

## Instituto Superior Técnico, University of Lisbon

### Integrated Master in Aerospace Engineering

Circuit Theory and Electronics Fundamentals

Joana Matos (95799), Ricardo Abreu (95842), Vasco Emídio (95856)

Fourth Laboratory Report

May 12, 2021

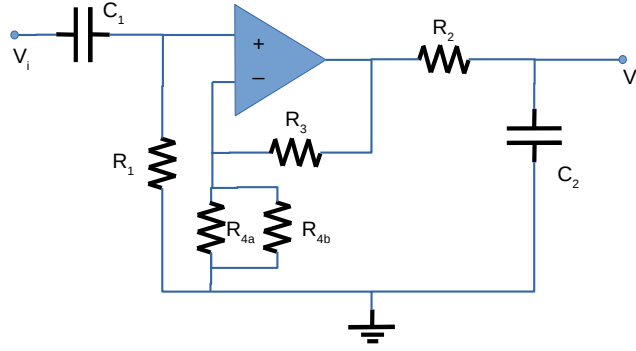
## Contents

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>Theoretical Analysis</b>	<b>2</b>
2.1	Gain, Input and Output Impedances at central frequency . . . . .	2
2.2	Frequency response $V_o(f)/V_i(f)$ . . . . .	3
<b>3</b>	<b>Simulation Analysis</b>	<b>4</b>
3.1	Frequency Analysis . . . . .	4
3.1.1	Magnitude Response . . . . .	4
3.1.2	Phase Response . . . . .	5
3.2	Simulation Results . . . . .	6
<b>4</b>	<b>Conclusion</b>	<b>6</b>

## 1 Introduction

The objective of this laboratory assignment is to simulate a Band Pass Filter using OPAMP as shown in Figure 1.

This way, we should choose the architecture of the Gain and Output amplifier stages, however, we must consider the cost of the components in the circuit. Its diagram is shown in Figure 1.



**Figure 1:** Band Pass Filter Circuit

This circuit is composed by 5 resistors, 2 capacitors and an OPAMP. The values of the components are exhibited in the table below.

Name	Value
$R_1$	9.090000e+02 Ohm
$R_2$	1.000000e+03 Ohm
$R_3$	1.000000e+05 Ohm
$R_{4a}$	1.000000e+03 Ohm
$R_{4b}$	1.000000e+04 Ohm
$C_1$	2.200000e-07 F
$C_2$	1.803279e-07 F

**Table 1:** Components Values

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

## 2 Theoretical Analysis

In this section, the circuit shown in Figure 1 is analysed theoretically. It is important to notice that the purpose is to build a band pass filter circuit using an OPAMP. This way, the considered circuit has a high pass filter, a signal amplifier (OPAMP) and a low pass filter in series, where the capacitor  $C_1$  and the resistor  $R_1$  act as a high pass filter, while capacitor  $C_2$  and resistor  $R_2$  function as a low pass filter.

Thus, We will begin by computing the gain, input and output impedances at the central frequency. Then, we will compute the frequency response  $V_o(f)/V_i(f)$ , using the incremental circuit, solving the circuit for a frequency vector in log scale with 10 points per decade, from 10Hz to 100MHz.

### 2.1 Gain, Input and Output Impedances at central frequency

In this subsection, we will compute the gain, input and output impedances at the central frequency.

First, we will calculate low and high cutoff frequencies,  $w_L$  and  $w_H$ , respectively, as well as central frequency,  $w_O$ , which are given by:

$$w_L = \frac{1}{R_1 \cdot C_1} \quad (1)$$

$$w_H = \frac{1}{R_2 \cdot C_2} \quad (2)$$

$$w_O = \frac{1}{\sqrt{w_L \cdot w_H}} \quad (3)$$

Their values are presented in the table below:

Name	Value
$w_L$	5.000500e+03 rad/s
$w_H$	5.545455e+03 rad/s
$w_O$	5.265933e+03 rad/s
$f_O$	8.380992e+02 Hz

**Table 2:** Frequencies

Now we can determine the gain at central frequency, which is given by the following equation:

$$T_{w_O} = \frac{R_1 \cdot C_1 \cdot jw_O}{1 + R_1 \cdot C_1 \cdot jw_O} \cdot \left(1 + \frac{R_3}{R_4}\right) \cdot \frac{1}{1 + R_2 \cdot C_2 \cdot jw_O} \quad (4)$$

, where  $R_4$  is the  $R_{4a}$  and  $R_{4b}$  equivalent resistor.

