Circuit Theory and Electronics Fundamentals

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

Second Laboratory Report

Group 20
João Maria De Carvalho Coelho Navarro Soeiro, 95803
Bruno Manuel Bento Pedro, 96363
Leonor João Dias Ferreira, 96422

April 7, 2021

Contents

1 Introduction

The objective of this laboratory assignment is to study a circuit containing a sinusoidal voltage source v_s connected to seven resistors (R_1 to R_7), a capacitor C, a dependent voltage source (current-controlled) and a dependent current source (voltage-controlled). The circuit can be seen in Figure 1.

The nodes are designated with numbers as seen in the Figure 1 and the node voltages will be represented with their respective numbers (ex. V_3 represents the voltage in node number 3). The characteristic equations of the dependent sources can also be seen in the Figure 1.

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 5.

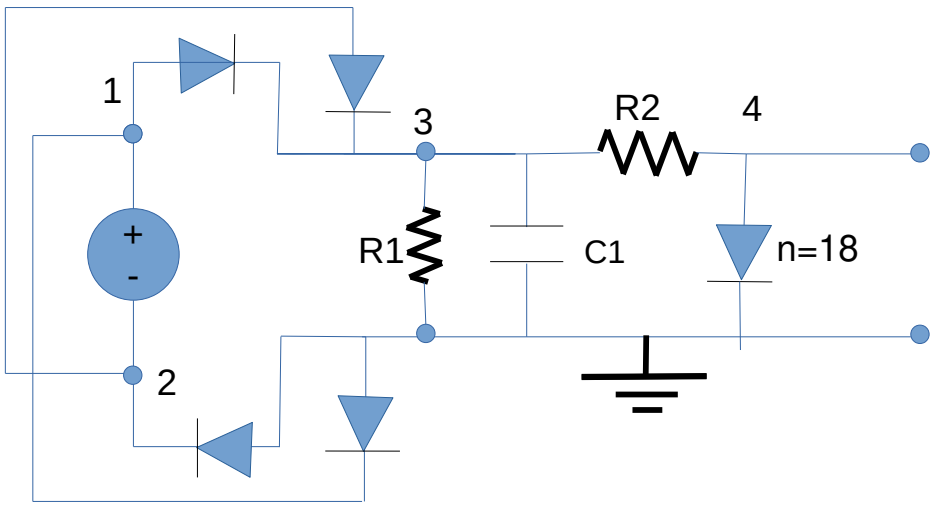

Figure 1: Circuit drawn to build the AC/DC Converter.

Name [unit]	**Value**
n	10.778589
$R_{envelope}$ [kOhm]	150.000000
$C_{envelope}$ [uF]	100.000000
$R_{regulator}$ [kOhm]	5.000000

Table 1: Values considered to build the AC/DC converter circuit.

2 Theoretical Analysis

Name [unit]	Value
$Avarage(v_O)$ [V]	12.000000
$Ripple(v_O)$ [V]	0.000900
$Cost$ [MU]	257.200000
$Merit$	4.313424

Table 2: Results from theoretical analysis.

