Circuits Theory and Eletronic Fundamentals

Aerospace Engineering Master's Degree

Laboratory 2 Report

Group 56

Alexandre Couto 95766
Bruno Pinto 95774
Hugo Aranha 95796

05/04/2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing a sinusodal voltage source V_s, a capacitor C, a linear voltage dependent current source I_b, a linear current dependent voltage source V_d and multiple resistors $R1, ..., R7$. The circuit can be seen in Figure 1.

In Section Theorical analysis and simulation results, the results of all the theoretical analysis of the circuit are presented while comparing these to the ones obtained in ngspice simulation. The conclusions of this study are outlined in Section Conclusion.

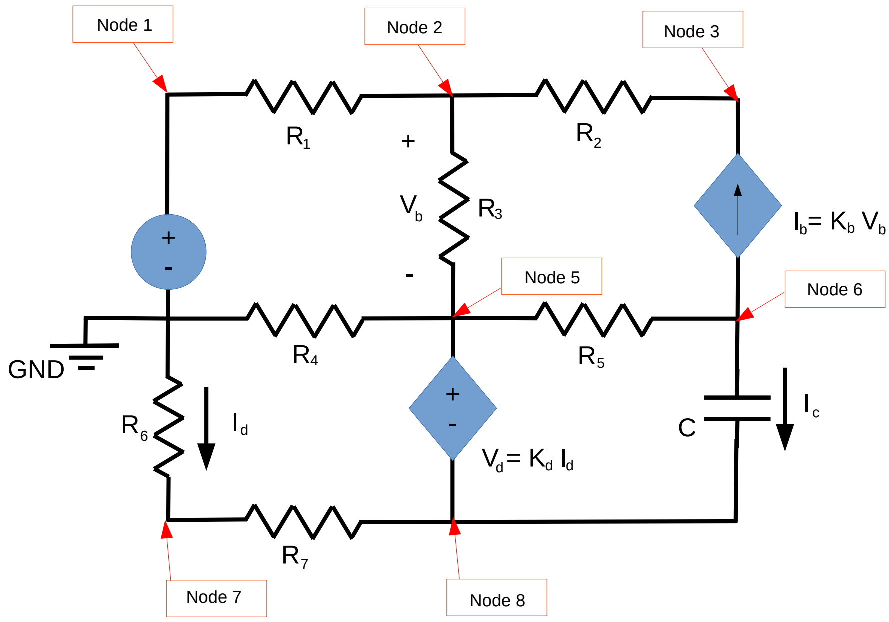


Figure 1: Circuit to be analyzed in this report.

2 Theoretical Analysis and Simulation Results

In this section, the circuit shown in Figure 1 is analysed theoretically and is simulated too, following the order of the topics that are in presentation of this laboratory.

2.1 Topic 1 - T/S

This topic corresponds to the first theoretical and the first simulation topics.

Using the nodal method (based on the Kirchhoff's Current Law (KCL)), since we have 8 different nodes we must have 8 different equations in order to have a solvable system of equations, therefore we obtain the following set of equations:

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

$$V_1 = V_S; \tag{2}$$

$$\frac{V_2 - V_1}{R_1} + \frac{V_2 - V_5}{R_3} + \frac{V_2 - V_3}{R_2} = 0; \tag{3}$$

$$\frac{V_3 - V_2}{R_2} - K_b \times (V_2 - V_5) = 0; \tag{4}$$

$$\frac{V_5 - V_0}{R_4} + \frac{V_5 - V_2}{R_3} + \frac{V_5 - V_6}{R_5} + \frac{V_8 - V_7}{R_7} - I_C = 0 \iff \frac{V_3 - V_0}{R_4} + \frac{V_5 - V_2}{R_3} + \frac{V_5 - V_6}{R_5} + \frac{V_8 - V_7}{R_7} = 0; \tag{5}$$

$$\frac{V_6 - V_5}{R_5} + K_b \times (V_2 - V_5) + I_C = 0 \iff \frac{V_6 - V_5}{R_5} + K_b \times (V_2 - V_5) = 0; \tag{6}$$

$$\frac{V_7 - V_0}{R_6} + \frac{V_7 - V_8}{R_7} = 0 \iff \frac{V_7}{R_6} + \frac{V_7 - V_8}{R_7} = 0; \tag{7}$$

$$V_5 - V_8 = K_d \times \frac{V_7}{R_6}; \tag{8}$$

being Equation 1 referent to node 0, Equation 2 to node 1, Equation 3 to node 2, Equation 4 to node 3, Equation 5 to node 5, Equation 6 to node 6, Equation 7 to node 7 and Equation 8 to node 8. In equations 5 and 6, considering $I_c = C * \frac{dVc}{dt}$ and being the circuit static, V_c is a constant so I_c is zero.

