Circuit Theory and Electronics Fundamentals - T3

João D. Álvares[1], João M. Teixeira[1], and Rui M. Martins[1]

[1]*Instituto Superior Técnico, Av. Rovisco Pais 1, 1049-001 Lisboa*

4th May 2021

Abstract

In this report, one analyses how AC/DC converters work with a proposal of a circuit made up of an envelope detector circuit (that converted the AC current into DC and smoothed it) and a voltage regulator circuit (which limited the voltage) that was built to convert an AC current input (with 230V and frequency of 50Hz) into a DC current output with a voltage of 12V. This analysis was made both theoretically and computationally by using *Ngspice* to simulate the circuit (this time the analysis was essentialy made through the *Ngspice* simulation). The results obtained (...)

Resumo

Neste relatório, foi proposta a utilização de um conversor AC/DC com o principal objetivo de converter um *input* de corrente AC (um sinal sinusoidal de frequência 50Hz e com uma tensão associada de 230V) num *output* de corrente DC (com uma tensão nos terminais de saída de 12V). Para tal, o conversor é constituído de um *envelope detector* cuja principal função é converter o sinal de alternado em contínuo e suavizar o mesmo e de um *voltage limiter*, que teve como função limitar superiormente a tensão nos terminais de saída do circuito. Os resultados obtidos (...)

Contents

List of Tables

List of Figures

1 Introduction

The main goal of this laboratory assignment is to analyse and better understand how AC/DC converters work and to find an optimal solution for the circuit built (i.e., a solution that allow one to get the result wanted at a minimal cost - having in mind that components like capacitors and resistors are very expensive comparatively with components like diodes).

AC/DC converters are circuits that act as transformers of the circuit's current from an alternating input (AC) - in this case of $230V$ and a frequency of $50Hz$ - to a direct output (DC) - which in the case will be of $12V$.

Essentially, these converters use rectifiers (which will turn the AC input into DC output - can be full-wave and half-wave as it will be seen), voltage regulators (that can limit positive and/or negative voltages and are made up of multiple diodes) and a reservoir capacitor which smooths the pulsating DC current.

The circuit used to simulate this was:

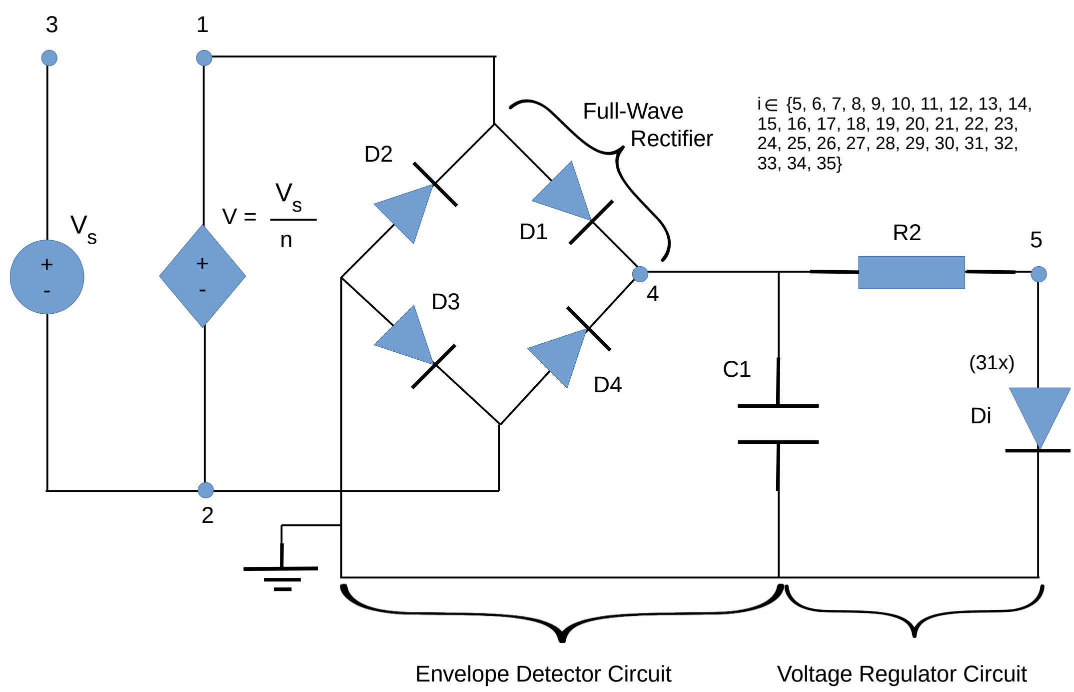


Figure 1: *Circuit analysed. For the voltage dependent voltage source, one used* $n = 17.78988$ *- although this is not an integer, the students were motivated to use rational numbers for* n*.*

The results were obtained through a theoretical analysis, in which one predicted the output by using a theoretical method that suited the real circuit and through a simulation that was made using *Ngspice*. The results obtained through both methods will be analysed throughout the report.

