

# Circuit Theory and Electronics Fundamentals 2020/2021

Integrated Masters in Aerospace Engineering, Técnico, University of Lisbon

## Third Laboratory Report

May 5th, 2021

Rui Rodrigues, 95844  
Tiago Silva, 95850  
Tomás Ribeiro, 95854

## Contents

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>Theoretical Introduction</b>	<b>2</b>
<b>3</b>	<b>Theoretical Analysis</b>	<b>2</b>
3.1	Envelope Detector Circuit . . . . .	2
3.2	Voltage Regulator Circuit . . . . .	3
3.3	Voltage Ripple . . . . .	4
3.4	The Output DC Level . . . . .	4
<b>4</b>	<b>Simulation Analysis</b>	<b>5</b>
4.1	Envelope Circuit Output . . . . .	6
4.2	Circuit Output . . . . .	6
<b>5</b>	<b>Relative Error and Graphic Analysis</b>	<b>8</b>
5.1	Topic I . . . . .	8
5.2	Topic II . . . . .	9
5.3	Topic III . . . . .	9
5.4	Topic IV . . . . .	10
5.5	Topic V . . . . .	10
<b>6</b>	<b>Conclusion</b>	<b>10</b>

## 1 Introduction

The objective of this laboratory assignment is to study and implement an AC/DC converter circuit while, at the same time, trying to maximize the figure of merit M. The AC/DC converter

circuit is composed of two circuits: an Envelope Detector circuit, an electronic circuit that takes a (relatively) high-frequency amplitude modulated signal as input and provides an output, which is the demodulated envelope of the original signal, and a Voltage Regulator circuit, a system designed to automatically maintain a constant voltage. The circuit can be seen in Figure 1.

In Section 2, a theoretical introduction is made in order to contextualize all the main principles that sustain our analysis of the circuit. This circuit is carefully analysed in Section 3, where the results are obtained in GNU Octave. Also, in Section 4, the circuit is analysed by simulation through the use of NGSpice to simulate the electric circuit behaviour. The results of the simulation of Section 4 are then compared to the theoretical results obtained in Section 3 and the comparative results are expressed in Section 5. The conclusions of this study are outlined in the final part of the report, in Section 6.

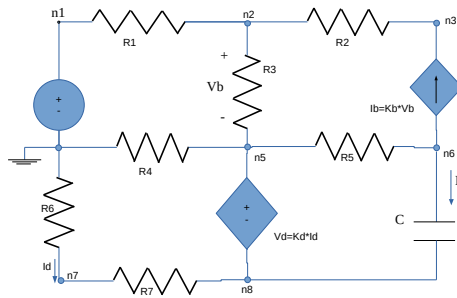


Figure 1: Third laboratory circuit.

## 2 Theoretical Introduction

## 3 Theoretical Analysis

THIS IS NOT ACTUAL In this section, we can find the results of each topic required in the theoretical analysis. The numeric results or graphics are presented alongside a short explanation of the interpretation of the problem.

$$\left\{ \begin{array}{l} v_t = v_n + v_f \\ f = 1kHz = 1000Hz \\ \text{NEEDTOBEIMPROVED} \\ A = 14; \\ T = 1/50; \\ w = 2 * \pi * (1/T); \\ R = 1000 \\ C = 100e - 6 \end{array} \right.$$

### 3.1 Envelope Detector Circuit

The voltage source  $v_S$  produces a sinusoidal wave and is defined by the following expression:  $v_S = A \cos(\omega t)$ .

The envelope detector circuit includes a full-wave bridge rectifier, whose voltage output  $v_{0_{rect}}$  corresponds to the absolute value of the voltage source  $v_S$ .

$$v_{0_{rect}} = \begin{cases} v_S, v_S \geq 0 \\ -v_S, v_S \leq 0 \end{cases}$$

Finally, the envelope detector circuit voltage  $v_0$  is the highest between  $v_{0_{rect}}$ , the full-wave bridge rectifier voltage, and the voltage  $v_{0_{exp}}$  induced when the capacitor  $C$  discharges through the resistor  $R$ , described by the following equation:  $v_{0_{exp}} = A \cos(\omega t_{OFF}) e^{\frac{-(t_{ON} - t_{OFF})}{RC}}$ .

