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

1 Introduction **2**

2 Theoretical Introduction **2**

3 Simulation Analysis **3**

3.1 Output Voltage Gain 4

3.2 Bandwidth and Cut-Off Frequencies 4

3.3 Input and Output Impedances 5

4 Relative Error and Graphic Analysis **5**

5 Conclusion **5**

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

The implementation of the AC-DC converter circuit was made according to the theoretical or simulation principles we were based on, with the construction of two simillar circuits with just a few differences in the components value (such as the resistance of some resistors and the capacitance of the capacitors) and also in the components number (such as the number of diodes). These small adjustments were made in order to compensate the changes in the behaviour of the electric circuits components verified while implementing both the theoretical circuit in GNU Octave and the simulation circuit in NGSpice.

In Section 2, a theoretical introduction is made in order to contextualize all the main principles that sustain our construction and analysis of the circuit. A theoretical AC-DC coverter is built and carefully analysed in Section **??**, where the results are obtained in GNU Octave. Also, in Section 3, another AC-DC converter is constructed and analysed by simulation through the use of NGSpice to simulate the real electric circuit behaviour. The results of the simulation of Section 3 are then compared to the theoretical results obtained in Section **??** and the comparative results are expressed in Section 4. The figure of merit, calculated according to the components used to build the simulation circuit, can also be found in Section 4. The conclusions of this study are outlined in the final part of the report, in Section 5.

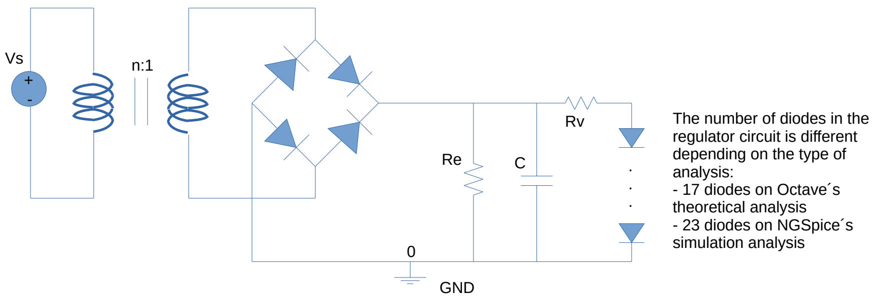

Figure 1: Third laboratory circuit.

2 Theoretical Introduction

Electric power is transported on wires either as a direct current (DC) flowing in one direction at a non-oscillating constant voltage, or as an alternating current (AC) flowing backwards and forwards due to an oscillating voltage. AC is the dominant method of transporting power because it offers several advantages over DC, including lower distribution costs and simple way of converting between voltage levels thanks to the invention of the transformer. AC power that is sent at high voltage over long distances and then converted down to a lower voltage is a more efficient and safer source of power in homes.

AC to DC converters are one of the most important tools in power electronics because a lot of real applications are based on this type of conversions. The AC current to DC current conversion process is known as rectification. This rectifier converts the AC supply into the DC supply at the load end connection. Normally, transformers are used to adjust the AC source to get the step down transformer to reduce the voltage amplitude, so that there is a better operation range for the DC supply.

The output voltage of the full-wave rectifier is not constant, it is always oscillating and thus can't be used in real-life applications. That's why it is required a DC supply with a constant output voltage. This need can be fulfilled by using an adequate filter with an inductor or a capacitor (envelope detector circuit) to make the output voltage smooth and constant.

This capacitor is connected in parallel to the load resistance in a linear power supply. The capacitor is used to increase the DC voltage and to reduce the ripple voltage of the output obtained. This capacitor is also called a reservoir or smoothing capacitor and it is generally followed by a voltage regulator which eliminates the remaining ripples so that the required output can be achieved.

While the rectifier conducts and the potential is higher than the charge across the capacitor, the capacitor stores the energy from the transformer. However, when the output of the rectifier falls below the charge across the capacitor, the capacitor naturally discharges its energy into the circuit. As the rectifier conducts current only in the forward direction, all the energy discharged by the capacitor will flow into the load. There are other types of filters, such as the half wave rectifier, studied in the theoretical classes. However, the efficiency of the full-wave rectifier is double that of a half-wave and the ripple voltage is lower using this four diodes bridge rectifier.

