Circuit Theory and Electronics Fundamentals

Aerospace Engineering, Técnico, University of Lisbon

T3 Laboratory Report

May 04, 2021

Authors

Duarte Brito, 96373
Henrique Caraça, 96393
Nuno Ribeiro, 96459

Contents

1 Introduction

The objective of this laboratory assignment is to study a converter circuit, in this case from alternate current (AC) to direct current (DC). This circuit uses a transformer, a full wave rectifier, an envelope detector and a voltage regulator, which were chosen from the lectures. The format of the circuit can be seen in Figure 1. And the circuit chosen can be seen in Figure 2.

In Section 2, the circuit is analysed by means of a ngspice simulation. In Section 3, a theoretical analysis of the circuit is presented. The results are then compared to the theoretical results obtained in Section 3. The conclusions of this study are outlined in Section 4.

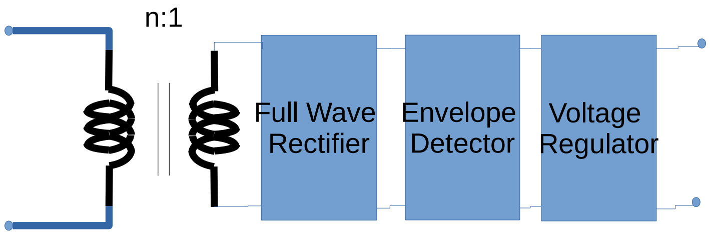


Figure 1: Assigned Circuit.

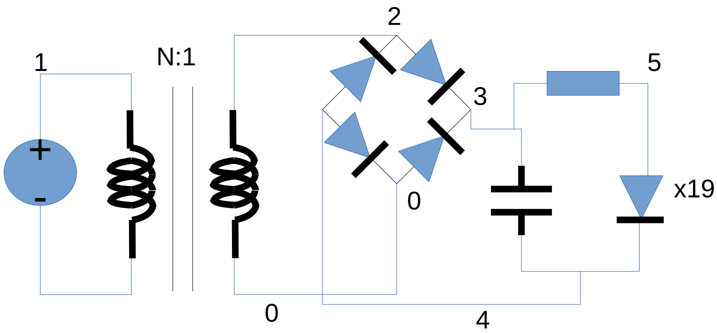


Figure 2: Chosen Circuit.

Note: the diode in the voltage regulator is merely representative, as there are in fact 19 diodes in this subcircuit

2 Simulation Analysis

2.1 Optimizer

In order to maximize the M value, we started with a simple circuit and tried to make small changes and evaluate their results. The main changes were the value of the parameters of the chosen components, their number, and their location.

We realised that the manual optimization was going to be a long and difficult process, so we decided to create a simple optimizer to do it autonomously.

It has the capacity to change the capacitance of the capacitor, the resistance of the resistor, the N value of the inductor, and the number of diodes. We found that it worked best if we fixed the values of some of these parameters. The combination that worked best for us was fixing the resistance value, the N value and the number of diodes both in the voltage regulator and in the full wave rectifier, and only optimize the the transformer and capacitor. That means that we still had to tinker with the number of diodes and the resistance by hand, but it was much less work intensive than tinkering with all the variables.

The program works by doing small variations of the values and evaluating their effect. It starts with a starting configuration. Then it chooses a parameter to variate at random and varies it a random amount. Then it checks if it is better than the current best one. The second variation is done to the best performer, the combination with the best M value. This means that the best M value keeps improving if the program finds a best combination at random.

We ended up leaving the program for a few minutes for short optimization sessions, and found that the program had a high tendency to get the average voltage on point with the 12v, as expected. It also ran much faster than we were expecting. It ended up being a win win situation in our eyes, and it can be found in our git repository. Note that upon executing the code, since the results are tied to random values, the final answer might not be exactly the same as ours. It might even be better if it is given enough time.

Component	Parameter Value
Resistor	45 kΩ
Capacitor	17.5150173805745 μ F
Number of diodes in the Voltage Regulator	19 Diodes
Number of diodes in the full wave rectifier	4 Diodes
N value of the transformer	7.29150093083668
Total cost	64.81501 MU

Table 1: Table of costs. Note that the total cost is sum of the the other values (in this units, with the number of diodes multiplied by 0.1).

2.2 Spice Results

Using the values stated above and using them in a simulated circuit in ngspice as shown in the circuit shown in Figure 2 we obtain the following results.

In the following image the blue line represents the voltage at the ends of the envelope detector, the red represents the voltage obtained by the transformer, and last, the orange represents the output voltage in the voltage detector.

Parameter	Value
mean	11.998890
Ripple	0.005566
Cost	64.815017
M	2.310727

Table 2: Spice results. The M value and the cost were calculated by an additional octave script included in the git.

