Circuit Theory and Electronics Fundamentals

MEAer (Integrated Master In Aerospace Engineering), Técnico, University of Lisbon

Laboratory 2: RC Circuit analysis

Group 3

Diogo Faustino, nº95782
Henry Machado, nº95795
Rúben Novais, nº95843

April 5, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study the behaviour of an RC circuit, as seen in Figure 1. In order to analyse it, we will make use of different techniques such as Thevenin and Phasor analysis in order to establish the natural, forced and total solutions of the circuit. A frequency response analysis then follows, where we look at the magnitude and phase as a function of frequency. Then, we compare the theoretical and simulation data.

In Section 2, we go through the various steps that are needed to obtain the total solution of the circuit (natural plus forced responses) and present the theoretical frequency response analysis. In Section 3, we present the circuit response as simulated in NGSpice. The results in operating point and transient analysis are compared to the theoretical results obtained in Section 2. The conclusions of this study are laid out in Section 4.

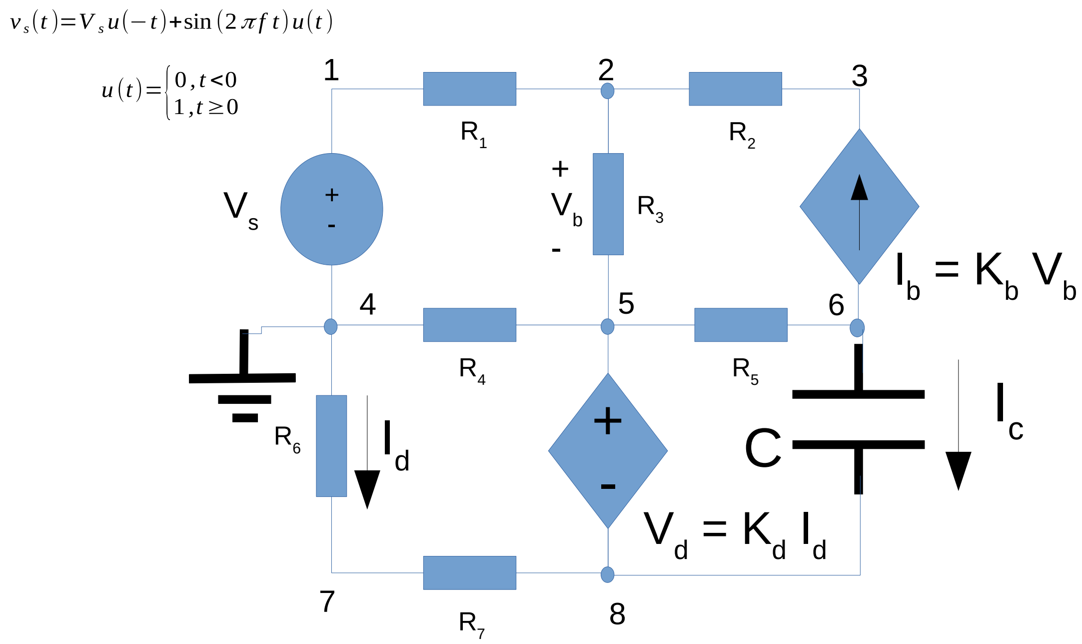

Figure 1: RC circuit.

2 Theoretical Analysis

We analyse the circuit shown in Figure 1 for t¡0 using the nodal method. The nodes are numbered according to what's shown in the picture. In this first instance we are working in a steady state, where no current is flowing through the capacitor: we can replace it with an open circuit. After doing this, it is clear that all the components we are working with are linear and so we will need to solve a system of linear equations to determine the initial values for the subsequent analysis. This way, we've ran the nodal method and solved the linear system on GNU Octave.

2.1 Node Analysis for t¡0

Knowing that V4=0 since it is connected to the ground:

$$\begin{pmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G1 & G1+G2+G3 & -G2 & -G3 & 0 & 0 & 0 \\ 0 & Kb+G2 & -G2 & -Kb & 0 & 0 & 0 \\ -G1 & G1 & 0 & G4 & 0 & G6 & 0 \\ 0 & 0 & 0 & 0 & 0 & -G6-G7 & G7 \\ 0 & 0 & 0 & 1 & 0 & G6*Kd & -1 \\ 0 & -G3 & 0 & G3+G4+G5 & -G5 & G6 & 0 \end{pmatrix} \begin{pmatrix} V1 \\ V2 \\ V3 \\ V5 \\ V6 \\ V7 \\ V8 \end{pmatrix} = \begin{pmatrix} Vs \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix} \tag{1}$$

