TÉCNICO LISBOA

CIRCUIT THEORY AND ELECTRONICS FUNDAMENTALS

[MEAER]

INTEGRATED MASTER'S DEGREE IN AEROSPACE ENGINEERING

LABORATORY 3

AC/DC CONVERTER

May 5, 2021

Group 1:

António Nunes, 95770
Francisco Branco, 95788
Pedro Alves, 95836

Professor: José Teixeira de Sousa

Contents

List of Tables

List of Figures

1 Introduction

In this laboratory, we analysed in a theoretical approach as well as using software simulation, an AC/DC converter, composed with an Envelope Detector Circuit and a Voltage Regulator Circuit. In this report, a software simulation and theoretical analysis will be stacked up against each other. This assignement allowed us to deal with important concepts such as **diodes** and its diverse utility in circuits. In Figure 1 the stated circuit is presented.

A theoretical analysis of the circuit will be presented combining DC and incremental analysis, as well as a RC natural solution (for the envelope detector and voltage ripple calculation), giving us some insights on the non-linearity behaviour of the circuit. Also, it is important to notice that for better teory-experimental marriage, an ideal diode model is combined with a voltage source V_{ON} to simulate the potencial drop across a diode. Regarding the DC and incremental analysis of the voltage regulator circuit, we superimposed both solutions referred and studied the precision (time average of V_{out} - which should be as close to $12V$ as possible) and the voltage ripple of the output which stands for the difference between maximum and minimum peaks of the output voltage.

Simultaneously, the circuit is analysed by computational simulation tools, via *Ngspice*, and the results are compared to the theoretical results obtained, in Section 2. The conclusions of this study are outlined in Section 4.

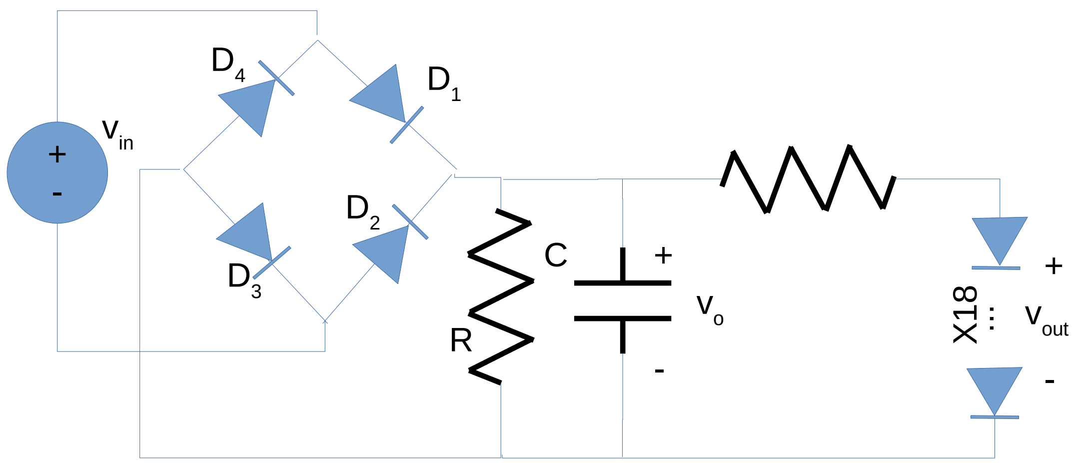

Figure 1: AC/DC Converter used.

2 Theoretical and Simulation Analysis

In order to compare *side by side*, we'll discuss the theoretical and simulation analysis at the same time.

However, one has to present the circuits implemented on the AC DC converter in order to fully understand the simulation. The constants values used during the theorectical analysis are expressed in the table below.

