Circuit Theory and Electronics Fundamentals

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

April 5, 2021

Laboratory Assignment - T3

Group nº59

José Miguel Goulão - 95814
Lourenço Pacheco - 95817
André Gomes - 96352

Contents

1 Introduction

The objective of this laboratory assignment is to optimize and study an AC/DC converter circuit. We were given total freedom to choose the architecture of the Envelope Detector and Voltage Regulator circuits. Our goal is to achieve the highest Merit (M) possibe. This value is obtained with the following equations:

$$M = \frac{1}{cost \times (Ripple(vout) + avg(vout - 12) + 10^{-6})}$$

$$cost = cost_{resistors} + cost_{capacitor} + cost_{diodes}$$

$$cost_{resistors} = 1MU/kOhm; cost_{capacitors} = 1MU/\mu F; cost_{diodes} = 0.1MU/diode$$

For reasons explained later, our circuit, shown in figure 1, contains, in total:

- one voltage source (V_1)
- two inductors (L_1,L_2)
- one resistor (R_1)
- one capacitor (C_1)
- twentyt two diodes (D_1-D_{22})

The circuit can be divided in three parts. The source, the Envelope Detector Circuit and the Voltage Regulator Circuit (Figure 1).

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

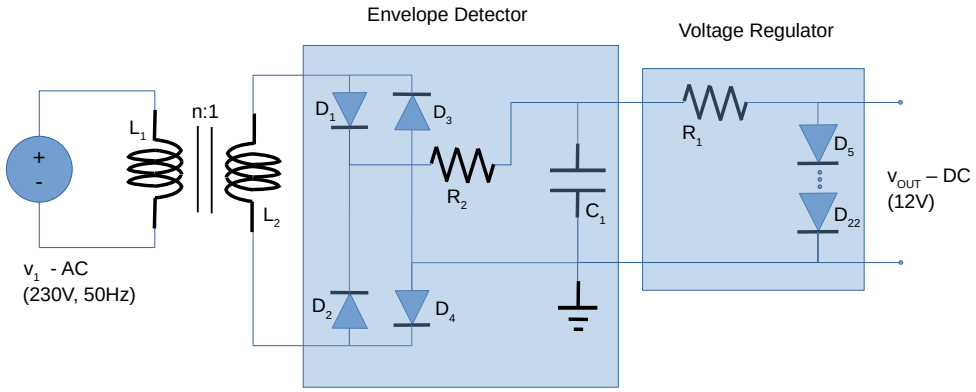

Figure 1: Circuit T3

2 Theoretical Analysis

In this section, the circuit in Figure 1 is analysed theoretically.

To begin, the optimization of the values for each component is presented. Afterwards, the theoretical analysis with those results is shown, with all the plots and tables.

2.1 Optimization

As a consequence of not being given any values, the group had to optimize the values for each component in order to obtain tha highest Merit possible. Thus, an Octave script was written to give us the optimal values.