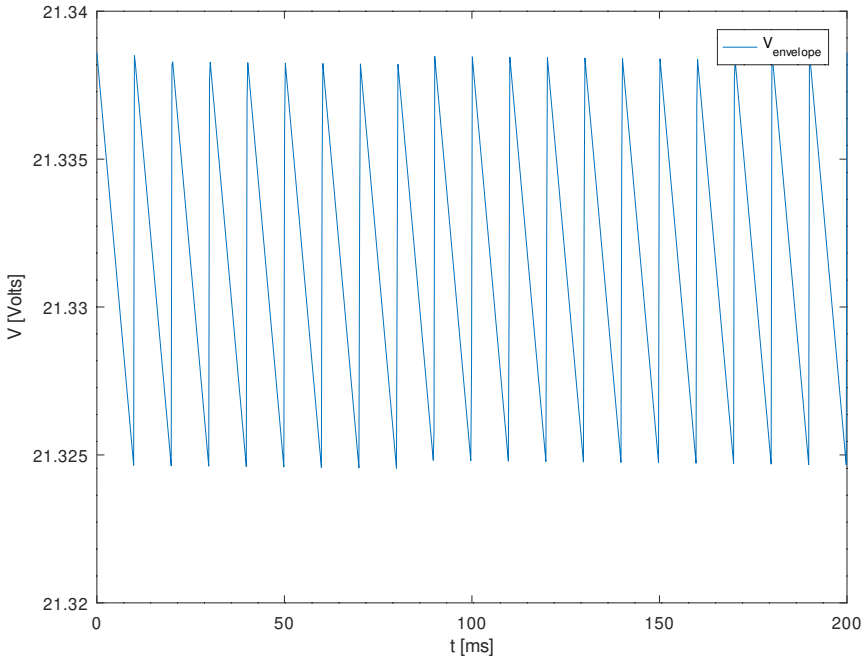

Figure 2: Output voltage at the Envelope Detector.

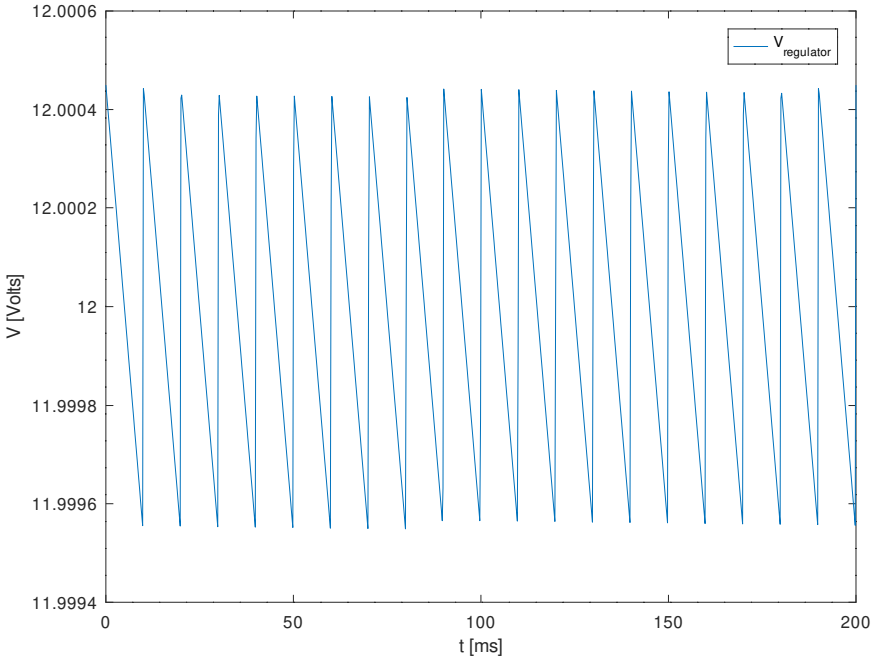

Figure 3: Output voltage at the Voltage Regulator circuit.

