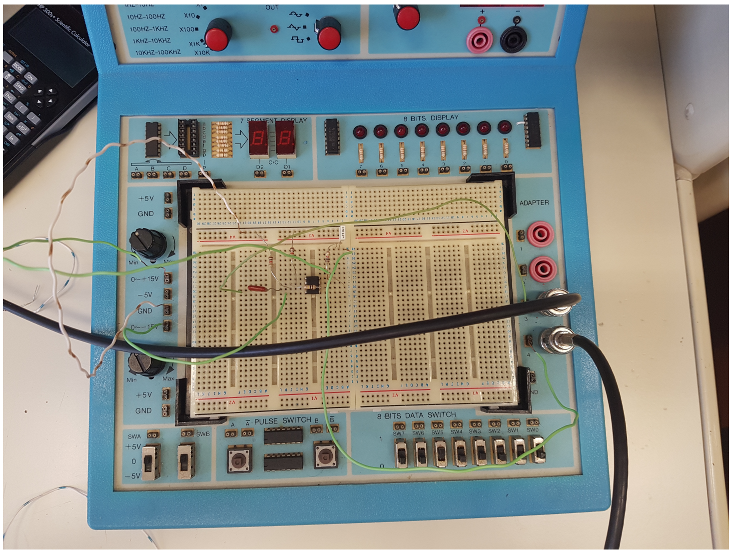
Circuit Theory and Electronics Fundamentals

Aerospace Engineering, Técnico, University of Lisbon

T5 Laboratory Report

Instituto Superior Técnico, Universidade de Lisboa

June, 2021



Authors
Duarte Brito, 96373
Henrique Caraça, 96393
Nuno Ribeiro, 96459

Contents

1 Introduction

The objective of this laboratory assignment is to make a Bandpass filter using an OP-AMP.
The equivalent circuit can be seen in Figure 1.

This circuit is made up of an amplifier (an OP-AMP and resistors R_3 and R_4), a high pass filter (the left part, that is C_1 and R_1), and a low pass filter (the right part, that is R_2 and C_2) .

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

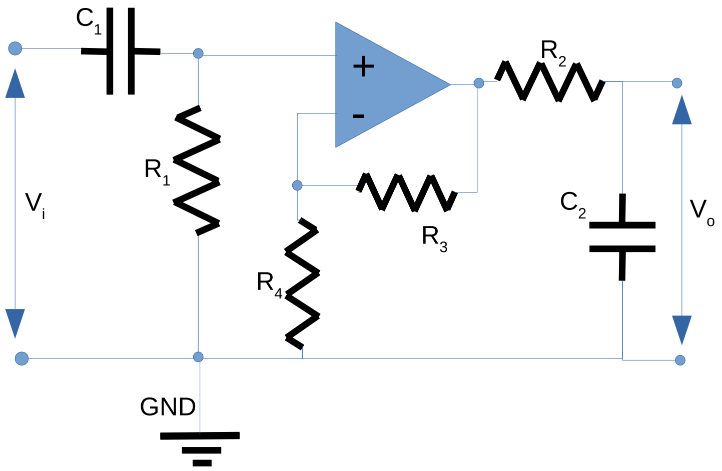


Figure 1: Used Circuit.

2 Simulation Analysis

2.1 Incremental Modifications

In order to maximize the M value, the provided circuit was used, and small changes were made to its parameters in order to evaluate their results. Since the number and type of components available were limited, not only were the best parameters evaluated but it was also looked into the best way of making the respective component from coupling other components in various ways.

In order to maximize the merit value, we started with the cheapest combination of C's and R's (C = 0.22 μF and R = 1 kΩ). Then we calculated the corresponding frequency with the formulas 1, 2 and 3 discussed in the theoretical analysis. We noted that it was lower than the target central frequency. That meant that we needed the appropriate higher frequency to keep the 1kHz central frequency (formula 3). Using once again formula 1 and keeping the same resistance we found that the C2 value we were looking for was 0.110 μF, that could be achieved by combining 2 0.220 μF capacitors in series.

For R3 and R4, we took a more trial and error focused approach. We fixed the value of R4 to $1k\Omega$ as it was the lowest value, and then played with the value of R3, knowing that a higher resistance means a higher gain, to make the gain as close to 40 dB while also trying to maximize the merit.

