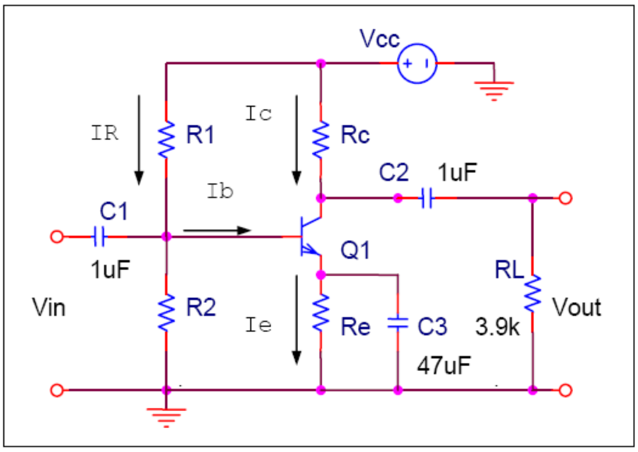
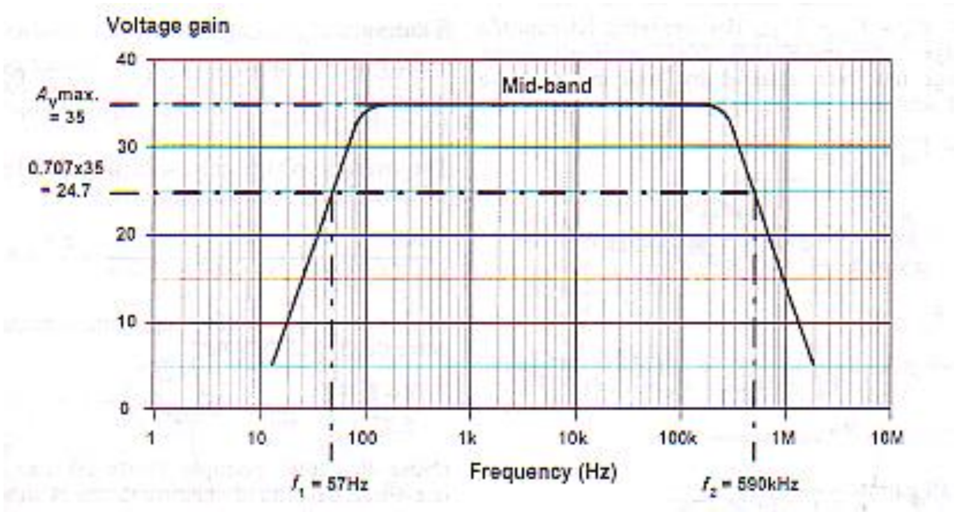
Xi'an Jiaotong Liverpool University

Department of Electrical and Electronic Engineering

EEE109 Lab 3 (S1)

FREQUENCY RESPONSE OF A BJT AMPLIFIER





<u>REMEMBER TO OBEY THE LAB SAFETY RULES TO AVOID INJURY AND</u>

<u>READ THROUGH THE SCRIPT AND UNDERSTAND IT BEFORE YOU GO INTO THE LAB!</u>

1. Objectives

1) To calculate the value of the resistors Rc, Re, R1 and R2 for the common emitter and common collector amplifiers circuits.
2) To measure upper and lower cutoff frequencies of a common-emitter amplifier (CE) and common collector (CC).
3) To simulate amplifier frequency response measurements using PSPICE software.
4) To study the frequency response of the common emitter (CE) and common collector (CC) BJT transistor amplifiers in the low to high frequency range (10 Hz to 300GHZ).

2. Components

1) Transistor - BJT 2N3904
2) Resistor R_L=3.9 kΩ
3) Resistors R_c, R_e , R_1 and R_2 according to prelab calculations
4) Capacitors 2 x 1μF, 1 x 47 μF.

