Laboratory Report 3: AC/DC Converter

Circuit Theory and Electronics Fundamentals

Department of Electrical and Computer Engineering, Técnico, University of Lisbon

April 8, 2021

Work by:

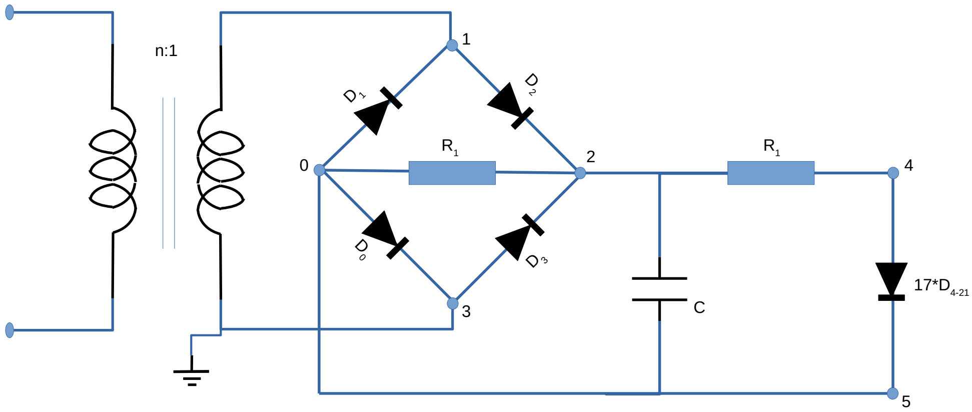
Beatriz Contente 95772

Francisco Fonseca 95789

Manuel Carvalho 95823

1 Introduction

The objective of this laboratory assignment is to transform an AC input voltage of 230V to a DC output voltage of 12V. To attain this goal, we used a circuit with three main parts: a transformer, an envelope detector (composed by a full-wave bridge rectifier with 4 diodes, 1 resistor and 1 capacitor) and a voltage regulator (containing 1 resistor and an undefined number of diodes that was computed by an algorithm which we had developed). The referred circuit is shown in the picture below.



As mentioned above, it is also important to refer that we have developed an optimization algorithm (in Octave) in order to find the number of diodes, the values of the resistors and capacitor and the consequent "n" (relation established by the number of coils in each side of the transformer) that would lead to the best value of merit, computed in Ngspice with the formula given by the Professor.

2 Theoretical Analysis

In this section we will discuss the theoretical analysis of our circuit. For this purpose, we will first explain seperately the envelope detector and the voltage regulator circuits on the AC/DC converter. The values used throughout this analysis are shown below. V_ON is achieved by using the Ngspice value for V_out. Theoretically, V_ON is given by equation **??**.

$$V_{ON} = \frac{V_{out}}{N_{diodes}}, \tag{1}$$

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

Symbol	**Value**
V_{ON}	988745938
A_f	$14.64891221288435V$
R_{ef}	$26k\Omega$
R_{rf}	$10k\Omega$
C	$3.3750e-05F$
n[1]	20
η	1

Table 1: Values for theoretical analysis

2.1 Envelope Detector

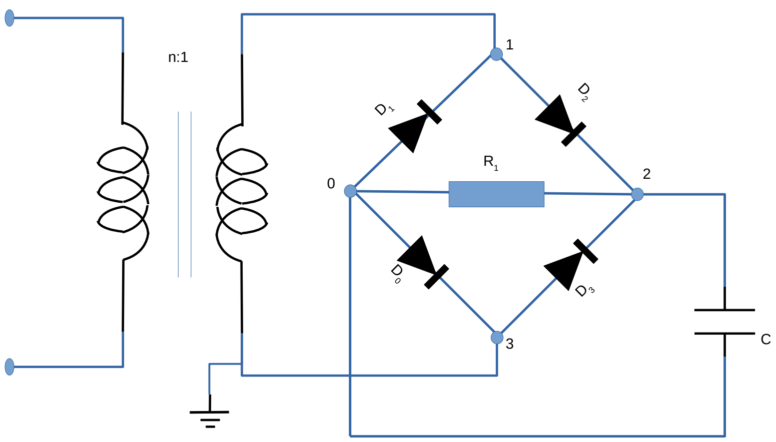


Figure 2: Envelope Detector Circuit.