As well as  $Z_{in}$  and  $Z_{out}$ , at the central frequency, are determined by:

$$Z_{in} = \frac{R_1 \cdot C_1 \cdot jw_O}{1 + R_1 \cdot C_1 \cdot jw_O} \cdot \left(1 + \frac{R_3}{R_4}\right) \cdot \frac{1}{1 + R_2 \cdot C_2 \cdot jw_O} \quad (5)$$

$$Z_{out} = R_1 + \frac{1}{jw_O \cdot C_1} \quad (6)$$

The results are shown in table 3:

Name	Value
$Gain$	3.532348e+01 dB
$Z_{in}$	9.090000e+02 + j-8.631813e+02 Ohm
$Z_{out}$	5.258371e+02 + j-4.993320e+02 Ohm

**Table 3:** Results at central frequency

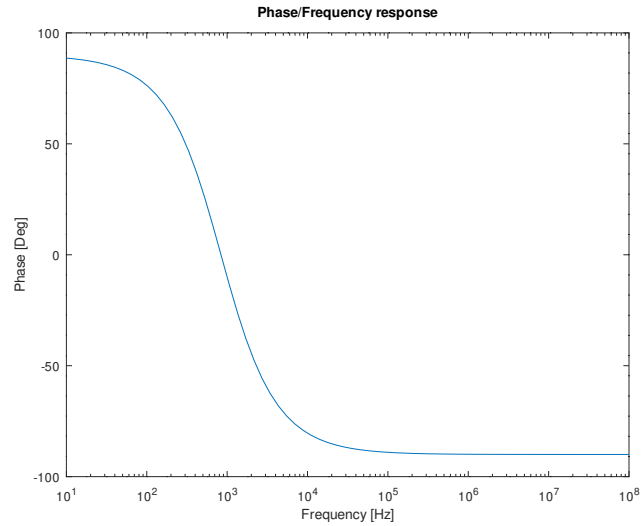
## 2.2 Frequency response $V_o(f)/V_i(f)$

Now, considering the transfer function  $T(s)$ , it is given by:

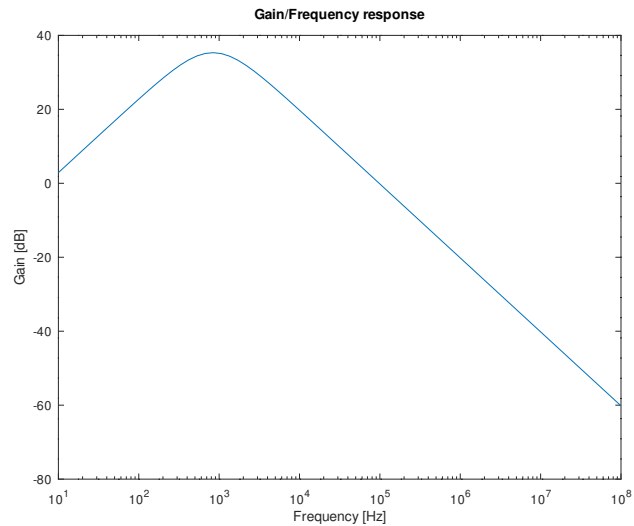
$$T(s) = \frac{V_{out}(s)}{V_{in}(s)} = 1 + \frac{Z_{in}(s)}{Z_{out}(s)} = \frac{R_1 \cdot C_1 \cdot s}{1 + R_1 \cdot C_1 \cdot s} \cdot \left(1 + \frac{R_3}{R_4}\right) \cdot \frac{1}{1 + R_2 \cdot C_2 \cdot s} \quad (7)$$

, where  $\frac{R_1 \cdot C_1 \cdot s}{1 + R_1 \cdot C_1 \cdot s}$ ,  $1 + \frac{R_3}{R_4}$  and  $\frac{1}{1 + R_2 \cdot C_2 \cdot s}$  represent the gain at high pass filter, OP-AMP and low pass filter, respectively.

This way, the gain frequency response is plotted in the figure below.



**Figure 2: Phase Frequency Response**



**Figure 3: Gain Frequency Response**

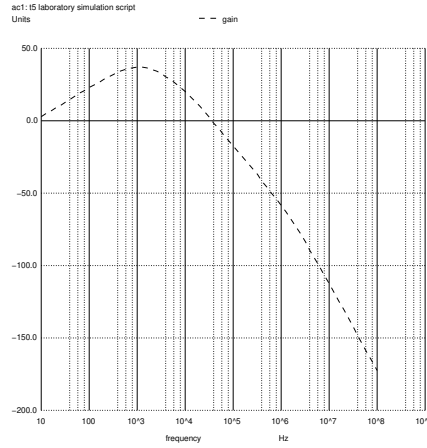
As we can see there is a maximum for the frequencies in the central frequency, which is near to 1KHz, otherwise low and high frequencies have a low gain. That was expected because both low and high frequencies were blocked by high and low pass filters.