To evaluate the built circuit, one used a merit score which depended on the cost of the circuit, the output voltage ripples and the distance from the average value of output voltage obtained to the desired value ($V = 12V$). Then, as the AC/DC converter is as efficient as lower the cost, the ripples and the difference to the desired value are, the formula used to evaluate the score obtained with the AC/DC converter was:

$$M = \frac{1}{c(r + d + 10^{-6})} \tag{1}$$

in which c represents the total cost of the circuit, r represents the output voltage ripples and d represents the average value for $|v_0 - 12|$ in which v_o is the output voltage.

To calculate the cost, one considered that the cost of the circuit equals the sum of the cost of the resistors, capacitors and diodes and the "price table" for this components is given as:

Component	Price
Resistors	1 MU $k\Omega^{-1}$
Capacitors	1 MU μF^{-1}
Diodes	0.1 MU diode^{-1}

Table 1: Price table for the components used

2 Simulation Analysis and the Path for the Greater Good

For this analysis, we used the default diode model available in *Ngspice*, having then created the circuit presented in Fig. 1 (it was using Ngspice that we determined, in fact, what was the best circuit for our purposes). So, we used the following chain of thought in order to do what we did. In the envelope detector part, we took the limit as $R \to \infty$, which is the same as no current passing through there, which is the same as nothing being there. This drastically reduces our ripples, for then we do not need the resistance at all and the cost is 0. Of course this is slightly counterbalanced by the presence of the 10^{-6}, in the merit calculation ((1)). Knowing that the voltage ripples reduce to (2)

$$V_{ripple} = v_o(0) - v_o(T) = A\left(1 - e^{-\frac{T}{2RC}}\right) \tag{2}$$

we can model very superficially the function that should describe this merit in terms of our components ((3)).

$$f(R, C) = \frac{1}{(R + C) * A * \left(1 - e^{-\frac{T}{2RC}} + 10^{-6}\right)} \tag{3}$$

From that, we can then plot what's the aspect of the function, shown in Fig. 10.

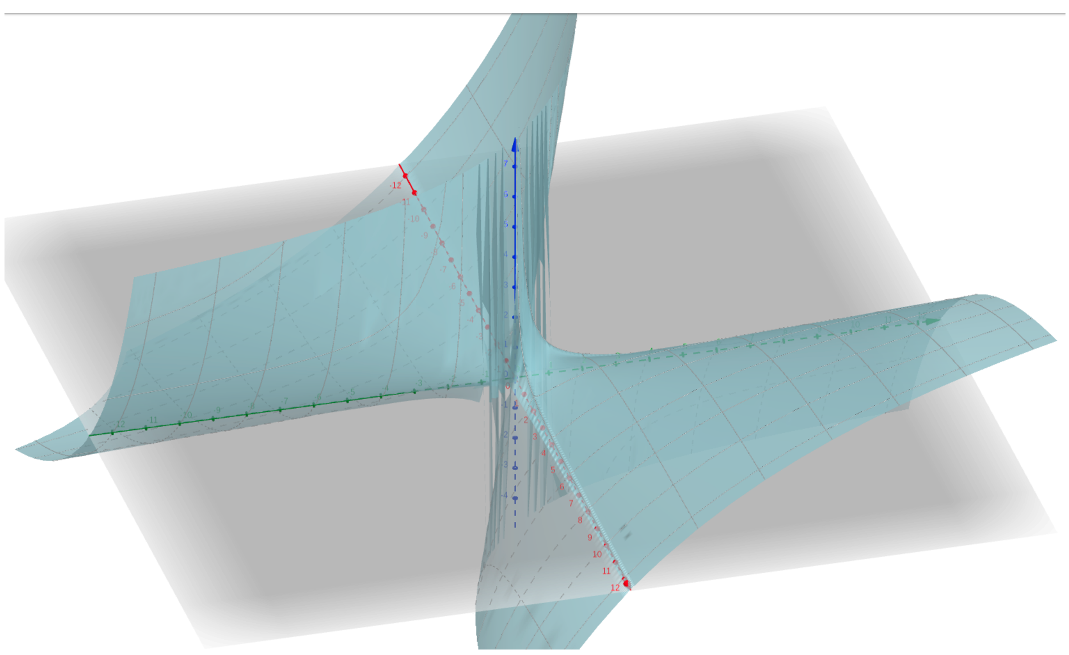


Figure 2: *Merit approximated function. Plot obtained using GeoGebra3D*

And thus we have the justification of our doings by removing the resistor. Then, we made use of the specifications of the non-linear part of the diodes and found that there is a very fortunate correlation between the amount of diodes in the voltage regulator and the ripples in the end voltage. What this means is that we can let go of the equation that says that $R2$ must be much greater than rd, for this is but a very rough approximation of the real life's diode behaviour. The only setback of this method is the stabilization time, but the merit does not take that into account, so we're good. This way, we used 31 diodes, for nonetheless we did not want to have an exaggerated stabilization time. WARNING: This will cause big differences between the theoretical analysis and the simulation. The model presented in class, when applied to Octave, will basically serve of nothing.