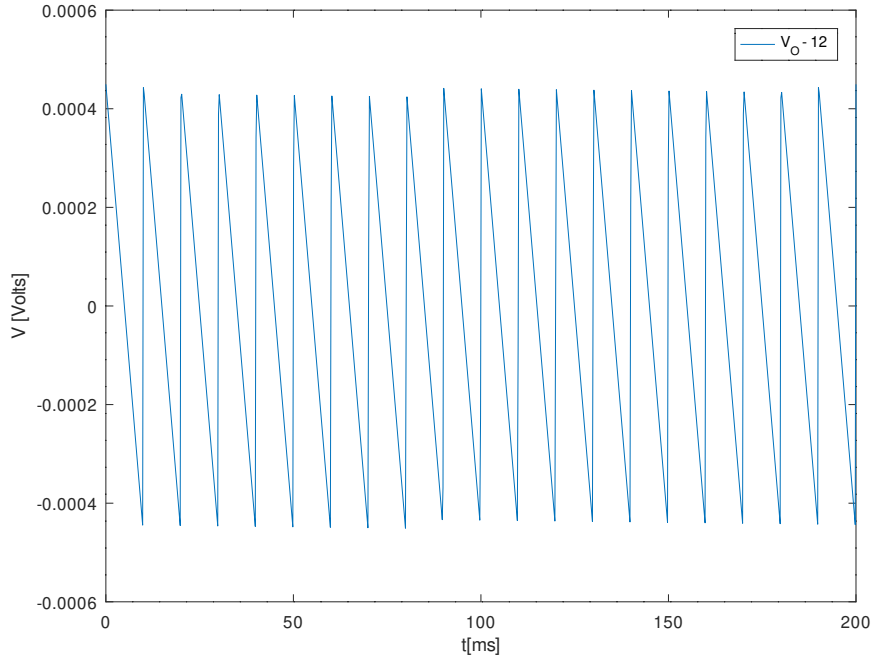

Figure 4: Deviation of output voltage at the Voltage Regulator from 12V.

3 Simulation Analysis

In this section, a Ngspice circuit will be analyzed and compared to the one made in the theoretical analysis using Octave. In order too obtain a decent merit value, we tried different combinations of values for the resistances and the capacitor, while trying to get a good average value and a relatively low cost. We used a full wave rectifier and a normal voltage regulator, with 18 diodes. The circuit is as follows:

The graphics for the voltage of the envelope detector, the voltage of the voltage regulator and Vo-12 are, respectively:

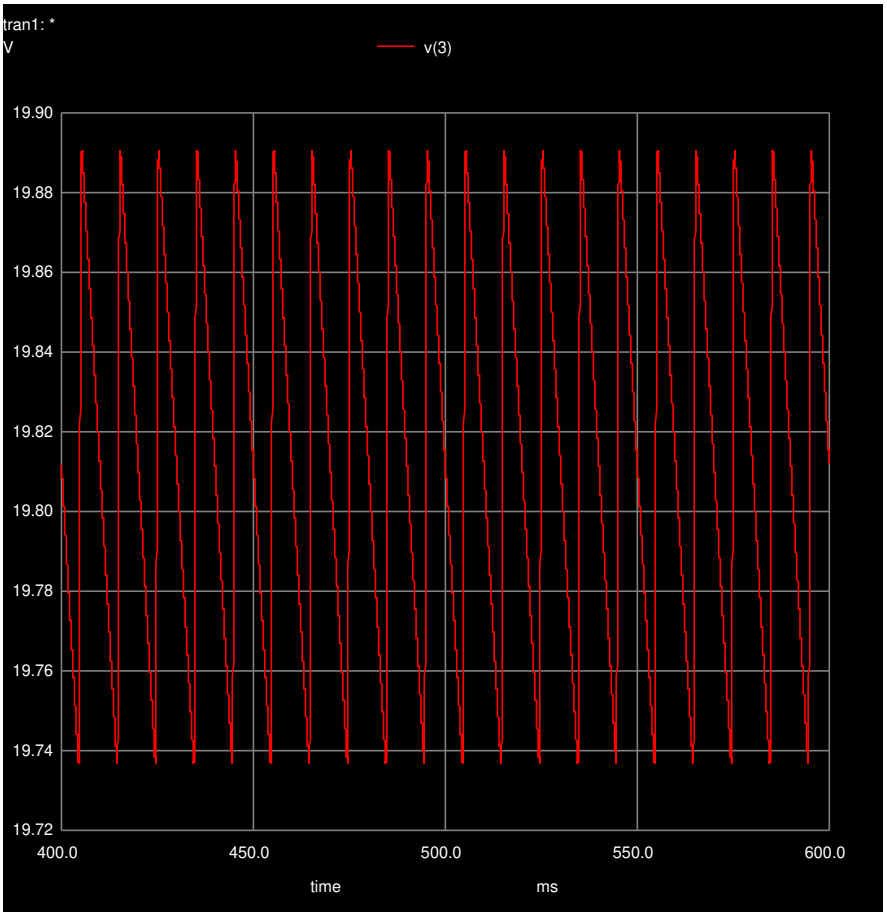

Figure 5: Output voltage at the Envelope Detector. The *x axis* represents the time in miliseconds and the *y axis* the voltage in Volts.

