allowed us to obtain the current through it.

NgSpice - Voltages (V)	
@gb[i]	-2.86455e-04
@r1[i]	2.732477e-04
@r2[i]	-2.86455e-04
@r3[i]	-1.32073e-05
@r4[i]	1.229564e-03
@r5[i]	-2.86455e-04
@r6[i]	9.563162e-04
@r7[i]	9.563162e-04
v(1)	5.216040e+00
v(2)	4.939074e+00
v(3)	4.361415e+00
v(5)	4.978786e+00
v(6)	5.853598e+00
v(7)	-2.00109e+00
v(8)	-2.97875e+00
v(9)	0.000000e+00

Octave - Voltages (V)	
V1	5.216040e+00 V
V2	4.939074e+00 V
V3	4.361415e+00 V
V4	-0.000000e+00 V
V5	4.978786e+00 V
V6	5.853598e+00 V
V7	-2.001094e+00 V
V8	-2.978754e+00 V

Table 3: Nodal Voltage Comparison

3.2 Calculus of Req - Simulation

Similarly to the last section, an operating point analysis to the circuit (referência ao circuito) was conducted, with the difference being that the voltage source vs was turned off and the capacitor was replaced by the independent voltage source Vx which corresponds to the value of $v(6)$-$v(8)$. This Vx is equivalent to the voltage in the capacitor's terminals. The values of currents and nodal voltages were then put in a table, while the equivalent thevenin resistor was calculated by the following equation:

$$Req = (v(6) - v(8))/vxbranch, \tag{8}$$

with $vxbranch$ corresponding to the current Ix that flows through the Vx's branch.

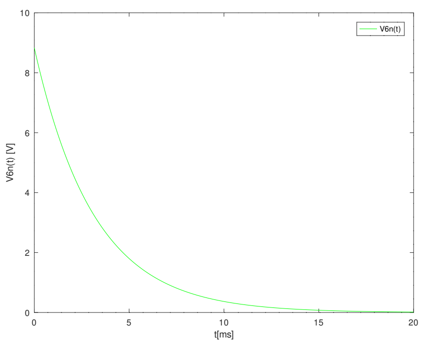
NgSpice - Voltages (V)	
@gb[i]	0.000000e+00
@r1[i]	0.000000e+00
@r2[i]	0.000000e+00
@r3[i]	0.000000e+00
@r4[i]	0.000000e+00
@r5[i]	-1.63724e-02
@r6[i]	0.000000e+00
@r7[i]	0.000000e+00
v(1)	0.000000e+00
v(2)	0.000000e+00
v(3)	0.000000e+00
v(5)	0.000000e+00
v(6)	5.000000e+01
v(7)	0.000000e+00
v(8)	0.000000e+00
v(9)	0.000000e+00

Octave - Voltages (V)	
V1	0.000000e+00 V
V2	-0.000000e+00 V
V3	-0.000000e+00 V
V4	-0.000000e+00 V
V5	0.000000e+00 V
V6	5.000000e+01 V
V7	0.000000e+00 V
V8	-0.000000e+00 V

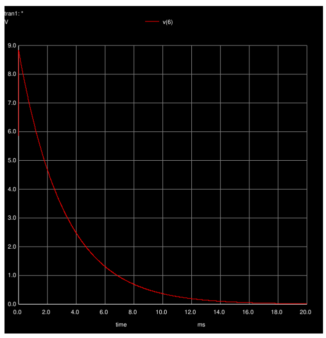
Table 4: Nodal Voltage Comparison

3.3 Transient Analysis for t $\geq$ 0 (Natural Solution)

In this section, a transient analysis was conducted in order to evaluate the natural response of the circuit, which means, the variation over time. Once that to calculate the natural response, the voltage source vs is turned off, the circuit simulated was equal to the one in the previous question. From this, the voltage in the capacitor's over time (time interval considered was [0,20]ms) was calculated and plotted, using Vx (calculated in the previous section) as the initial condition for the capacitor's voltage.