3. Plan for the lab:

Before the Lab: Read through the lab script beforehand and calculate the value of the resistors Rc, Re, R1 and R2 for the common emitter and common collector amplifiers circuits. You are required to use PSpice Software to simulate the CE or CC amplifier circuits.

After the Lab: You are required to complete **all** PSpice simulations and complete all the outstanding calculation.

---

4. Introduction

In this lab, two BJT amplifier configurations will be investigated: the common-emitter, and the common collector amplifier. Both amplifiers typically use a self-biasing scheme and have a relatively linear output.

4.1. Common-Emitter Amplifier

The common emitter amplifier in Figure 1 is characterized by high voltage (Av) and current gain (Ai). The amplifier typically has a relatively high input resistance (1 - 10 KΩ) and a fairly high output resistance. Therefore it is generally used to drive medium to high resistance loads. The circuit for the common-emitter amplifier can be seen in Figure 1. It is typically used in applications where a small voltage signal needs to be amplified to a large voltage signal. Since the amplifier cannot drive low resistance loads, it is usually cascaded with a buffer that can act as a driver.

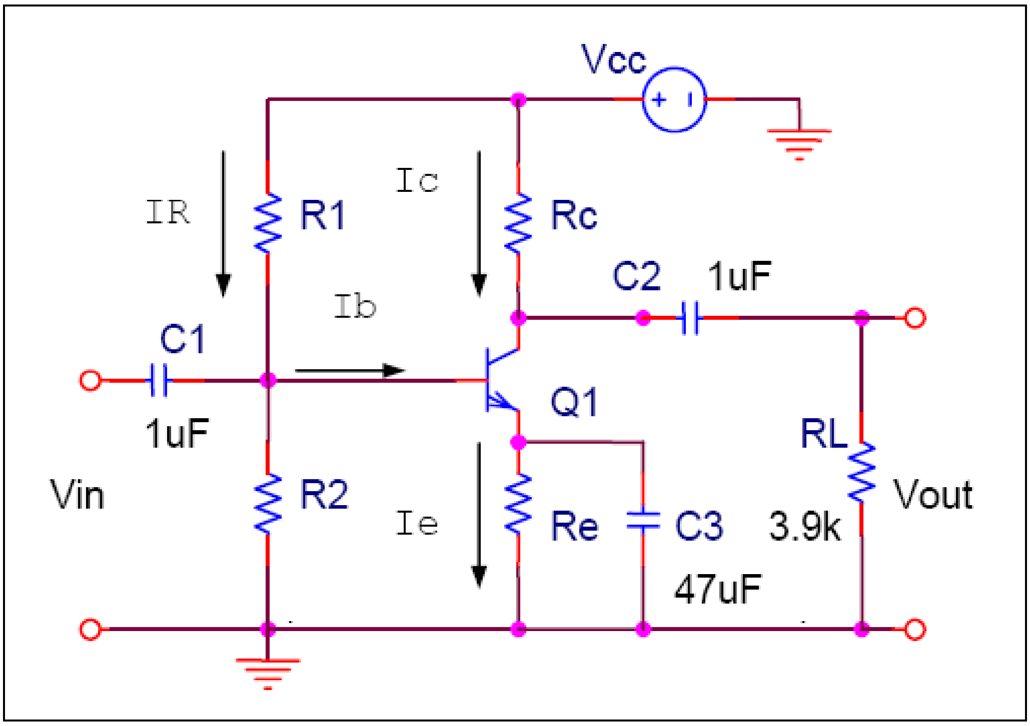


Figure 1: The common emitter amplifier

4.2 Common-Collector Amplifier

The common-collector (CC) amplifier, or the emitter-follower as it is sometimes called, is a unity voltage gain, high current gain amplifier. The input resistance for this type of amp is usually 1KΩ to 100KΩ. A typical CC amp can be seen in Figure 2. Because the amplifier has a voltage gain of one, it is useful as a buffer amplifier, providing isolation between two circuits while providing driving capability for low resistance loads.

