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

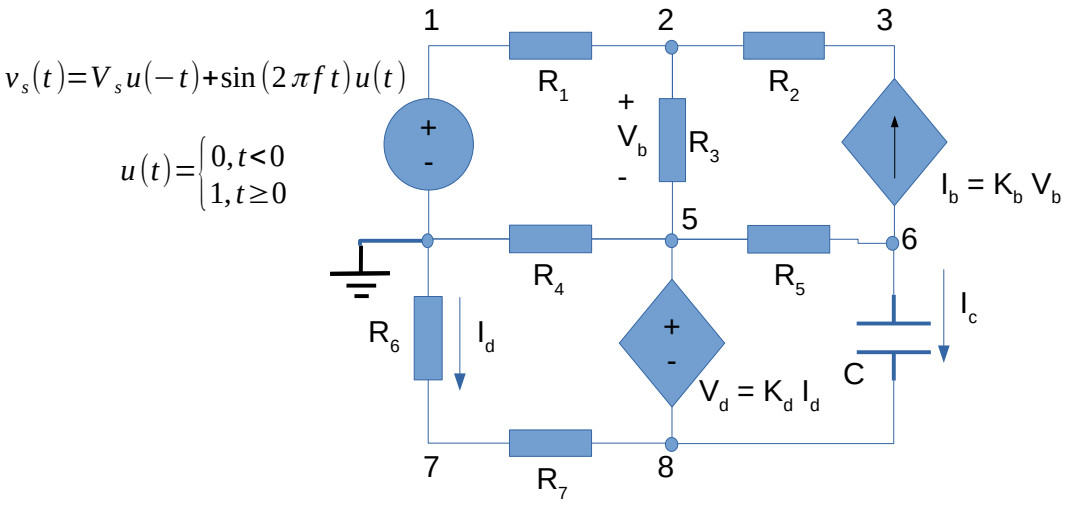
Contents

1 Introduction

The objective of this laboratory assignment is to analyze a RC circuit to find the natural and forced response as well as doing a frequency analysis. Furthermore, it is asked to run a simulation using NgSpice to detect small diferences between the different approaches and understand why said differences happen. The circuit can be seen in Figure **??**.

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

Figure 1: RC Circuit with alternate voltage source (V_s), linear dependent sources (V_d-linear current controlled voltage source and I_b-linear voltage controlled current source) and capacitor C



The values given for this report can be found in table **??**(Obtained with the number).

Name	Values

Table 1: Values obtained by using the Python program that can be found in folder $python$.

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 **??**.

$$V_{ON} =$$

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 **??**:

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

Table 2: Obtained values by optimization ocatve script

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 2: Envelope detector voltage

