TÉCNICO LISBOA

Circuit Theory and Electronics Fundamentals

Mestrado em Engenharia Física Tecnológica, Técnico, University of Lisbon

Ana Martins (96506), Miguel Moreira (96556), Sara Oliveira (96566)

T4

May 5, 2021

Contents

1 Introduction

The circuit described in the present report is displayed in Figure 1.

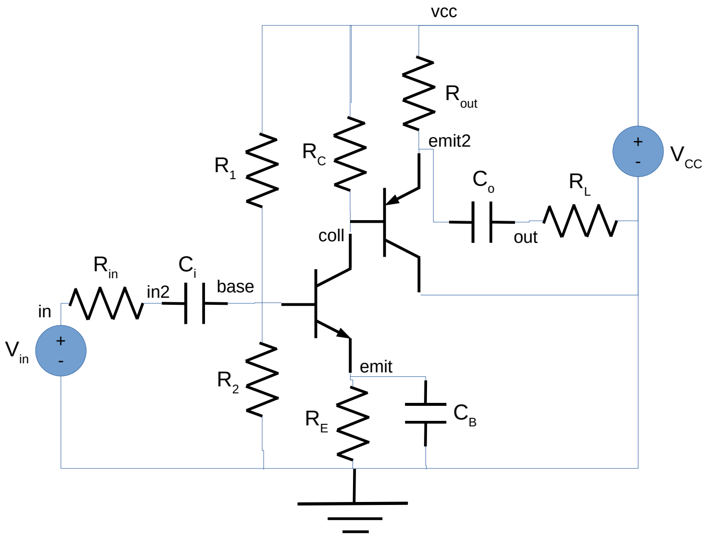


Figure 1: The circuit we will be working with.

Firstly, in Section 2, the circuit is analysed by simulating it as a whole, using the *software Ngspice*.

After that, we proceed to the theoretical analysis of this same circuit to verify and compare results with the simulation data. In Section 3, the theoretical analysis is executed in two distinct steps: the first step consists of a operating point analysis, where it's only considered the DC component of the current; the second step, called incremental analysis, takes into account the frequencies of the current in time, therefore, the AC component of the current.

Lastly, in Section 4 the results obtained in the two previous Sections are compared, focusing on the currents and tensions obtained in key points of the circuit. as well as, impedances and gain. The conclusions of this study are outlined in Section 5.

2 Simulation Analysis

The circuit, as is presented on Figure 1, was simulated using *NGSpice*.

2.1 Voltage Gain

We first took a look at what happened in node *out* compared to node *in2*, which suffered the effect from both of the stages of the circuit, using frequency analysis. The result is presented in Figure 2.

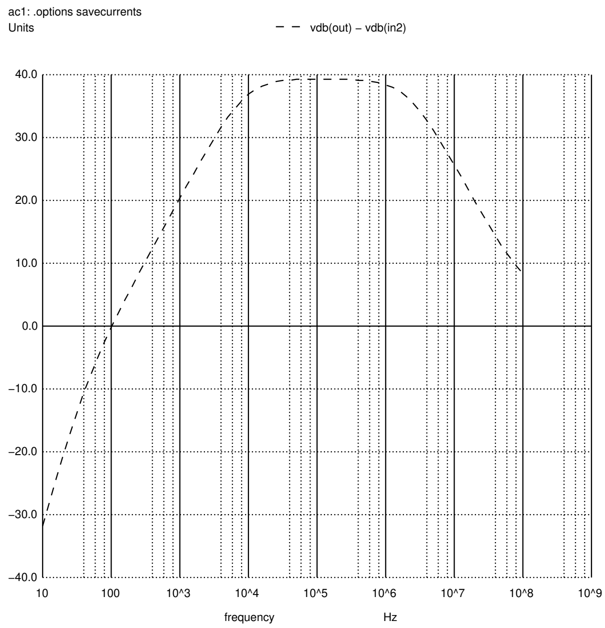


Figure 2: Output Voltage Gain Simulated

This graph is the consequence of a passband filter, which means that there is only a specific set of values - a band, if you will - where the circuit works the way it is supposed to. In this case, the passband filter works between around 10^4 and 10^6 Hz, which means the amplifier barely works for the human ear.