Finally, one found the values considered as the best to find an optimal merit score which are given by:

Component	Amount	Value	Cost
Resistors	1	$0.2k\Omega$	0.2
Capacitors	1	1.33μF	1.33
Diodes	31+4	-	3.5

Table 2: Components values shown in Fig. 1.

2.1 Cost and Merit Score

Evaluating the merit, and knowing that the cost is given by (4)

$$c = cost = 35 \cdot 0.1 + 0.2 + 1.33 = 4.85\text{MU} \tag{4}$$

Using the values obtained for c, d and r, one can finally calculate the merit score obtained with this circuit (calculated using equation (1)), which allows one to get the following score:

$$M = 3.31752e + 02 \tag{5}$$

2.2 Envelope Detector and Voltage Regulator output voltages

To understand the role played by the envelope detector and the voltage regulator circuits in our circuit one plotted the results obtained for the output voltages in each one of them. So, starting by the plot obtained at the output terminals of the envelope detector circuit

$$V = v_4 \tag{6}$$

Plotting this function:

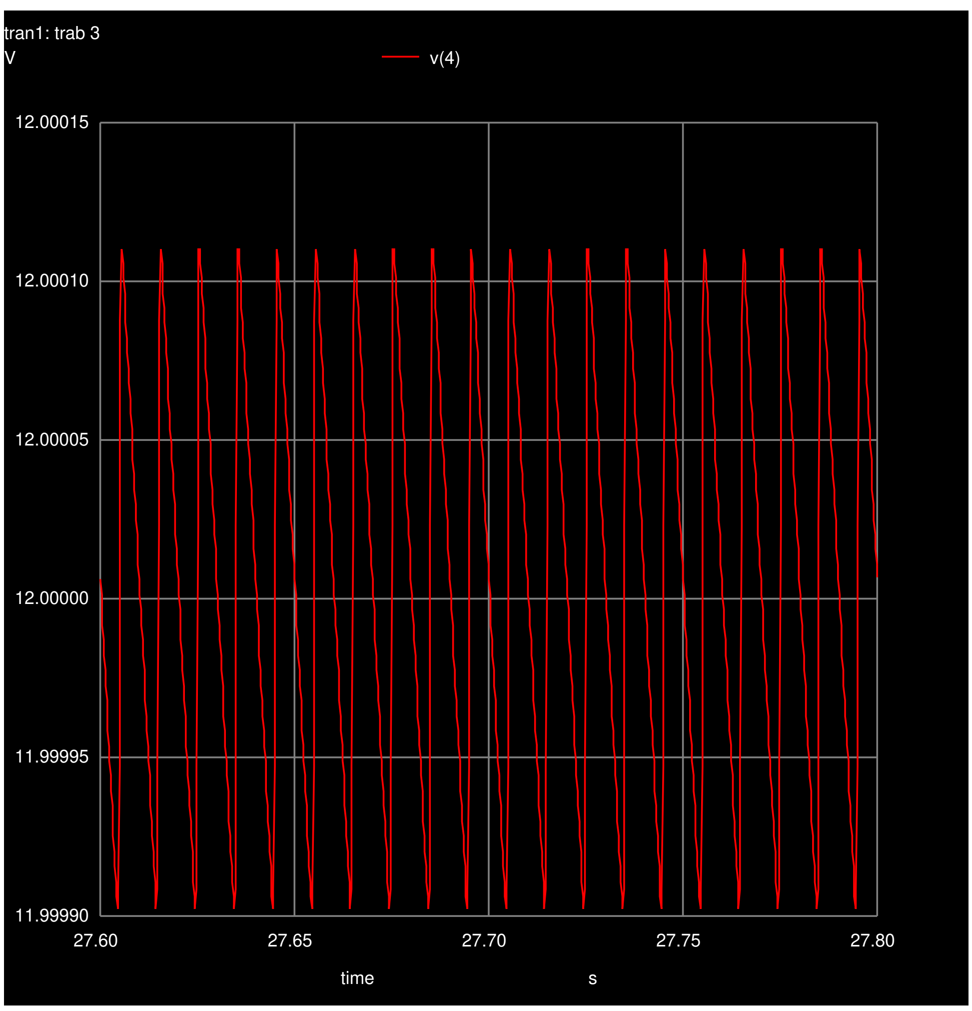


Figure 3: *Plot of the output voltage registered at the output terminals of the envelope detector circuit*