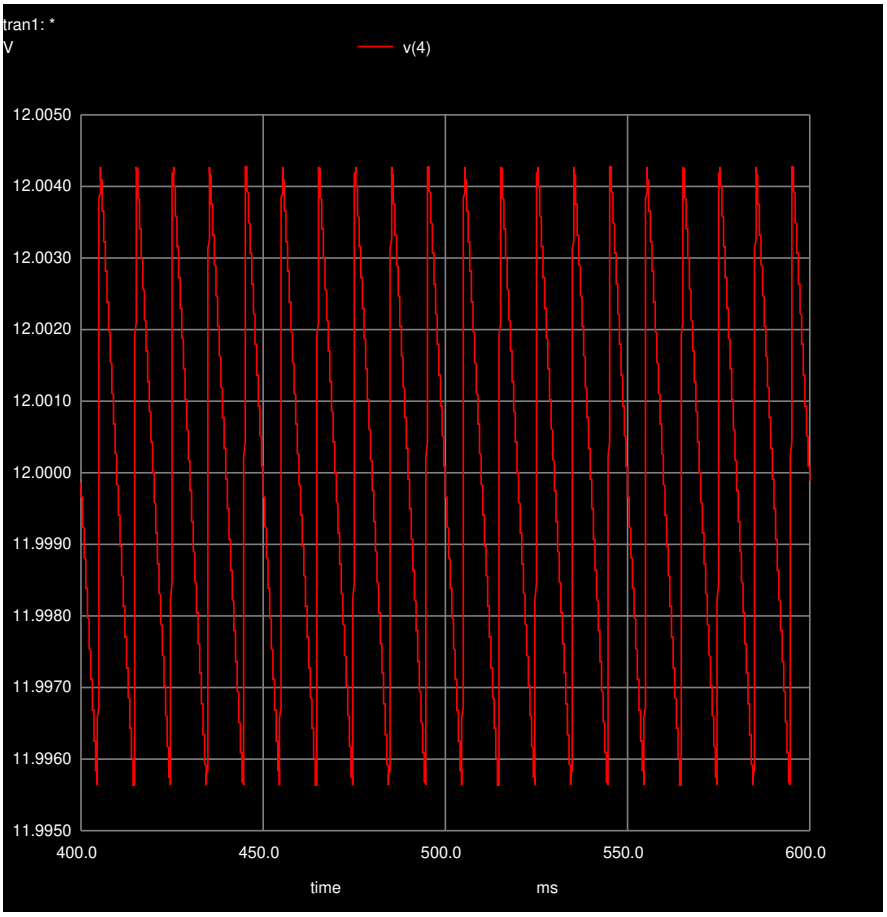

Figure 6: Output voltage at the Voltage Regulator circuit. The *x axis* represents the time in miliseconds and the *y axis* the voltage in Volts.

