

## **Circuit Theory and Electronics Fundamentals**

### **Lab 4 - Audio Amplifier**

### **Aerospace Engineering**

Laboratory Report

May 23, 2021

Eva Claro, 95785

Miguel Isidoro, 95834

Pedro Braz, 95837

## **Contents**

<b>1</b>	<b>Introduction</b>	<b>3</b>
<b>2</b>	<b>Simulation Analysis</b>	<b>4</b>
<b>3</b>	<b>Conclusion</b>	<b>6</b>

# 1 Introduction

This report is being made for the subject of Circuit Theory and Electronics Fundamentals and is related to the forth laboratory being its objective to develop an audio amplifier circuit (made of Bipolar Junction Transistors) by choosing the architecture of the Gain and Output amplifier stages. The circuit is shown in ??.

In Section ?? a theoretical analysis will be made and it can be decomposed in two stages: the gain stage where the objective is to have the maximum gain possible and the output stage whose objective is to lower the impedance of the amplifier. Secondly, in Section 2 it will be simulated the circuit using ngspice tools. Following with both results from Section ?? and Section 2 being compared and commented in Section ??.

Also, it is important to notice that were used two types of BJTs transistors: BC557A (PNP type) and BC547A (NPN type).

Finally, the conclusions of this study are outlined in Section 3.

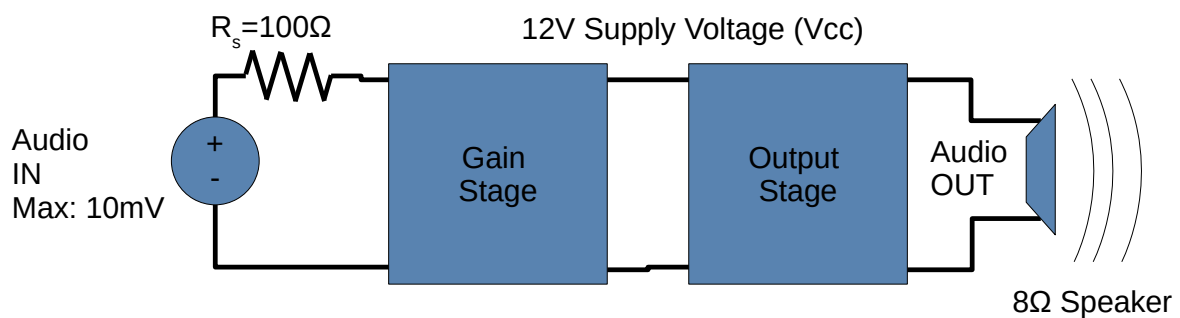


Figure 1: AC/DC converter circuit

## 2 Simulation Analysis

This section covers the audio amplifier circuit simulation using the Ngspice tool.

As asked in the lab assignment, a NPN transistor and a PNP transistor were used in gain stage and output stage respectively. The goal was to calculate the impedances ( $Z_I$  and  $Z_O$ ), the cut off frequencies (), the bandwidth (the difference between the cut off frequencies) and the total gain.

It was also confirmed if the BJTs are on the F.A.R. (forward active region) by comparing  $V_{CE}$  and  $V_{BE}$  for NPN type and  $V_{EC}$  and  $V_{EB}$  for PNP type.

Later in this report, we will compare this results with the theoretical ones but for now we will just show them.

The Table 1 and 2 shows the BJTs voltages and their F.A.R. confirmation.

Name	Value [A or V]
V(CE)	2.78156
V(BE)	0.70931
V(CE) > V(BE)	Yes

Table 1: F.A.R. confirmation - BC547A (NPN type)

Name	Value [A or V]
V(EC)	4.49605
V(EB)	0.817257
V(EC) > V(EB)	Yes

Table 2: F.A.R. confirmation - BC557A (PNP type)

In the next table it is presented the results asked.

Name	Value [A or V]
VGain	37.9181
Bandwidth	1.55393E+06
COFreq	8793.49

Table 3: Ngspice simulation results

The merit obtained by the groups is presentend in the following table.

Name	Value [A or V]
Cost	123.208
merit	54.3849

Table 4: Cost and merit results

Name	Value [A or V]
Zin	-548.062 + 82.7641 j

Table 5: Merit values

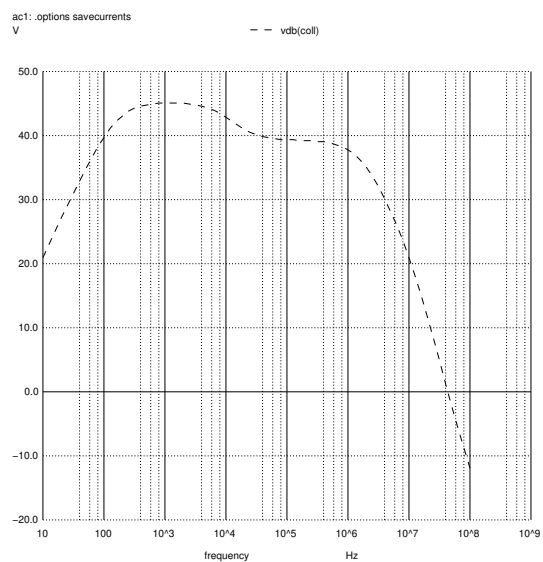


Figure 2: Output Voltage of the envelope detector  $v(4)$

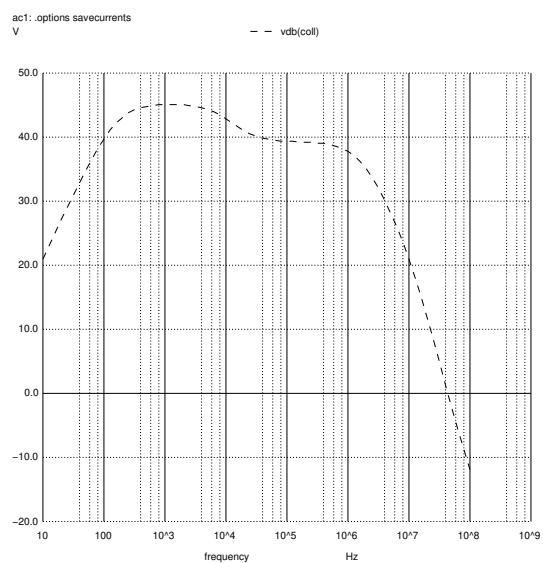


Figure 3: Input Voltage of the secondary circuit  $(v(2)-v(3))$

### 3 Conclusion

The objective of this laboratory assignment was to develop an audio amplifier circuit and the main goal was achieved. However by observing analysis and simulation results it can be seen a difference between the two. This is the result of using a non-linear circuit whereas the model used by NGSpice is far more complex than the theoretical model used. Regarding this one and despite the differences, the theoretical model gives good results and can be used when there is no simulation tools to use or to quickly confirmed the simulation results obtained.

The objective of this laboratory assignment was to develop an audio amplifier circuit and the main goal was achieved. However it was achieved not having the best merit. The merit of the circuit was obtained by trial and error, a method that is not perfect and does not result in the best possible results. In this way, we concluded that in order to obtain good results, we were obliged to "yield" part of the merit.

We also note that this time, the results were not equal and exactly the same comparing both NGSpice and Octave.

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, processes that were also explained on our lectures. 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.

The error obtained between the average theoretical value and average simulated value is 4.86% which wouldn't be significant in a real life scenario but for a online simulation is a bit significant.

This way, the objective should have never been to have equal results, but rather, have results that seemed reasonable, which we believe it was achieved. The merit obtained was 1.538149e-01.