2.2 Theoretical Results

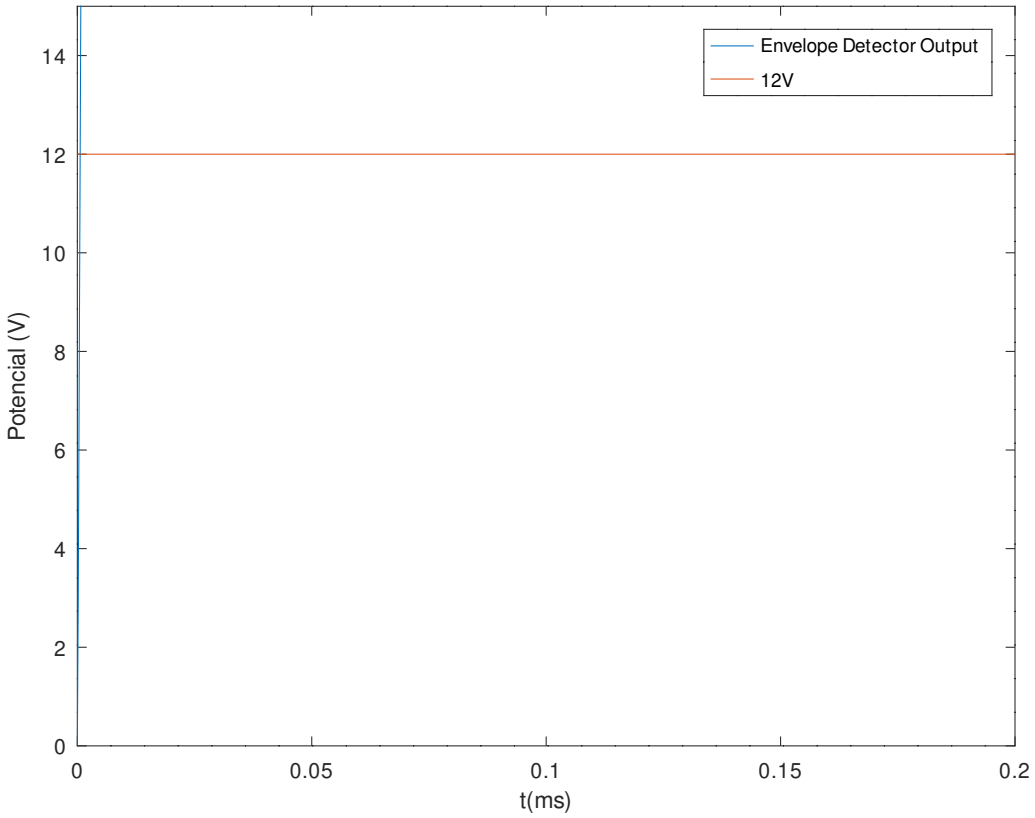

Figure 2: Envelope Detector Circuit v_{out}

3 Simulation Analysis

In this section, Circuit T3 is reproduced with the help of Ngspice.

Ngspice is a simulator for eletronic circuits that can output a variety of results. This emulator computes the voltages in every node, as well as the potential difference between two given nodes. Apart from that, the group made use of the command *.options savecurrents* which also enables the output of the currents that pass through all branches. Moreover, function to help determine de minimum, maximum and average of the plots were also used.

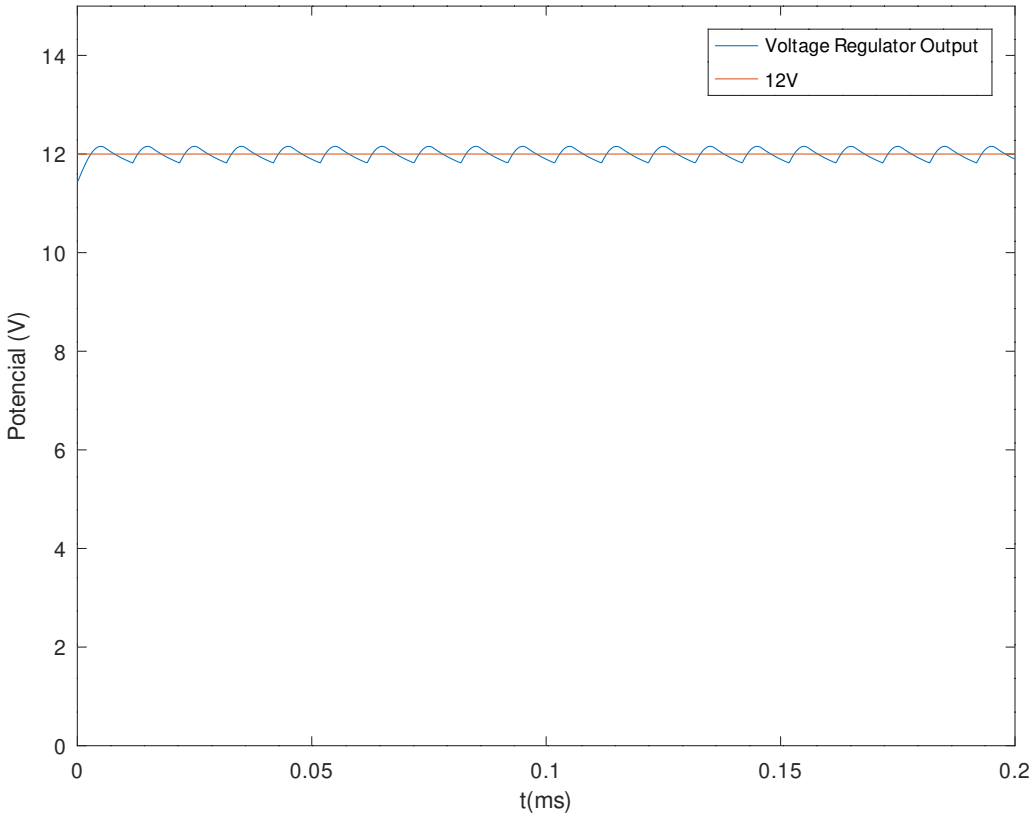

Figure 3: Voltage Regulator Circuit v_{out}

Firstly, the outcome of the simulation is shown, as well as a brief explanation on how it was achived. Afterwards, a comparison is done between those values and the ones attained in Section 2.

