Circuit Theory and Electronics Fundamentals

BSc Aerospace Engineering, Técnico, University of Lisbon

Lab 1: Circuit analysis methods

March 25, 2021

Group 48
Dinis Papinha, 84379
Filipa Gonçalves, 89662
Carlos de Vasconcelos, 90227

Contents

1 Introduction

The objective of this laboratory assignment is to design an AC/DC converter circuit, that receives an AC voltage of 230V and 50Hz and outputs a DC voltage of 12V. In order to achieve the goal, it was possible to choose the winding ratio of the transformer, and the architecture of the Envelope Detector and Voltage Regulator circuits. A *Merit* score is obtained as a function of the chosen architecture's cost and how accurate the output is (relative to the aimed 12V DC output).

The chosen circuit is presented in Figure !!!

in which R2 is (describe parameters)

In Section 2, using *Octave*, a theoretical analysis is presented, in which we describe the theoretical model used to the predict the output of the Envelope Detector and Voltage Regulator circuits and the results of the described analysis are presented. In this Section's last subsection (Subsection !!!!), the cost of the circuit and Merit score are determined. In Section 3, the circuit is analyzed by simulation, but using the Ngspice software, instead of a chosen theoretical model. The results are compared in Subsection 3.1 (the last of this third Section). The conclusions of this study are outlined in Section 4.

2 Theoretical Analysis

2.1 Envelope Detector Circuit Output

In this subsection, we present the theoretical approach used for determining the output of the Envelope Detector Circuit, and the resulting plot.

Firstly, the v_{in} voltage needs to have its amplitude reduced. To achieve that, a transformer with 1:17 ratio is used and the AC voltage dropping form node 1 to node 2 is given by equation 1.

$$v_{trans}(t) = \frac{A_{in}}{n} \cdot cos(\omega \cdot t) \tag{1}$$

where A_{in} is the amplitude of the original signal v_{in}, $n = 17$, $\omega = 2 \cdot pi \cdot f$, and $f = 50Hz$.

Secondly, a envelope detector circuit is used, composed of a full-wave bridge rectifier circuit and a capacitor. The full-wave bridge rectifier consists of four diodes and a resistor, and the ideal diode model is used.

As shown in theoretical classes, the output voltage of the envelope detector v_3 from $t = 0$ to t_{off} is equal to the absolute value of v_{trans}. The instant t_{off} can be determined using equation 2.

$$t_{off} = \frac{arctan(\frac{1}{\cdot R \cdot C \cdot \omega})}{\omega} \tag{2}$$

From then on, v_3 is computed as equation 3

$$v_3(t) = \frac{A_{in}}{n} \cdot cos(\omega \cdot t_{off}) \cdot e^{\frac{t - t_{off}}{R_1 \cdot C}} \tag{3}$$

until the absolute value of v_{trans} is greater than equation 3. After that, v_3 equals again the absolute value of v_{trans}, until the next t_{off}, which comes half the wave period after the previous one, and so on. Note that, in v_3, the frequency has doubled.

In consequence, we are able to obtain the plot in Figure !!!!