Name	**Value**	**Units**
V_{ON}[1]	0.63158	V [Volts]
I_s	$1 \cdot 10^{-14}$	A [Amperes]
R_{env}	140	$k\Omega$ [kOhms]
R_{vreg}	120	$k\Omega$ [kOhms]
C	600	μF [μFarad]
V_{in}	201.135	V [Volts]
N_i	1.14351	Transformer Ratio
V_T	$25.88794 \cdot 10^{-3}$	V [Volts]
η	1	Constant

Table 1: Constants Values

2.1 Envelope Detector Circuit

Firstly, we must discuss the first half of the circuit that was used. A Full Wave Bridge Rectifier along side a Resistor and a Capacitor in paralell were used to envelope the V_{in} sinusoidal wave. An important characteristic we should bear in mind is that, because it is best wanted to have the minimum ripple possible, the time constant τ of the capacitor discharge should be as high as possible, so that V_o stays as constant as possible during this time. To achieve this, because $\tau = RC$, a simple solution is to increase the resistor impedance (R) and the capacitor capacity (C). Of course, a more intelligent solution is to use a full-wave bridge rectifier insted of a half-wave, so that the time of discharge of the capacitor is decreased to half (the frequency of the wave is doubled). It should be noted that V_{in} is the transformed voltage from the plug: $\frac{230}{N_i}$.

A scheme of this circuit is presented below.

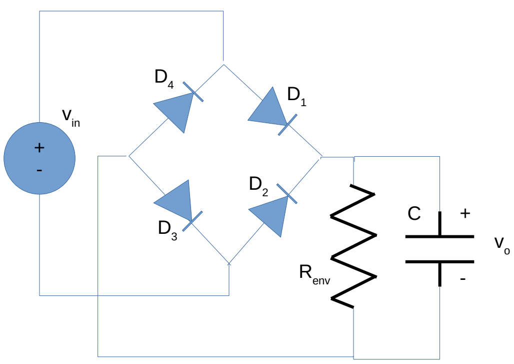

Figure 2: Envelope Detector Circuit

[1] V_{ON} value is computed using *Ngspice* results for V_{out}. By definition, $V_{ON} = \frac{V_{out}}{N_{diodes}}$

2.2 Voltage Regulator Circuit

The second half of the AC/DC converter is composed by a Voltage Limiter in series with a resistor, both in parallel to the Envelope Detector. The input voltage of this circuit is V_o of the Envelope Detector. The purpose of this circuit is, besides limiting the voltage, atenuating the rough ripple of V_o, see equation (4). This circuit is analysed in the following way: first, one should separate in DC and AC components. The DC simulation using the diode model described yields that the condition $V_o = N \cdot V_{ON}$ which ensures forward bias at all the diodes, and so, each one can be substituted by an equivalent voltage source V_{ON}. Of course, if $V_o < N \cdot V_{ON}$, the output voltage V_{out} is V_o, since the diodes are off, which means the circuit can be considered to be open. The AC incremental analysis is explained in further detail in the next section.

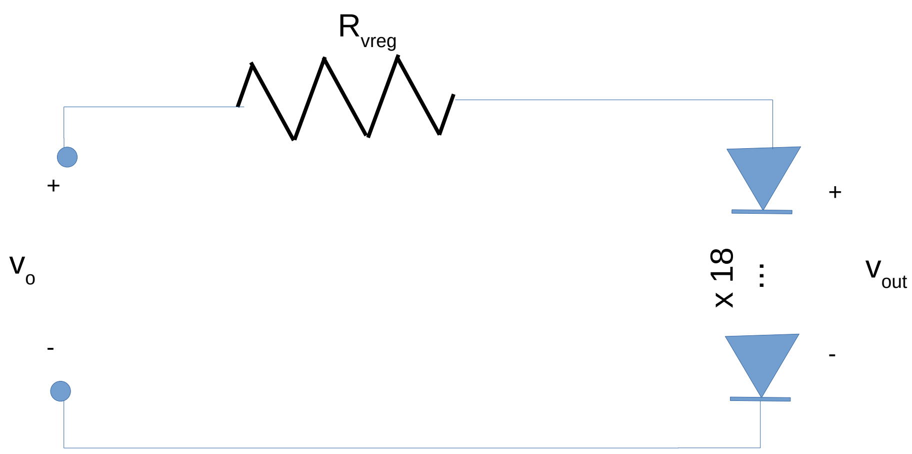

Figure 3: Voltage Regulator Circuit

In conclusion, when we merge these two circuits, we end up with our AC/DC converter in Figure 1.