The results of the simulation and the theoretical results are presented in the following tables:

Name	Value [A or V]
V0	0.00000000000
V1	5.19519931250
V2	4.98987457574
V3	4.55661925037
V5	5.01921537838
V6	5.65928593115
V7	-2.01261748847
V8	-3.01962752440

Table 1: Theoretical Results for topic 1

Name	Value [A or V]
@c1[i]	0.000000e+00
@gb[i]	-2.08664e-04
@r1[i]	1.992363e-04
@r2[i]	2.086637e-04
@r3[i]	-9.42740e-06
@r4[i]	1.200363e-03
@r5[i]	-2.08664e-04
@r6[i]	1.001127e-03
@r7[i]	1.001127e-03
v(1)	5.195199e+00
v(2)	4.989875e+00
v(3)	4.556619e+00
v(5)	5.019215e+00
v(6)	5.659286e+00
v(7)	-2.01262e+00
v(8)	-2.01262e+00
v(9)	-3.01963e+00

Table 2: Simulation Results for topic 1

As we can see both tables have the same results and the reason for that is that ngspice uses the same models (KCL/KVL) as we used in octave.

2.2 Topic 2-T/S

This topic corresponds to the second theoretical and the second simulation topics. Using the nodal method (based on the Kirchhoff's Current Law (KCL)), since we have 8 different nodes we must have 8 different equations in order to have a solvable system of equations, therefore we obtain the following set of equations:

$$V_0 = 0; \tag{9}$$

$$V_1 = 0; \tag{10}$$

$$\frac{V_2 - V_1}{R_1} + \frac{V_2 - V_5}{R_2} = 0; \tag{11}$$

$$\frac{V_3 - V_2}{R_2} - K_b \times (V_2 - V_5) = 0; \tag{12}$$

$$\frac{V_7}{R_6} + \frac{V_7 - V_8}{R_7} = 0; \tag{13}$$

$$V_6 - V_8 = V_x; \tag{14}$$

$$V_5 - V_8 + K_d \times \frac{V_7}{R_6} = 0; \tag{15}$$

$$\frac{V_5}{R_4} + \frac{V_5 - V_2}{R_3} + \frac{V_5 - V_6}{R_5} + \frac{V_8 - V_7}{R_7} + \frac{V_6 - V_5}{R_5} + K_b(V_2 - V_5) = 0 \iff \frac{V_5}{R_4} + \frac{V_5 - V_2}{R_3} + \frac{V_8 - V_7}{R_7} + K_b(V_2 - V_5) = 0; \tag{16}$$

The results of the simulation and the theoretical results are presented in the following tables:

Name	Value [A or V]
Ic	-0.00282933503
Req	-3067.47464193000
Tau	0.00319445744
V0	0.00000000000
V1	0.00000000000
V2	-0.00000000000
V3	0.00000000000
V5	0.00000000000
V6	8.67891345555
V7	-0.00000000000
V8	-0.00000000000

Table 3: Theoretical Results for topic 2

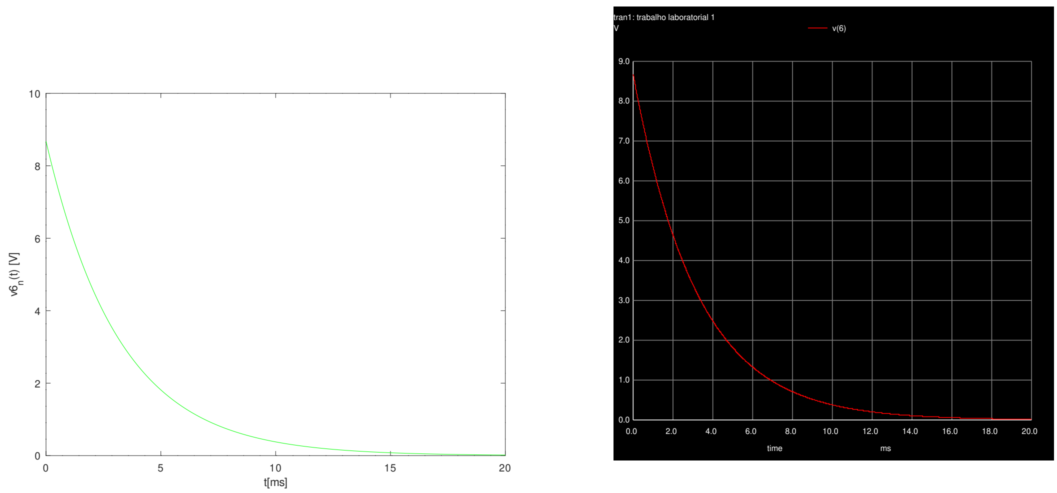
As we can see both tables have the same results and the reason for that is that ngspice uses the same models (KCL/KVL) as we used in octave. This procedure is important to allow us to compute the equivalent resistor in the circuit which will be used in the calculation of the natural solution. In ngspice, this allows us to present the natural solution in node 6 as will be presented in the next topic.