To plot the output voltage observed on the terminals of the voltage regulator circuit, one plotted the function given by

$$V = v_5 \tag{7}$$

which allowed to get

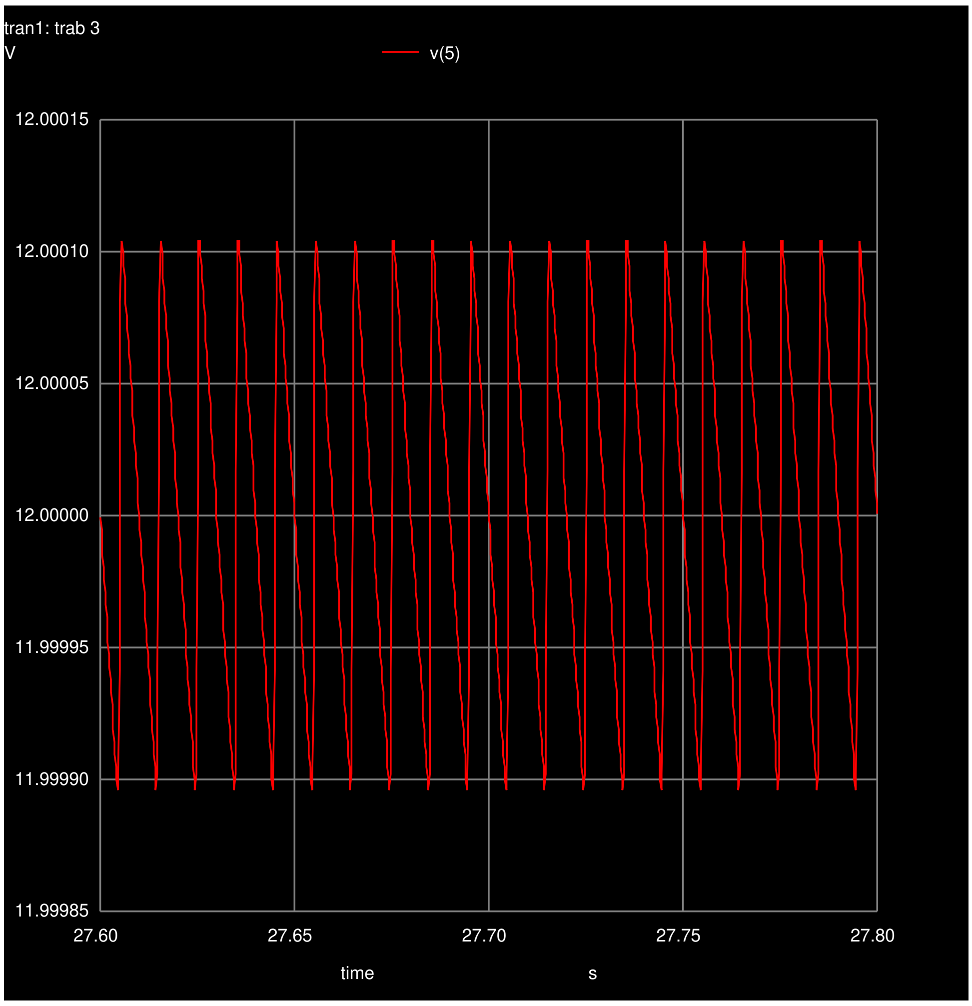


Figure 4: *Plot of the output voltage registered at the output terminals of the voltage regulator circuit*

Comparing the two plots one can conclude that the plots are perfectly similar except on the y axis scale. In fact, the difference between the two plots was caused by the action of our voltage regulator circuit. As the number of diodes was chosen to set an upper limit to the output voltage of the circuit to the desired value $V = 12V$ the difference registered between the two graphs only reflects the effect of that part of the circuit. And in fact, as one can see, the voltage regulator circuit is working as it is supposed to because the voltage that was intially oscillating around $14V$ is oscillating around $12V$ at the output terminals of the voltage regulator. This translates, numerically, into: INSERT HERE

$$v_{6average} = \tag{8}$$

2.3 Plotting $v_o - 12$

Finally, to conclude the simulation analysis (which one can antecipate that allowed to get precise and satisfactory results) and to better understand how the output voltage behaved next to the desired value of $V = 12V$, one plotted the function $V = v_o - 12$:

