

Circuit Theory and Electronics Fundamentals

Department of Electrical and Computer Engineering, Técnico, University of Lisbon

May 23, 2021

Laboratory Assignment - T4

Group nº59

José Miguel Goulão - 95814

Lourenço Pacheco - 95817

André Gomes - 96352

Contents

1	Introduction	2
2	Theoretical Analysis	3
2.1	Gain stage - Degenerated Common Emitter Amplifier	3
2.2	Output stage - Common Collector Amplifier	3
2.3	Circuit frequency response	4
2.4	Theoretical Results	4
3	Simulation Analysis	5
3.1	Simulated results	5
3.2	Simulated analysis	6
3.2.1	Coupling Capacitors	6
3.2.2	Bypass capacitor	6
4	Conclusion	7

1 Introduction

The objective of this laboratory assignment is to create an audio amplifier circuit by choosing the architecture of the Gain and Output amplifier stages whilst optimizing the circuit on its Merit (M), that is given by the equation:

$$M = \frac{\text{voltageGain} \times \text{bandwidth}}{\text{cost} \times \text{lowerCutoffFreq}}$$

$$\text{cost} = \text{cost}_{\text{resistors}} + \text{cost}_{\text{capacitors}} + \text{cost}_{\text{transistors}}$$

$$\text{cost}_{\text{resistors}} = 1\text{MU}/\text{kOhm}; \text{cost}_{\text{capacitors}} = 1\text{MU}/\mu\text{F}; \text{cost}_{\text{diodes}} = 0.1\text{MU}/\text{transistor}$$

The circuit studied, displayed in figure 1 utilizes the following components:

- two voltage sources (V_1 and V_{cc})
- two transistors (Q_1, Q_2)
- six resistors ($R_1, R_2, R_e, R_c, R_{in}, R_{out}$)
- three capacitors (C_i, C_b, C_o)

Theoretical and simulation analysis are presented in Section 2 and Section 3, respectively, and the results of each are then compared. Finally, in Section 4 the conclusions of the laboratory assignment are outlined.

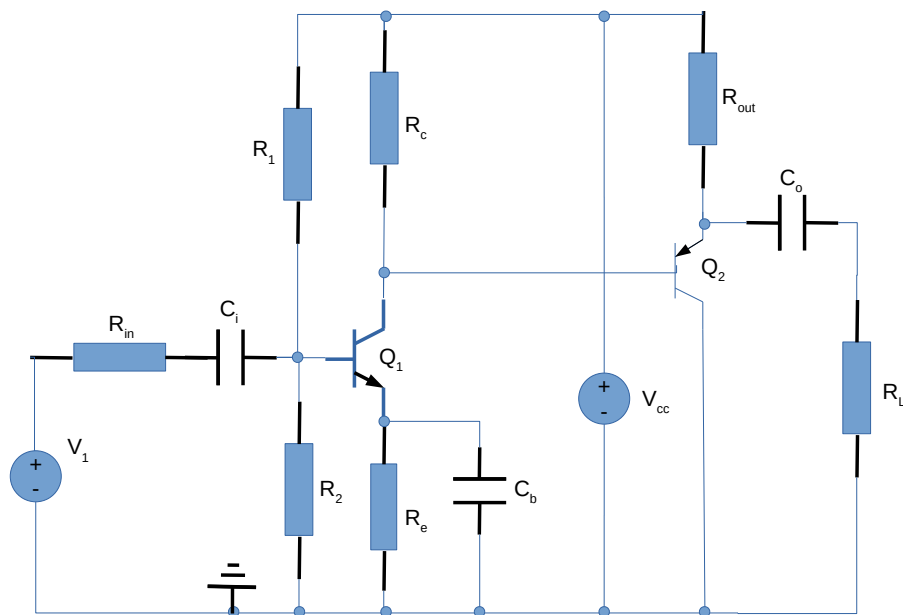


Figure 1: Circuit T4

2 Theoretical Analysis

In this section we will explain the inner workings of the class A audio amplifier created and perform a theoretical analysis of each component. The common Kirchoff laws and the simplified BJT equation were used to perform the analysis.

2.1 Gain stage - Degenerated Common Emitter Amplifier

It is in this stage of the amplifier that most of the signal amplification takes place. This stage is comprised of a NPN transistor, two capacitors (coupling and bypass) and four resistors (two for the voltage divider, R_1 and R_2 , one for the temperature stabilization R_E and another to set the no signal current, R_C).

The base voltage of the transistor needs to be forward biased in order for the NPN transistor to work in the forward-active region. For that reason, a voltage divider is added to make sure that the base voltage is always larger than $0.7V$. A coupling capacitor is added between the voltage divider and the signal source in order to add the input AC signal to the voltage divider DC signal (this will be explained in more detail in the simulation section).