The value for the highest point in the graoh is shown below.

Variable	Value (dB)
gaindb	3.929296e+01

Additionaly, we worked out the bandwidth, which is shown below.

Variable	Value
bandwidth	1.073353e+06

The value of the bandwidth shows for how long the passband filter works - i.e. for how much of the frequencies tested the circuit actually amplifies.

2.2 Input and Output Impedances

To work out the input impedance, we simply measured the voltage in node *in2*, the node of the input stage of the circuit, and divided it by the current going through it. The result is shown in the table below, with the first value being the real part and the second value being the imaginary part.

Variable	Value ($k\Omega$)
zin	5.638527e-01

For measuring the output impedance, we had to change the circuit used in *NGSpice*. We turned off source V_{in}, leaving it in short circuit. Furthermore, we substituted the load, which was the component that was overseeing the whole circuit, with a dummy voltage source equal to the one we turned of, calling it V_L as it is in the place of the load. This circuit can be observed in Figure 3.

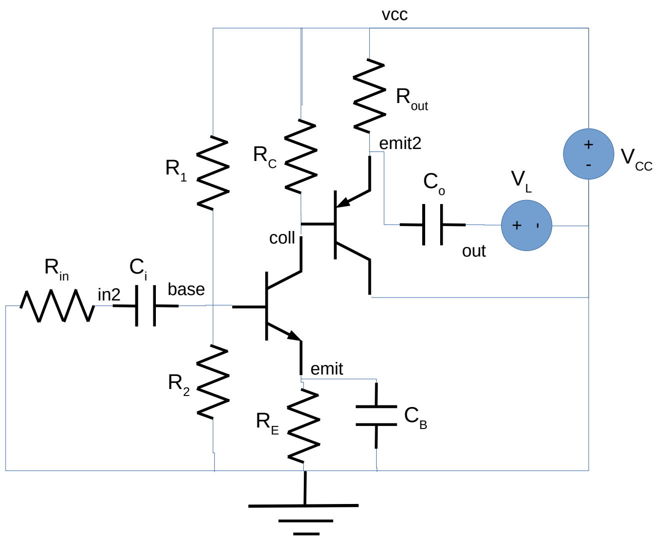


Figure 3: Circuit used for measuring the output impedance.

The final result of the output load is seen is below, dividing the values we got for the voltage in node *out* and dividing it through the current going through it.

Variable	Value ($k\Omega$)
zout	1.005540e-02

2.3 Merit

Lastly, we calculated the merit, which is shown below.

Variable	Value
merit	-6.86406e+00,-2.86839e-01

3 Theoretical Analysis

3.1 Operating Point

When it comes to operating point analysis, the capacitors behave as open circuits, for their impedance is infinite, therefore, for the gain stage we can compute the Thévenin's equivalent of the bias circuit, making it easier to analyse. Running simple mesh analysis, and knowing the NPN transistor works in FAR, we can calculate every current and voltage. To compute the input and output impedance regarding this stage and its gain, one has to analyse the incremental circuit for medium frequencies, the ones that belong to the passband, using the calculations shown in the class slides. The table below shows all these results.

Variable	**Value**
IB1	5.0044e-05
IC1	8.9429e-03
IE1	8.9929e-03
VE1	8.9929e-01
VO1	3.0571e+00
VCE1	2.1578e+00
gain1 (dB)	4.8392e+01
ZI1	4.8443e+02
ZO1	8.8628e+02

For the output stage we do the same analysis but with the supermesh and this time the transistor is PNP. We calculate essentially the same variables:

Variable	**Value**
VI2	3.0571e+00
IE2	8.2429e-02
IC2	8.2068e-02
VO2	3.7571e+00
gain2 (dB)	-7.0225e-02
ZI2	8.5989e+03
ZO2	3.0217e-01

Combining both stages, we get the total gain, imput and output impedances:

Variable	**Value**
gain (dB)	4.7959e+01
ZI	4.8443e+02
ZO	3.9820e+00

3.2 Incremental analysis

This time, we create our incremental circuit that combines both stages as shown in figure 4.