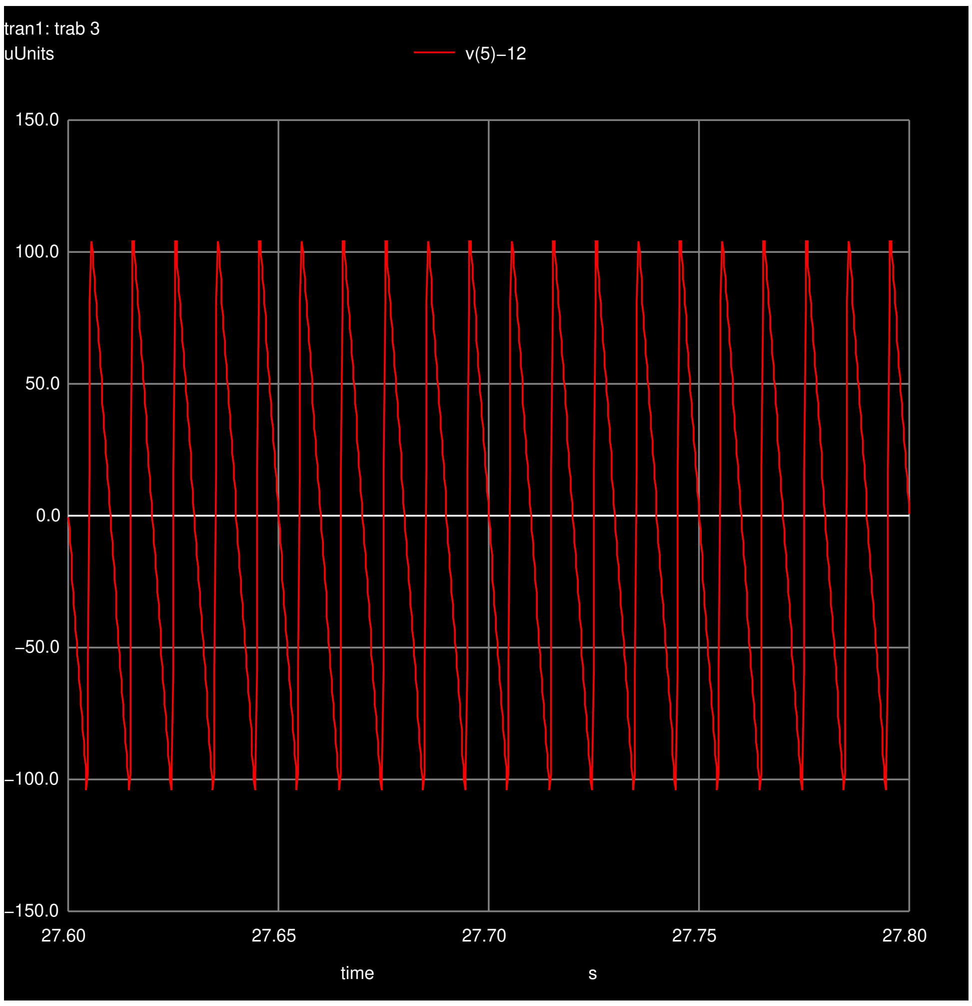


Figure 5: *Plot of the difference between the circuit's output voltage and the desired voltage, $V = 12V$*

which, numerically, gives: INSERT HERE

$$|v_6 - 12| = \tag{9}$$

3 Theoretical Analysis

In this analysis, we have some differences from the theoretical analysis for some of the standard parts of an AC-DC converter, those being the high number of diodes and the absence of the resistor in parallel with the capacitor.

Initially we started with a voltage source with the standard $\frac{230}{n}V$ and $50Hz$, in which n was the one experimentally determined to produce the best merit in ngspice, in this case being 17.78988, which produces an amplitude of 12.9287.

The first part of the circuit to be analysed is the full-wave rectifier, which has a behaviour that is well described by the absolute value of the input wave, because the voltage is way bigger than V_{ON} of the diode.

$$V_{FullWave} = \left|\frac{V_i n}{n}\right| \tag{10}$$

The next component is the capacitor has a envelope detector. The formula for calculating the envelope voltage in each period is

$$V_{envelope} = A\cos(\omega t_{OFF})e^{-\frac{t_{ON}-t_{OFF}}{RC}} \tag{11}$$

with

$$t_{OFF} = \frac{1}{\omega}\arctan\left(\frac{1}{\omega RC}\right) \tag{12}$$

But in our circuit we did not included the resistor so, with it being in parallel it will behave as a if the resistance is infinite, but we chose to, in octave use a very high number to represent it, in this case $10^9\Omega$, not only because it is easier for the Octave to operate withe the real number but also because if can represent the high resistance that the diodes are offering, even bellow V_{ON}.