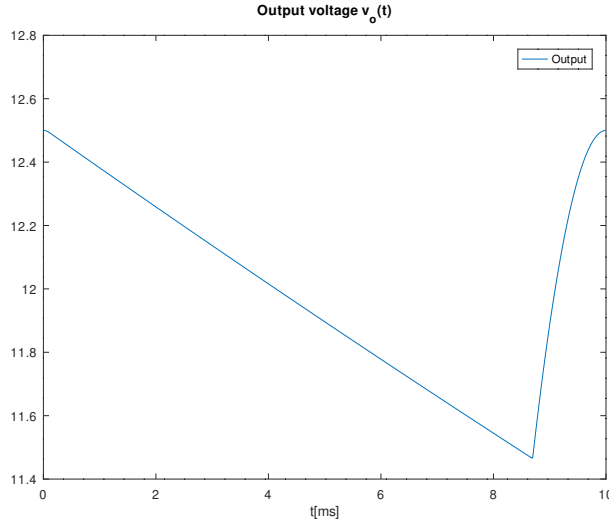


Figure 2: The final envelope detector circuit voltage  $v_0$  during a half-period interval.

where  $t$  is expressed in milliseconds (ms) along the x-axis and  $v_0$ , the envelope output voltage, is expressed in Volts (V) along the y-axis.

### 3.2 Voltage Regulator Circuit

The voltage regulator attenuates oscillations in the input signal without frequency dependence and takes advantage of the non-linear characteristic of the  $N$  diodes included in the positive voltage limiter.

The voltage analysis of the regulator applies the incremental analysis method, separating the DC and incremental components.

The diode incremental resistance is calculated using the following expression:  $r_d = \frac{\eta v_t}{I_s e^{\frac{v_d}{\eta v_t}}}$ .

Applying the voltage divider rule, the AC increment  $v_{out}$  is defined using the following relation:  $v_{out} = \frac{Nr_d}{Nr_d + R2} v_O$ .

INCLUDE A PARAGRAPH HERE TO EXPLAIN HIGH VALUE OF R2.

On the other hand, the DC voltage  $V_{OUT}$  is obtained multiplying the diode voltage  $v_d$  by the number of diodes  $N$  used in the voltage regulator circuit:  $V_{OUT} = N v_d$ .

$$\left\{ \begin{array}{l} \text{NEED TO BE IMPROVED} \\ n = 17; \\ R2 = 10e3; \\ \eta = 1; \\ v_t = 25e - 3; \\ v_d = 0.706; \\ I_s = 1e - 14; \end{array} \right.$$

The final voltage of the regulator circuit  $v_{OUT}$  is obtained adding the results of the incremental analysis and the DC analysis:  $v_{OUT} = v_{out} + V_{OUT}$ .

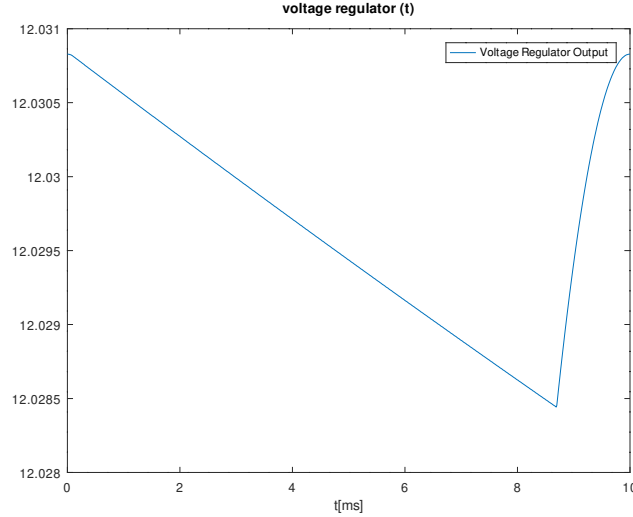


Figure 3: The final voltage  $v_{OUT}$  of the voltage regulator circuit during a half-period interval.

where  $t$  is expressed in milliseconds (ms) along the x-axis and  $v_{OUT}$ , the voltage of the regulator, is expressed in Volts (V) along the y-axis.

