Lab 5: Bandpass filter using OP-AMP

Circuit Theory and Electronics Fundamentals
Master's programme in Engineering Physics, Técnico, University of Lisbon

Diogo Costa | 96187
Ana Sousa | 96508
Isabel Alexandre | 96537

June 06, 2021

Contents

1 Introduction

The main objective of this laboratory assignment is to design a bandpass filter, using an OP-AMP. This filter should be centered at $1\ kHz$ and produce a gain of $40\ dB$ at this frequency. There are also some limitations in terms of resources: maximum 3 $1\ k\Omega$ resistors, 3 $10\ k\Omega$ resistors, 3 $100\ k\Omega$ resistors, 3 $220\ nF$ capacitors and 3 $1\ \mu F$ capacitors.

The secondary goal is to achieve the best performance at the lowest cost. This is quantified by maximizing the following function:

$$M = \frac{1}{cost \cdot gainDeviation \cdot centralFrequencyDeviation + 10^{-6}} \tag{1}$$

$$cost = cost\ of\ resistors + cost\ of\ capacitors + cost\ of\ transistors$$
$$cost\ of\ resistors = 1MU/k\Omega$$
$$cost\ of\ capacitors = 1MU/\mu F$$
$$cost\ of\ transistors = 0.1MU/transistor$$

In Section 2 a basic filter circuit is introduced and the study performed in the laboratory, using this circuit, is presented. This is followed by Section 3, where the same circuit is simulated. Furthermore, in an effort to improve our results, an optimized circuit is obtained and presented. A theoretical analysis is then performed, considering either circuit, in Section 4. The results obtained for the optimized circuit via simulation and theoretical analysis are compared in Section 5. Here, M is presented, allowing for some considerations to be made about the studies performed. The simulation was run on **Ngspice** and the theoretical calculations were performed with **Octave**.

2 Real circuit

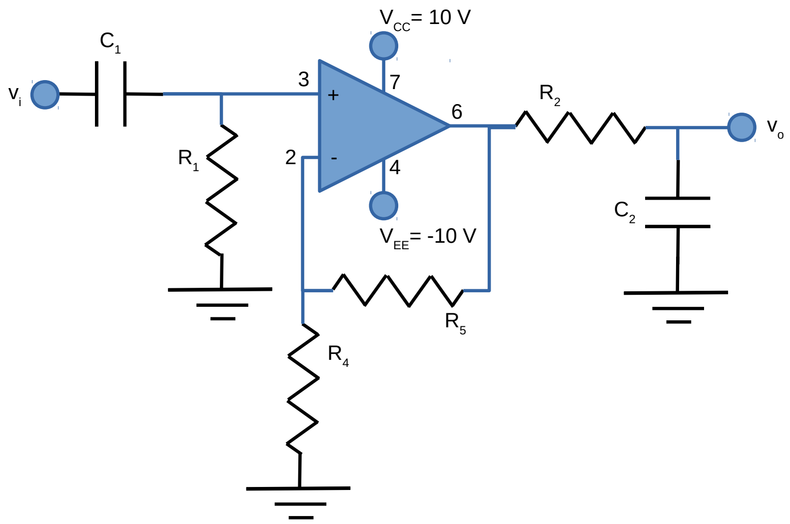

Figure 1: Bandpass filter

For the purpose of trying out this circuit, we chose the following parameters:

Parameter	Value
$C_1(nF)$	220
$R_1(k\Omega)$	1
$C_2(nF)$	220
$R_2(k\Omega)$	0.5
$R_4(k\Omega)$	1
$R_5(k\Omega)$	100

Table 1: Parameters for the simple circuit

By varying the frequency using a signal generator and measuring the peak to peak amplitudes of v_i and v_o, the gain as a function of the frequency could be determined. Below is a table which includes the data we retrieved (f, $2|v_1|$ and $2|v_2|$) and the calculated gain, in dB.