The resistor R_C is used to set the zero signal current that passes through the BJT.

The resistance R_E adds temperature stabilization and removes the temperature dependency of the gain in the circuit but has the drawback of decreasing the AC gain of the amplifier. A bypass capacitor is used to decrease the gain degenerating effects of R_E on higher frequencies (this will be explained in more detail in the simulation section).

By analysis of the circuit using the mesh method, it is possible to find expressions for the input and output impedances of this stage. The derivation of these equations was performed in lecture 16.

Input impedance:

$$Z_i = \frac{(r_o + R_C + R_E)(R_B + r_\pi + R_E) + g_m R_E r_o r_\pi - R_E^2}{r_o + R - R_E}$$

Output impedance:

$$Z_o = Z_x || R$$

$$Z_x = r_o \frac{\frac{1}{R_E} + \frac{1}{r_{pi} + R_B} + \frac{1}{r_o} + \frac{g_m r_\pi}{r_{pi} + R_B}}{\frac{1}{R_E} + \frac{1}{r_{pi} + R_B}}$$

The Common Emitter has the drawback of having high output impedance. For that reason the Output stage is added as way of reducing the output impedance and to supply more current to the load.

2.2 Output stage - Common Collector Amplifier

The output stage is comprised of a PNP transistor, a resistor and a capacitor.

This stage has a gain level of almost unity so it is not responsible for the amplification. The main purpose is to decrease the output impedance to work better with the 8Ω load and allow larger currents to flow.

The capacitor is used to remove the DC component of the signal and only allow for the AC signal to reach the output terminals.

By analysis of the circuit using the node method, it is possible to find expressions for the input and output impedances of this stage. The derivation of these equations was performed in lecture 17.

Input impedance:

$$Z_i = \frac{g_\pi + g_E + g_o + g_m}{g_\pi(g_\pi + g_E + g_o)}$$

This value is higher than the output impedance of the gain stage. Since both the stages are connected in series, they follow the voltage divider law. For that reason, to achieve as large as possible gain, it is important that the output impedance of the gain stage is smaller than the input impedance of the output stage.

Output impedance:

$$Z_o = \frac{1}{g_{pi} + g_E + g_o + g_m}$$

This value is low so it can be connected to the load efficiently.

2.3 Circuit frequency response

Due to the use of capacitors in the circuit, the gain is dependent on the frequency of the input. This response was achieved by calculating the equivalent impedance of the branches with capacitors for a set of frequencies.

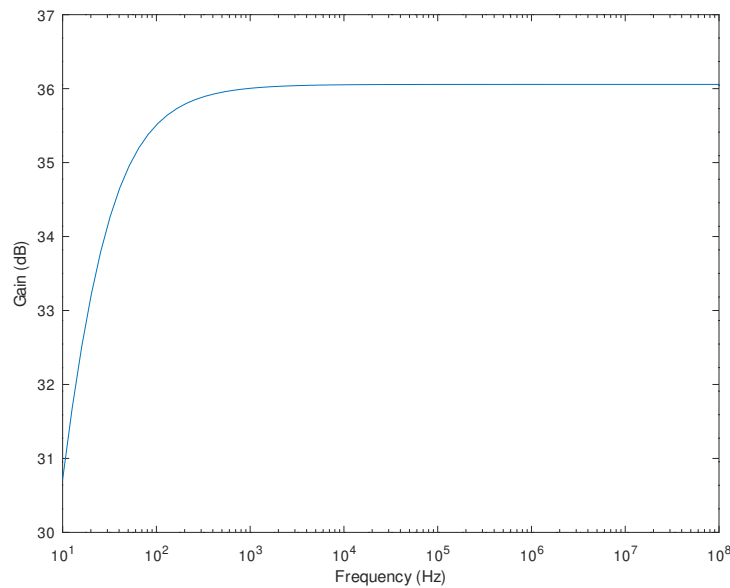


Figure 2: Frequency Response V_o/V_i .

2.4 Theoretical Results

The obtained results were as follows:

Name	Value
V_e	1.093271e+00
V_c	9.912854e+00
V_{ce}	8.819583e+00
V_{ec2}	1.061285e+01

Table 1: Values from Octave.

Name	Value[Ohm]
$Z1_{In}$	7.822336e+02
$Z1_{Out}$	3.728356e+02
_____	_____
$Z2_{In}$	2.725047e+03
$Z2_{Out}$	2.653788e-01
_____	_____
Zt_{In}	7.822336e+02
Zt_{Out}	1.682064e+00

Table 2: Values from Octave.

3 Simulation Analysis

In this section, Circuit T4 is reproduced with the help of Ngspice.