We can also easily obtain the current values in the various branches through Ohm's law. This yields the following results:

Name	Value [A and V]
@c[i]	0.000000e+00
@g[i]	-2.29771e-04
@r1[i]	2.191669e-04
@r2[i]	2.297712e-04
@r3[i]	1.060424e-05
@r4[i]	1.185502e-03
@r5[i]	2.297712e-04
@r6[i]	9.663348e-04
@r7[i]	9.663348e-04
v(1)	5.113399e+00
v(2)	4.889448e+00
v(3)	4.428712e+00
v(4)	-1.95390e+00
v(5)	4.921444e+00
v(6)	5.620908e+00
v(7)	-1.95390e+00
v(8)	-2.92781e+00

Table 1: Theoretical analysis results. (A variable preceded by @ is of type *current*)

2.2 Equivalent Resistor

In order to compute the equivalent resistor we ran a nodal analysis, making Vs=0 and replacing the capacitor with a voltage source Vx=V6-V8 as calculated in the previous step. This is made to ensure that the voltage in the capacitor is continuous since it does so in reality: this is a capacitor discharging through a resistance, any discontinuity in voltage would require an infinite amount of current.

$$\begin{pmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ -G1 & G1+G2+G3 & -G2 & -G3 & 0 & 0 & 0 \\ 0 & Kb+G2 & -G2 & -Kb & 0 & 0 & 0 \\ -G1 & G1 & 0 & G4 & 0 & G6 & 0 \\ 0 & 0 & 0 & 0 & 0 & -G6-G7 & G7 \\ 0 & 0 & 0 & 1 & 0 & G6*Kd & -1 \\ 0 & 0 & 0 & 0 & 1 & 0 & -1 \end{pmatrix} \begin{pmatrix} V1 \\ V2 \\ V3 \\ V5 \\ V6 \\ V7 \\ V8 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ Vx \end{pmatrix} \tag{2}$$

With the following definitions:

$$V_x = V_6 - V_8; \tag{3}$$

$$I_x = \frac{V_6 - V_5}{R_5} + \frac{V_3 - V_2}{R_2}; \tag{4}$$

$$Req = \frac{V_x}{I_x} \tag{5}$$

We were able to obtain the following values, laying the foundation for the natural solution:

Name	Value
@Ix	0.002808 A
Vx	8.548721 V
Req	3044.174022 Ohm
Tau	0.003060 s

Table 2: Nodal analysis results.

2.3 Natural solution

The general solution for RC circuits, making use of the computations in the previous section, yields:

$$V_{6n}(t) = V_x exp(-\frac{t}{RC}) \tag{6}$$

From this equation, we obtained the following plot, which depicts the first twenty milisseconds of the circuit's natural response.

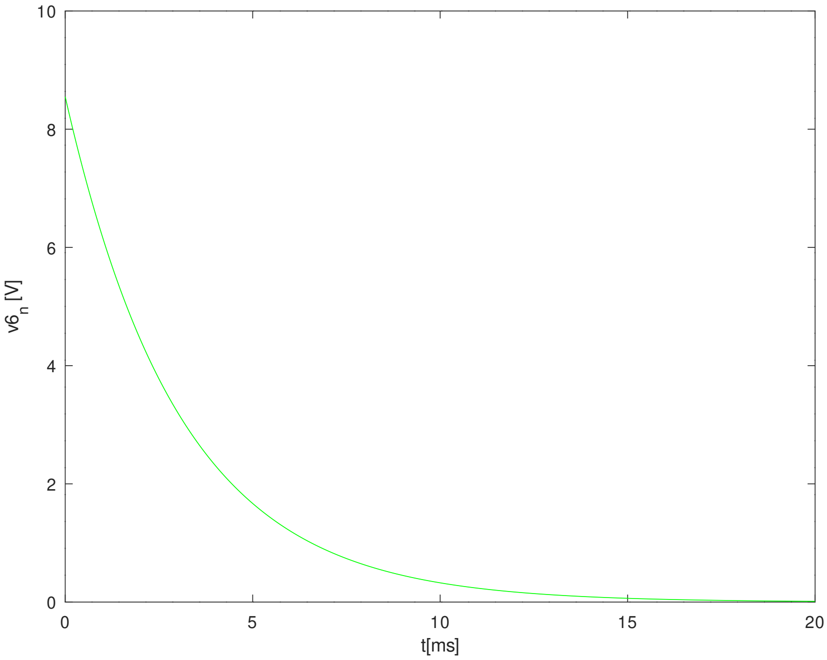

Figure 2: Natural solution to V_6 node voltage.