### 3 Simulation Analysis

#### 3.1 Frequency Analysis

##### 3.1.1 Magnitude Response

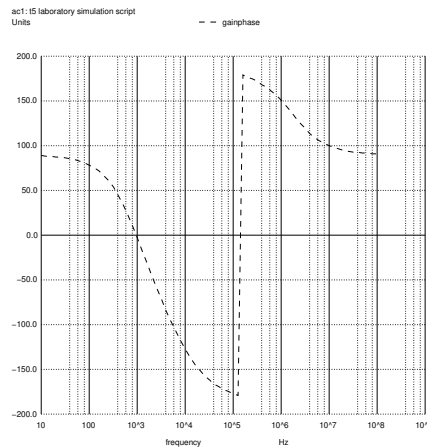
Figure 4 shows the frequency magnitude response of the circuit, for the voltage gain, which is expressed in dB. Comparing to the theoretical graphic, we notice that the curves are very similar. We only notice the differences by taking a closer look to both curves, and see that the maximum voltage gain is not the same for the theoretical and simulated results. Both results approach very well the reference value, 40dB, and the differences between these two results might rely on the fact that NGSpice uses a real OPAMP model.



**Figure 4:** Magnitude response for the voltage gain  $V_{out}/V_{in}$ , in dB.

### 3.1.2 Phase Response

Figure 5 shows the frequency phase response of the circuit, for the voltage gain, which is expressed in degrees. Comparing to the theoretical graphic, we notice that the curves are very similar. We only notice the differences by taking a closer look to both curves, and see that the maximum voltage gain is not the same for the theoretical and simulated results. Both results approach very well the reference value, 40dB, and the differences between these two results might rely on the fact that NGSpice uses a real OPAMP model.



**Figure 5:** Phase response for the voltage gain  $V_{out}/V_{in}$ , in degrees.

### 3.2 Simulation Results

Table 4 shows the simulated results, using NGSPICE, for the input and output impedances, at the central frequency. Remind that the values on the right are the theoretical ones obtained by Octave, which are reproduced in this section to help compare them to the simulated results. Comparing both the values, we conclude that the results are very close, some differences resulting on the fact that, on NGSpice, we used an approximate value of the central frequency to calculate the impedances and because in this simulating tool, it considers the components real (the OPAMP, most importantly).

Input impedance	Ohms	Output impedance	Ohms
zin	1.161859e+03	zout	8.251525e+02

**Table 4:** Input and Output impedances

Finally, Table 5 shows the values obtained for the gain and central frequency of the circuit under analysis. The table shows also the merit result and the values used for the calculation of the merit figure, which are the cost, the central frequency deviation (which should be ideally 1kHz) and the linearly voltage gain deviation (which should be ideally 100).

Name	Values
Gain(dB)	37.0419
Gain	71.1371
Central Frequency(Hz)	1051.2
Gain deviation	28.8629
Central frequency deviation(Hz)	51.2041
Cost(MU)	13446.7
Merit	9.28821E-07

**Table 5:** Values for the calculation of the Merit.

As one can notice, we achieved a very low merit value. However, if we analyse the restrictions imposed on the uses of certain components and the results for the gain and the central frequency, which are quite close to the reference values, we conclude that, despite the low merit value, we achieved a very efficient circuit whose quality would degrade significantly if we changed randomly the values of the components to achieve the highest merit figure possibly.

## 4 Conclusion

In this laboratory assignment the objective of building and analysing an Audio Amplifier Circuit, made of a gain stage and an output stage, has been achieved. The theoretical analysis was performed with the help of the Octave math tool and the circuit simulation using the Ngspice tool. For both analysis, we determined the gain and input and output voltages of the circuit, as well as, lower cut-off frequency. We also plotted the frequency response  $V_o(f)/V_i(f)$ . At the end, we calculate the merit of our work.

This way, in theoretical analysis, we explained why the two stages could be connected without significant signal loss. And, in simulation analysis, we also explained the purpose of the coupling capacitors and their effect on the bandwidth, the purpose of the bypass capacitor and its effect on the gain and the effect of resistor  $R_C$  on the gain.

As previously mentioned, the simulation results had slight differences from the theoretical ones. However, we designed an acceptable Audio Amplifier Circuit - furthermore, we obtained a decent figure of merit, despite the cost being too high.