EEE109 Lab 3

Frequency Response of A BJT Amplifier

Group members: Kai-Yu Lu

Student IDs: 1614649

Group ID: 72

Submission date: 02/12/2017

Abstract

Amplifier is a significant component in modern microelectronics. Common-emitter amplifier is one of the application of the transistor. In order to further study the frequency response of the common-emitter amplifier, we choose NPN transistor as the subject investigated and use PSPICE to simulate it. Subsequently, we could understand the basic characteristics of the common-emitter transistor. Before the lab, students should read through lab script. During the experiment, students are encouraged to execute requirements of the experiment. After the experiment, students should arrange the data and write a report. Through this experiment, students could learn how to use the PSPICE to simulate the experiment, which might improve the future learning efficiency. This report will give introduction, theory and the circuit design. Certain experimental method results and discussion will be also provided.

1. Introduction

Common-emitter circuit is a wide application used in the amplifier circuit. Due to the input signal of the common-emitter amplifier is from the base and the emitter and the output signal is from the emitter, therefore, this design is called common-emitter amplifier. In addition, the emitter is connected to the ground. This circuit is normally used to amplify the small voltage to the application which requires the large voltage. During this experiment, we will find β which is the parameterized relationship between the current in the base and the current in the collector. Then we will use β measure and calculate the frequency response.

2. Theory

During this experiment, we will apply Thevenin Theorem the simplify the circuit for convenience for calculating. The value of β should be 50-300. In Figure 1, it is the tested response curve in this experiment. There are three stages in the frequency response which are low-frequency range, midband and high-frequency range. In more detail, we call the low-frequency range midband and high-frequency range when the condition satisfies $f_0 < f_L$, $f_L < f_0 < f_H$ and $f_0 > f_H$ correspondingly.

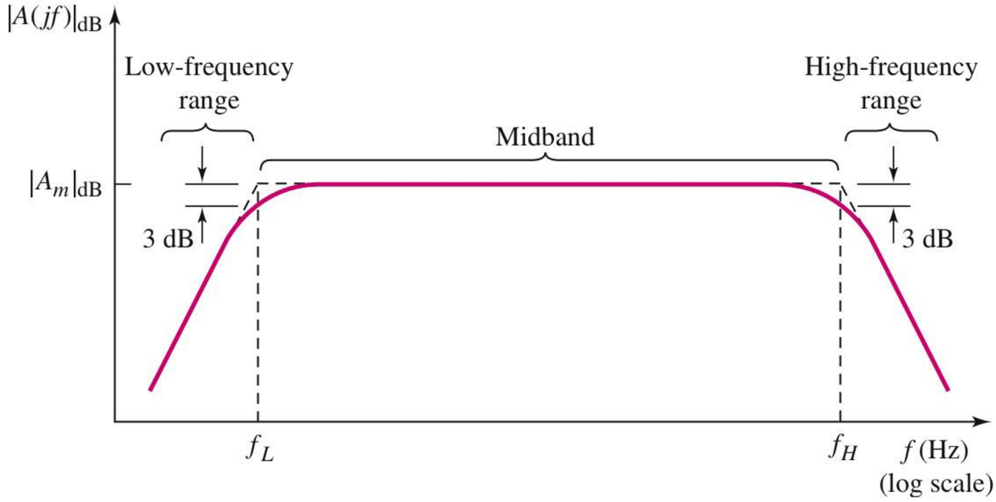

Figure 1: The curve of the gain of amplifier with changing frequency [1]

3. Circuit design

The first task was to calculate β. The circuit we should construct is shown in Figure 2.