Figure 1: Voltages in diferent components

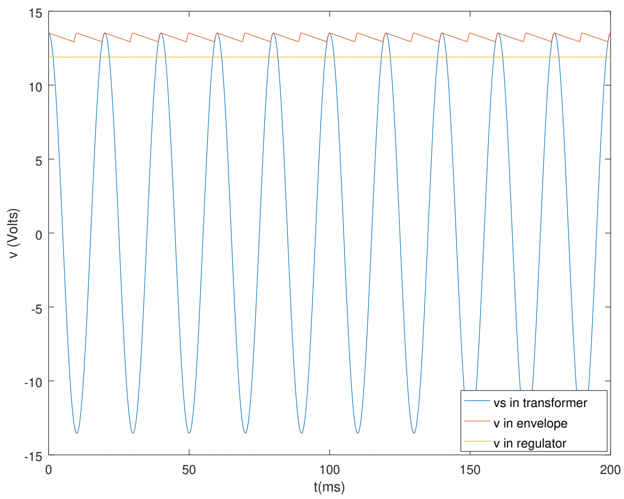
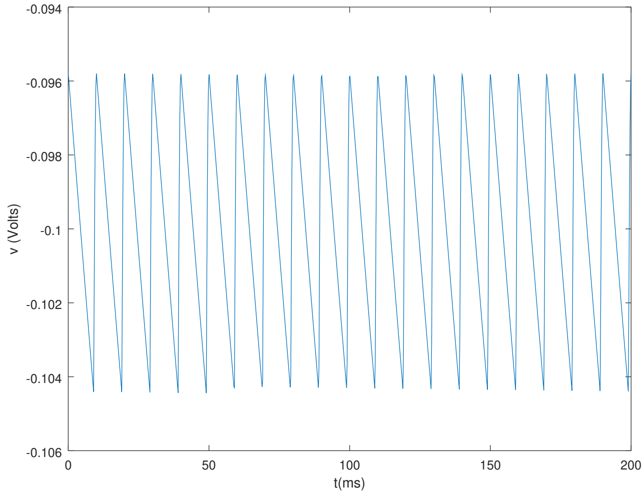


Figure 2: Graphical representation of output voltage with 12 V



2.2 Voltage Regulator Circuit

In this subsection, incremental analysis is used to compute the AC component of v_4 ($v_{4_{AC}}$) and, subsequently, $v_4(t)$ as a whole. Incremental analysis was used. In order to achieve the subsection's goal, we need to obtain r_d (incremental diode resistance) and, as learned in theoretical classes, it is as follows

$$r_d = \frac{\eta \cdot V_T}{I_S \cdot e^{\frac{V_{ON}}{\eta \cdot V_T}}} \tag{4}$$

in which $I_S = 1e - 14$, $V_T = 0.025$, $\eta = 1$ and $V_{ON} = 0.7$ (as an estimated approximation).

To obtain $v_{4_{AC}}$, equation 5 is used,

$$v_{4_{AC}}(t) = \frac{17 \cdot r_d}{17 \cdot r_d + R_2} \cdot v_3(t). \tag{5}$$

since there are 17 diodes in series.

Adding to the aforementioned signal its DC component, which is V_{ON} multiplied by the number of diodes (17), the following plots can be obtained in Figure !! and Figure !!.

Lastly, the voltage ripple is calculated through equation 6.