2.4 Forced and final total solution

ye

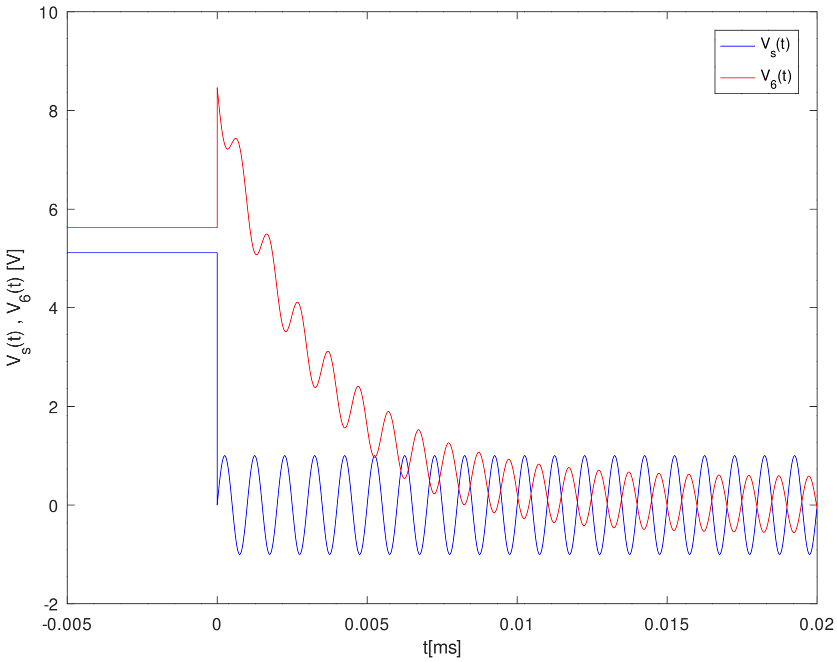

Figure 3: Total solution to V_6 node voltage.

Applying the Kirchhoff Voltage Law (KVL) in the different loops, we get four different equations, which we can then solve as a matrix:

$$I_D = I; \tag{7}$$

$$(R_1 + R_3 + R_4)I_A - R_3 I_B - R_4 I_C = -V_A; \tag{8}$$

$$(R_4 + R_6 + R_7 - K_C)I_C - R_4 I_A = 0; \tag{9}$$

$$(R_3 K_B - 1)I_B - R_3 K_B I_A = 0. \tag{10}$$

$$\begin{pmatrix} 0 & 0 & 0 & 1 \\ R1 + R3 + R4 & -R3 & -R4 & 0 \\ -R4 & 0 & R4 + R6 + R7 - Kc & 0 \\ -R3Kb & R3Kb - 1 & 0 & 0 \end{pmatrix} \begin{pmatrix} Ia \\ Ib \\ Ic \\ Id \end{pmatrix} = \begin{pmatrix} I \\ -V \\ 0 \\ 0 \end{pmatrix} \tag{11}$$

2.5 Node Analysis

As in the previous section, we extract a set of equations from the circuit, this time using the node method. This yields 8 equations, one for each node. Then, we can solve those 8 equations in the form of the matrix presented here.

$$\begin{pmatrix} G7 & 0 & 0 & 0 & 0 & 0 & G6 & 0 \\ 1 & -1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & 0 & -G2 & G1+G2+G3 & -G1 & 0 & -G3 \\ 0 & 0 & G5 & 0 & G3 & 0 & G4+G6 & -G3-G4-G5 \\ 1 & 0 & 0 & 0 & 0 & 0 & Kc*G6 & -1 \\ 0 & 0 & 0 & 0 & G1 & -G1 & -G4-G6 & G4 \\ 0 & 0 & 0 & -G2 & Kb+G2 & 0 & 0 & -Kb \end{pmatrix} \begin{pmatrix} V1 \\ V2 \\ V3 \\ V4 \\ V5 \\ V6 \\ V7 \\ V8 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ V \\ 0 \\ I \\ 0 \\ 0 \\ 0 \end{pmatrix} \tag{12}$$

3 Simulation Analysis

Since this circuit is a steady one, the voltage or current values of the various components doesn't vary in time. Therefore, only the Operating Point Analysis is necessary in this circuit simulation

3.1 Operating Point Analysis

For this circuit's simulation on Ngspice, there was a need to introduce a null voltage source between the ground node and the R7 resistor. The voltage related to node V2 will not appear in following table 3 because it has the same voltage as node V1 (its omission is necessary for the simulation to run correctly).