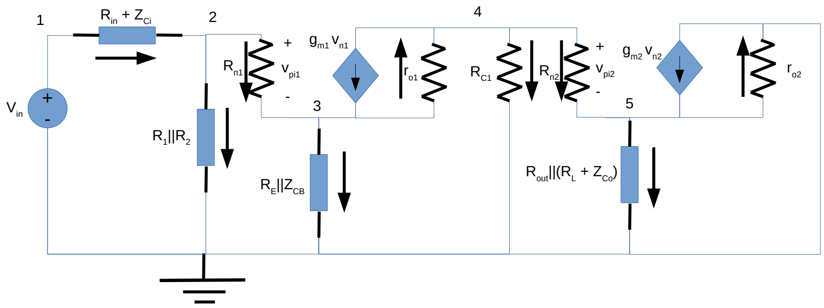


Figure 4: Incremental analysis circuit

Performing nodal analysis for frequency $\in [1Hz, 100MHz]$, with 10 points per decade, we get a vector with the several gain values, which we plot in a dB graph.

$$\begin{cases} V_1 = V_{in} \\ \frac{V_1 - V_2}{R_{in}+Z_{C_i}} - \frac{V_2}{R_1||R_2} - \frac{V_2-V_3}{R_{\pi_1}} = 0 \\ \frac{V_2-V_3}{R_{\pi_1}} + g_{m_1} v_{\pi_1} - \frac{V_3}{R_E||Z_{C_B}} - \frac{V_3-V_4}{r_{o_1}} = 0 \\ g_{m_1} v_{\pi_1} + \frac{V_4}{R_{C_1}} + \frac{V_4-V_5}{R_{\pi_2}} - \frac{V_3-V_4}{r_{o_1}} = 0 \\ \frac{V_4-V_5}{R_{\pi_2}} + g_{m_2} v_{\pi_2} - \frac{V_5}{R_{out}||(R_L+Z_{C_o})} - \frac{V_5}{r_{o_2}} = 0 \\ v_{\pi_1} = V_2 - V_3 \\ v_{\pi_2} = V_4 - V_5 \end{cases} \quad (1)$$

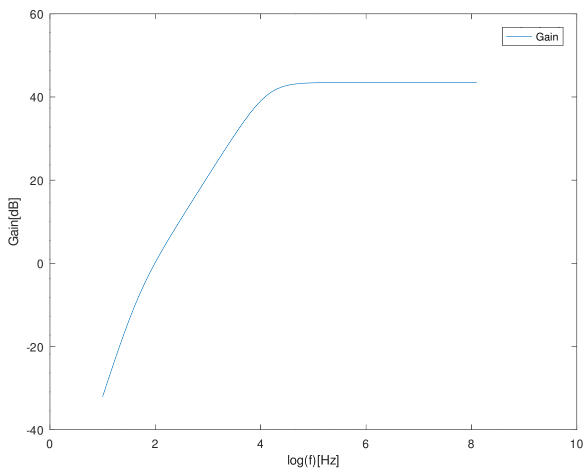


Figure 5: Gain in dB

4 Comparing the Results

In this section, we will be comparing the results obtained by both the Theoretical Analysis and the Simulation.

In the following tables and plots, we can observe the results already presented in previous sections now side by side, with the first table being the result of the simulation and the second one being the result from the theoretical analysis. These are displayed in this way as a means of easier representation.