2.3 Theoretical Analysis

For the Theoretical approach, we used the Ideal Model + Voltage Source, V_{ON}. This choice was made because it gives a pretty good approximation of the Diode Equation, and the latter is more time-consuming due to the need of solving numerically non-linear equations.

By applying KVL to the Envelope Detector, either in its positive cycle (when D_1 and D_3 are ON) or its negative cycle (when D_2 and D_4 are ON), we end up with,

$$V_o = -2V_{ON} + V_{in} \tag{1}$$

The diodes, however turn off when the current on the resistor, i_R, equals the current on the capacitor, i_C, which leads to,

$$\frac{Acos(\omega t|_{OFF}) - 2V_{ON}}{R_{eq}} = A\omega sin(\omega t|_{OFF}) \tag{2}$$

where $R_{eq} = R_{env}||R_{vreg}$

Solving this equation yields t_{OFF}, which allows us to know when the diodes turn off and an exponential decay, due to the the capacitor discharge, occurs.

$$V_o = V_o|_{t_{OFF}} e^{-\frac{t-t_{OFF}}{R_{eq}C}} \tag{3}$$

The Diodes turn on again when $V_{in} = V_o + 2V_{ON}$. With this non-linear equation one can estimate t_{ON}. By knowing that this behaviour repeats in each half wave period, we have all the needed information to theoricaly preview the behaviour of this circuit.

For the Voltage Regulator, we perform a DC + AC analysis, the latter using incremental analysis. If we separate both components, we have $V_o = V_O + v_o$

Concerning DC analysis, if $V_O > N \cdot V_{ON}$, then $V_O = N \cdot V_{ON}$, where $N \equiv$ the number of diodes.

Concerning the incremental analysis,

$$v_{out} = \frac{N \cdot r_d}{N \cdot r_d + R_{vreg}} v_o \tag{4}$$

where $r_d \equiv$ diode resistance using incremental analysis.

$$r_d = \frac{\eta V_T}{I_s e^{\frac{V_D}{\eta V_T}}} \tag{5}$$

where $V_T = \frac{kT}{q}$, $k \equiv$ *Boltzmann* constant, $T \equiv$ temperature (in K) and $q \equiv$ electron charge.

By making r_d smaller, it is expected that the v_{out} ripple should go down. A simple solution to decrease this resistance is simply to add layers of parallel diodes in series, so that I_s decreases if we take the relation for diodes in parallel: $I_{s_{eq}} = N \cdot I_s$. However, as we discuss in further detail in 3, it led us to a worse quality/cost ratio.

2.4 Comparison

We are now ready to make a comparison of the two approaches.

Let us start by seeing the overall panorama with the plots of V_{in}, V_o and V_{out} as well as the values of the deviation and ripple. We will firstly present the theoretical plots of these values, as well as a table with some important results.