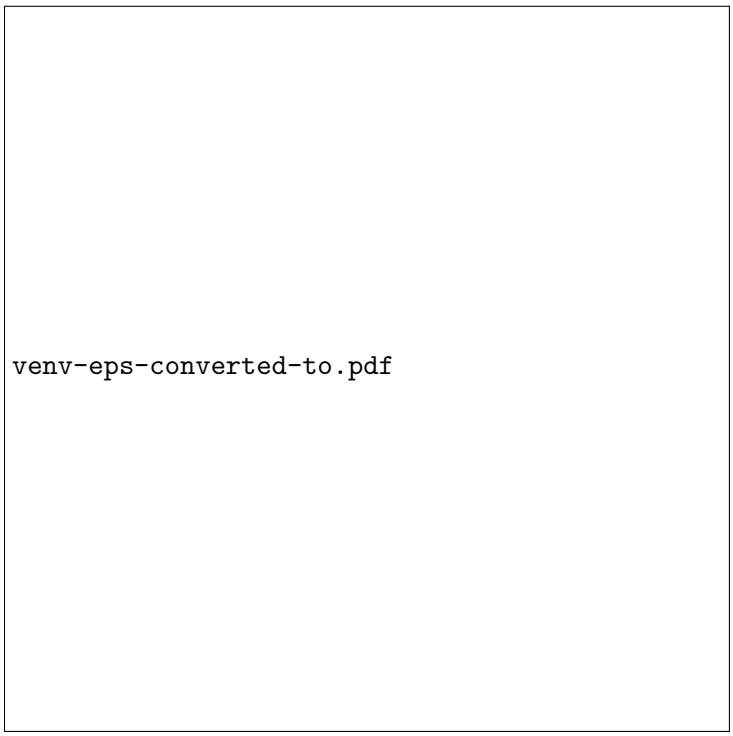


Figure 3: Output voltage - AC + DC component

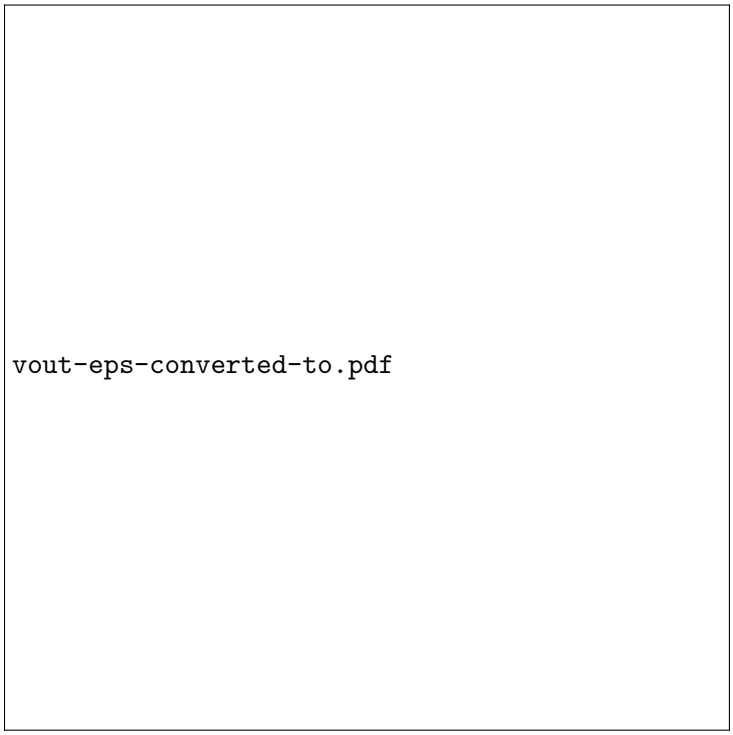


Figure 4: Output voltage ripple

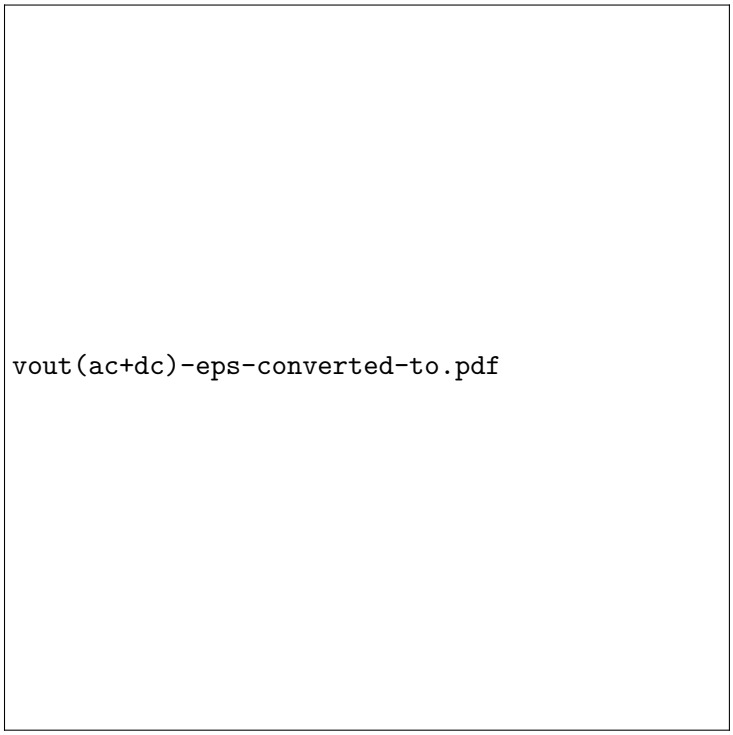
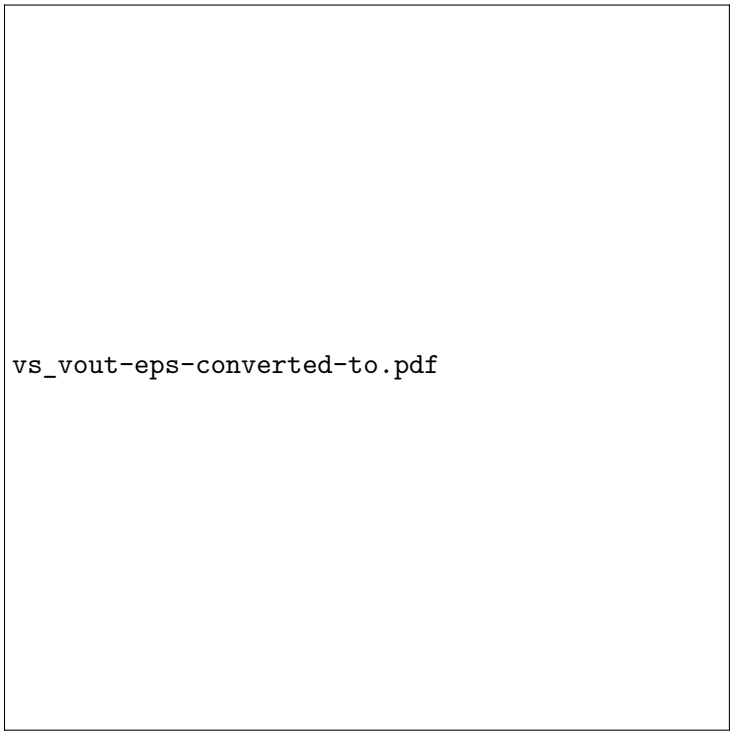


Figure 5: Input, Envelope and Output voltages