Finnaly, a voltage regulator is a circuit that creates and maintains a fixed output voltage, regardless of changes to the input voltage or load conditions. A simple voltage regulator can be made from a resistor in series with a diode or a series of diodes. The voltage regulators keep the voltages from a power supply within a range that is compatible with the other electrical components.

3 Simulation Analysis

In this section, we can find the results of the topics 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 three different subsections.

3.1 Output Voltage Gain

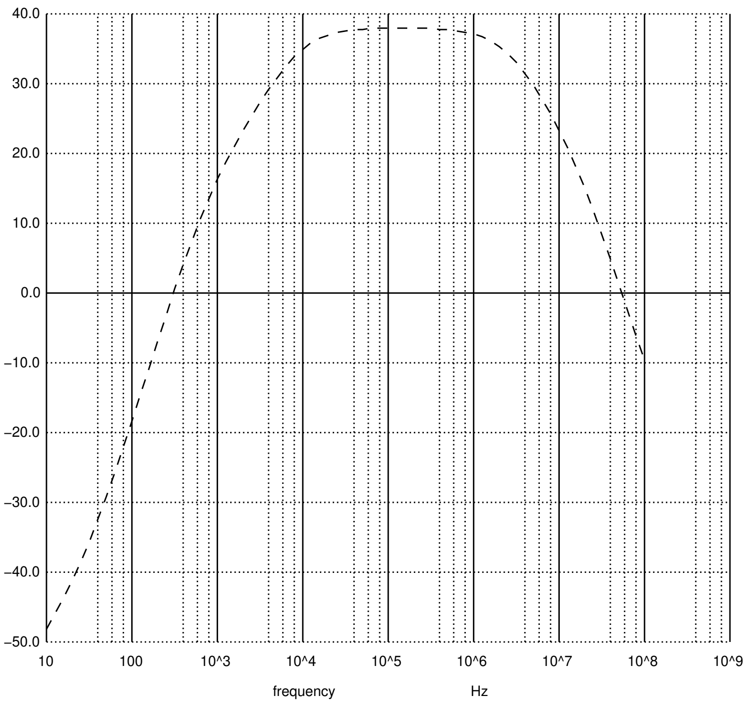

Figure 2: The Output Voltage of the Output Stage of a Common Collector Amplifier.

where f is expressed in Hertz (Hz) along the x-axis
and $v_{dB_{out}}$, the output voltage of the amplifier, is expressed in in Volts (V) using a decibel scale along y-axis.

NGSpice Formula	Output Voltage Gain (V)
vdb(out)[40]	3.793151e+01

3.2 Bandwidth and Cut-Off Frequencies

Variable	Frequency Value (Hz)
f1	1.004524e+04
f2	2.189576e+06
f2-f1	2.179531e+06

where f_1 is the lower cut-off frequency,
af_1 is the upper cut-off frequency and
$f_2 - f_1$ is the bandwith (the difference between the upper and lower cut-off frequencies).

3.3 Input and Output Impedances

NGSpice Formula	**Input Impedance Value (Ω): a, b from $Z_{in} = a + bj$**
v(in2)[40]/(v(in)[40]-v(in2)[40])*100/1000	6.077977e-01,-1.08849e-01

NGSpice Formula	**Output Impedance Value (Ω)**
1/i(vin)	-6.93273e+00

4 Relative Error and Graphic Analysis

5 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 construction and analysis of the circuit was also finished with success through simulation and theoretical interpretation, which allowed a good comparative analysis between the differences presented in the behaviour of a simulated and theoretical electric circuit.

In fact, this assignment allowed us to build two AC-DC converters which were able to transform the current and achieve an output voltage as close to $12v$ as possible. Even though we needed to construct two different electric circuits according to the simulation or theoretical principles we were based on, both of them were succesfuly implemented and achieved the pretended output voltage. The simulation results about the output voltage matched the theoretical results precisely. This accuracy was confirmed by the matemathical calculation of relative errors, which were proved to be really small. Also, the comparative analysis of graphics ploted by both theoretical and simulation tools confirmed the similarity of the results, obtained successfully and with notorious precision.