### 3.3 Voltage Ripple

The voltage ripple can be improved using a full-wave rectifier circuit in the envelope detector circuit.

The value of the voltage ripple  $v_{ripple}$  can be calculated relating the maximum and the minimum values of the final voltage of the regulator circuit  $v_{OUT}$ , using the following expression:  
 $v_{ripple} = \max(v_{OUT}) - \min(v_{OUT})$ .

<b>THIS SHOULD PRINT THE RIPPLE BELOW</b>	
THE OUTPUT VOLTAGE RIPPLE IS	0.00238568 V

### 3.4 The Output DC Level

The  $t_{ON}$  is calculated using the Newton-Raphson iterative method, starting with a function obtained through the following equation:  $A\cos(\omega t_{ON}) = A\cos(\omega t_{OFF})e^{\frac{-(t_{ON}-t_{OFF})}{RC}}$ .

The DC level output average is obtained integrating the sinusoidal and exponential functions during a half-period interval. The calculations are shown below.

$$t \in [0, t_{OFF})$$

$$\int_0^{t_{OFF}} = \frac{Nr_d}{Nr_d + R_2} \frac{A}{\omega} \sin(\omega t_{OFF}) + Nv_d t_{OFF}$$

$$t \in [t_{OFF}, t_{ON})$$

$$\int_{t_{OFF}}^{t_{ON}} = \frac{Nr_d}{Nr_d + R_2} (-\text{ARCCos}(\omega t_{OFF}) (e^{\frac{-(t_{ON}-t_{OFF})}{RC}} - 1)) + Nv_d (t_{ON} - t_{OFF})$$

$$t \in [t_{ON}, \frac{T}{2}]$$

$$\int_{t_{ON}}^{\frac{T}{2}} = \frac{Nr_d}{Nr_d + R_2} \frac{A}{\omega} (\sin(\omega \frac{T}{2}) - \sin(\omega t_{ON})) + Nv_d (\frac{T}{2} - t_{ON})$$

$$\bar{V} = \frac{\int_0^{t_{OFF}} + \int_{t_{OFF}}^{t_{ON}} + \int_{t_{ON}}^{\frac{T}{2}}}{\frac{T}{2}}$$

<b>THIS SHOULD PRINT THE AVERAGE BELOW</b>		
THE OUTPUT VOLTAGE AVERAGE LEVEL IS		12.0224 V

The Output DC Level accuracy can be evaluated through comparing the final voltage of the circuit  $v_{OUT}$  and the 12V constant function.

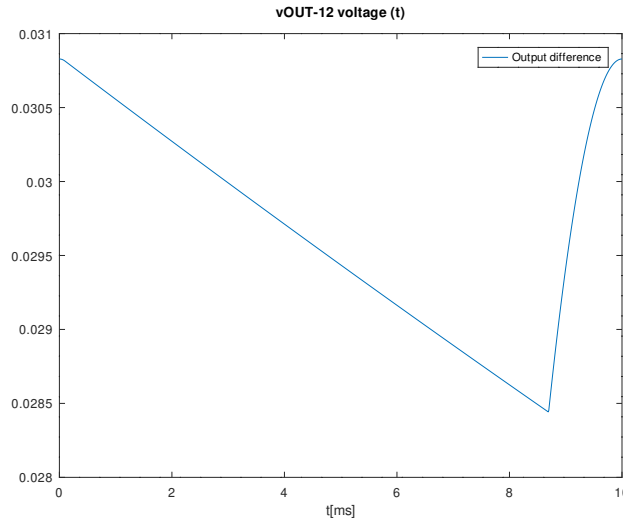


Figure 4: The voltage  $v_{OUT} - 12$  during a half-period interval.