Table 3 below shows the simulated operating point results for the circuit under analysis.

Using the results of the theoretical analysis for mere guidance, we know the voltage values for the nodes: that should follow from our simulation. We know then in which direction the voltage drops happen and we use this order for all branches except the current sources. In the current sources the order follows the current flow of such sources, as it is norm in Ngspice. Following the order we stated for the other branches means that the current that flows through every resistor must have a positive value in table 3, because in resistors the voltage drop and current flow have the same direction (resistors always consume energy).

Analysing table 3, we notice that the current flowing through every resistor has a positive value, as it should be. I_a is the current flowing through R1, I_b is the current flowing through R2, I_c is the current flowing through R6 (and R7), and, finally, I_d is equivalent to Idd. Comparing the simulation results for the voltage and current values with the theoretical analysis values, we notice that the values for voltage in nodes V_1 to V_9 and the current for all four mesh currents are exactly equal, if we exclude all roundings carried out by NgSpice, since its precision (up to 7 significant figures) can be slightly different than the precision we used in GNU Octave (a maximum of 2 significant figures of difference). Different digits only start to appear by the 6^{th} decimal case, which is a disposable calculation error (Shrinking the theoretical results to Ngspice precision, we get equal theoretical and simulation results).

With that being said, the theoretical results are equal to the simulation results (the complete accuracy is fruit of the staticness of the circuit), which confirms our theoretical analysis. The fact that the voltage and current results are equal obviously results in equal power values for each branch.

4 Conclusion

In this laboratory assignment the objective of studying the presented circuit has been achieved. All the calculations for the Mesh and Node methods have been carried out using the Octave

Name	Value [A or V]
@c[i]	0.000000e+00
@g[i]	-2.29771e-04
@r1[i]	2.191669e-04
@r2[i]	2.297712e-04
@r3[i]	1.060424e-05
@r4[i]	1.185502e-03
@r5[i]	2.297712e-04
@r6[i]	9.663348e-04
@r7[i]	9.663348e-04
v(1)	5.113399e+00
v(2)	4.889448e+00
v(3)	4.428712e+00
v(4)	-1.95390e+00
v(5)	4.921444e+00
v(6)	5.620908e+00
v(7)	-1.95390e+00
v(8)	-2.92781e+00

Table 3: Operating point analysis. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

Maths tool and the circuit simulation has been done using the Ngspice tool. In a real, presential lab class, all sorts of experimental errors would be present: the resistance of the wiring, internal resistance of the sources, the temperature, external noise, etc. Besides that, reading and assembly errors would be expected. In this case, the simulation results matched the theoretical results: the reason for this match is the fact that this is a straightforward circuit containing only linear components, so if done correctly, the theoretical and simulation models cannot have noticeable differences. For more complex components, the theoretical and simulation models could differ greatly but this is not the case in this work. Despite the proximity of the values, there is still some difference between them (around the 6th decimal place). This can be considered negligible for the experiment in case, which doesn't require a higher degree of accuracy since the whole purpose is just to study a simple circuit (it could be a problem if this belonged to a bigger functional structure where every small discrepancy between the expected and real data mattered). Nevertheless, these differences may have been caused by the employment of different numerical methods for solving the linear system and for different rounding criteria employed by the two different tools.

Name	Value [A or V]
@g[i]	-4.25263e-18
@r1[i]	4.056361e-18
@r2[i]	4.252625e-18
@r3[i]	1.962643e-19
@r4[i]	-8.55795e-19
@r5[i]	2.808224e-03
@r6[i]	-8.67362e-19
@r7[i]	-1.78493e-18
v(1)	0.000000e+00
v(2)	-4.14491e-15
v(3)	-1.26722e-14
v(4)	1.753779e-15
v(5)	-3.55271e-15
v(6)	8.548722e+00
v(7)	1.753779e-15
v(8)	3.552714e-15

Table 4: Operating point analysis. A variable preceded by @ is of type *current* and expressed in Ampere; other variables are of type *voltage* and expressed in Volt.