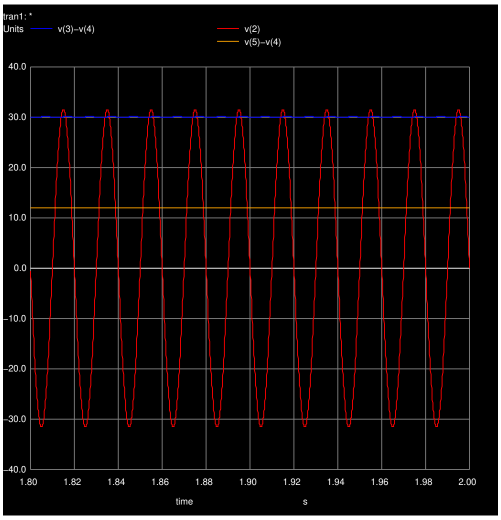


Figure 3: Detailed view

2.3 Simulation values for the theoretical analysis

In the Figure 5 we can see the results of small variations of the voltage values in the current that passes through the diodes (simulated using ngspice).

By definition, the incremental resistance in the diode can be calculated by $\frac{dv}{di} = \frac{1}{\frac{di}{dv}}$.

We then approximate the value of the derivative to its finite derivative, which is the inverse of the value of the slope in 5.

Lastly, to calculate the value of the incremental resistance of one diode, we divide the value obtained previously by the number of the diodes, leading to a final 65.7894736842105 Ω.

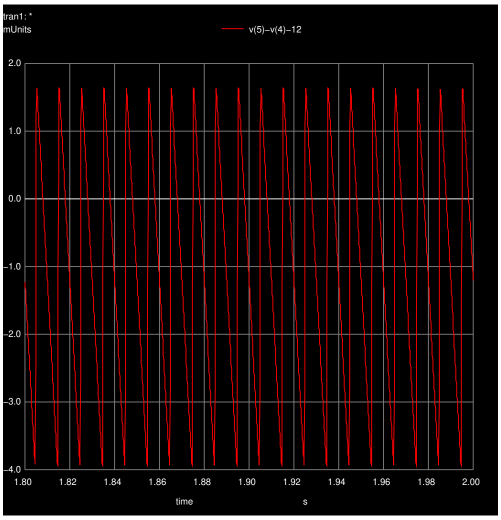


Figure 4: Detailed view

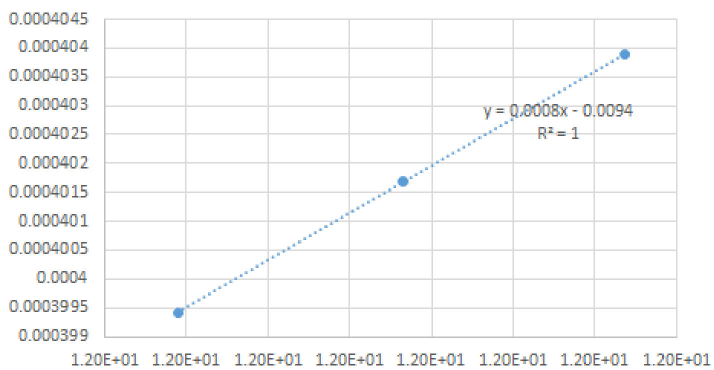


Figure 5: Current (in A) in function of the voltage (in V)

3 Theoretical Analysis

3.1 Voltage rectifier

First the current resulting from the transformer, which is considered ideal is calculated using the formula $V_2/n_2 = V_1/n_1$, thereupon, the circuit is analyzed to understand which diodes from the rectifier are on and off for each signal of current. Next, by using KVL, it is possible to define the tension on the capacitor without addressing its discharge $V_c = Acos(w.t) - (n_{diodes} \cdot V_{on})$, being A the amplitude of the voltage after being transformed in the inductor.

3.2 Envelope detector

Capacitor discharge: By using KCL, it is possible to write that $i_d = i_c + i_r$ (i_r being the current if the equivalent resistor of voltage regulator). Once the capacitor is charged and because there is fluctuation due to the sinusoidal nature of the voltage coming from the rectifier, there will be a point where $i_r = -i_c$. At this point, the diodo will turn off and the capacitor will begin to discharge. This point is called t_{off} and it can be calculated like

$$t_{off} = (1/w) \cdot atan(1/w/(CR_{v_{reg}})).$$

The function that defines the discharging is calculated using the formula obtained in the lectures:

$$V_{exp} = (A - 2.V_{on}).cos(w \cdot t_{off})exp(-(t - t_{off})/(CR_{v_{reg}}))$$

3.3 Voltage at the Capacitor

The voltage at the capacitor will be the maximum between the sinusoidal function coming from the voltage rectifier and the function of the discharge.

We can decompose this voltage, v_C, in its DC component, V_c, and its AC component, v_c (V_c is the mean of the function and v_c is the function minus V_c).

3.4 Voltage Regulator

Output Voltage: The output voltage, v_O, will be the voltage measured in the nodes of the first and last diodes. This voltage can be decomposed in two components:

$$v_O = V_o + v_o$$

V_o will be equal to number of diodes, n_d times V_{on}.

To get v_o, it is used the incremental model of the diode, where it can be replaced by a resistor r_d. After using voltage dividir, one can get the following:

$$v_o = \frac{n_d \cdot r_d}{n_d \cdot r_d + R} \cdot v_s$$

After doing the sum of V_o and v_o, the output voltage, v_O, has been calculated.