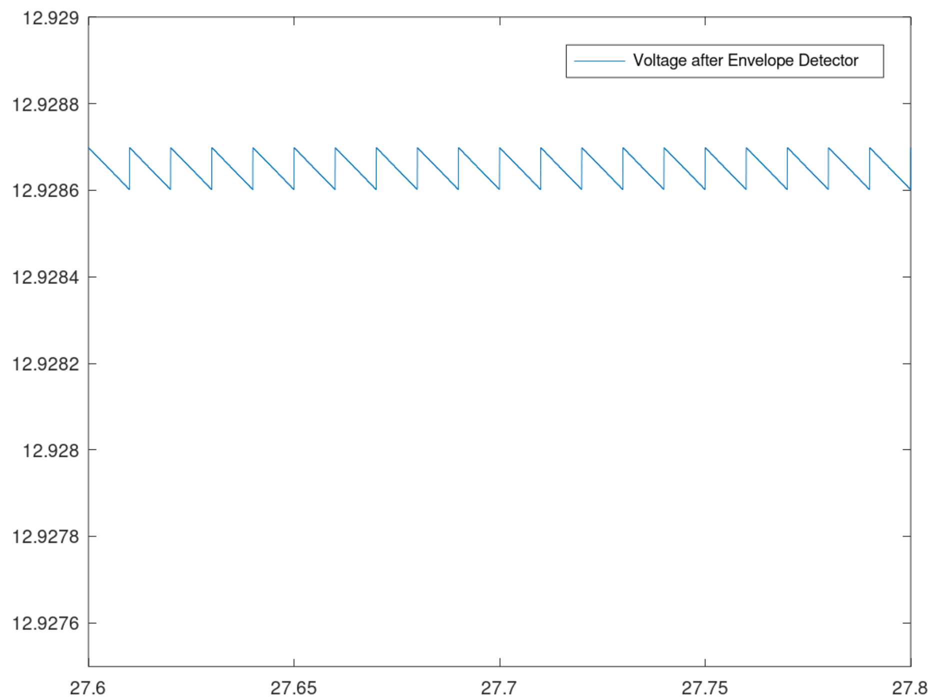
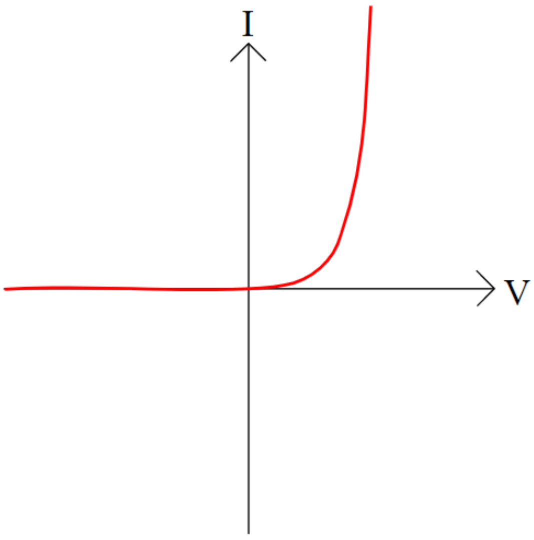


Figure 6: *Signal as it exits the envelope detector according to the theoretical analysis.*

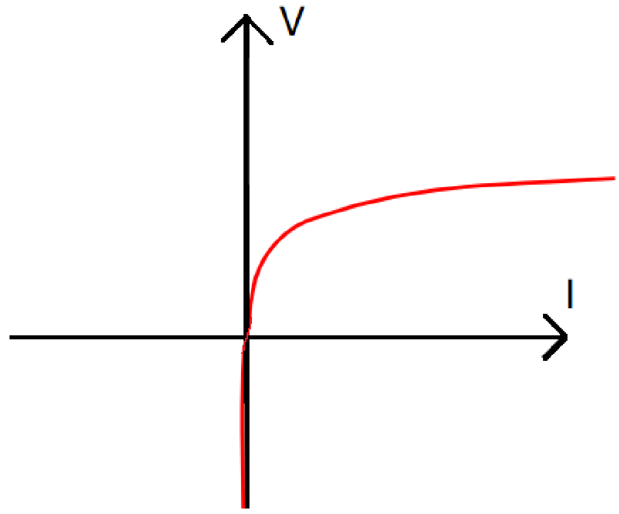
The last part of the circuit is the voltage regulator, in our case it's a set of 31 diodes in series, all in parallel with the capacitor. In this analysis we followed the expected formulae, although, because of the heavy difference between simulation and theoretical analysis we can begin disprove the usage of these equations for higher number of diodes.

First we must understand the behavior of the diodes and the best way to understand it it through it's incremental resistance, it starts of as really high but than it decreases to more constant and low values and this change is exponential. That being said, the formula is the following.

$$r_d = \frac{\eta V_T}{I_S e^{\frac{V_D}{\eta V_T}}} \tag{13}$$



(a) Current as function of voltage



(b) Voltage as function of current

Figure 7: I-V characteristic of a diode

In this figure we can both understand that the diodes have different regions depending on the voltage but, considering $R = \frac{dV}{dI}$, from the second figure we conclude that the resistance if extremely high for the OFF zone of the diode and it can be used as a high resistance instead of actual resistors.