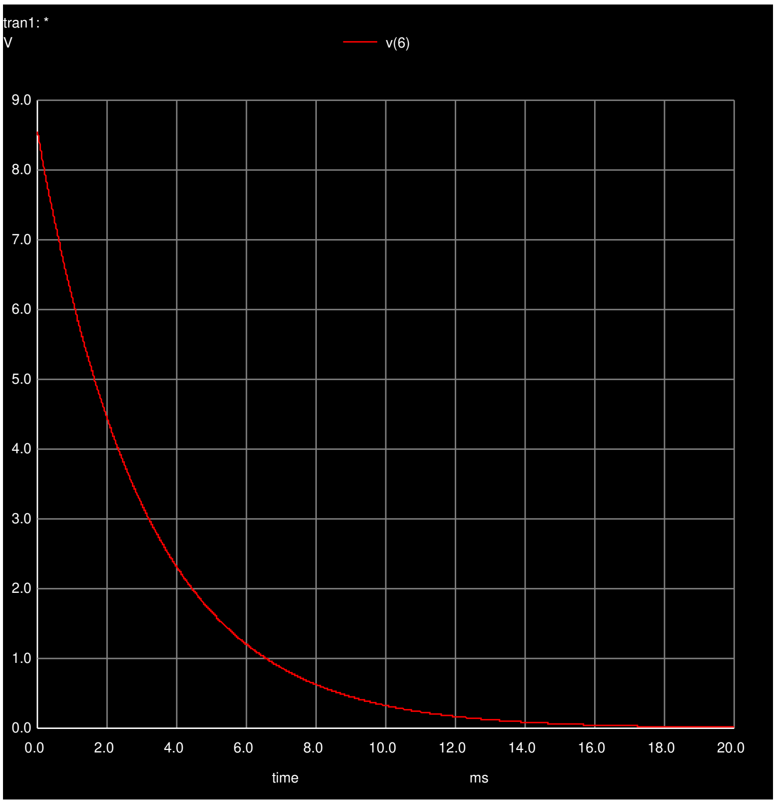

Figure 4: Simulated natural response for V_6 node voltage.

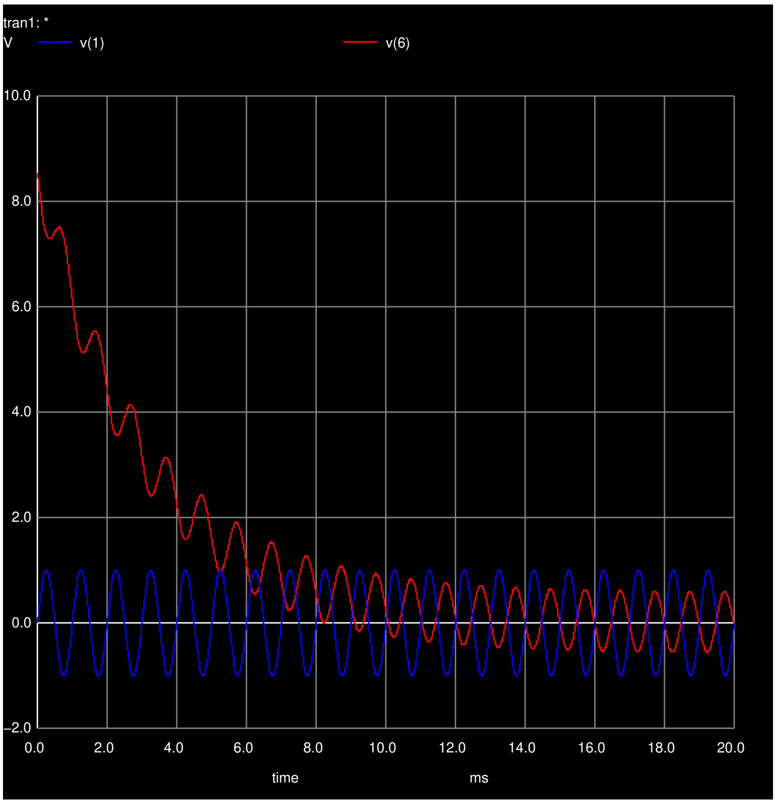

Figure 5: Simulated response for V_6 node voltage and stimulated voltage V_S.