Where  $t$  is expressed in seconds (s) along the x-axis and  $v_{OUT} - 12$ , the output voltage difference, is expressed in Volts (V) along the y-axis.

## 4 Simulation Analysis

In this section, we can find the results of each topic required in the simulation analysis. The numeric results or graphics are presented alongside a short explanation of the interpretation of the problem. All of the results were obtained using NGSpice and the section is divided in two different subsections.

In this part of the lab, we tried to build the circuit in such a way that it would be as similar as possible as the one used in the Octave theoretical part. The most important difference between both approaches is that in Ngspice we had to use more diodes in the regulator circuit than in Octave. This is because while in Octave we used the standard diode voltage (0.7V), requiring 17 diodes, in Ngspice that wasn't possible because diodes have varying voltages. Given that, our Ngspice voltage regulator circuit is made of 23 diodes.

Another difference from the Octave model, is that the ideal diode model used in the envelope circuit is not possible to replicate in Ngspice. However, this does not affect the final results because the transformer conversion factor was updated to compensate the fact that there is going to be a voltage drop in the envelope circuit diodes.

## 4.1 Envelope Circuit Output

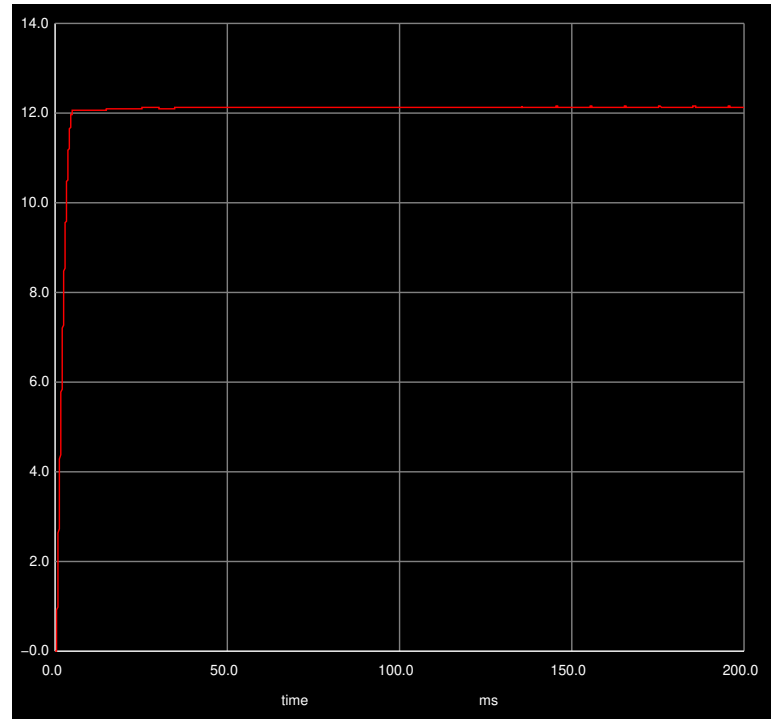


Figure 5: The final envelope detector circuit voltage  $v_0$  during 10 periods

In the graphic above, it is possible to see that, even though it is not the final result of the whole converter, it is already very close to  $12V$ . It is important to note that, if we zoom in, there will be a certain level of ripple that may affect our desired result, given that the signal hasn't gone through the voltage regulator yet. Still on this topic, a particularity of Ngspice is the rising line at the very beginning which clearly stands for some transient the software uses.

## 4.2 Circuit Output

In this part the total output of the circuit is analysed.