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 **??**).

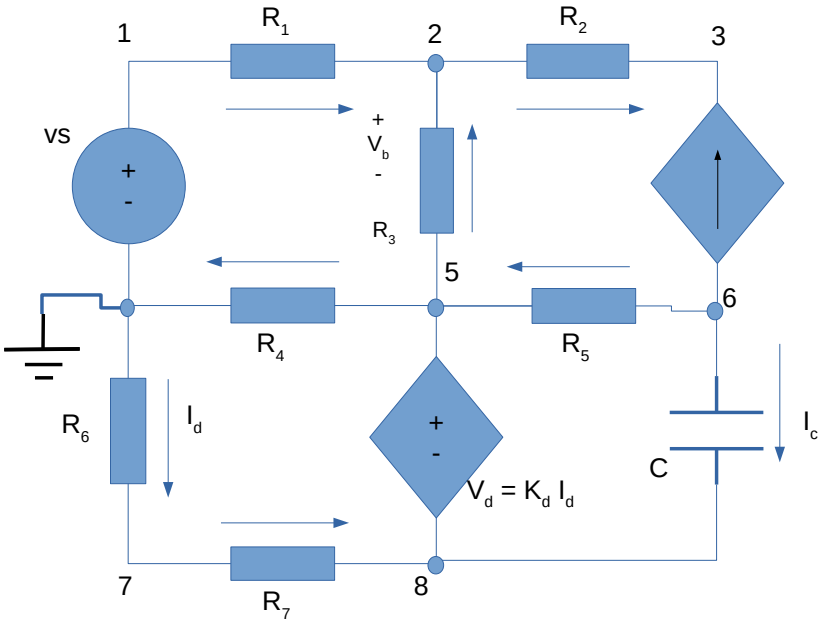


Figure 6: Representation of the final circuit with all the correct directions for the currents.

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.