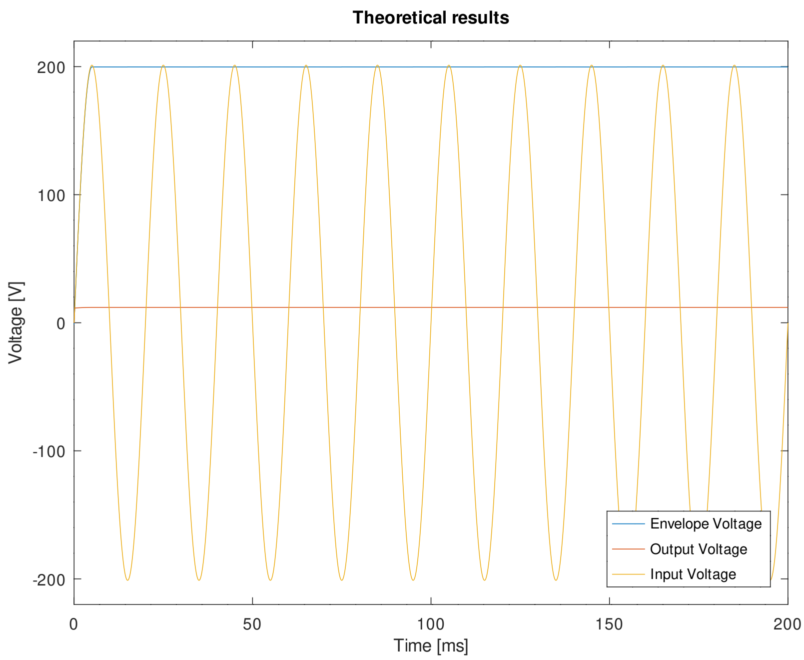

Figure 4: Theoretical Results

Name	Value [V]
Deviation	0.004618
DC level	12.004618
Ripple	0.000130

Table 2: Theoretical Values

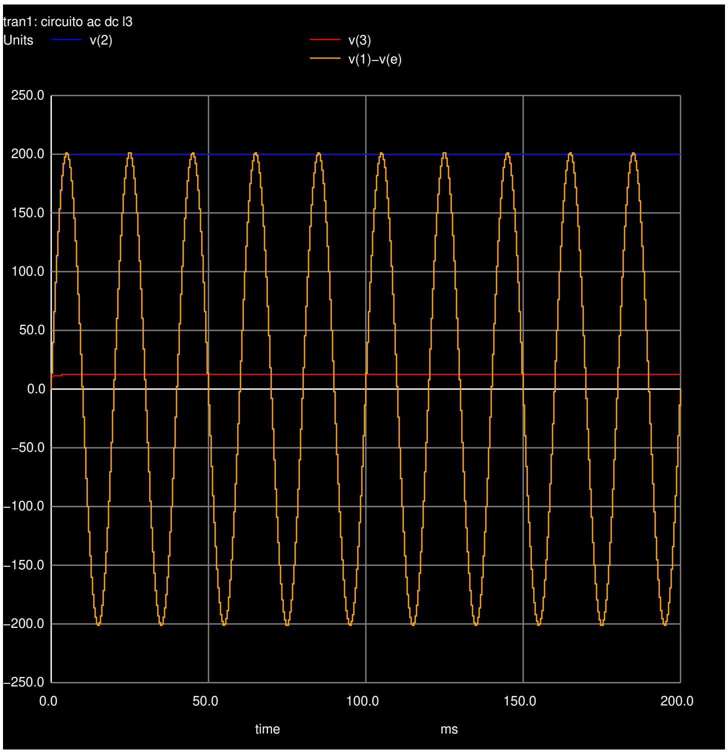

Figure 5: Simulation Results

Name	**Value [V]**
ripple	0.00012
deviation	0

Table 3: Simulation Values

Let us zoom in to analyse with further detail the ripple and the deviation. Again, we firstly present the plots obtained through the theoretical analysis, followed by the Ngspice simulation.