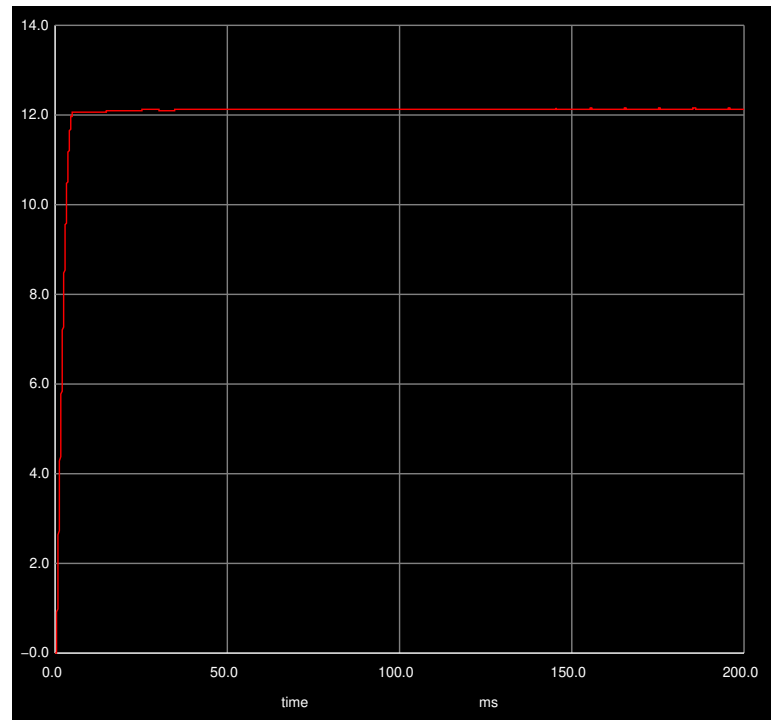


Figure 6: The final output voltage  $v_0$  during 10 periods

In the graphic above, the output voltage is represented. Looking at it is enough to understand that the final voltage is very close to the desired  $12V$ , however the difference that remains will be discussed below.

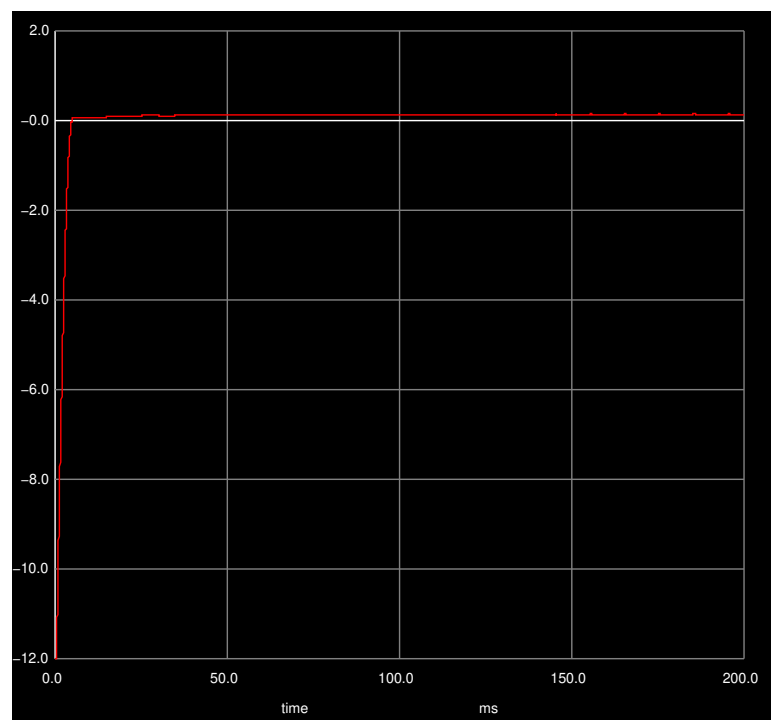


Figure 7: The final output voltage  $v_0$  subtracted by 12 during 10 periods

The graphic above essentially shows the ripple of the circuit, which is basically how much the output voltage deviates from  $12V$  and, once again, the results are pretty satisfying, with the line being very close to zero for most of the 10 periods. Still, below we can find a table with more accurate ripple data at a numerical level, which will confirm the theory that the simulation was well-succeeded.