Name	Value [A or V]
@gb[i]	1.053111e-17
@r1[i]	-1.00553e-17
@r2[i]	-1.05311e-17
@r3[i]	4.757942e-19
@r4[i]	2.124111e-18
@r5[i]	-2.82934e-03
@r6[i]	1.734723e-18
@r7[i]	1.830908e-18
v(1)	0.000000e+00
v(2)	1.036259e-14
v(3)	3.222868e-14
v(5)	8.881784e-15
v(6)	8.678914e+00
v(7)	-3.48740e-15
v(8)	-3.48740e-15
v(9)	-5.32907e-15

Table 4: Simulation Results for topic 2

2.3 Topic 3-T/S

This topic corresponds to the third theoretical and the third simulation topics.



(a) (b)

Figure 2: (a) Natural solution of V6 using octave (b) Natural solution of V6 using ngspice

Looking to the graphics, we can see that both produce similar results.

2.4 Topic 4-T

This topic corresponds to the fourth theoretical.

Name	Value [A or V]
V0	0.00000000000+(0.00000000000)j
V1	0.00000000000+(-1.00000000000)j
V2	-0.00000000000+(-0.96047798662)j
V3	-0.00000000000+(-0.87708266349)j
V5	-0.00000000000+(-0.96612566265)j
V6	-0.08302511326+(0.57709770658)j
V7	0.00000000000+(0.38739947544)j
V8	0.00000000000+(0.58123420157)j

Table 5: Theoretical results of topic 4

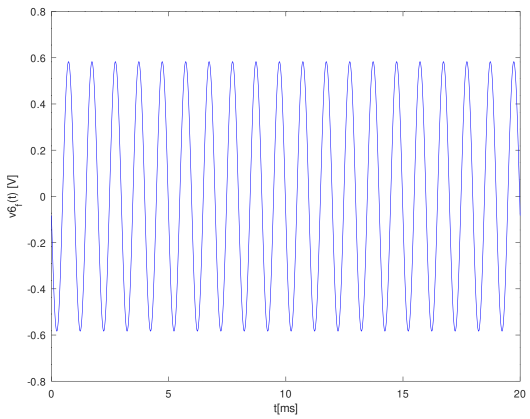
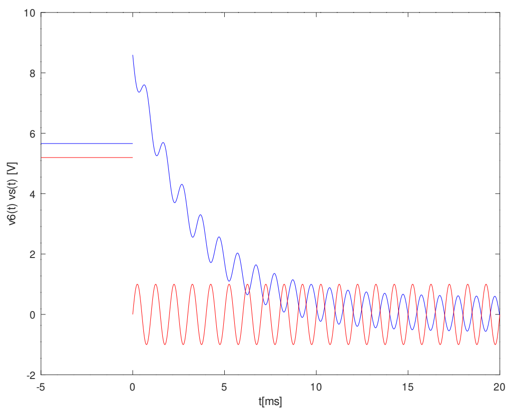


Figure 3: Forced solution of V6

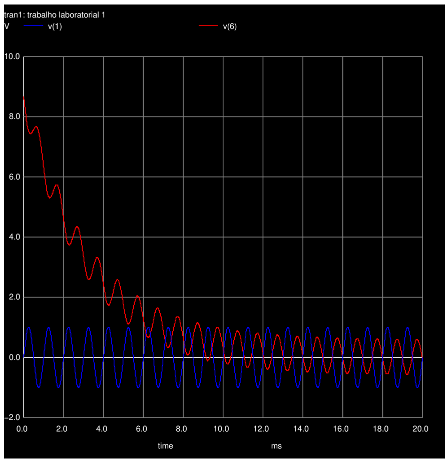
Using phasors and apllying the nodal method, we obtained the phasors of all nodes, in particular the node 6 phasor which when multiplied by $exp(jwt)$ corresponds to the forced solution. The aspect of the graph is like we expected, a sinuosoidal wave.

