

Circuit Theory and Electronics Fundamentals

Masters of Aerospace Engineer, Técnico, University of Lisbon

Laboratory Report

Group 37

Afonso Magalhães, nº95765
Fábio Monteiro, nº95786
Leonardo Encarnação, nº95816

June 6, 2021

Contents

1	Introduction	3
2	Theoretical Analysis	4
2.1	Transfer function computation	4
2.1.1	High pass filter stage	4
2.1.2	Gain stage	5
2.1.3	Low pass filter stage	6
2.1.4	Overall Transfer function	6
2.2	Cutoff and Central Frequencies	7
2.3	Input and Output Impedances	7
3	Simulation Analysis	8
3.1	Circuit simulation	8
3.2	Input and Output Impedances	10
4	Comparison	11
5	Merit Results	12
6	Conclusion	12

1 Introduction

The objective of this laboratory assignment is to choose the best architecture of the circuit in order to build a Bandpass filter using OP AMP. This assignment allowed us to deal with important concepts such as OP AMP and its diverse utility in circuits. We did this while paying attention to the merit of the project designed. This merit is calculated exactly as the next equation:

$$M = \frac{1}{cost * (gaindeviation + centralfrequencydeviation + 1e - 6)} \quad (1)$$

Being the cost the following:

- cost = cost of resistors + cost of capacitors + cost of transistors
- cost of resistors = 1 monetary unit (MU) per kOhm
- cost of capacitors = 1 MU/ μ F
- cost of transistors = 0.1 MU per transistor

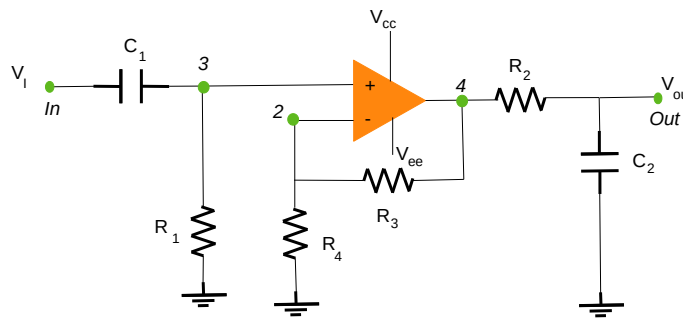


Figure 1: Main circuit

The constants values used are expressed in the following table.

Name - Value	
C1	2.200000e-07
C2	1.100000e-07
R1	1.000000e+03
R2	1.000000e+03
R3	1.000000e+05
R4	1.000000e+03
Vcc	1.000000e+01

Table 1: Initial Values

In Section 2, a theoretical analysis of the circuit is presented. In Section 3, the circuit is analysed by simulation, and in Section 4 the results are compared to the theoretical results obtained in Section 2. Next in Section 5 we show the cost and merit attributed to our circuit. To finish, the conclusions of this study are

outlined in Section 6.

2 Theoretical Analysis

In this section we will analyse theoretical our bandpass filter using OP AMP.

To do so, and because there were several things to be analysed, we divided the theoretical analysis in the following subsections that explain the different sectors that our circuit has and also each one will be detailed separately.

2.1 Transfer function computation

In this section, a theoretical analysis of the circuit shown in section 1 was conducted. The OP Amp was considered ideal (internal impedance between v_+ and v_- is infinite, which means no current flows through it and $v_{os}=0$ or in other words $v_+ = v_-$). So, in order to build a bass pand filter, a capactor was connected in series (C_1) with the input voltage which will function as a high pass filter (for low frequencies, the impedance goes to infinity and the capacitor is basically an open circuit) and other capacitor (C_2) was connected in parallel with the output voltage, functioning as a low pass filter (for high impedances, the impedance goes to 0 and the capacitor is basically a short circuit). To sum up, this circuit consists of a high pass filter, a signal amplifier and a low pass filter in series.