with $\eta = 1$, $V_T = 0.025V$, $I_S = 10^-14$ and V_D being the the voltage difference in each diode that the estimated as being the value in the input of the set of diodes divided by the number of diodes. And, as expected we obtained a big value that is 247280.745039 Ω. The biggest difference to the simulation analysis starts here: the high number of diodes seems to help stabilize the signal, but the following formula indicates that for high number of diodes and diode incremental resistance the oscillation after the voltage regulator must be the same as the one leaving the envelope detector

$$v_o = \frac{Nr_d}{Nr_d + R} V_{envelope} \tag{14}$$

The only thing left to discuss is the DC component of this approximation for the voltage regulator which will be $V_{envelope}$ unless $V_{envelope}$ is bigger than NV_{ON}, but V_{ON} is some value between 0.65 and 0.7 $k\Omega$ so it's upper limit with $N = 31$ is way higher than any value in $V_{envelope}$, so the DC component is only change, and just a little, in the oscillation amplitude from the previous formula.

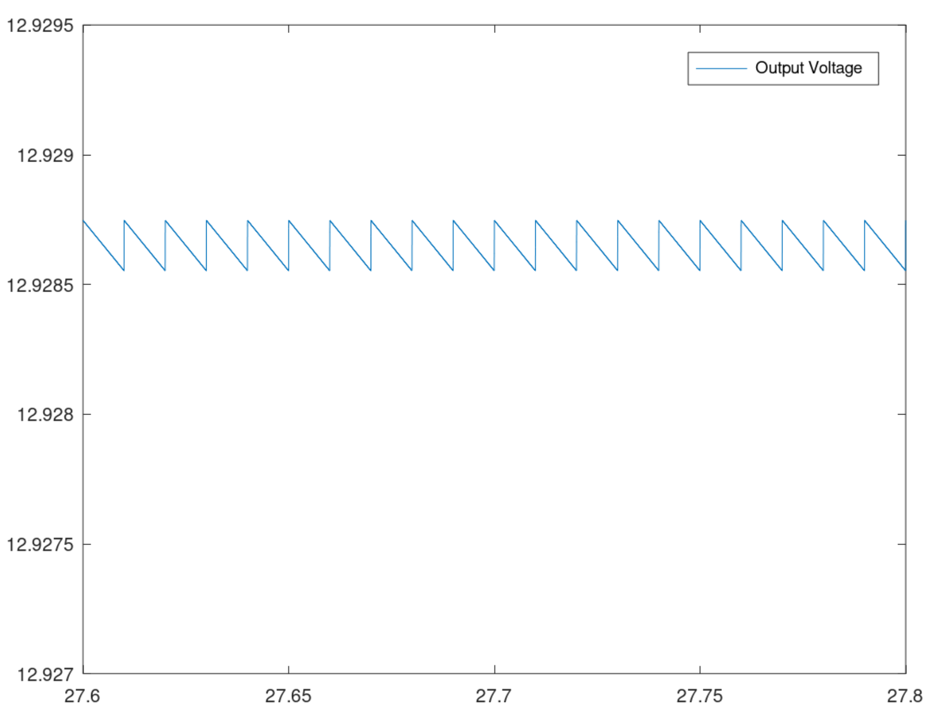


Figure 8: *Output voltage according to the theoretical analysis.*

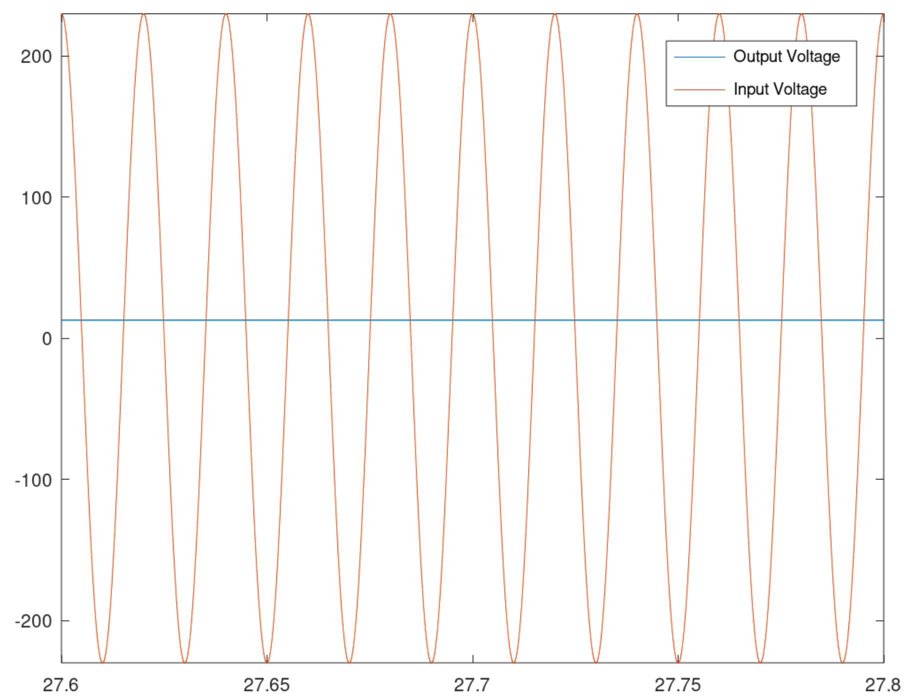


Figure 9: *Comparison between output and input voltage.*

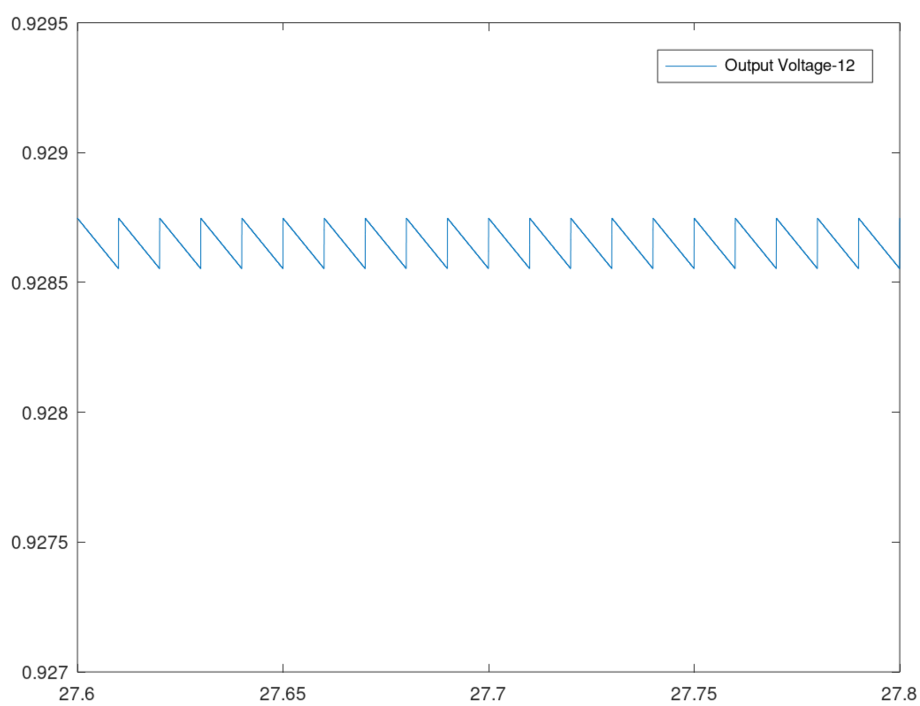


Figure 10: *Output signal minus the 12V goal.*

With this output and the components used the final relevant values are the average of the signal 12.928650 V, the ripple 0.000194 and finally 0.232528 , but as we know the value of the average can be easily adjusted by n in the transformer so, removing that term, the corrected merit in the theoretical analysis is 1106.692837 .

4 Conclusion

In this laboratory assignment, as presented above, we were proposed to build and study the behaviour of an AC/DC converter. To build it, one used an envelope detector circuit (composed by a full-wave rectifier which turns the AC input into a DC output and a capacitor which smooths the oscillating current) and a voltage regulator circuit (that uses a resistor and multiple diodes which limit the output voltage to the desired value - 12V).