2.5 Topic 5-T/S

This topic corresponds to the fifth theoretical and the fourth simulation topics.



(a)



(b)

Figure 4: (a) Solution to V6 and Vs using octave (b) Solution to V6 and Vs using ngspice

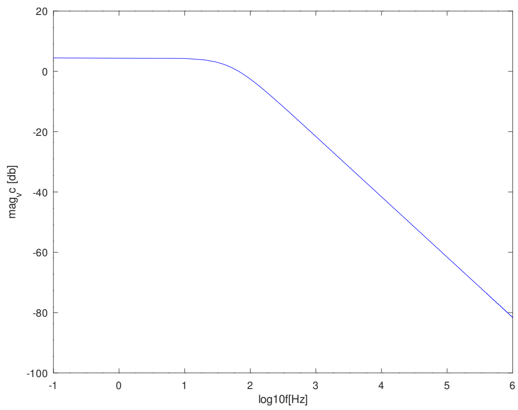
In octave, to obtain the solution for V6 we just add the natural and forced solution. In ngspice, we did a transient analysis to obtain V6 solution. As we can see, both figures look the same for V6 and obviously for Vs.

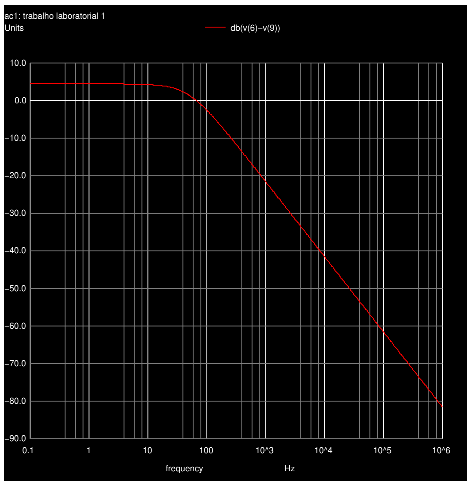
2.6 Topic 6-T/S

This topic corresponds to the sixth theoretical and the fifth simulation topics.

For the frequency analysis, in octave we chose to solve the nodal method with frequency as a symbolic variable. This procedure takes a few more seconds but ensures that we obtain the results without any approximation. Another advantage of this method is that we obtain all nodes frequency dependence.

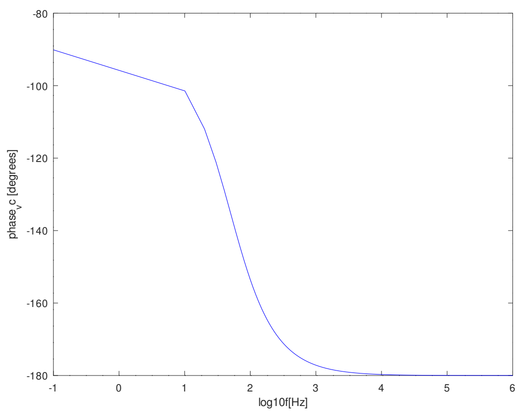
Using octave and ngspice we obtain very similar graphics for phase and magnitude of phasor Vc. Although, they have a few differences in the beginning of the graphic because we have a logaritmic scale so in the beginning we have much less points that we have in the end.

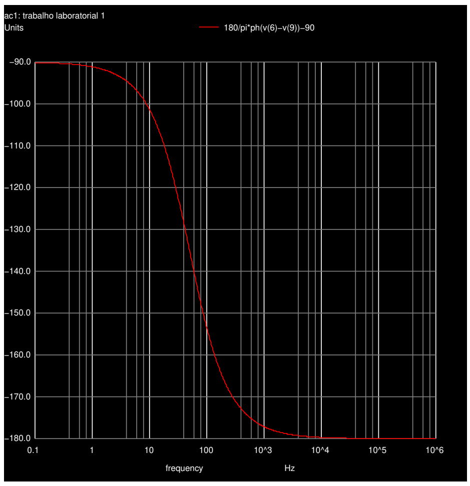




(a) (b)