4.1 Table Comparison

Variable	Value (dB)
gaindb	3.929296e+01

Variable	Value ($k\Omega$)
zin	5.638527e-01

Variable	Value ($k\Omega$)
zout	1.005540e-02

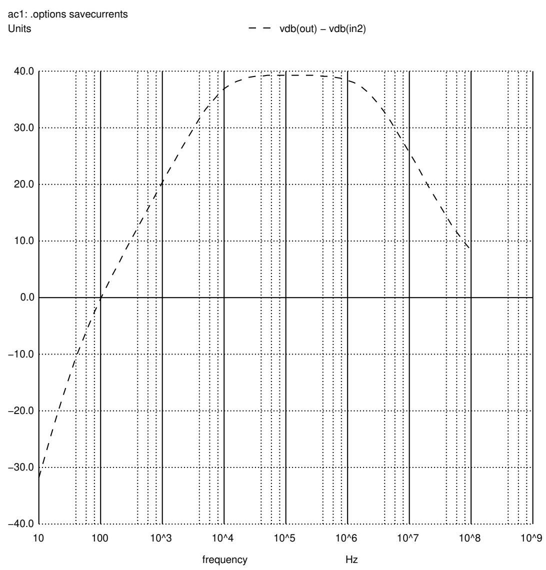
Variable	Value
gain (dB)	4.7959e+01
ZI	4.8443e+02
ZO	3.9820e+00

Figure 6: Simulation Values (LEFT) and Theoretical Values (RIGHT)

In the tables above we can compare the gain (in dB), the input impedance (on the left - $k\Omega$, on the right - Ω) and the output impedance (on the left - $k\Omega$, on the right - Ω)).

The components used in the simulation have, overall, non-linear behaviours. As in the theoretical analysis we used linear approximations, the estimated values for impedances and gain are slitghly off when comparing to the ones obtained by the *NGSpice* simulation.

4.2 Plot Comparison



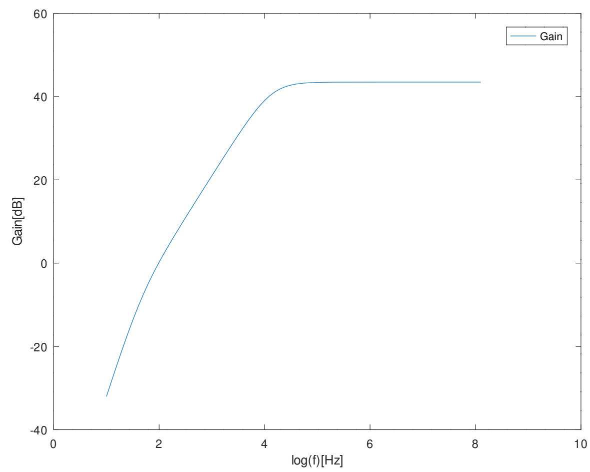


Figure 7: Simulation Gain Values (LEFT) and Theoretial Gain Values (RIGHT)

Straightforwardly, we can verify that the theoretical plot, for low frequencies, closely resembles the plot obtained by simulation. However, for high frequencies the plots are clearly different. It's thought that, due to linear approximations, the theoretical model isn't ideal for highfrequency analysis, whereas the simulation, using non-linear components closer to reality, isn't prone to this kind of deviation.

5 Conclusion

In conclusion, it was verified that, while the theoretical predictions are good to have an understanding of how the components interact with each other, it only works under specific conditions - in this case for lower frequencies - whereas the simulation always gives a better approximation of reality. This happens because the theoretical analysis is an oversimplification of the real circuit, in this case, switching the transistors for all linear components, in such a way that the circuit still, ultimately works as a filter. However, as we could see in the graph for the gain in the theoretical analysis, the circuit used for the theoretical analysis works more closely as a highpass filter instead of a bandpass one.

As such, we do believe that the analysis of this amplifier circuit was successful, even considering the differences between the two analysis.

References

[1] Phyllis R. Nelson, *Introduction to SPICE Source Files* Slides

[2] *SPICE 'Quick' Reference Sheet*, Stanford University

[3] Holger Vogt *et al*, *Ngspice's User Manual*, Version 34

[4] *GNU Octave* Documentation Files

[5] José Teixeira de Sousa, *Circuit Theory and and Eletronic Fundamentals* Class Slides