To compute the overall transfer function for this circuit we've computed the tansfer function for these three stages and multiplied them all to obtain the overall transfer function

2.1.1 High pass filter stage

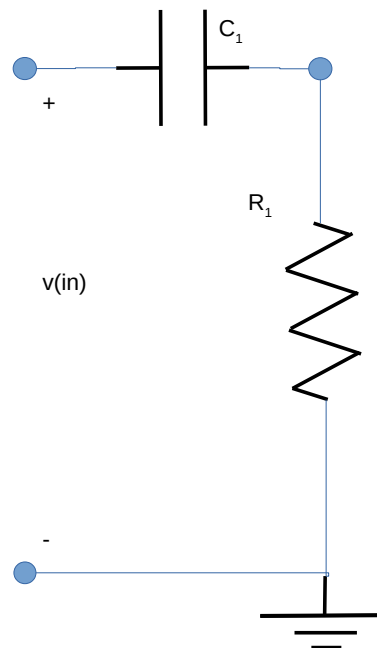


Figure 2: High Pass Filter stage circuit

Observing the image above and knowing that the input impedance at the op-amp is infinite, by just doing the voltage divider we've obtained the transfer function for these stage:

$$T_1(s) = \frac{R_1}{R_1 + \frac{1}{sC_1}} = \frac{R_1 C_1 s}{1 + R_1 C_1 s} \quad (2)$$

2.1.2 Gain stage

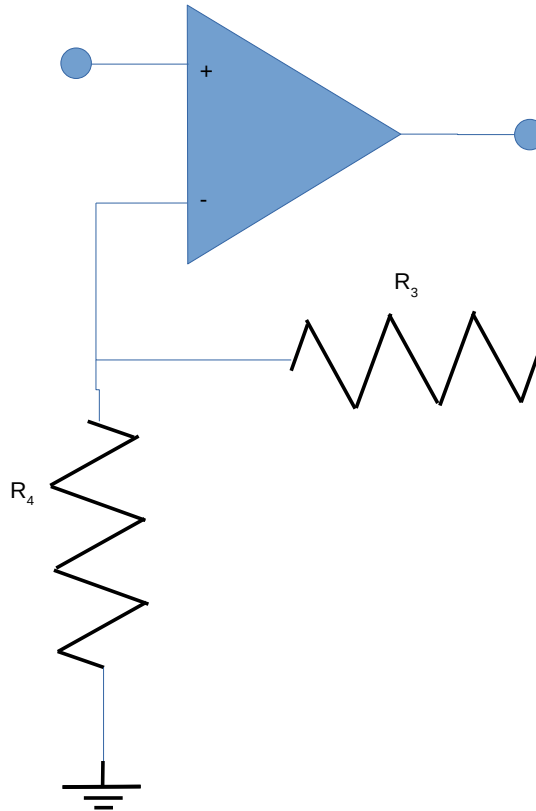


Figure 3: Gain stage circuit

To analyse these part of the circuit, first we need to specify some equations for the op-amp. Firstly the voltage in the two terminals is equal, this is, $v_3 = v_2$, and the other important thing to know is that the input impedance of this device is infinite. By knowing this, we can very quickly determine that the output voltage will be:

$$v_o = v_4 = v_2 + v_{R_3} = v_3 + v_{R_3} = v_i + v_{R_3} \quad (3)$$

We also know that:

$$v_i = v_{R_4} = R_4 i \iff i = \frac{v_i}{R_4} \quad (4)$$

$$v_{R_3} = R_3 i = \frac{R_3}{R_4} v_i \quad (5)$$

So the transfer function for this stage is:

$$T_2(s) = \frac{v_o}{v_i} = \frac{v_i + v_{R_3}}{v_i} = \frac{(1 + \frac{R_3}{R_4})v_i}{v_i} = 1 + \frac{R_3}{R_4} \quad (6)$$

2.1.3 Low pass filter stage

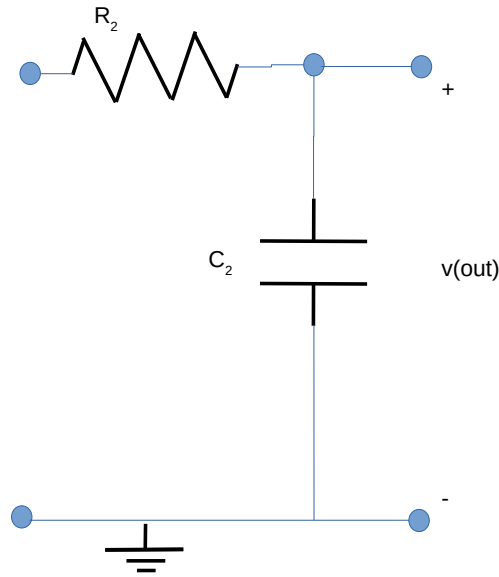


Figure 4: Low pass filter stage circuit

Once again, considering the figure above, it's just making the voltage divider formulae and we've obtained the transfer function for this stage

$$T_3(s) = \frac{\frac{1}{sc_2}}{R_2 + \frac{1}{sc_2}} = \frac{1}{1 + C_2 R_2 s} \quad (7)$$