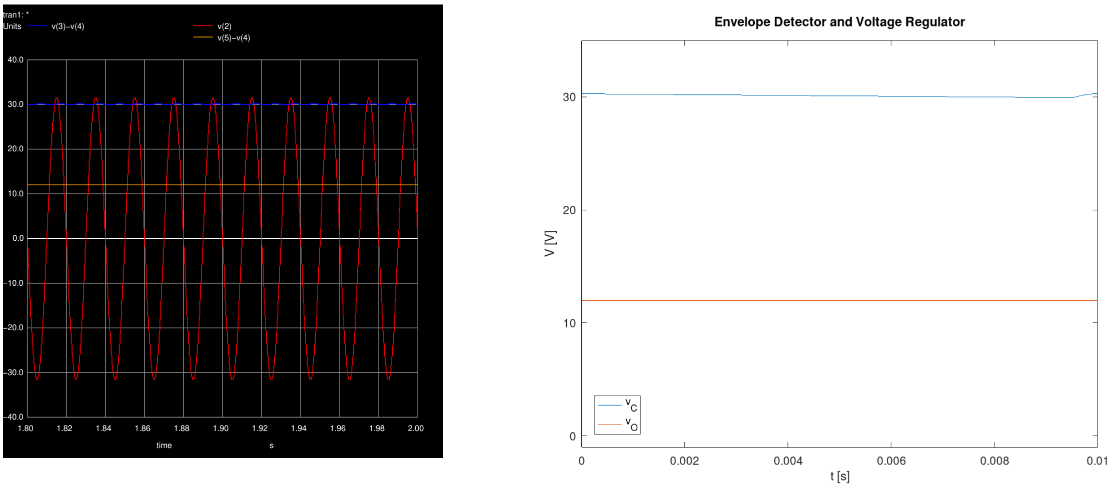
3.5 Results and Comparisons

The results obtained in the simulation and theoretical analysis were similar (as it can be seen in the graphics bellow):

- The values of the voltage drop of the envelope detector are close
- The ripple's order of magnitude is the same

- Thes values of M are similar

Note that the DC component of the voltage regulator in the theoretical analysis is equal to the one obtained in the simulation because the value of V_{on} used in the theoretical analysis was taken from NGSpice.



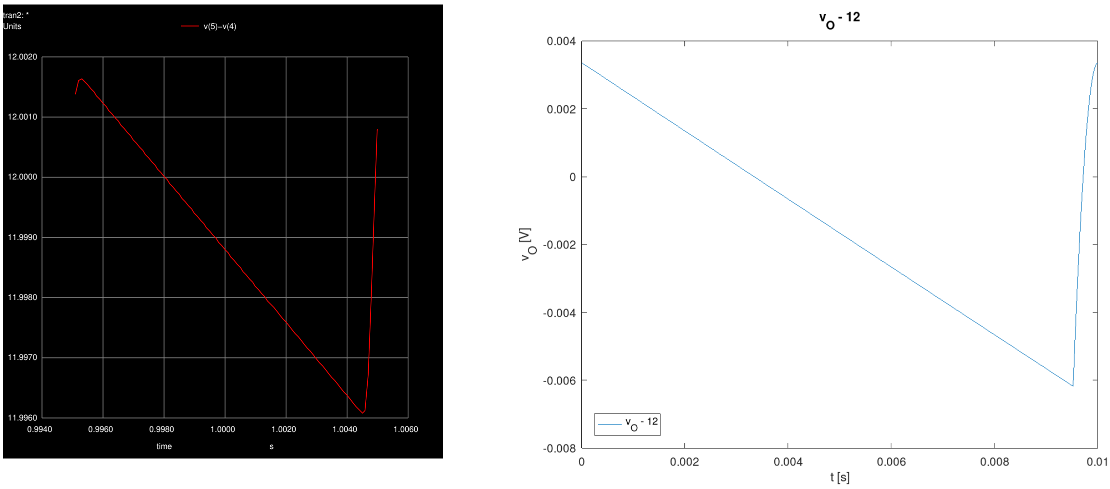
(a) Simulation Results

(b) Theoretical Results

Figure 7: Envelope Detector and Voltage Regulator

Parameter	Value
mean	11.998890
Ripple	0.005566
Cost	64.815017
M	2.310727

Table 3: Spice results. The M value and the cost were calculated by an additional octave script included in the git.



(a) Simulation Results

(b) Theoretical Results

Figure 9: Deviation from the objective of 12 V

Parameter	Value
Output DC level	11.998660 V
Ripple	0.009528 V
Cost	64.815017 MU
M	1.884083

Table 4: Spice results. The M value and the cost were calculated by an additional octave script included in the git.

4 Conclusion

Unlike previous laboratories, this time, the results were not equal.

However, we believe that the differences are not that significant and they can be explained by how NGSpice solves the circuit compared to how it was done in the theoretical analysis. To solve the circuit, NGSpice used far more advanced simulation methods for the diodes, with many more parameters, while we used an approximated model with V_{on} and an incremental resistor.

This way, the objective should have never been to have equal results, but rather, have results that are ”close enough”, which we believe it was achieved.