Ngspice is a simulator for electronic circuits that can output a variety of results. This emulator computes the voltages in every node, as well as the potential difference between two given nodes. Apart from that, the group made use of the command `.options savecurrents` which also enables the use of the currents that pass through all branches. Moreover, function to help determine the maximum and interception of the plots were also used.

Firstly, the outcome of the simulation is shown, as well as a brief explanation on how it was achieved. Afterwards, a comparison is done between those values and the ones attained in Section 2.

3.1 Simulated results

In this laboratory assignment, the Ngspice script made use of the same values considered for the Octave script.

Figure 3 displays v_{out} . This voltage signal has no visible distortion of the input sine waves.

Figure 4 shows v_{coll} .

Figure 3 displays v_{out} .

Table 3 shows important values to make sure the BJTs operate in the F.A.R.

Name	Value
v(coll)	1.005359e+01
v(emit)	1.117079e+00
v(emit2)	1.086574e+01
v(emit2)-v(coll)	8.121456e-01
v(coll)-v(base)	8.244599e+00
v(coll)-v(emit)	8.936511e+00
v(base)-v(emit)	6.919123e-01

Table 3: Checking values for F.A.R.

Table 4 displays the total impedances of the circuit (In and Out).

Name	Value[Ohm]
Z_{in}	923.912
Z_{out}	3.49807

Table 4: Total impedance values from Ngspice.

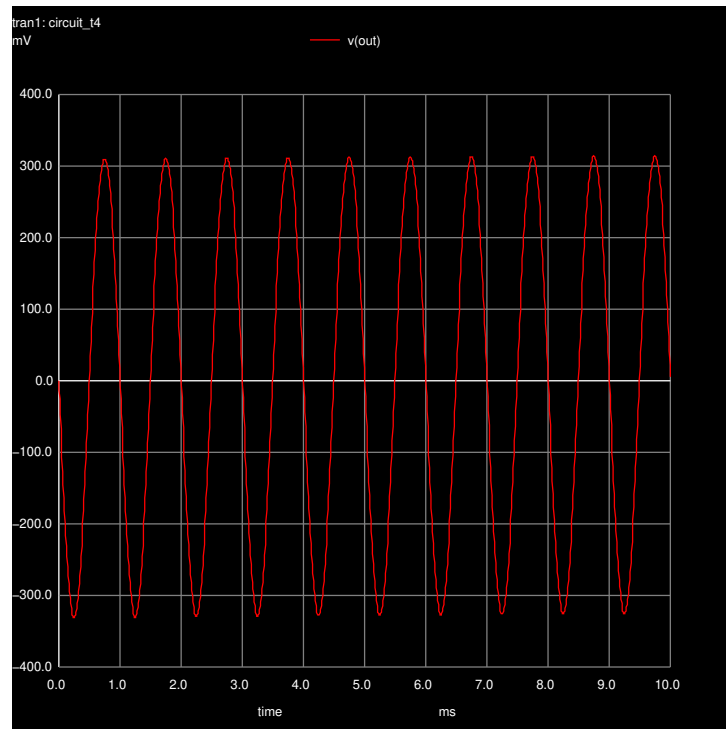


Figure 3: v_{out}

Lastly, the group also used Ngspice to compute the Merit. Table 5 shows all the values necessary to compute the Merit, as well as the Merit itself.

Name	Value
vgain	3.262849e+01
bandw	3.118611e+06
cost	8.160017e+03
lcof	7.972204e+00
merit	1.564187e+03

Table 5: Merit and other variables.

3.2 Simulated analysis

3.2.1 Coupling Capacitors

The purpose of the coupling capacitors on the audio amplifier circuit is to eliminate the DC component of a signal and only pass the AC component, which in the case of an audio amplifier is what carries the sound information.

This arrangement of capacitors is normally used to link/couple (that's where it's name comes from) only the AC signal of one circuit to another.

3.2.2 Bypass capacitor

A bypass capacitor's purpose is to remove the AC component of a signal by shorting it to ground. For this to work, the bypass is placed in parallel with a resistance. Since the capacitor's impedance follows the equation $|Z_c| = 1/\omega C$, for DC signals ($\omega = 0$) the impedance is infinite