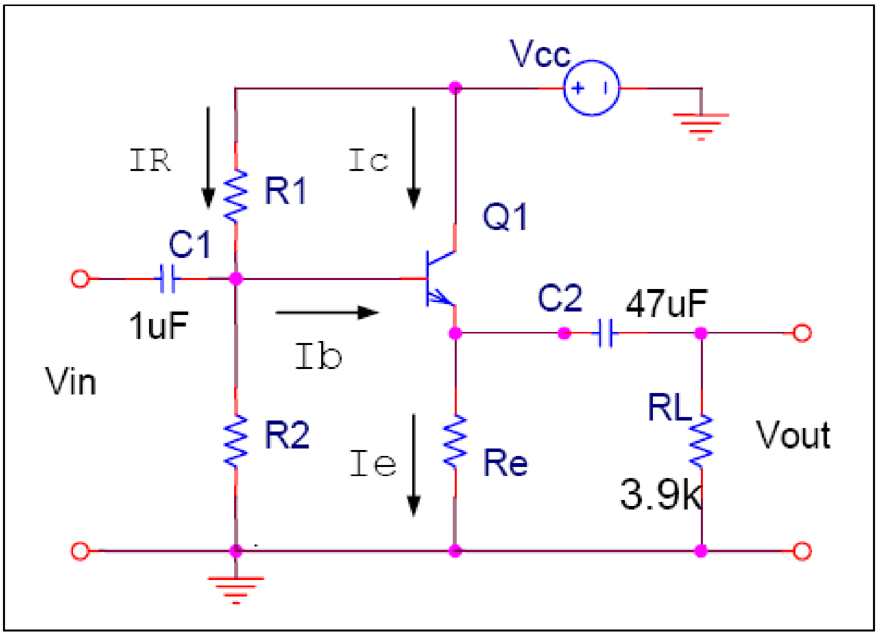


Figure 2: The common collector amplifier

5. PSPICE Simulations

5.1 Common-Emitter amplifier

1) Let's find β of the transistor (details see appendix). You will use this value for determining the unknown resistor values.

2) For the circuit in Figure 1 calculate the values of R_c, R_e, R_1 and R_2. DC Bias this circuit for V_{cc} = 10V, V_{ce} = 5V and I_c = 5mA.

The resistors R_1 and R_2 form a potential divider, which will fix the base potential of the transistor. The current I_R through the R_1 is usually set at least to 10 times the base current I_b required by the transistor. The base emitter voltage drop of the transistor is approximated as 0.7 volts. There will also be a voltage drop across the emitter resistor R_e. The inclusion of this resistor also helps to stabilize the bias: If the temperature increases, then extra collector current will flow. If I_c increases, then so will I_e as $I_e = I_b + I_c$. The extra current flow through R_e increases the voltage drop across this resistor reducing the effective base emitter voltage and therefore stabilizing the collector current.

Assume that $V_e = V_{cc}/10$ and $I_R = 10I_b$ and use Equations (1) to (5) to obtain Rc, Re, R_1 and R_2 values. For your Lab setup choose the closest standard resistor values.

$$V_{cc} = I_c R_c + V_{ce} + I_e R_e \tag{1}$$

$$I_e = I_b + I_c \quad \text{as} \quad I_c >> I_b, \quad \text{then} \quad I_c \sim I_e \tag{2}$$

$$V_b = V_e + 0.7 \tag{3}$$

$$R_2 = \frac{V_b}{9I_b} \tag{4}$$

$$R_1 = \frac{V_{cc} - V_b}{I_R} = \frac{V_{cc} - V_b}{10I_b} \tag{5}$$

3) Calculate the voltage gain, the current gain, the input resistance for this amplifier. All the calculations must be shown.