$f\ (Hz)$	$2\lvert v_1\rvert\ (V)$	$2\lvert v_2\rvert\ (V)$	$(\frac{v_2}{v_1})_{dB}$
500	0.105	5.5	34.28
600	0.105	5.9	34.99
700	0.105	6.2	35.42
800	0.105	6.2	35.42
900	0.105	6.2	35.42
1000	0.105	6.2	35.42
1100	0.105	6	35.14
1200	0.104	5.9	35.08
1300	0.104	5.7	34.78
1400	0.104	5.55	34.55
1500	0.104	5.4	34.31
2000	0.104	4.6	32.91
2500	0.104	3.9	31.48
3000	0.104	3.35	30.16
3500	0.104	2.9	28.91
4000	0.104	2.6	27.96
4500	0.104	2.3	26.89
5000	0.104	2	25.68
5500	0.104	1.8	24.76
6000	0.104	1.6	23.74
10000	0.103	1	19.74

Table 2: Data collected

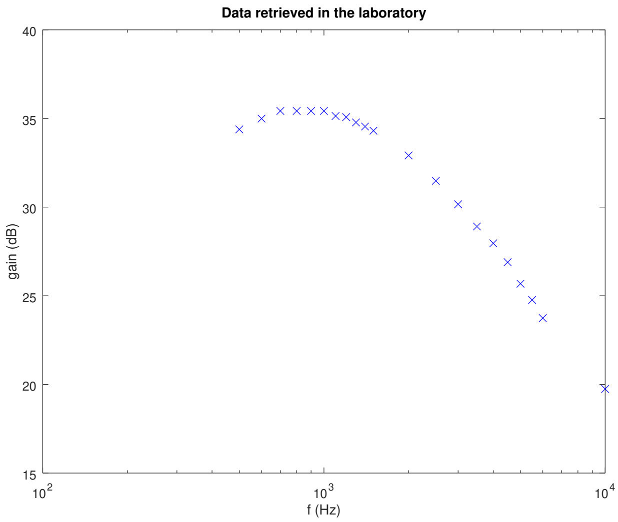

Figure 2: Data collected

3 Simulation Analysis

3.1 Simple circuit

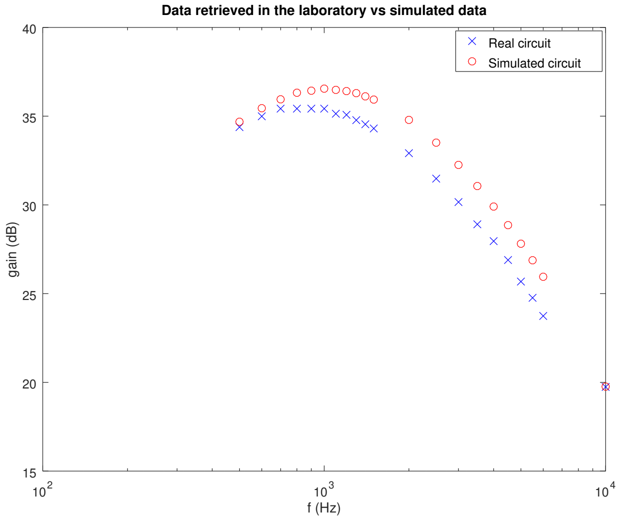

Figure 3: Data collected versus simulated data

It is obvious, from the above graph, that there are slight deviations between the real data and the simulated data, as expected. This can be due to any number of experimental causes:

- Errors in the peak to peak amplitude determined by the oscilloscope
- Errors in the resistors used in the lab (had gold tolerance)
- Errors in the capacitors used in the lab
- For the $500\ \Omega$ resistor, two $1\ k\Omega$ resistors were employed, in parallel
- All signals went through the breadboard
- V_{CC} was not exactly $10\ V$ and V_{EE} was not exactly $-10\ V$
- Discrepancies in the **Ngpisce** models

Using these parameters, the following was obtained:

Name	Value
$f_c\ (Hz)$	1006.492067637197
$gain(f_c)\ (dB)$	36.55104
z_{in}	0.9999928 + i (-0.723536)
z_{out}	0.341692 + i (-0.233043)