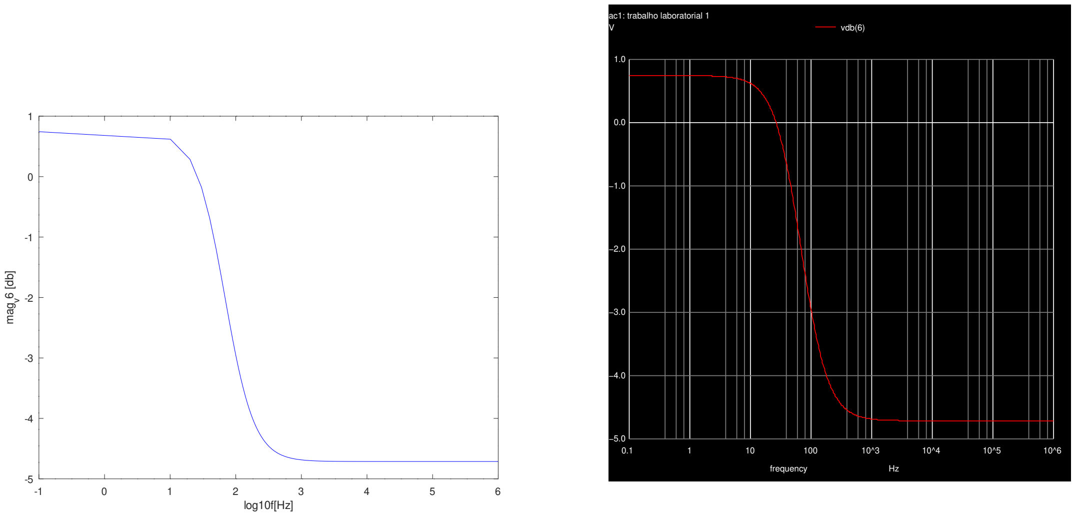
Figure 5: (a) Magnitude of Vc in function of frequency using octave (b) Magnitude of Vc in function of frequency using ngspice





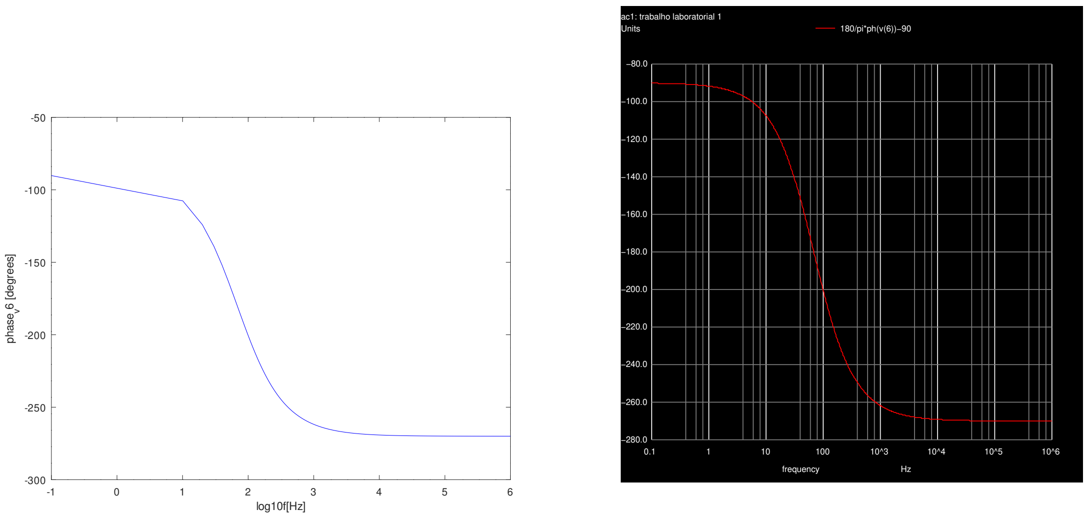
(a) (b)

Figure 6: (a) Phase of Vc in function of frequency using octave (b) Phase of Vc in function of frequency using ngspice



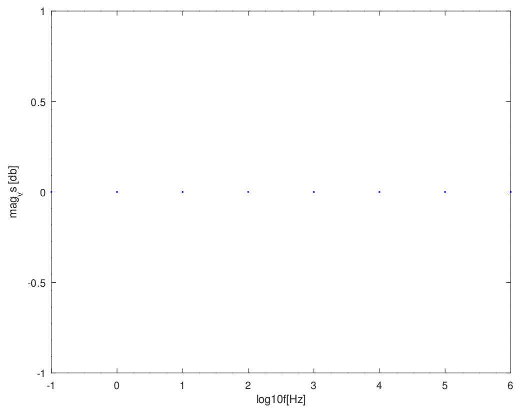
(a) (b)