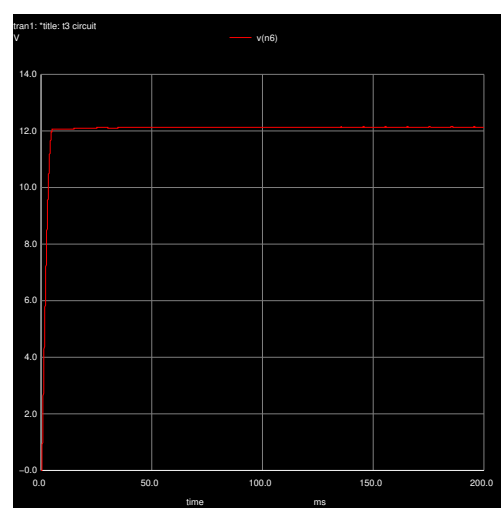
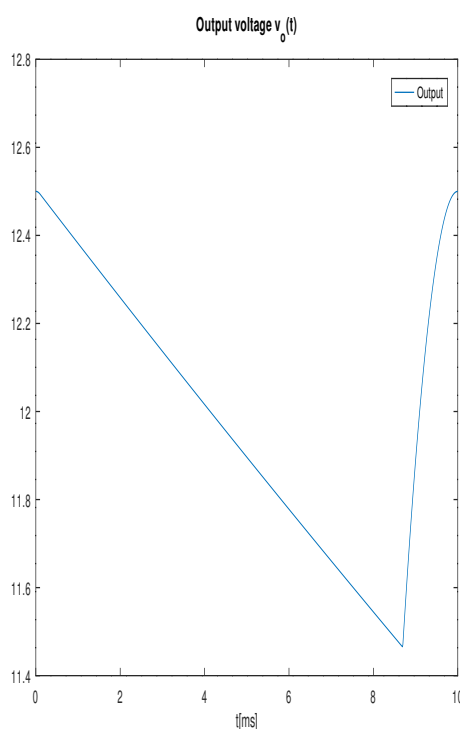
Data	Value
max	1.214333e+01
min	1.212654e+01
ripple	1.679523e-02
average	1.200008e+01

Table 1

From the table above, we can see that the average has a pretty good result being practically  $12V$ . When it comes to the ripple, it also has a very good result, with an order of significance in the hundredths.

## 5 Relative Error and Graphic Analysis

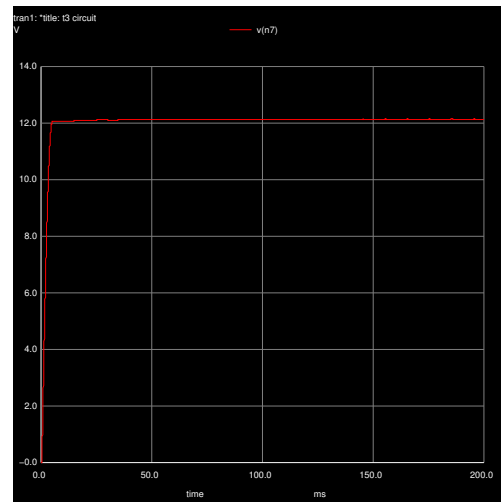
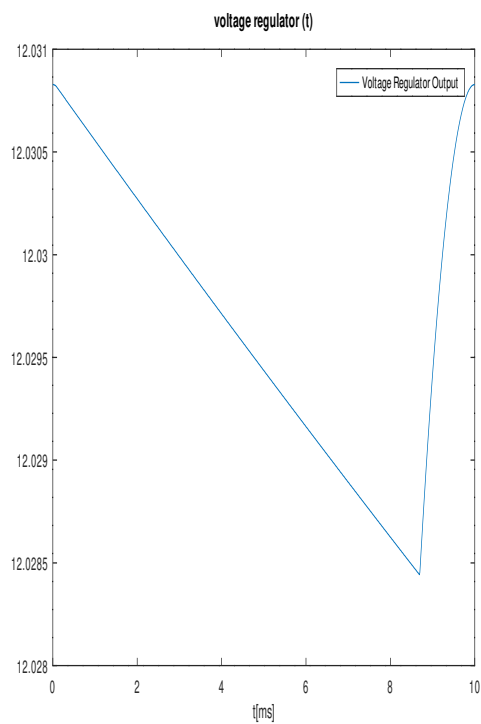
### 5.1 Topic I