2.1.4 Overall Transfer function

The overall transfer function is given by:

$$T(s) = T_1(s) * T_2(s) * T_3(s) = \frac{R_1 C_1 s}{1 + R_1 C_1 s} * \left(1 + \frac{R_3}{R_4}\right) * \frac{1}{1 + C_2 R_2 s} \quad (8)$$

Using octave to plot this transfer function bode plots, we've obtain the following.

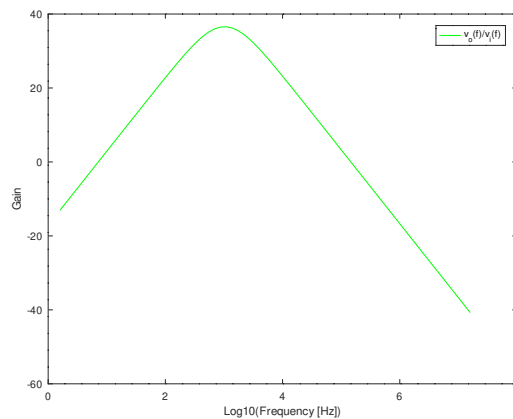


Figure 5: Gain

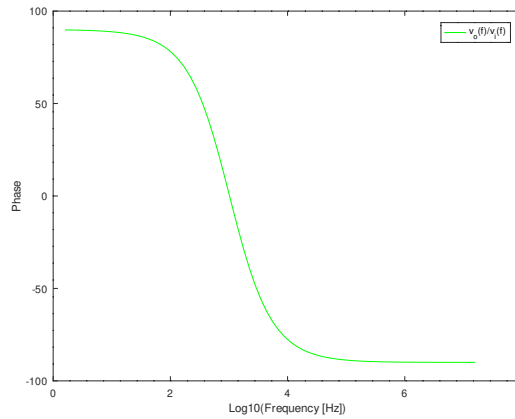


Figure 6: Phase

2.2 Cutoff and Central Frequencies

To obtain the cutoff frequencies we just need to determine the poles of the transfer function. By looking at its formulae we obtain the following poles:

$$w_L = \frac{1}{R_1 C_1} \quad (9)$$

$$w_H = \frac{1}{R_2 C_2} \quad (10)$$

After obtaining these frequencies, in order to obtain the central frequency we just need to obtain the frequency exactly in middle of these two cutoff frequencies in the gain plot in dB. This means that the central frequency is given by the following formulae:

$$w_0 = \sqrt{w_L w_H} \quad (11)$$

Using octave we've reached the following values for these 3 frequencies.

Name - Value	
LowFreq BandPass	4.545455e+03
HighFreq BandPass	9.090909e+03
Central Freq	6.428243e+03

Table 2: BANDPASS FREQUENCY

With respect to the central frequency we've also obtained its value in Hz and its gain in dB

Name - Value	
Central Freq (Hz)	1.023087e+03
Gain Central Freq (dB)	3.656460e+01

Table 3: WO FREQUENCY GAIN

2.3 Input and Output Impedances

In order to compute the input impedance we take the circuit and turn off all independent sources. After these we connect an independent voltage source in the input of the circuit and determine the current in this source. Observing the circuit in the introduction and knowing that the input impedance of the op-amp is infinite, we get

that the input impedance is just the resistor R_1 and the capacitor C_1 in series, so the input impedance for the central frequency is:

$$Z_{in} = R_1 + \frac{1}{j\omega_0 C_1} \quad (12)$$

To compute the output stage, we start in the same way as the input impedance, by turning off all the independant sources. After these, instead of connecting a independant voltage source at the input of the circuit, we connect it to the output, and determine the current at these source in the same way as before. Because the output impedance of the op-amp is 0, the output impedance of the circuit is just the resistor R_2 and the capacitor C_2 in parallel. So the output uimpedance for the central frequency is given by:

$$Z_{out} = \frac{R_2}{1 + jR_2 C_2 \omega_0} \quad (13)$$

Using octave, we've obtain the following values for the impedances:

Name - Value	
Z in	1.000000e+03 + -7.071068e+02j
Z out	6.666667e+02 + -4.714045e+02j

Table 4: IMPEDANCES

3 Simulation Analysis

In this section, Ngspice was used in order to simulate the Bandpass Filter. A brief description of the circuit modeled in NGspice is going to be presented and a comparison between the values obtained in NGspice and the ones in Octave is going to be done as well. In order to do that, we used the model $\mu A741$ from Texas In-struments to represent the OP AMP. The measurement of these parameters and the overall performance of the circuit is represented in the beggining of the 5.