Figure 7: (a) Magnitude of V6 in function of frequency using octave (b) Magnitude of V6 in function of frequency using ngspice

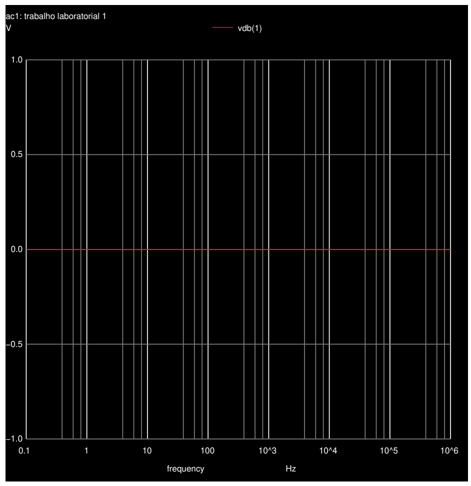


(a) (b)

Figure 8: (a) Phase of V6 in function of frequency using octave (b) Phase of V6 in function of frequency using ngspcice

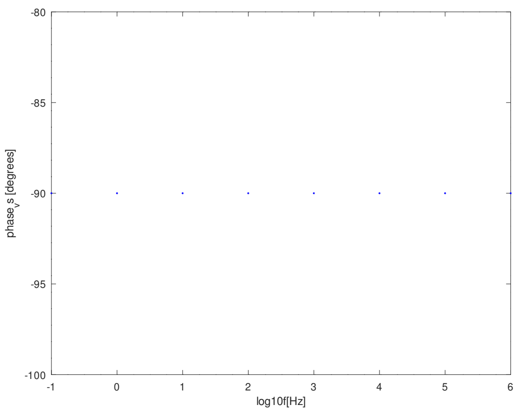


(a)

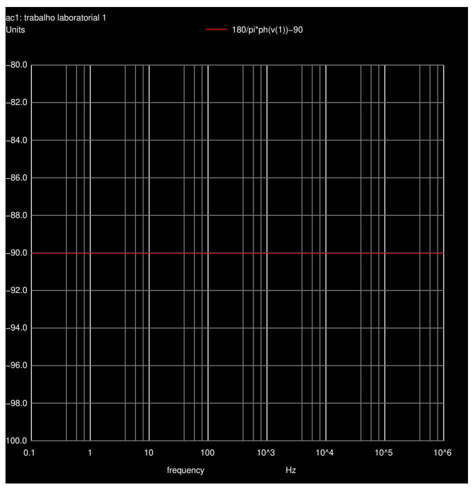


(b)

Figure 9: (a) Magnitude of Vs in function of frequency using octave (b) Magnitude of Vs in function of frequency using ngspice



(a)



(b)

Figure 10: (a) Phase of Vs in function of frequency using octave (b) Phase of Vs in function of frequency using ngspice

To compare V_c and V_s, we can use the following equation, because this represents the RC equivalent transfer function equation.

$$\frac{V_c}{V_s} = \frac{1}{1 + j * w * Req * C}$$

Analysing the equation we could understand the differences between the plots of magnitude and phase of V_c and V_s. The magnitude and phase of V_s don't change with the frequency. To low frequencys, we can see that w is almost zero, so the phase difference is almost none $ph(V_C) = atan(w * Req * C)$ and the magnitude is almost the same. To high frequencys, we can see that w is tending to infinite, so the phase difference is close to $\frac{\pi}{2}$ and the magnitude of V_c is close to 0.

To compare V_c and V_s with V_6, we must remember that $V_c = V_6 - V_8$. If we understand that V_8 don't change with frequency, so we know that the graphs of V_c will have the same form of the graphics of V_6 despite some vertical displacements. And knowing the differences between V_c - V_s and V_c - V_6 the differences between V_s - V_6 will be direct.

3 Conclusion

As a pack, the results predicted by the Theoretical Analysis of the circuit match the ones obtained when simulating the circuit in NGSpice. This was already expected since the circuit is only built with linear components. However, we can see small differences in tables values mainly due to rounding and also some differences may occur in the graphs because of different solving methods of the circuit mainly in Topic 5, since NGSpice solves differencial equations and we solved the problem using a numerical method. Summing everything up, we can say that the laboratory objective of analysing the circuit has been achieved due to the match of results.