As seen in figure **??**, the envelope detector part of the circuit consists on a full wave bridge rectifier, a resistor (R_ef) and a capacitator (C), the last two in paralell. To reach the mininum ripple possible, we can increase the resistor impedance and the capacitor capacity, so that the time constant τ of the capacitor discharge is high enough ($\tau = RC$). Another method was to use a full wave rectifier, as said above, since the time of discharge of C is half of what it would

[1]This refers to the n constant in the transformer.

be using a half wave rectifier. This part of the circuit was used to envelope the V_{in} sinusoidal wave, being V_{in} achieved from the following equation.

The envelope detector can be on a positive cycle where D_0 and D_2 let current pass or a negative cycle that corresponds to D_1 and D_3 letting the current pass.

$$V_{in} = \frac{230V}{n}, \tag{2}$$

Regardless of the cicle the envelope detector is on, using KVL we achieve the following equation.

$$V_o + 2V_{ON} - V_{in} = 0, \tag{3}$$

When the current on the resistor is equal to the current on the capacitor, the diodes shut off, leading to the following equation.

$$\frac{Acos(\omega t|_{OFF}) - 2V_{ON}}{R_{eq}} = A\omega sin(\omega t|_{OFF}) \tag{4}$$

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

$$V_o = V_o|_{t_{OFF}} e^{-\frac{t-t_{OFF}}{R_{eq}C}} \tag{5}$$

2.2 Voltage Regulator

This part of this circuit was implemented so as to limit the voltage value and to soften the riple of the voltage that came from the previous part of the circuit (**??**). It involves a resistor and seventeen diodes in series.

The analysis has to be made by seperating the DC and AC components. The DC simulation uses the condition $V_o = N * V_{ON}$, so that we can replace each diode with V_{ON}.

In this section, the theoretical analysis requires we think about the DC and AC components separately ($V_o = V_O + v_o$).

The DC analysis is fairly simple since if $V_O > 17 * V_{ON}$, V_O is equal to $17 * V_{ON}$. The number 17 can be replaced by any number of diodes chosen.

As for AC, we achieve equation **??** by using equations **??** and **??**. The variable k is the Boltzmann constant, T is the temperature in Kelvin and q is electron charge. To lower the v_{out} ripple, the value for r_d should also be low (see equation **??**).

$$v_{out} = \frac{nr_d}{nr_d + R_{vreg}} v_o, \tag{6}$$

$$r_d = \frac{\eta V_T}{I_s e^{\frac{V_D}{\eta V_T}}}, \tag{7}$$

$$V_T = \frac{kT}{q}, \tag{8}$$

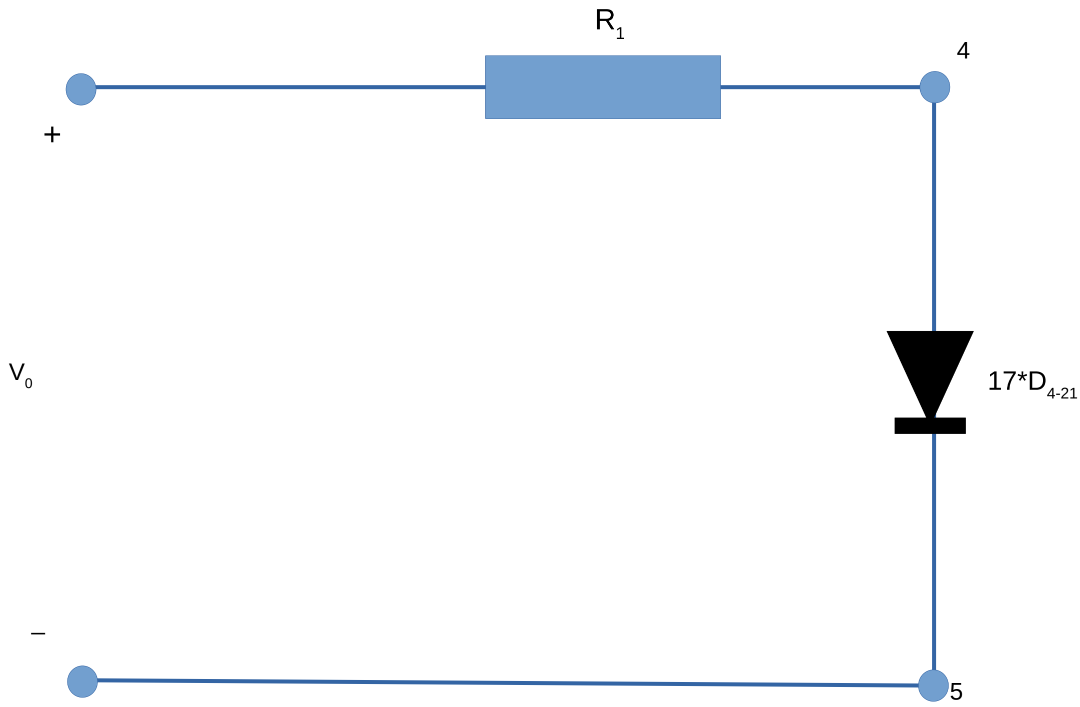


Figure 3: Voltage Regulator Circuit.