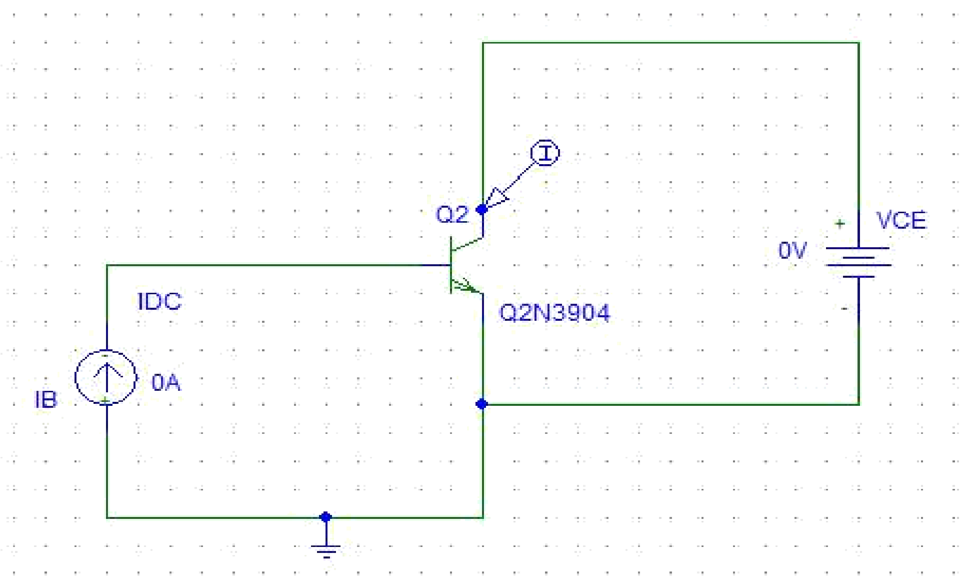

Figure 2: The circuit to find β

The next task was to construct the common-emitter circuit as it can be indicated in Figure 2.

4. Experimental and Method

1) Construct the circuit as shown in Figure 1. Then we needed to set the parameters for the circuit and these procedures could be referred in Figure 3.

- 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 ~ 5 mA.

** The unit 'µA' is represented by 'uA' in PSpice.*

Figure 3: The steps to set parameters [2]

2) Calculate β.

3) Construct the common-emitter circuit as indicated in Figure 4.

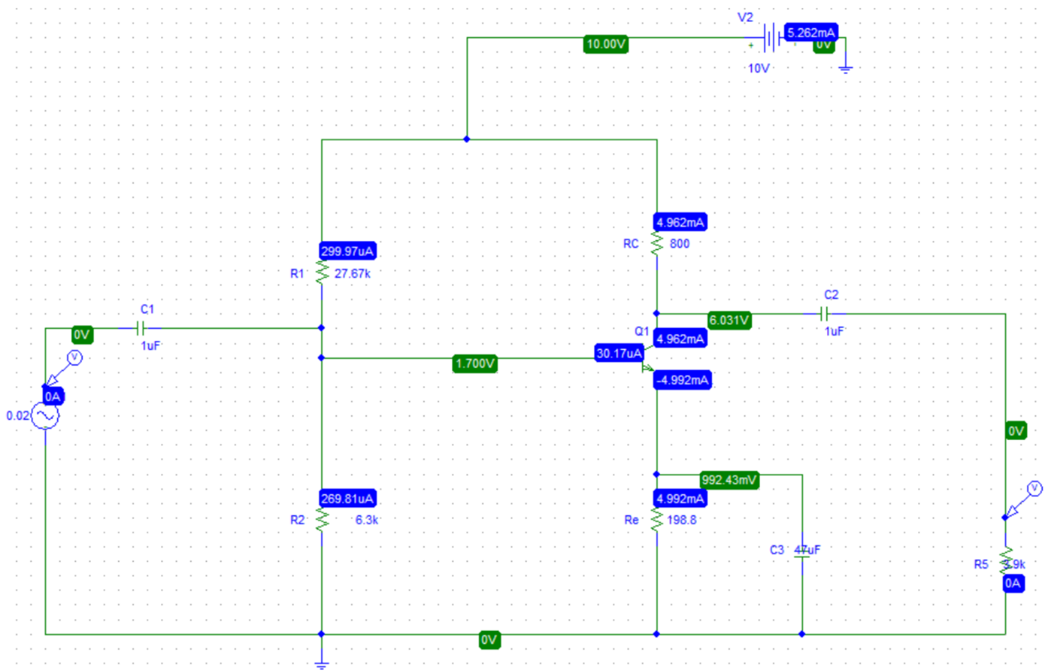

Figure 4: The circuit of the common emitter

4) Use the PSPICE AC analysis function to obtain the gain and phase frequency response for common-emitter amplifier form 10Hz to 10GHz and find the 3db point [2].

5) Add two markers to measure the phase of input voltage and output voltage. Click ‘Simulate’ and add Y axis and add trace. For detail, it can be seen in Figure 5.

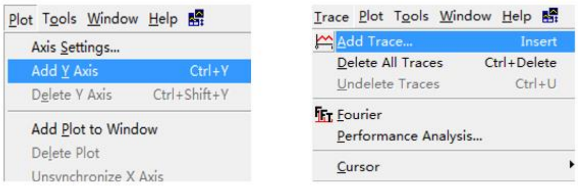

Figure 5: The detailed steps.