$$V_{Ripple} = max(v_4(t)) - min(v_4(t)) \tag{6}$$

and we are able to obtain

$$V_{Ripple} = 0.008643V \tag{7}$$

$$DCLevel = 11.9V \tag{8}$$

2.3 Cost and Merit

Using the cost information described in Section 1, the cost of the circuit is

$$cost = 33.74MU \tag{9}$$

Considering the Merit equation given, the Merit score is

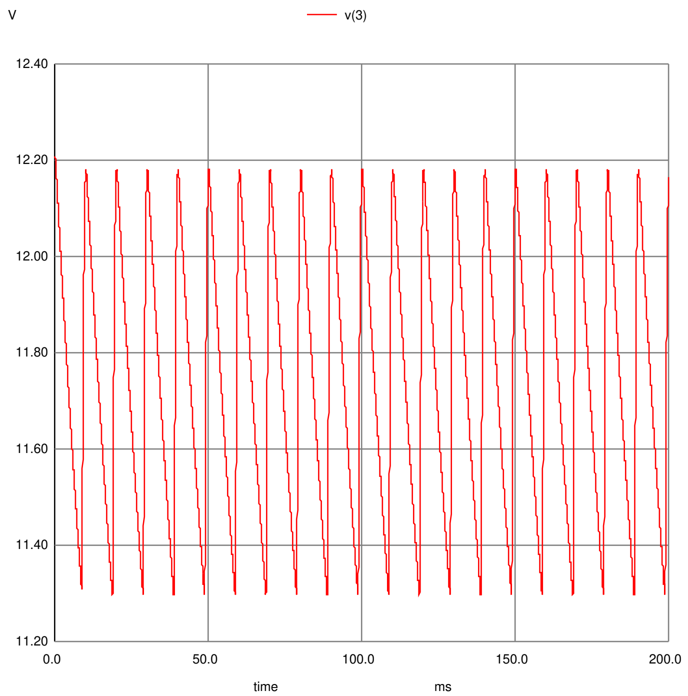
$$M = 0.2728 \tag{10}$$

3 Simulation Analysis

In this section, the intent is to simulate the circuit, using *Ngspice*.

Firstly, the AC/DC converter was subjected to a transient simulation for 10 periods. Then the outputs of the envelope detector and voltage regulator circuits were plotted as shown in the following figures, along with the output AC component plus DC Deviation plot.

Figure 3: Voltage after the envelper detector



The output voltage ripple and the DC level were simulated aswell and are as follows.

$$V_{Ripple} = 0.2933V \tag{11}$$

$$DCLevel = 10.798 \tag{12}$$

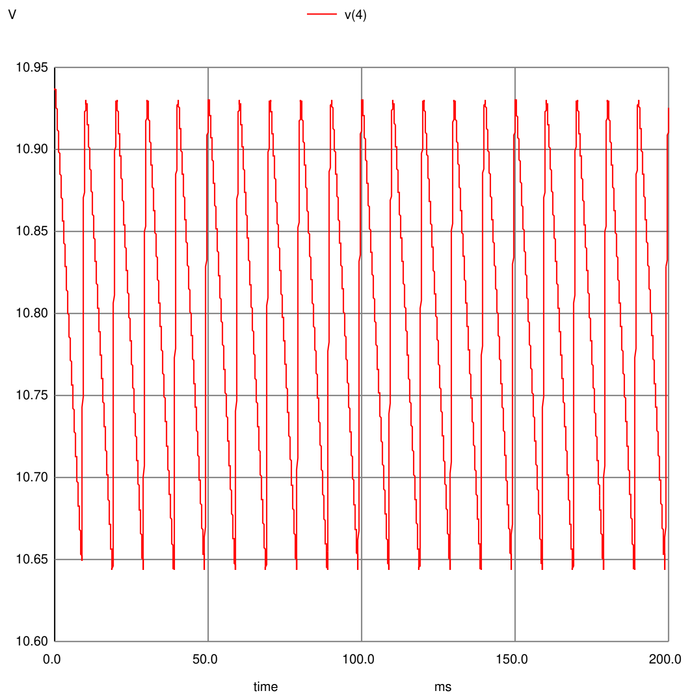
3.1 Comparisons

In this last subsection, we present the values and plots obtained from the Theoretical Analysis and the Simulation Analysis that refer to the same sets of evaluations, for visualization purposes, and comment on their similarities or discrepancies.

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!1111

All in all, there is an obvious difference between the theoretical and simulation results. The results obtained using *Octave* are closer to the aimed values and oscillate less. The values obtained with *Ngspice* are smaller ($v_4(t)$ is closer to 11V, for example). Nevertheless, the overall wave forms are similar, the frequency of the signals is the same, and, by both approaches, the results satisfyingly accurate. The discrepancies stem from the different considerations made in the theoretical model and in the Ngspice software's models. In the Ngspice models, the approximations used when diodes are involved are much more complex and accurate.

Figure 4: Voltage after the regulator



4 Conclusion

In this laboratory assignment, the objective of analysing and designing an AC/DC converter circuit has been achieved. Measures and plots of the output and ripple voltages have been computed, theoretically using the Octave maths tool and by circuit simulation using the Ngspice tool, and the final output came close to a DC 12V voltage. The discrepancies between the results acquired from the different tool are the consequence of different models used in calculations, being the models used in Ngspice tool regarding diodes much more precise, exact and complex.

Figure 5: Voltage diference between the regulator and the 12 V DC