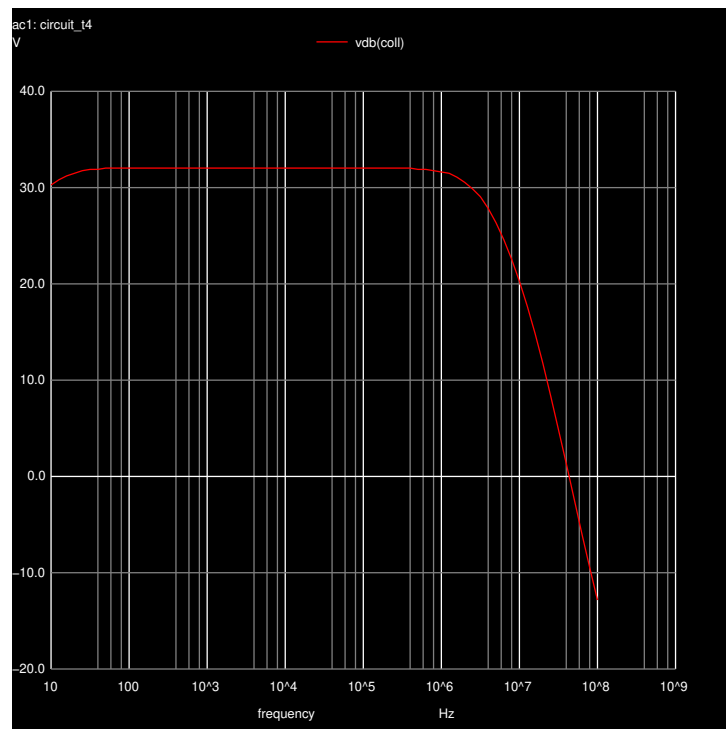


Figure 4: v_{coll} in dB

and so all the current flows through the resistor. When the frequency increases, the capacitor's impedance decreases and so a larger part of the signal passes through the capacitor directly to ground. This way, the DC signal is the main contributor to the voltage difference between the components, while the AC signal is directed to ground without any resistance.

In our application, the bypass capacitor is used to allow to have the resistance R_E without degenerating the gain. The resistance R_E adds temperature stabilization and removes the temperature dependency of the gain in the circuit but has the drawback of decreasing the AC gain of the amplifier. For this purpose, only the DC component needs to be affected by the resistance R_E and for this reason, the bypass capacitor is used so that higher frequencies are not affected by R_E . This way we can achieve the best of temperature stabilization without gain degeneration.

4 Conclusion

In order to perform theoretical and simulational analysis of the circuit Octave and Ngspice were used, respectively.

Theoretical methods were used to compute the gain, impedances and frequency response of both of the stages. Contrary to past lab assignments the theoretical results differ a lot from the simulated results.

Comparing the frequency response graphs we can see that they are quite different. The most noticeable difference is that the theoretical method does not predict the higher cutoff frequency. This is explained by the fact that the theoretical method considers all the components to be ideal when in reality all of the components have some residual capacitive characteristics which can become noticeable at really high frequencies. In addition it is possible that the BJT model simulates the speed at which a transistor can be responsive.

The impedance and gain also differ significantly.

In conclusion, our simulated circuit was able to achieve a decent gain and Merit value and so we considered it a success. In addition we were able to understand the functioning principles of

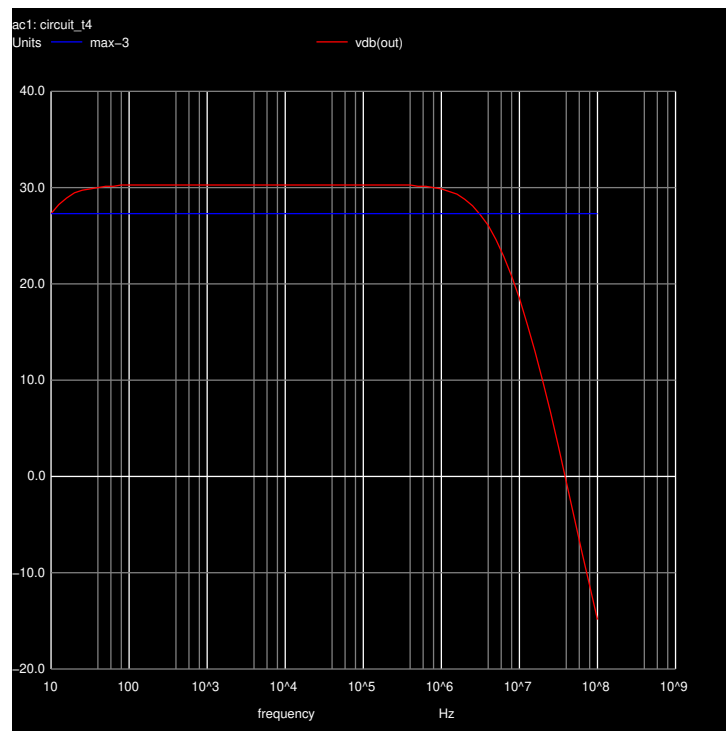


Figure 5: v_{out} in dB

a class A amplifier but also were exposed to the difficulty of choosing parameters that optimize the results of complex circuits.