6) Use ‘toggle cursor’ to obtain the 3db points at the curve of voltage gain A_V in db.

5. Results and calculation

The result of the first task is shown in Figure 6.

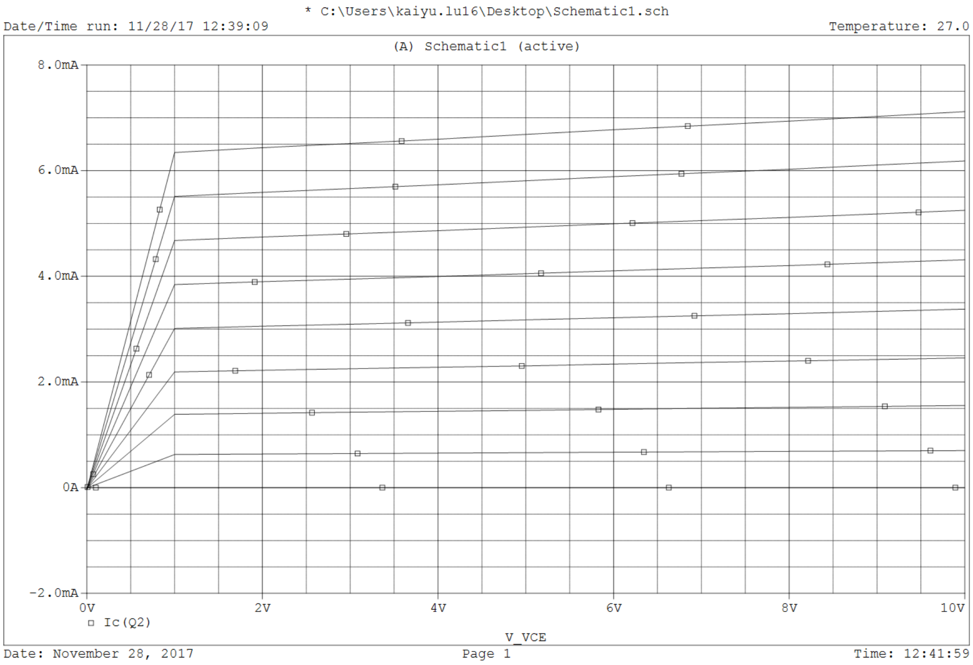

Figure 6: The output characteristics of the transistor

Assumed that V_{CE} = 5V and $I_C = 5$ mA, therefore, we could find the specific line for the output characteristics to find I_B which was 30 uA and calculate β.

$$\beta = \frac{I_C}{I_B} = \frac{5\ mA}{30\ uA} = 167 \tag{1}$$

Due to the given condition provided be the lab materials, we could know the value of V_{CC}, V_{CE}, I_C. By the formulas below, we could calculate R_1 and R_2.

$$I_C = \beta I_B \tag{2}$$

$$V_{CC} = I_C R_C + V_{CE} + I_E R_2 \tag{3}$$

$$I_C = I_E \tag{4}$$

$$V_B = V_E + 0.7 \tag{5}$$

$$R_2 = \frac{V_B}{9I_B} = 6.3\text{k}\Omega \tag{6}$$

$$R_1 = \frac{V_{CC} - V_B}{I_R} = \frac{V_{CC} - V_B}{10 I_B} = 27.67\text{k}\Omega \tag{7}$$

Considering I_E is approximately equal to I_c due to I_B is much less than I_E. Therefore, we could also calculate R_C and R_E.

$$V_{RC} = V_{CC} - V_C = 4\text{V} \tag{8}$$

$$R_C = \frac{V_{RC}}{I_C} = 800\Omega \tag{9}$$

$$R_E = \frac{V_{RE}}{I_E} = 199\Omega \tag{10}$$

Therefore, we could put the calculated resistors into the circuit as show in Figure 4. After obtaining β, we could complete the next step. The frequency response of the common-emitter amplifier could be simulated by PSPICE as it can be indicated in Figure 7.