Table 3: Results

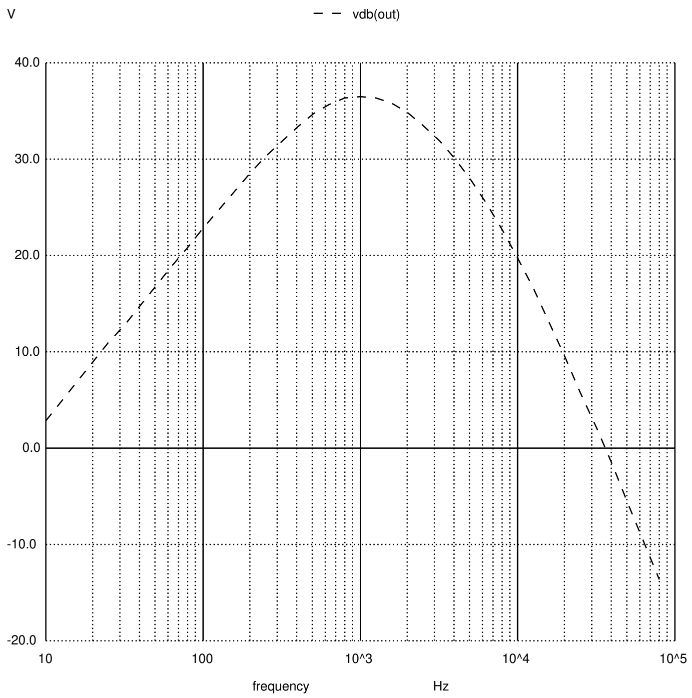

Figure 4: Frequency response - gain

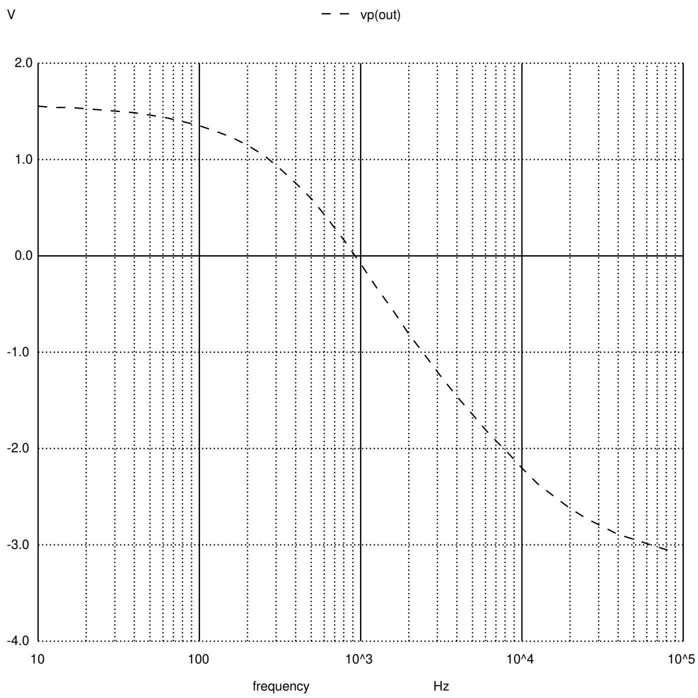

Figure 5: Frequency response - phase

3.2 Optimized circuit

Name	Value
$f_c\ (Hz)$	1006.492067637197
$gain(f_c)\ (dB)$	36.55104
z_{in}	0.9999928 + i (-0.723536)
z_{out}	0.341692 + i (-0.233043)