4) Simulate the above circuit in PSPICE using standard resistors values and attach the bias point results. For this you must show the values of the all the bias currents and voltages on your schematic.*

5) Using the PSPICE ***AC analysis*** function to obtain the gain and phase frequency response for this amplifier from 10Hz to 10 GHz and find the 3dB point. Attach the results and plots on your lab report.** Attach the print-out of the DC quiescent point values and the Bode plots of the magnitude (in dB) and phase angle (in degrees) of the gain ratio into the lab report. Please provide the comments based on your simulation results.

Attention: You must plot the Bode plots, i.e. the ratio of the output voltage over the input voltage!

<u>PSPICE simulations tips:</u>

* To provide a power supply to the circuit, use the “Battery” source from the *PSPICE* library and set it to a 10V value .

** For a sine wave signal source (used for simulating the Vin), use a Vac= **0.02**(V)

** To obtain the gain and phase frequency response plots for this circuit, you must run “AC ANALYSIS”. To get best results for your plots, set the AC Analysis Limits as follows:

f [Hz]	V_{in} [V]	V_{out} [V]	Θ[deg]	A_V[dB]
Selected frequencies				

$$A_V(dB) = 20\log\left|\frac{V_{out}}{V_{in}}\right|$$

5.2 Common-Collector amplifier

1) For the circuit shown in Figure 2 calculate the value of R_e, R_1 and R_2. Bias this circuit for V_{cc} = 10V, V_{ce} = 5V and I_c = 5mA.

The procedure for calculation of R_e, R_1 and R_2 values is very similar to that used for common-emitter amplifier. The current I_R through the R_1 is usually set at 10 times the base current I_b required by the transistor. The base emitter voltage drop of the transistor is approximated as 0.7 volts. There will also be a voltage drop across the emitter resistor R_e.

Assume that $I_R = 10I_b$ and use Equations (6) to (10) to obtain R_e, R_1 and R_2 values. For your Lab setup choose the closest standard resistor values.

$$V_{cc} = V_{ce} + I_e R_e \quad \textbf{(6)}$$

$$I_e = I_b + I_c \ \text{ as } \ I_c >> I_b, \ \text{ then } \ I_c \sim I_e \quad \textbf{(7)}$$

$$V_b = V_e + 0.7 \quad \textbf{(8)}$$

$$R_2 = \frac{V_b}{9I_b} \quad \textbf{(9)}$$

$$R_1 = \frac{V_{cc} - V_b}{I_R} = \frac{V_{cc} - V_b}{10I_b} \quad \textbf{(10)}$$

2) Calculate the voltage gain, the current gain, the input resistance, and the output resistance for this amplifier. All the calculations must be shown.

3) Simulate the above circuit in PSPICE using standard resistors values and attach the bias point results. For this you must show the values of the all the bias currents and voltages on your schematic.*

4) Using the PSPICE AC analysis function, obtain the gain and phase frequency response for this amplifier from 1Hz to 300GHz and find the 3dB point. Attach the results and plots on your lab report.** Attach the print-out of the DC quiescent point values and the Bode plots of the magnitude (in dB) and phase angle (in degrees) of the gain ratio into the lab report. Please provide the comments based on your simulation results.

Attention: You must plot the Bode plots, i.e. the ratio of the output voltage over the input voltage!

PSPICE simulations tips:

** For a sine wave signal source (used for simulating the Vin), use a Vac= **1** (V)

** To get better results for your plots set the AC Analysis Limits as follows:

f [Hz]	V_{in} [V]	V_{out} [V]	Θ[deg]	A_V[dB]
Selected frequencies				

$$A_V(dB) = 20\log\left|\frac{V_{out}}{V_{in}}\right|$$

6. Important Notice on Lab 3 Arrangement

1) Please obey the **lab rule** and **safety regulations** when working in the computer lab.
2) Please read the lab script before the lab.

6) The lab is conducted in groups. The **group size** is **TWO**. Please follow the same team arrangement (same partner) as in Lab #1and Lab #2. If you change the partner, please notify our Chief TA (Fei CHENG, fei.cheng@xjtlu.edu.cn).