3 Simulation Analysis and Results Comparison

In this section, we will the obtained results by simulating the referred circuit in Ngspice.

In order to analyze the circuit and achieve the main goal of this laboratory assignment, we have developed an optimization ocatve algorithm that would give us the values which would leat to the greater merit value. The obtianed values were the following ones, presented in table **??**:

Then, we can automatically compute the cost: $Cost =$!!!!!!!!.

The plots shown below are related to the variables V_{input}, $V_{envelope}$ and V_{output} in function of time. The values for the ripple and deviation are in table **??**.

Figure 4: Envelope detector voltage

Element	Value
n (Transformer)	!!!!!!!
Number of diodes	!!!!!!!
C F	!!!!!!!
Renvelope	!!!!!!!
Rregulator	!!!!!!!

Table 2: Obtained values by optimization ocatve script

Figure 5: Output voltage - AC + DC component

Figure 6: Output voltage ripple

Figure 7: Input, Envelope and Output voltages

Element	**Value**
Average Output Voltage [V]	!!!!!!!
Output Voltage Ripple [V]	!!!!!!!
Merit	!!!!!!!

Table 3: Simulation Values for Average, Ripple and Merit.

4 Conclusion

By analysing the circuit theoretically and then simulating the circuit using Ngspice, we can verify that the values of the unknown components match almost perfectly and all approaches agree on the final currents' directions across the circuit's branches (which can be seen below in figure **??**).

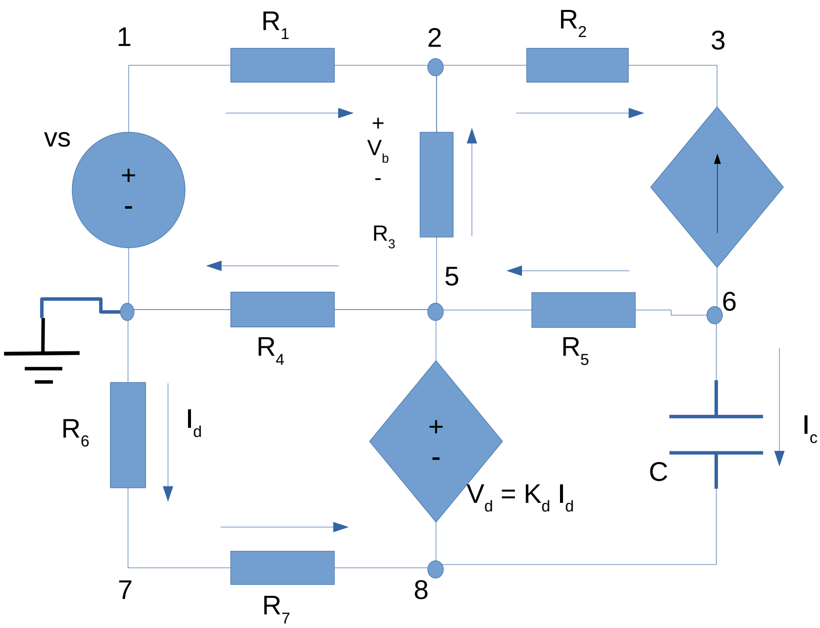


Figure 8: Representation of the final circuit with all the correct directions for the currents.

To better understand the discrepancies and compare results we present the several tables obtained side by side on octave and ngspice, respectively, in tables **??** to **??**.

Some small discrepancies are due to the different number of decimal places considered by Octave and Ngspice leading to slight inaccuracies. However, considering that the circuit complexity is still not considered, the differences are negligible. Furthermore, any differences in the order of 10^{-15} (or lower), are very likely related to the way the computer programs deal with mathematical operations (seen that 10^{-15} is extremely close to the precision of a double's mantissa). Note that the format of the data presented in the Ngspice tables are automatically chosen by the program.

All of this leads to the conclusion that the making of this laboratory assignment was coherent and that the main goal was attained: to achieve the circuit analysis through a theoretical and simulated approach.