Table 4: Results

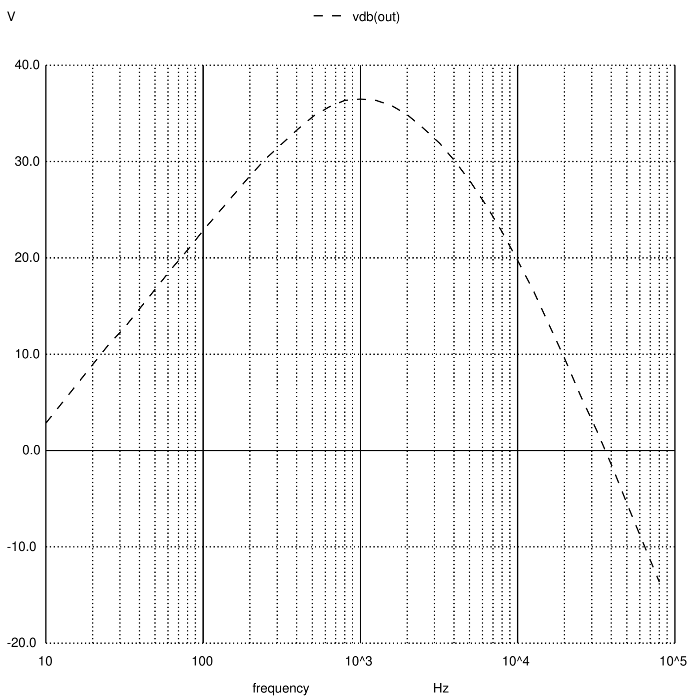

Figure 6: Frequency response - gain

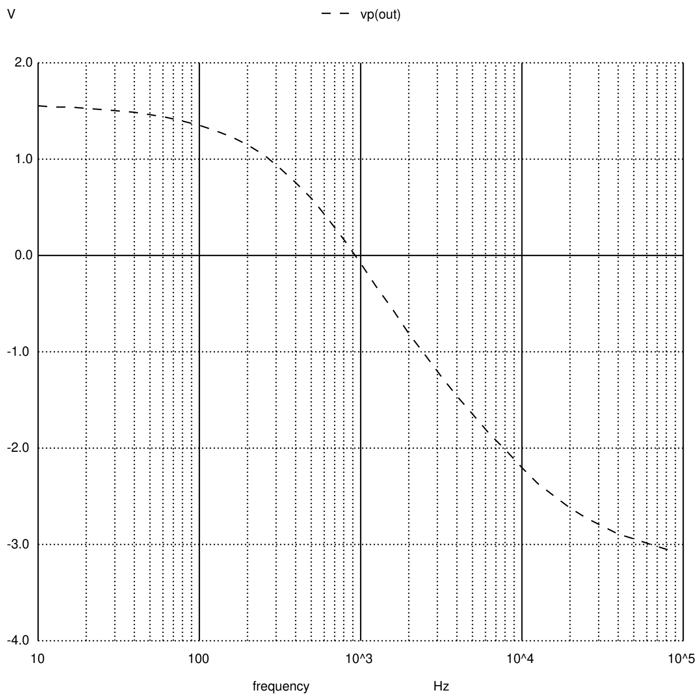

Figure 7: Frequency response - phase

4 Theoretical Analysis

First, the real circuit was analysed. As already mentioned, to perform the theoretical analysis **Octave** was used. This analysis was conducted considering a sine wave of amplitude 1 and phase 0 as the input. In order to compute the gain, the input impedance and the output impedance, the circuit was analysed using phasors.

It is possible to observe that the OP-AMP is inserted in a non-inverting amplifier with resistive feedback loop. Considering the ideal model for the OP-AMP, the voltage in node 3 must be equal to voltage in node 2. Also, voltage in node 2 must be:

$$\widetilde{V_2} = \frac{R_4}{R_4 + R_5}\widetilde{V_6} \tag{2}$$

An ideal OP-AMP has infinite input impedance, so no current goes in the entries of the OP-AMP. Considering this, we can perform nodal analysis and, for node 3, we obtain:

$$(\widetilde{V_3} - \widetilde{V_i})jwC_1 + \frac{\widetilde{V_3}}{R_1} = 0 \tag{3}$$

For the node of the output voltage, we get:

$$\frac{\widetilde{V_o} - \widetilde{V_6}}{R_2} + \widetilde{V_o}jwC_2 = 0 \tag{4}$$

With this, it's possible to write the following matrix:

$$\begin{bmatrix} 1 & -1 & 0 & 0 \\ 1 & 0 & -\frac{R_4}{R_4+R_5} & 0 \\ 0 & jwC_1 + \frac{1}{R_1} & 0 & 0 \\ 0 & 0 & -\frac{1}{R_2} & jwC_2 + \frac{1}{R_2} \end{bmatrix} \begin{bmatrix} \widetilde{V_2} \\ \widetilde{V_3} \\ \widetilde{V_6} \\ \widetilde{V_o} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ jwC_1\widetilde{V_i} \\ 0 \end{bmatrix} \tag{5}$$

By solving the matrix it is then possible to compute the gain for every frequency.