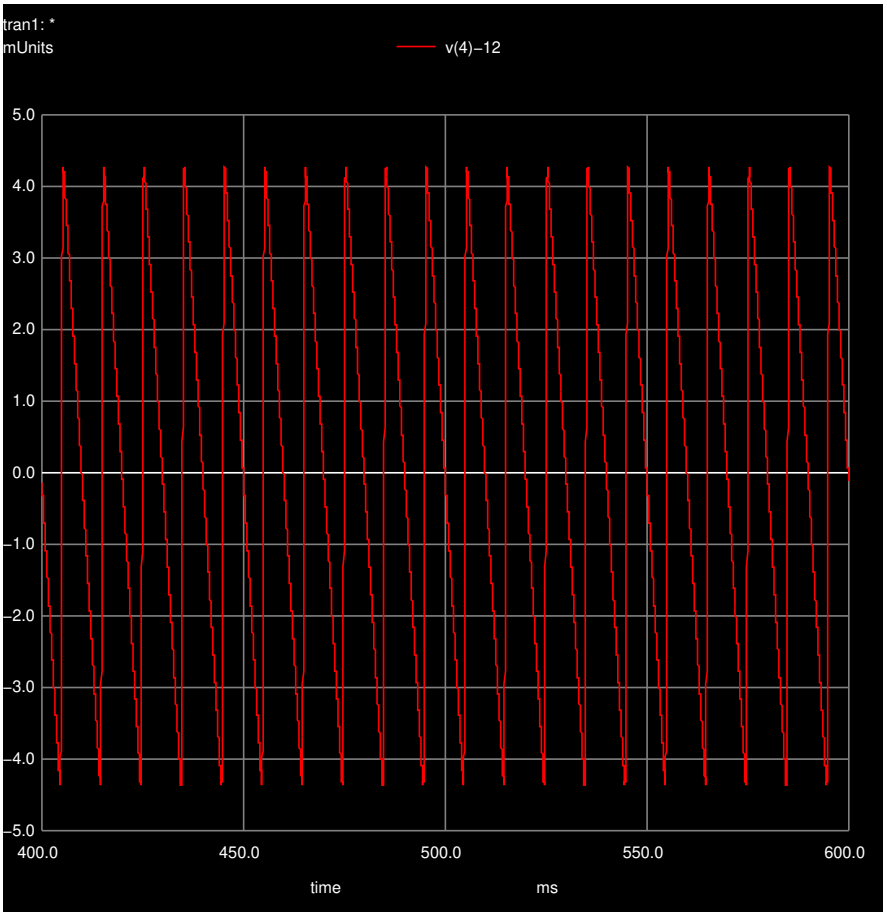

Figure 7: Deviation of the output voltage at the Voltage Regulator from 12V. The *x axis* represents the time in miliseconds and the *y axis* the voltage in Volts.

4 Comparison

We will now compare the results obtained in the theoretical analysis and the simulation analysis.

First, we obtained the values of both the voltages and currents of the circuit for t<0 using the nodal method in the theoretical analysis, and using the command ".op", Operating Analysis in the Ngspice engine in the simulation analysis, which gives us the voltages and currents of the circuit. The results obtained are extremely similar (Table **??**), which makes us believe the Ngspice engine uses for its calculations the same method used theoretical, the node method. The error obtained is minimal, with the first differences in the values appearing in the 4th/5th decimal number, and only by a unite, which gives us a error inferior to 0,01%.

We then analysed the circuit for t = 0, so with v_s = 0, and with V_x = V_6 - V_8, this voltage is replacing the capacitor so we can obtain R_{eq}, which is achieved with calculating I_x. Once again, given the similarity of the process to obtain the results with the one in the first section, the differences between the theoretical and simulation analysis are, once again, minimal and with errors inferior to 0,01% (Table **??**).

Thirdly, we were asked to acquire the natural solution of V_6, V_{6n}(t). Again, the graphics obtained by both type of analyses made are identical. The same happens with the forced solution V_{6f}(t) and the total solutions for both V_6(t) and V_s(t), asked in the next topics. In the theoretical analysis we used again analytic methods to solve the problem, while one the Ngspice engine, we used the Transient Analysis command. Given the similarity, it's once again safe to assume that the Ngspice uses in same way the same mathematical expressions and methods used theoretically,

Finally we did a frequency analysis, in which we studied how V_6(f), V_s(f) and V_c(f) vary with the frequency, again this analysis was made both using Octave and Ngspice and the graphics obtained for both the magnitude and the phase of the voltages is identical, with minimal differences, if any.

5 Conclusion

Given the comparison made previously and the results obtained along the work, we can safely say that the objectives for this laboratory assignment were successfully achieved. In this assignment we were able to analyse and study a circuit containing multiple resistors, independent and dependent sources and a capacitor, that varies with time.

Different types of analysis were effectuated, time, static and frequency analyses, both using the Octave and Ngspice engines. And the results obtained are within the expected and the similarities between the different types of analyses are satisfactory, but also expected given the nature of the circuit and its components, which were all linear, besides the capacitor, and all straightforward, which does not give many room for differences between the analyses.

Given all this the objectives for this laboratory assignment were all achieved.