2.1.1 Cost

Component	Parameter Value
Op Amp	13323 Mu
C_1	0.220 μF
C_2	0.110 μF (2x 0.220)
R_1	1 kΩ
R_2	1 kΩ
R_3	110 kΩ
R_4	1 kΩ
Total cost	13436.66 MU

Table 1: Table of costs. Note that the total cost is sum of the the other values.

2.1.2 Results

Parameter	Value
Voltage Gain	37.344430
lowerCutoffFreq	406.289000
higherCutoffFreq	2463.671000
f0	1000.481098
input impedence	999.000600 + i-7.319790 ohm
output impedence	0.199367 + i-14.466600 ohm
Cost	13437.052039
M	0.000040

Table 2: Spice Results

3 Theoretical Analysis

3.1 Results at Central Frequency

Because this a band pass filter there is a lower cut off frequency, f_L and a high cut off frequency, f_H. With these frequencies, we can define ω_L and ω_H.
The former can be calculated like so,

$$\omega_L = \frac{1}{R_1 C_1} \tag{1}$$

and the latter can be calculated like so,

$$\omega_H = \frac{1}{R_2 C_2} \tag{2}$$

The central frequency can then be obtained through the following formula:

$$\omega_0 = \sqrt{\omega_L \omega_H} \tag{3}$$

Theoretical results	value (Hz)
f_L	723.431560
f_H	1446.863119
f_0	1023.086723

Simulation results	value (Hz)
f_L	406.289000
f_H	2463.671000
f_0	1000.481098

Despite nott being able to predict accurately the lower and higher cut off frequency, the central frequency can be predicted quite well.

To compute gain of the circuit at the central frequency, the circuit was analysed in its three parts, the high pass filter, the amplification stage and the low pass filter, resulting in the respective formulae:

$$gain_{HPF} = \frac{j\omega_0 R_1 C_1}{1 + j\omega_0 R_1 C_1}; \tag{4}$$

$$gain_{Amplifier} = 1 + R_3/R_4; \tag{5}$$

$$gain_{LPF} = \frac{1}{1 + j\omega_0 R_2 C_2}. \tag{6}$$

The gain can then be obtained by the multiplication of each of the previous gains.

In order to obtain the input and output impedances the following formulae were used:

$$Z_i = \frac{1}{j\omega_0 C_1} + R_1 \tag{7}$$

$$Z_o = \frac{1}{1/R_2 + j\omega_0 C_2} \tag{8}$$

This results in the following results:

Theoretical results	value
gain	37.384634 (dB)
input impedance	1000.000000 Ohm
output impedance	666.666667 Ohm

Simulation results	value
gain	37.344430 (dB)
input impedence	999.000600 + i-7.319790 Ohm
output impedence	0.199367 + i-14.466600 Ohm

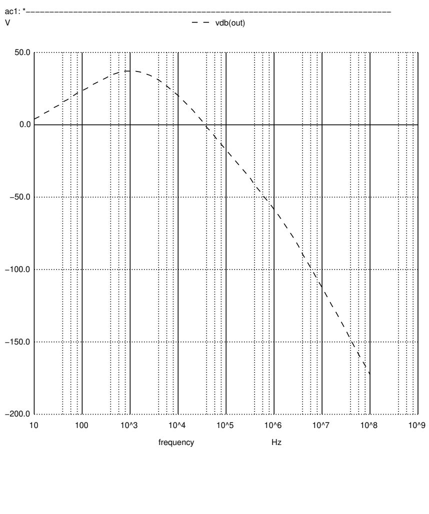
As it can be seen, the gain and input impedances have similar values, but the output impedance does not. This can be explained by the fact that the AM-POP is considered to be ideal in theoretical analysis, while in the simulation it acts like a real AM-POP.