3.1 Simulated results

In this laboratory assignament, the Ngspice script made use of the sames values considered for the Octave script.

Firstly, a transient analysis between $t = 0ms$ and $t = 150ms$ was performed. This operation only had the goal to provide us a plot (Figure 4) for the v_{out} in the beginning compare it to the desired outup voltage $(12V)$.

Aditionally, a transient analysis between $t = 150ms$ and $t = 350ms$ was carried out. Three plots were obtained. Figure 5 shows the output voltage of the Envelope Detector Circuit. Figure 6 shows the output voltage of the Voltage Regulator Circuit. Finaly, 7 shows the output voltage of the Voltage Regulator Circuit minus $12V$ $(v_{out} - 12)$.

As mentioned before, function to determine the minimum, maximum and average of the plots were also used. Table 1 shows the average, maximum and minimum of the plot in Figure 6, in that order. In adition, the Ripple (Vout(MAX) - Vout(MIN)) is also presented.

Lastly, the group also used Ngspice to compute the Merit. This was done, only to confirm the value obtained in Subsection 2.2.

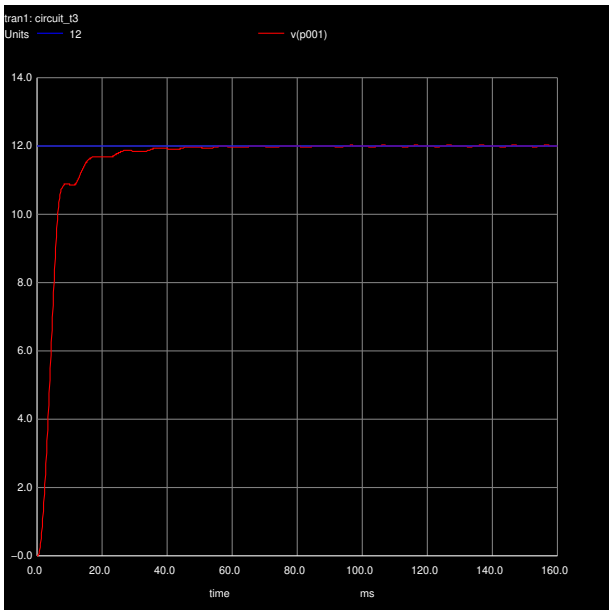

Figure 4: Inicial v_{out}

Name	Value [V]
vout(avg)	1.200000e+01
vout(max)	1.201992e+01
vout(min)	1.197959e+01
ripple(vout)	4.033000e-02

Table 1: Values from Ngspice related to the Voltage Regulator Circuit.

3.2 Comparison

4 Conclusion

For this laboratory assignment, we were given a circuit composed by resistors, dependent and independent current and voltage sources and had the objective of analyzing and simulating it and then compare the results obtained.

Static analyses were performed theoretically, through mesh and node analysis and by circuit simulation, using the Octave math tool and Ngspice tool, respectively. The simulation results matched the theoretical results very precisely. Therefore, we can conclude that our objective was achieved successfully.

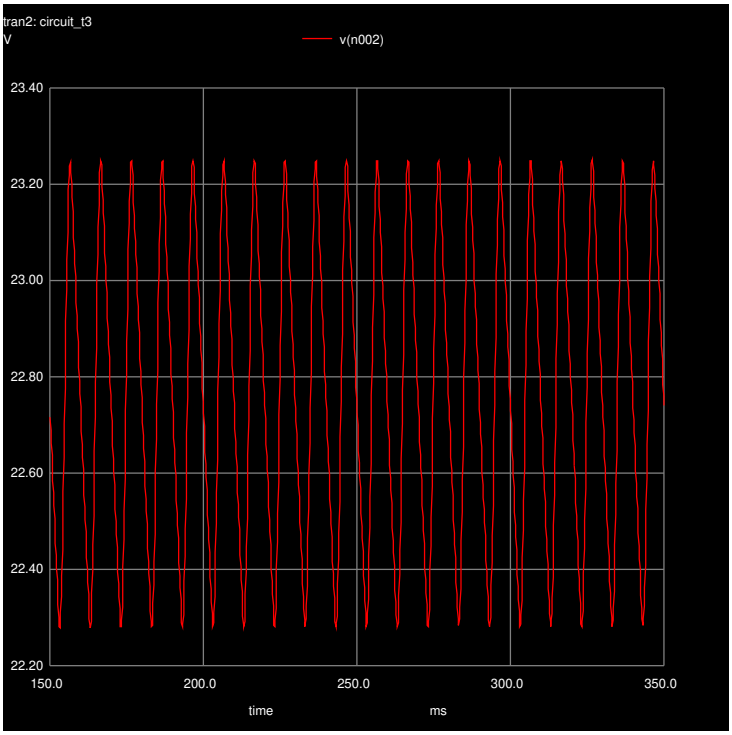

Figure 5: Envelope Detector Circuit v_{out}

Name	**Value [MU]**
cost(resistor)	7.884500e+00
cost(capacitor)	1.000000e+01
cost(diode)	2.100000e+00
cost	1.998450e+01
———————	———————
merit	1.240703e+00