(a) Natural Response (Octave)

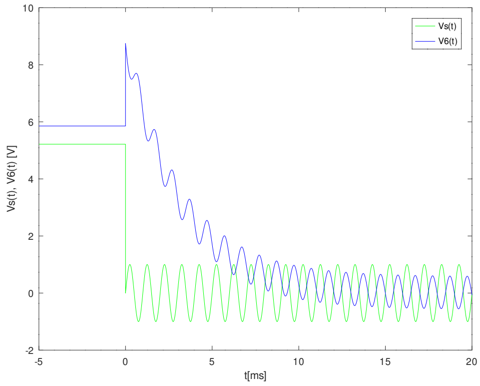


(b) Natural Response (NGSpice)

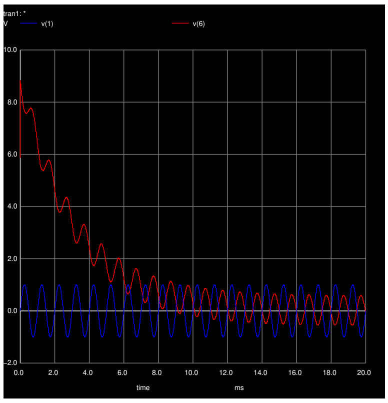
When observing the graph obtained from the simulation in Ngspice, we see that the capacitor's voltage overtime is a negative exponential matching the one obtained from the theorethical analysis in octave.

3.4 Operating Point Analysis for t $\geq$ 0 (Natural and Forced Solution)

In this section, as previously, a transient analysis was conducted in order to evaluate the natural and forced response of the circuit. In order to achieve this, the procedure adopted was the same as the one in the previous step, but with the voltage souce $vs(t)$ consisting of a sinusoidal wave sin(2*π*f).



(a) Natural and forced response (Octave)



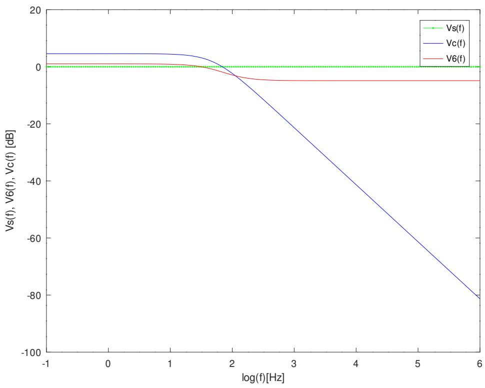
(b) Natural and forced response

When observing the graph obtained from the simulation Ngspice, it is possible to conclude that, over the period of time considered, the voltage in the capacitor tends to diminuish until its phase differs π from the phase of the voltage source, such as in the graph obtained from the theorethical analysis in octave.

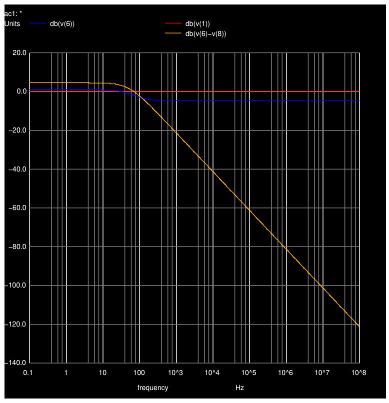
3.5 Frequency Responses

In this part of the assignment, an AC (Alternating Current) Analysis was conducted, in order to match the goal mentioned above. This type of analysis allows to study the frequency response of the circuit. For this, there is no frequency variation (steady-state analysis). 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.

3.5.1 Frequency Responses - Amplitude



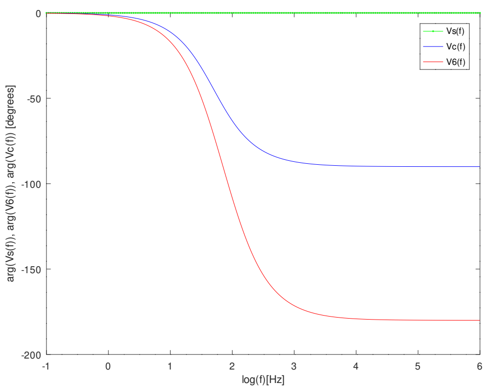
(a) Amplitude (Octave)



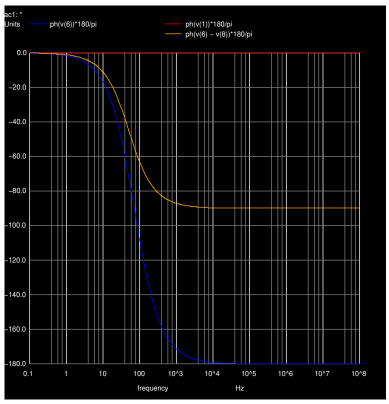
(b) Amplitude (NgSpice)

As expected, the results obtained from the ngspice simulation and the theorethical analysis in octave match.

3.5.2 Frequency Responses - Phase



(a) Arguments (Octave)



(b) Arguments (NgSpice)

As expected, the results obtained from the ngspice simulation and the theorethical analysis in octave match.

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. When comparing these last two we conclude that there aren't any disparity between the results and therefore no errors associated.
So we conclude that the methods utilized to analyse the circuit in question can be validated.