In first set of graphics, we have the comparison of the envelope circuits. The graphics do match our expectations in terms of results, with both already having very close numbers to  $12V$

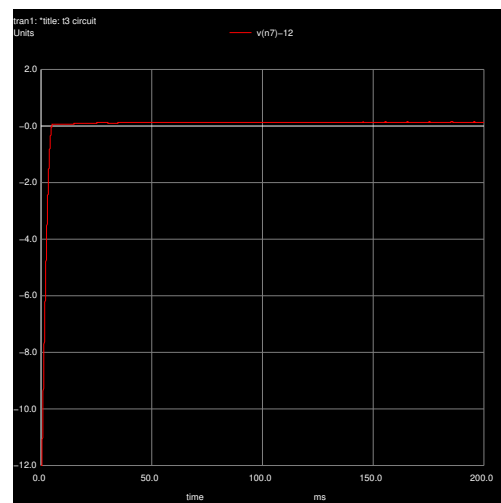
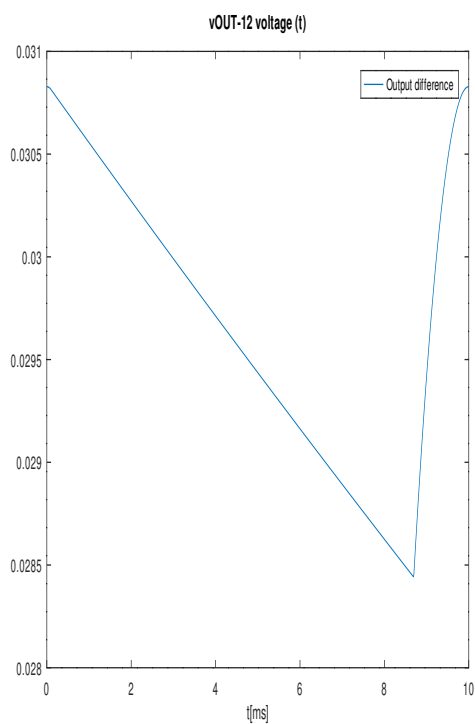


## 5.2 Topic II



In this second set of graphics, we have the comparison of the output between Octave and Ngspice. Once again, the results are very satisfying.

## 5.3 Topic III



In this last set of graphics, we have the comparison between the output difference (relative to 12V) obtained by Octave and Ngspice, which also correspond to what was required (as close to zero as possible).

#### 5.4 Topic IV

Comparison of various data between Octave and Ngspice			
DATA	Octave	Ngspice	Percentual Relative Error
Max	12.030828	12.143330	0.935111
Min	12.028443	12.126540	0.815545
Voltage Ripple	0.002386	0.016795	604.000781
Average Output Voltage	12.022394	12.000080	0.185602

In the table above, we have the comparison between Octave and Ngspice of the maximum output values, the minimum output values, the voltage ripple and the average. The percentual relative error is also included to make the interpretation of these values easier. With the exception of the Voltage Ripple, it is safe to say that all the values are very good. When it comes to the voltage ripple, while it is true that the relative error is of 5 times of difference, it is important to note that they are still very small values. The possible explanation for this discrepancy is that Ngspice uses a diode model which does not have fixed parameters (Namely  $V_D$ ), making it very hard to match the Octave results.

#### 5.5 Topic V

Figure of Merit	
Merit Figure	0.000410

To end our report, we present the figure of merit. It is important to remind that this figure is obtained upon the devices and values used in Ngspice and not the ones used in Octave. During our work, we struggled a bit trying to understand what the ideal value would be, given that we did not have any reference however, we did our best to make it as high as possible.

## 6 Conclusion

In this third laboratory assignment, all the major goals of the project were achieved. We concluded with success a further interaction with a new software (Ubuntu), with a simulation software (Ngspice), with a computational language program (GNU Octave) and with a text report editor (LaTeX). The analysis of the circuit was also finished with success through simulation and theoretical interpretation, which allowed a good comparative analysis between these two methods.