3.1 Circuit simulation

We started the simulation by designing the circuit in NGspce, having as start point the circuit presented in Section 1. Then, the results were calculated and are represented in the following table:

Name - Value	
Gain	36.5323
Central Frequency	1000
Gain deviation	32.9164
Central frequency deviation	0

Table 5: Results NGSpice

Consequently, the phase and the gain were computed. Is also important to remember that the main goal of this assignment was to design a band pass filter, meaning that this filter should cut both low an very high frequencies.

The following graphs represent the results obtained from simulation in NGSpice of the gain and phase:

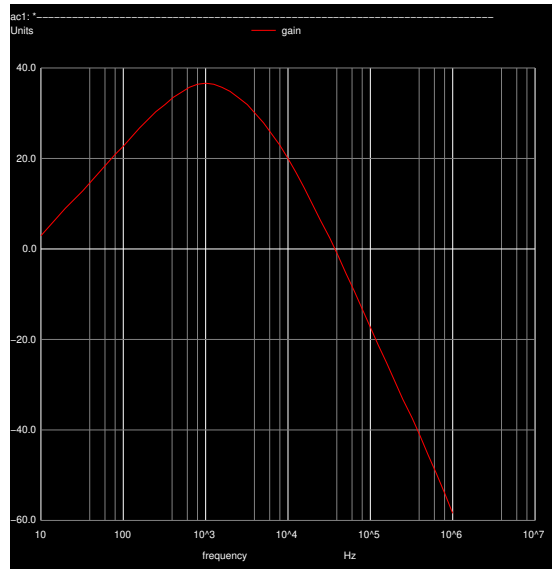


Figure 7: Gain

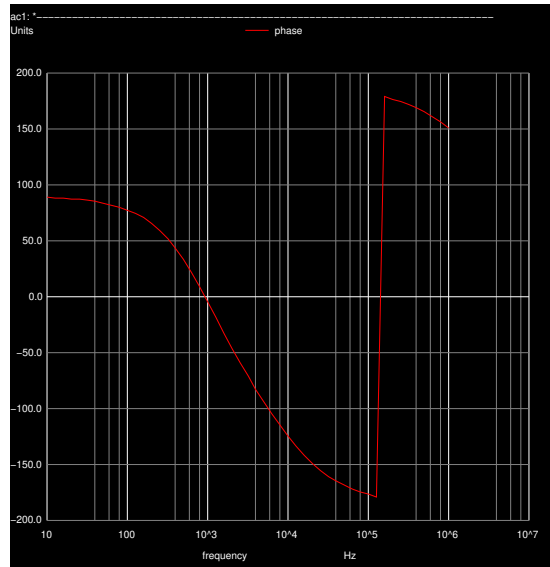


Figure 8: Phase

3.2 Input and Output Impedances

The following table represents the input impedance obtained from simulation in NGSpice:

Name - Value	
Zin	999.002 + -7.3282 j

Table 6: Zin

The result obtained for the input impedance, considering the value in Ohm's, is quite high, which turns out to be very beneficial for the gain, because the voltage in node 2 must be as similar to V_{in} as possible. Using a voltage divider, the only way to achieve this was to have a very high resistance value.

The following table represents the output impedance obtained from simulation in NGSpice:

Name - Value	
Zo	0.0522978 + -7.23396 j

Table 7: Zout

In regard to the output impedance, an opposite deduction to the one made for the input impedance is needed. Using again a voltage divider, the output impedance must be as low as possible, in order for the output voltage to be as high as possible.