Table 2: Simulated values for the Costs and Merit.

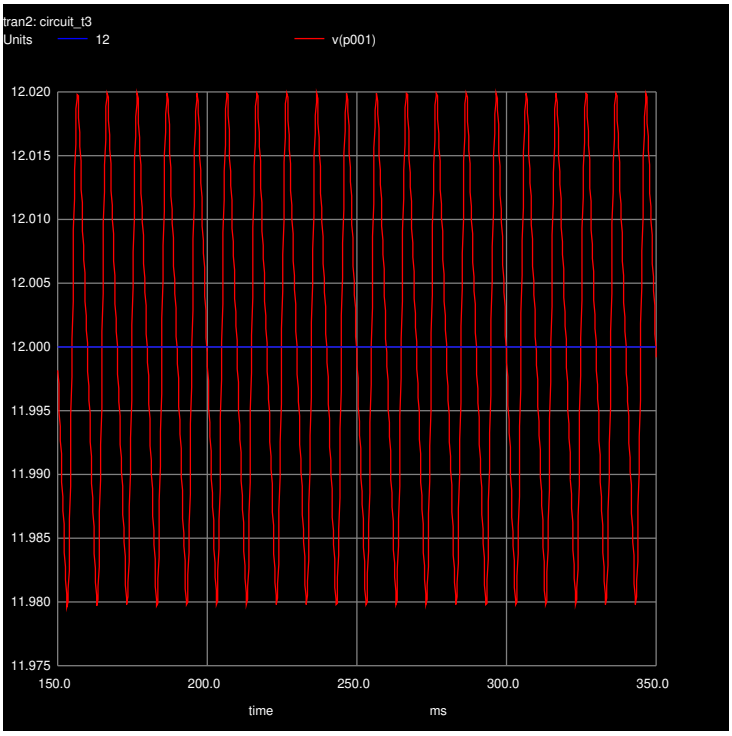

Figure 6: Voltage Regulator Circuit v_{out}

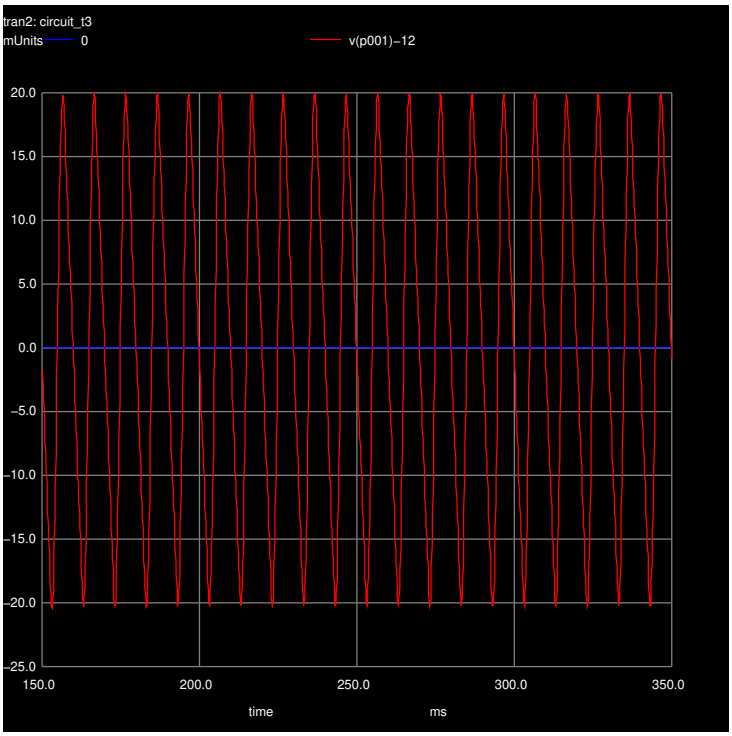

Figure 7: Output AC component + DC deviation
Vout(MAX) - Vout(MIN)