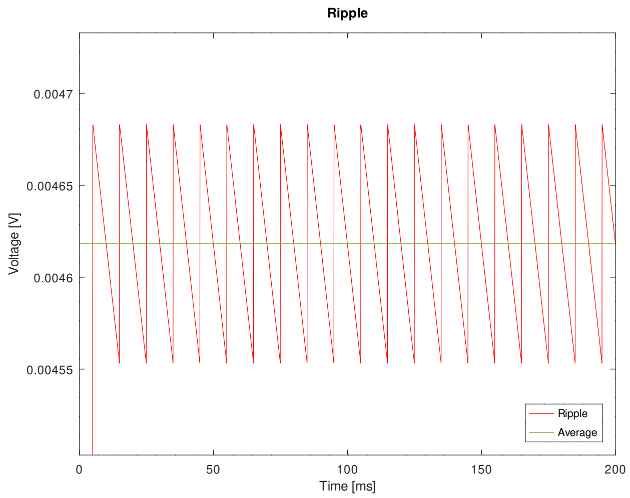

Figure 6: Theoretical Ripple and Deviation Results

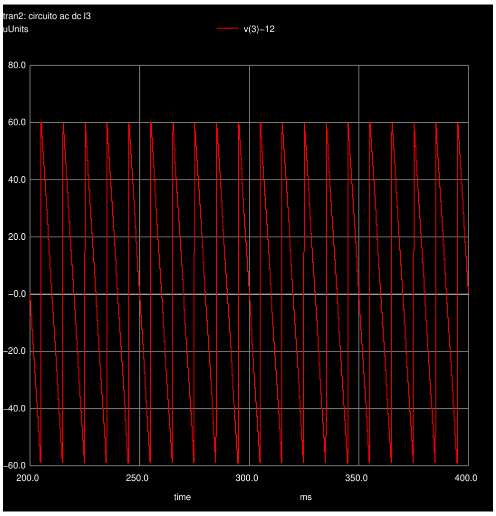

Figure 7: Simulation Ripple and Deviation Results

The overall behaviour and shape of both plots is **similar**. As important features we can underline the fact that V_o has **twice the frequency** of V_{in}, due to the full wave rectifier. We can also notice in both approaches an **exponential decay** due to the capacitor as seen in Section 2.3. It is important to notice that there is some kind of **distortion**, eventhough not a significant one, something expected due to the satisfying results we had.

With greater detail, we can easily realize, as seen in tables above as well as plot comparison, that the **theoretical ripple is bigger than the one presented on the simulation model**. However, when we experimented several circuits, sometimes we saw the opposite results. This occurance is yet just another example of the non-linearity and complexity of the model used by *Ngspice*, for small differences on the circuit can alter the behaviour of diodes themselves. Eventhough this happens, both results are interesting and satisfying, for they provide a somewhat stable output.

When it comes to the deviation (offset), a value of 0V was obtained (or smaller than $1E-06$ for Ngspice represents it as 0). On the theoretical side, we had a bigger offset. This difference, again, is not surprising, because there are a lot of different parameters involved on the simulation model, and just a few on the theoretical ones. Besides, we don't know for sure the value o V_{ON} used in *Ngspice*, as it changes depending on the circuit. However, we approximated it to a constant by dividing the Ngspice output average by the number of diodes used, as we needed a somewhat correct value to use in the theoretical analysis. As a remark, we can state that a deviation of 0V was a factor easier to achieve than the ripple, as we could just change V_{in} (see Section 3).

In a practical point of view, we can conclude that we had better results on *Ngspice* which represents the real circuit, so its positive. This has important applications in technologies that require precision.

3 Merit Results

From the results obtained through the Ngspice simulation (see Section 2.4) and considering we used the data shown in Section 1, we can compute the price and the merit using the *formulae* given in the lab assignment:

Name	Value
price	862.2
merit	9.58532

Table 4: Total Price and Merit

On the choice of the circuit, we started using a half wave rectifier circuit, which has a worse ratio quality/cost, which led us to choose the full wave. We also tried the option of multiple diodes in parallel, in order to reduce the resistance required. However, we saw that it gave us a higher cost as well as higher ripple, so it was not beneficial.

For our strategy, we opted to firstly give more attention to minimizing the ripple and the deviation, leaving the cost as a second thought. We then made small adjustments to further perfect our results, which in turn made the merit figure rise. Due to this, we saw that it was useful to increase V_{in} to values close to the ones on the primary. We even realise that for some circuits, if we had a higher secondary voltage, than primary, we could have bigger merits for some circuits (*e.g.* 24.5). To conclude, we began decreasing the cost until we found out the perfect compromise for us, giving the results shown in table 4.

4 Final Conclusion and General Notes

As a conclusion, we can state that, unlike previous lab assignments, there is not a major degree of similarity between both analysis. This was expected due to the fact that the circuit is non-linear because of the presence of diodes. The model used by *Ngspice* is far more complex than the theoretical model used. Despite these differences, the theoretical analysis can be quite accurate. Both *Ngspice* and *Octave* plots are similar. The only substancial error is in the voltage ripple and deviation calculations, however, because this AC/DC converter characteristics are so prone to subtle deviations, this is expected.

Although we tried to use exactly the the same values for the diode parameters (V_{on}, η, Is, V_T) that Ngspice uses for its diode model, the ideal linear model implemented in the theoretical analysis still was no match to the non-linear one implemented by *Ngspice*, which gave off results that can be assumed to be more in par with reality.

With all this in mind, this laboratory enabled us to deepen our knowledge regarding the diode, its different models and how it can be implemented to develop circuits with various purposes.