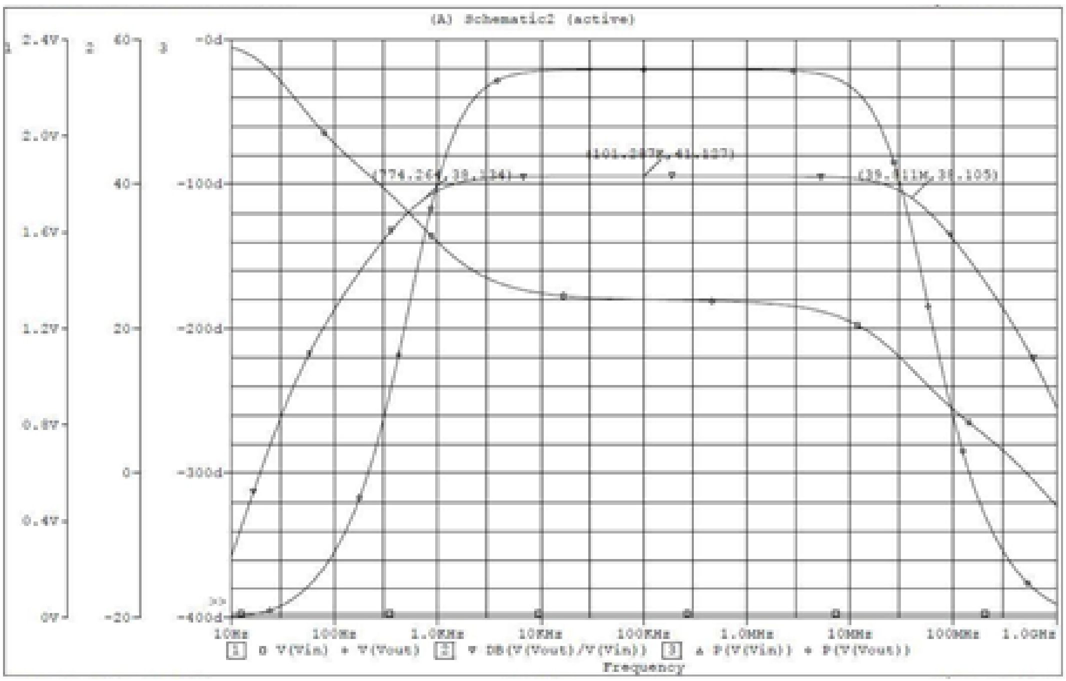

Figure 7: The frequency response of the common-emitter amplifier

In order to be more convenient to analyze Figure 7, we could draw Table 1.

f(Hz)	V_{in}(V)	V_{out}(V)	Θ(deg)	A_V(dB)
50	0.02	0.111	-47.513	14.813
75	0.02	0.192	-62.332	19.670
100	0.02	0.270	-71.705	22.640
200	0.02	0.565	-91.346	28.980
500	0.02	1.240	-117.767	39.113
1k	0.02	1.806	-139.843	40.532
2k	0.02	2.129	-157.675	41.102

10k	0.02	2.269	-175.362	41.102
20k	0.02	2.273	-177.698	41.122
50k	0.02	2.274	-179.144	41.127
100k	0.02	2.274	-179.687	41.128
200k	0.02	2.274	-180.069	41.128
500k	0.02	2.274	-180.665	41.126
1M	0.02	2.274	-181.467	41.128
7.10G	0.02	0.030	-336.000	3.000

6. Discussion

From the calculation of the first step, we accurately obtained β, because we selected the specific line when V_{CE} was equal to 5V and I_C was equal to 5 mA. From Figure 7, we could observe that f_L = 774.264 Hz and f_H = 39.811 MHz. Due to the increment of every parameters is not small enough, it might form the declinational curve. However, we could roughly find the cutoff frequency which is at -3 dB. In general, PSPICE could simulate the real experiment and obtain the accurate results.

7. Conclusion

From this experiment, we have learned how to calculate β when certain required parameters are known and simulate amplifier frequency response by using PSPICE. Through the simulation of PSPICE, we could simply find the -3 dB cutoff frequency of the transistor and three stages of the frequency response: the low-frequency range, midband and high-frequency range. Furthermore, we have learned how to use PSPICE to construct and simulate the circuit, which could improve our understanding and efficiency for our future learning. If we want to operate certain experiments with the limited condition, such as the problem of lacking materials or the time limitation, using PSPICE is a suggested method.

8. Reference

[1] *EEE109: Electronic Circuits Frequency Response- Capacitor Effect and Examples*. Department of Electrical and Electronic Engineering, XJTLU.

[2] *PSpice Guidance (Additional) - PSpice Settings.* Department of Electrical and Electronic Engineering, XJTLU.