To evaluate how good the built converter was, we recurred the merit score presented in (1). The results obtained in the simulation analysis (section 2) were very satisfactory except the stabilization time of the circuit, but that wasn't taken into account in the score obtained for the circuit although we had in mind that the circuit shouldn't have a very long stabilization time. The results given by the simulation analysis show an output signal with an average value displaced $\sim 10^{-7}$ from the desired value of 12V and with ripples $\sim 10^{-4}$. The output voltage obtained is thus almost perfectly equal to the desired output voltage for this converter, which shows that the goal of this simulation was successfully achieved.

Comparing the simulation and the theoretical results one can notice a significant difference between the plots and the results obtained with both as anticipated on section 2. In fact, this phenomenon can be justified by the multiple approximations made in the theoretical analysis, namely on the diode model used. In the analysis made using *Octave*, one used ideal diode models which neglect the non-linear behaviour of these components which ends up introducing discrepancies between the analysis methods used.

However, it is also important to notice that the results obtained on the theoretical analysis were affected of very small ripples. Moreover, adjusting the value for n associated with the transformer, we were able to obtain a merit score for the theoretical analysis even higher than the one obtained for the simulation analysis.

Therefore, and having explained the differences registered between the theoretical and the simulation analysis, it can be stated that the goals for this laboratory assignment were successfully achieved.

References

[1] Ngspice official website, *http://ngspice.sourceforge.net/*