The central frequency was calculated by considering that:

$$w_c = \sqrt{w_L w_H} \tag{6}$$

The analysis of the circuit allows to conclude that $w_L = \frac{1}{R_1C_1}$ and $w_H = \frac{1}{R_2C_2}$.

The input impedance will be:

$$z_{in} = \frac{\widetilde{V_i}}{\widetilde{I_i}} = \frac{\widetilde{V_i}}{(\widetilde{V_i} - \widetilde{V_3})jwC_1} \tag{7}$$

The output impedance is calculated considering that the input source is off, is like a short-circuit. That results in no current in C_1, R_1 and R_4. R_5 is in parallel with the OP-AMP. Ideally, the OP-AMP has 0 output impedance, so the parallel with R_5 results in 0 impedance. So, in the end, what is left is the parallel of R_2 and C_2. Consequently, the output impedance is:

$$z_{out} = \frac{1}{\frac{1}{R_2} + jwC_2} \tag{8}$$

With this analysis, the following results were obtained (gain and impedances were calculated for 1000 Hz):

Name	Value
$f_c\ (Hz)$	1023.086723
$gain(f_c)\ (dB)$	36.562591
z_{in}	1000.000000 + i (-723.431560)
z_{out}	338.366226 + i (-233.861947)

Table 5: Results

The frequency response was also calculated and the following was obtained:

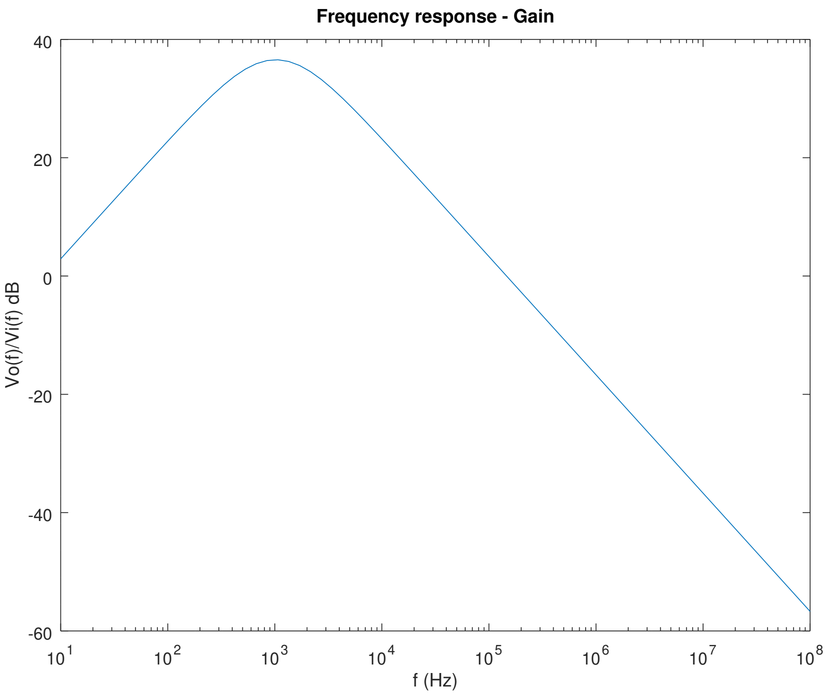

Figure 8: Frequency response - Gain

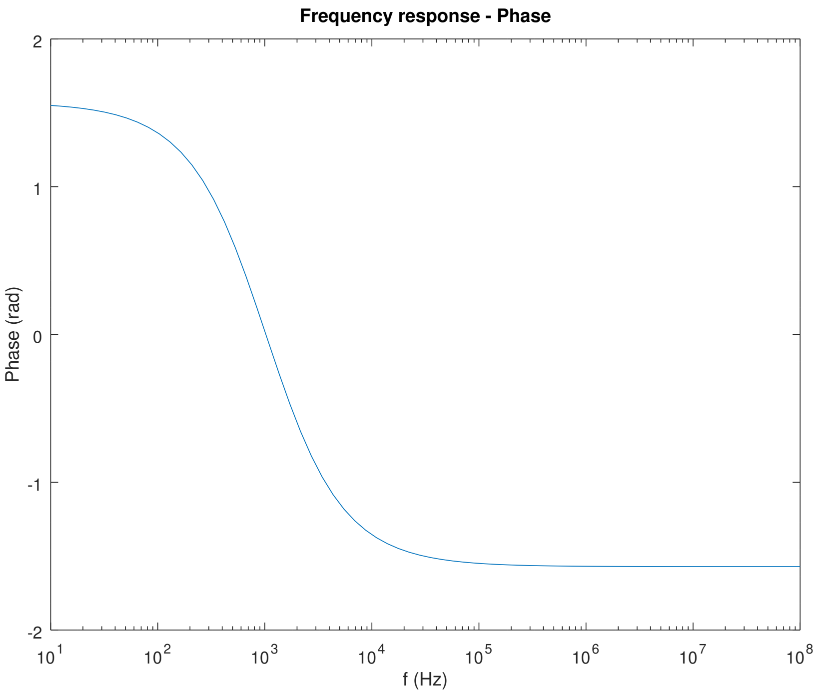

Figure 9: Frequency response - Phase

5 Conclusion

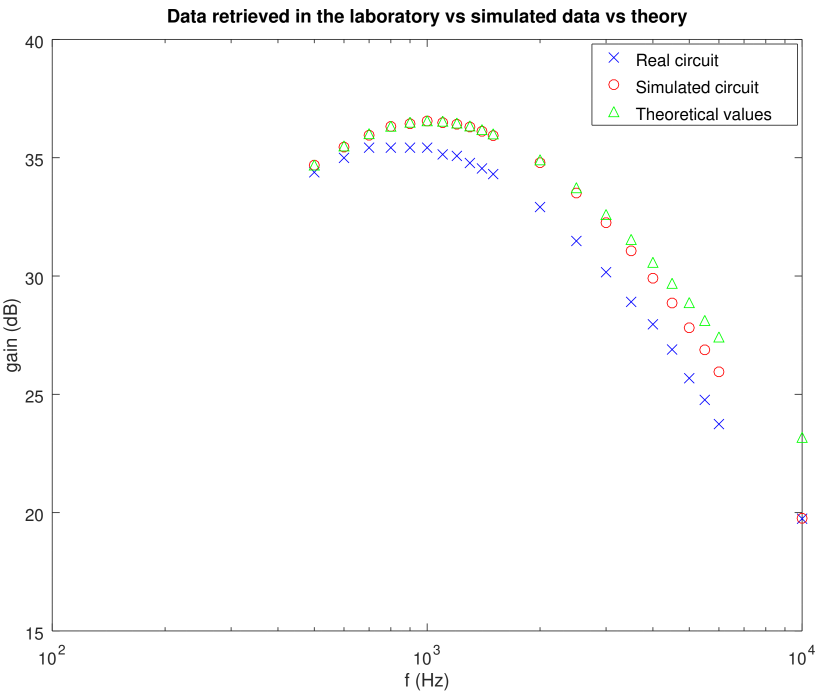

Figure 10: Data comparison

V	V_t	V_s	$\|V_t - V_s\|$	$Error(\%)$
$f_c\ (Hz)$	1023.086723	1006.49206764	20	2
$gain(f_c)\ (V)$	36.562591	36.55104	0.02	0.04
$Re(z_{in})$	1000.0	999.9928	0.008	0.0008
$Im(z_{in})$	-723.43156	-723.536	0.2	0.02
$Re(z_{out})$	338.366226	341.692	4	1
$Im(z_{out})$	-233.861947	-233.043	0.9	0.4

Table 6: Theoretical values (V_t) and Simulation values (V_s) of the operating point analysis (table produced with **Python**) - The absolute deviation and error presented here are rounded up to one significant digit, for ease of interpretation.