4 Comparison

In this section a global comparison of the two approaches is created, with the chosen values for the constants. The following table represents the comparison between the results obtained from Octave and from simulation in NGSpice:

Name - Value	
Gain	3.656460e+01
Central frequency	1.023087e+03
Gain deviation	3.266667e+01
Central frequency deviation	2.308672e+01

Table 8: Octave

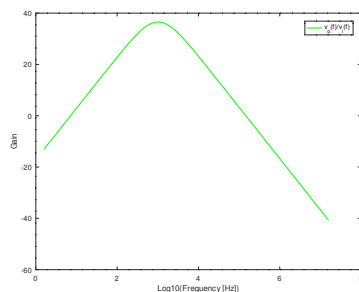
Name - Value	
Gain	36.5323
Central Frequency	1000
Gain deviation	32.9164
Central frequency deviation	0

Table 9: NGSpice

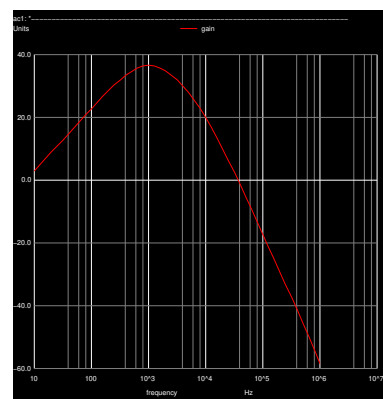
Table 10: Simulation results

As seen in the previous table, as requested, we were able to compute the passband frequency in a simulation analysis using the measure funtion and the central frequency in the theoretical analysis as explained in Section 2. We have to consider that the OP-AMP model chosen is far more complex in Ngspice than in Octave. The different calculation methods lead to the difference between both methods, being that the calculated and simulated impedances have the same issues.

Next we present both theoretical and simulation graphs of the frequency response of the gain and the phase side by side.

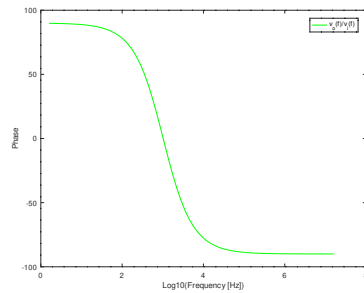


(a) OCTAVE

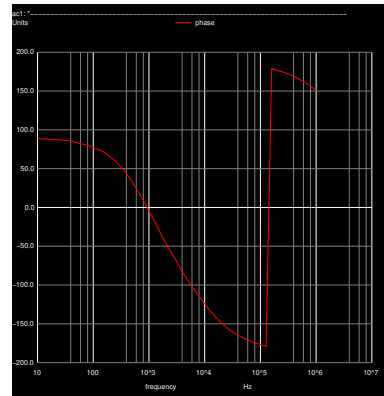


(b) NGSPICE

Figure 9: Gain



(a) OCTAVE



(b) NGSPICE

Figure 10: Phase

In general, in terms of the graphs, we achieved satisfactory results. When an initial comparison of the gain response is executed, using the shape of the graph, isn't difficult to realise that they have a similar behaviour. As seen in the upper graphs, in the initial section, there is a constant slope of +20dB/dec. After that, exists a small pass band, that is also the region where the central frequency is achieved, and a last part with a slope of -20dB/dec.

When it comes to the lower graphs, in the phase response, both plots differ. This is because our theoretical model does not predict the existence of capacitors in the OP AMP, as it idealises the gain as being purely real, i.e, there's no shift in phase. In reality, because the OP AMP has two capacitors, it is expected that in the phase frequency response plot each would introduce a shift of -90° , making the overall phase to go down to -270° , in other words, $+90^\circ$. As we can observe from the initial table of this Section (4) we have small relative errors for the values of interest. This somehow proves that, eventhough the OP AMP's are made of dozens of components, being some of them non-linear, we can predict its behaviour.

5 Merit Results

From the results obtained through the Ngspice simulation and considering we used the data shown in table 1, we can compute the merit using the formula given in the lab assignment, represented in the Introduction.

The values of cost and merit are represented in the next table:

Name - Value	
Cost	2.639850e+02
Merit	6.794373e-05

Table 11: Octave

Name - Value	
Cost	263.985
Merit	0.000115082

Table 12: NGspice

Table 13: Cost and Merit

6 Conclusion

In this laboratory assignment, the goal especified in the introduction has been achieved with a great merit. All analyses have been performed both theoretically using the Octave maths tool and by circuit simulation

using the Ngspice tool. When comparing these last two we conclude that there aren't any disparity between the results and therefore no errors associated. So, we conclude that the architerture that we used can be validated.