3.2 Frequency Response

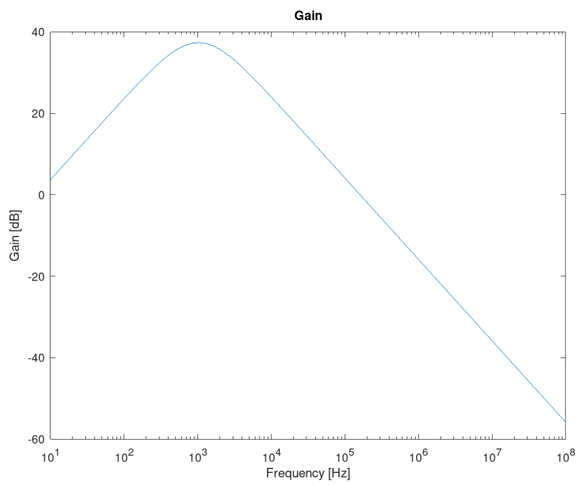
The frequency response can be was obtained the same way as it was done with the gain, but by applying the formula to many frequencies. This way, the transfer function will be:

$$f_{res} = \frac{R_1 C_1 \omega j}{1 + R_1 C_1 \omega j} \times (1 + \frac{R_3}{R_4}) \times \frac{1}{1 + R_2 C_2 \omega j} \tag{9}$$

With this function we can obtain the following plots:

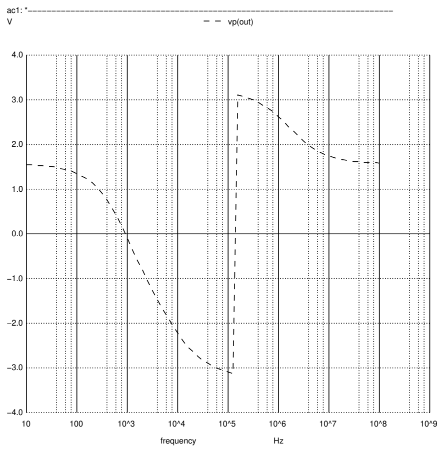


(a) Simulation Results

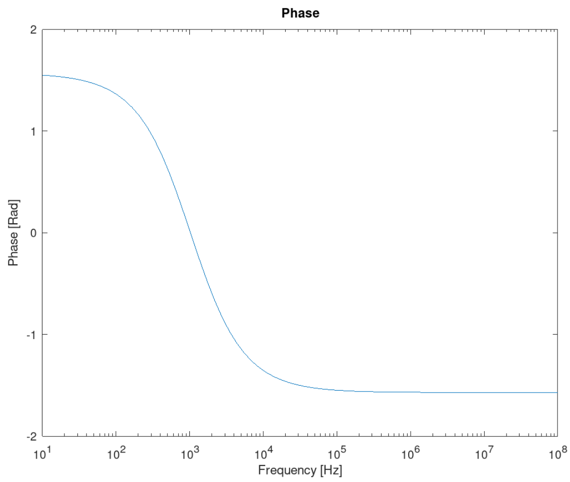


(b) Theoretical Results

Figure 3: Gain



(a) Simulation Results



(b) Theoretical Results

Figure 5: Phase

By looking at the previous results, it is possible to see that the theoretical model can predict quite well the value of the gain given a certain frequency. However, the same can't be said about the phase plots. Although, they start similar, as the frequencies get bigger, the results become completely differet.

This way, the merit will be

M	1.5142e-06

Table 3: Theoretical M

M	4.04744e-05

Table 4: Simulation M

Even with the previous mistakes, the theoretical analysis yields a decent estimate for the merit value with a similar order of magnitude.

4 Conclusion

Similar to the previous laboratory, the results of the theoretical ant simulation analysis were not equal. 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.

To solve the circuit, NGSpice used far more advanced simulation methods for the OP-AMP, with many more parameters.

This way, the objective should have never been to have equal results, but rather, have results that are "close enough", which we believe it was achieved.

Furthermore, one detrimental goal was to analyse and choose values for the circuit's elements that assure it's best and most optimal performance, which by taking into consideration the obtained data, we believe it was achieved.

We would also like to add that this lab enabled us to have the experience of going to the lab and build the real circuit that it was used. It showcased the advances of using simulators like NGSpice over the physical circuit, because they solve the problem of havinf broken pieces and allow much more flexibily in the analysis. Nevertheless, we still think that is crucial to have the experience of going to the lab if building the circuits "by hand" ,in order to learn how to work with real equipment and how to deal with real world problems.