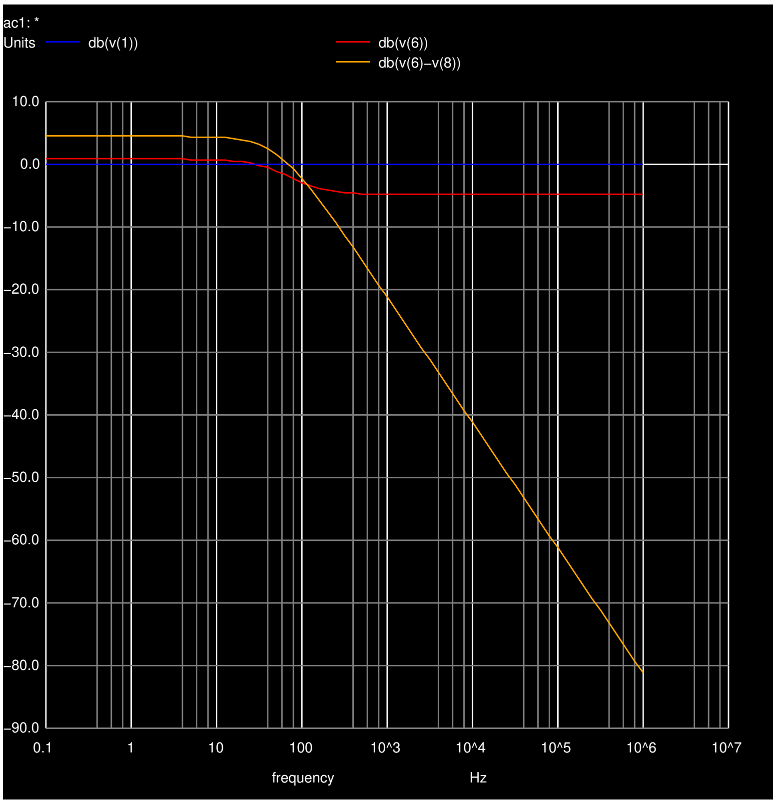
ac1: *
Units
db(v(1))
db(v(6))
db(v(6)−v(8))
10.0
0.0
−10.0
−20.0
−30.0
−40.0
−50.0
−60.0
−70.0
−80.0
−90.0
0.1
1
10
100
10^3
10^4
10^5
10^6
10^7
frequency
Hz

Figure 6: ...

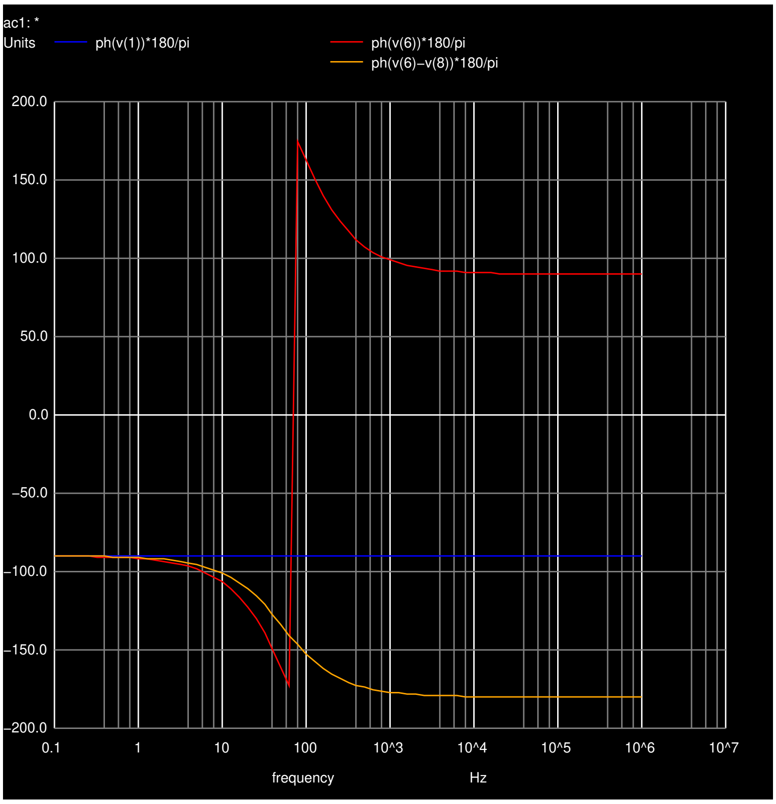
ac1: *
Units
ph(v(1))*180/pi
ph(v(6))*180/pi
ph(v(6)−v(8))*180/pi
200.0
150.0
100.0
50.0
0.0
−50.0
−100.0
−150.0
−200.0
0.1
1
10
100
10^3
10^4
10^5
10^6
10^7
frequency
Hz

Figure 7: ...