7) Only **ONE report** is required **per group**. DO NOT submit twice on ICE - *Nominate one student to submit on ICE.*

8) Please use the **template** provided for your lab #3 report (attached along with the lab script on ICE).

9) **Team work** and **individual efforts** are assessed in Lab #3 as each student in a team will be in charge of one specific amplifier configuration (either common-emitter or common-collector). Please refer to the marking scheme in Appendix B of the report template.

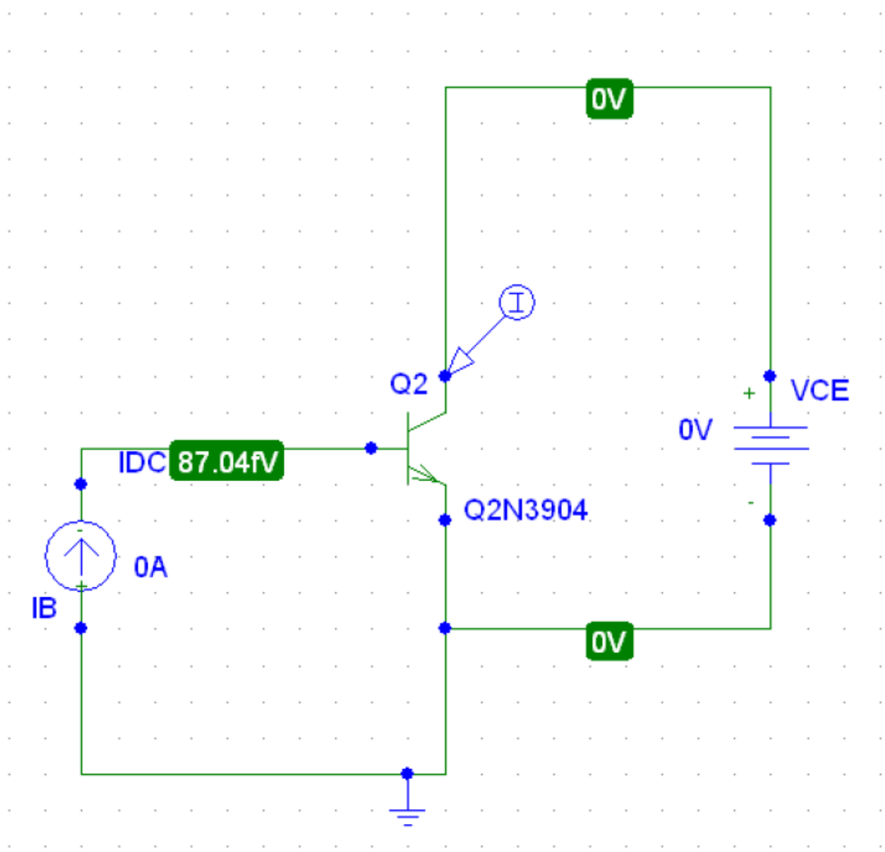
10) The lab report is due by **Dec. 18, 2016 (Sunday, Week 14)**.

Appendix

Transistor Output Characteristics

To obtain a plot of the output characteristics of the 2N3904 transistor i.e. I_C vs V_{CE} (0 to 10 V in steps of 1V) for a range of I_B (0 to 40uA in steps of 5uA). First, input the circuit schematic below. To obtain the family of characteristics in one simulation, pull down the 'Analysis' window, select 'Setup..' check the 'DC sweep' option and input the variable as 'Vce' with the desired range. Use the 'Nested sweep' option to set the base current steps, I_B. Thus for each value of I_B (0, 5, 10.....40 uA), V_{CE} is swept from 0 to 10 V.

Calculate the d.c. current gain, beta (also known as h_{FE}) at $I_C \sim 5$ mA.


0V
Q2
VCE
0V
IDC
87.04fV
Q2N3904
0A
IB
0V