Xi'an Jiaotong-Liverpool University
西交利物浦大學

EEE109 Lab 3: Frequency Response of A BJT Amplifier

Additional Lab Instructions

Safety at the Lab

- **REMEMBER TO OBEY THE LAB SAFETY RULES TO AVOID INJURY**

- **READ THROUGH THE SCRIPT AND UNDERSTAND IT BEFORE YOU GO INTO THE LAB !**

Components

1. Transistor - BJT 2N3904
2. Resistor R_L = 3.9 kΩ
3. Resistors Rc, Re , R1 and R2 according to prelab calculations
4. Capacitors 2 x 1μF, 1 x 47 μF.

Objectives

- To calculate the **value** of the resistors Rc, Re, R1 and R2 for the **common emitter** and **common collector** amplifiers circuits.

- To measure upper and lower **cutoff frequencies** of a common-emitter amplifier (CE) and common collector (CC).

- To simulate amplifier **frequency response** measurements using **PSPICE** software.

- 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)**.

Plan for the lab

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

Amplifier Configurations

- 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.

Common-Emitter Amplifier

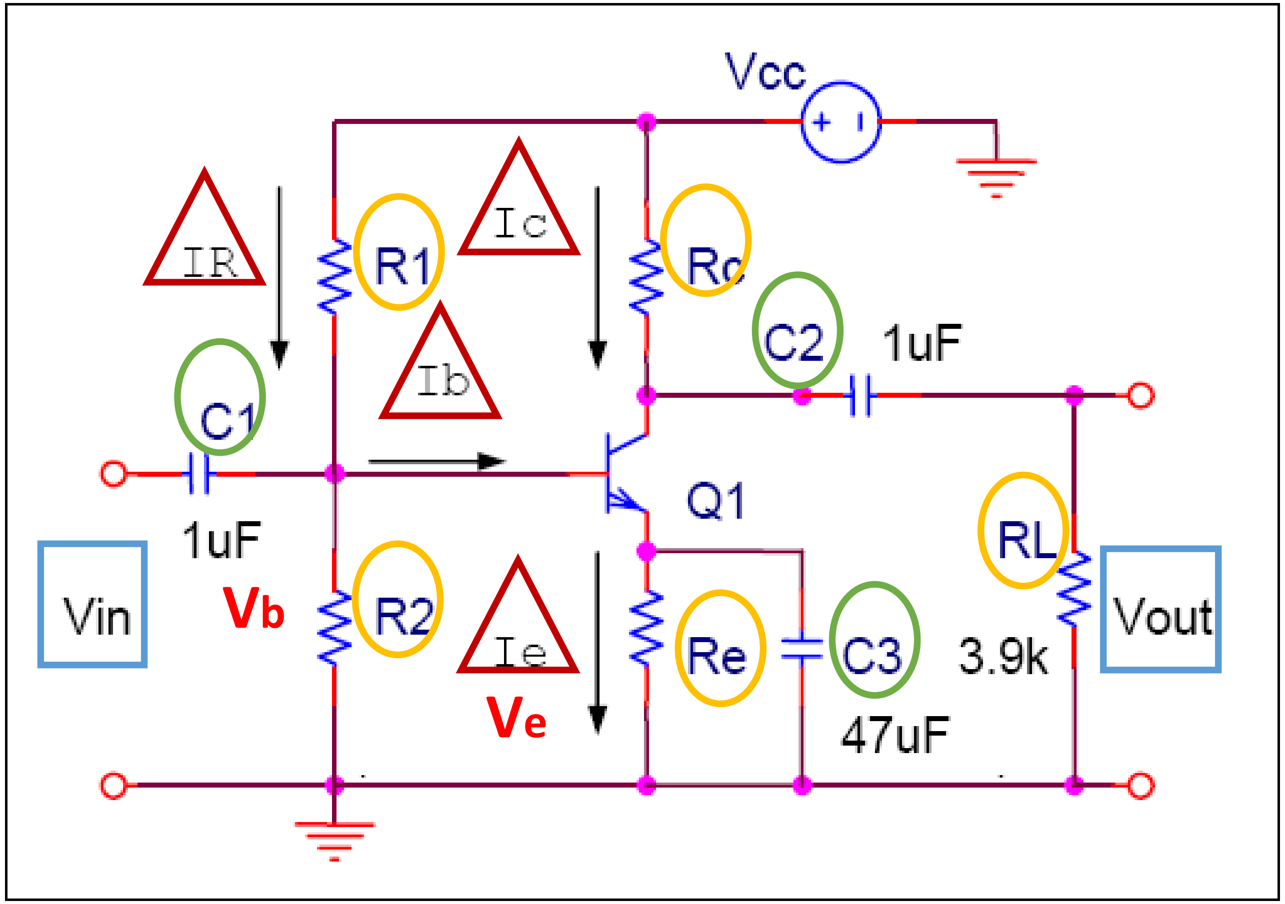


Figure 1

Common-Collector Amplifier

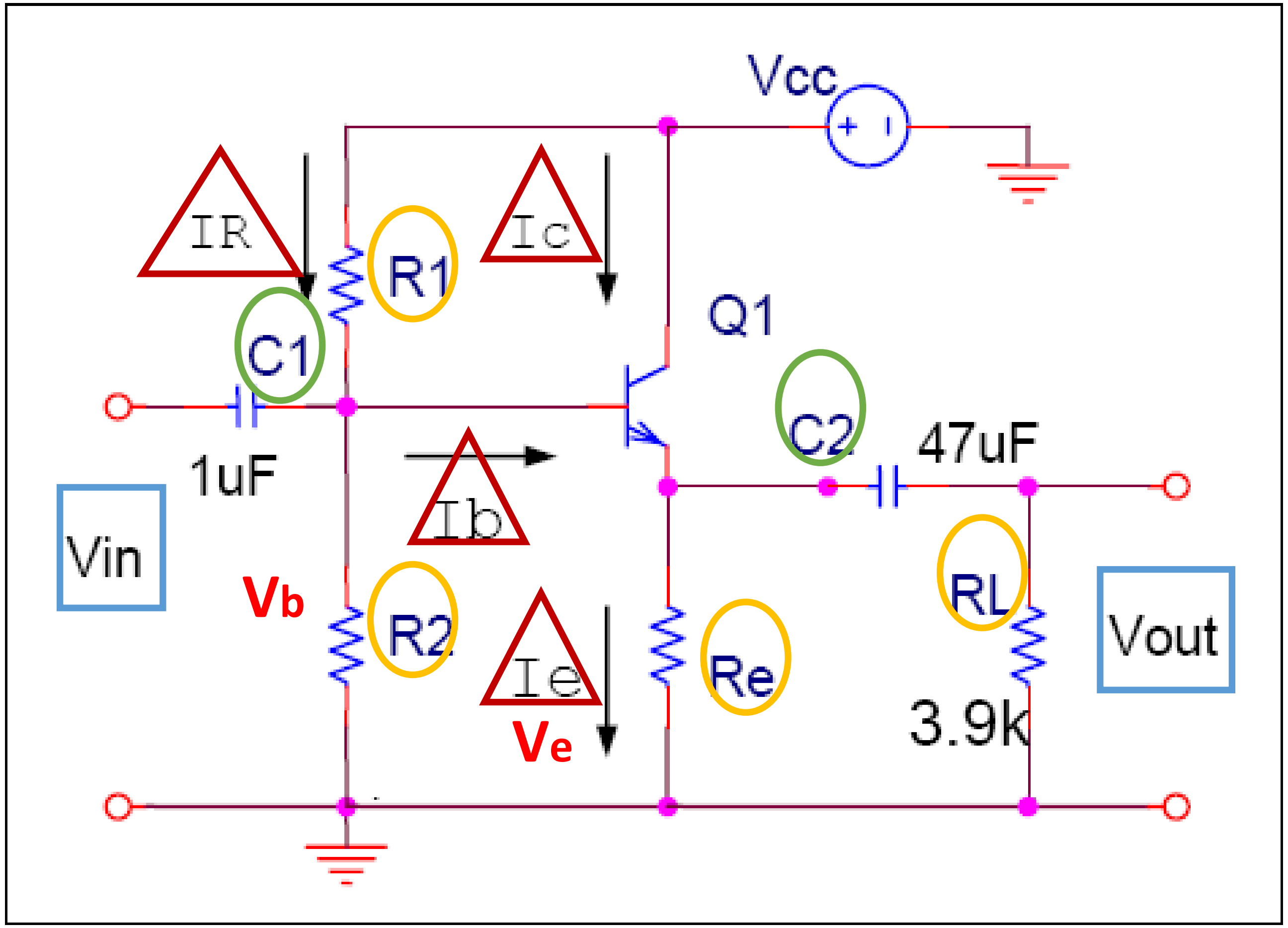


Figure 2

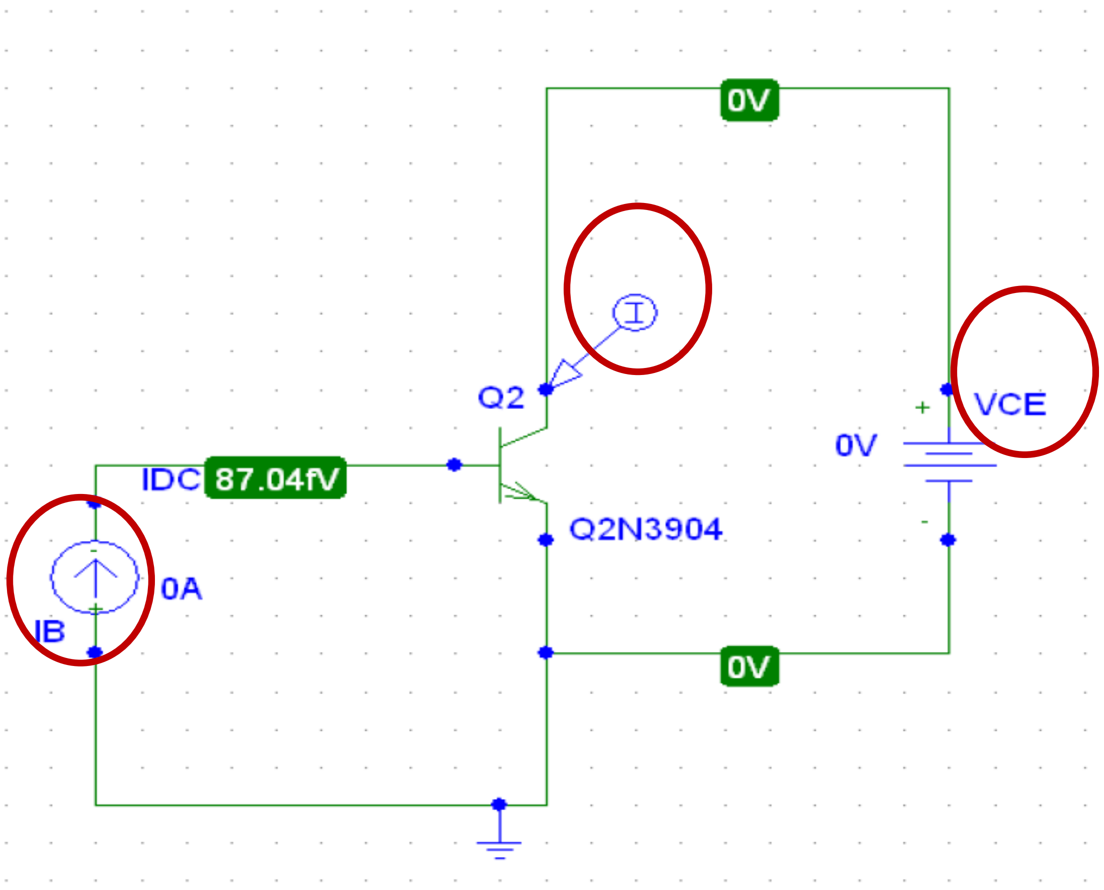
PSPICE Simulations – Common-Emitter Amplifier (1)

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

Transistor Output Characteristics (1)

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.

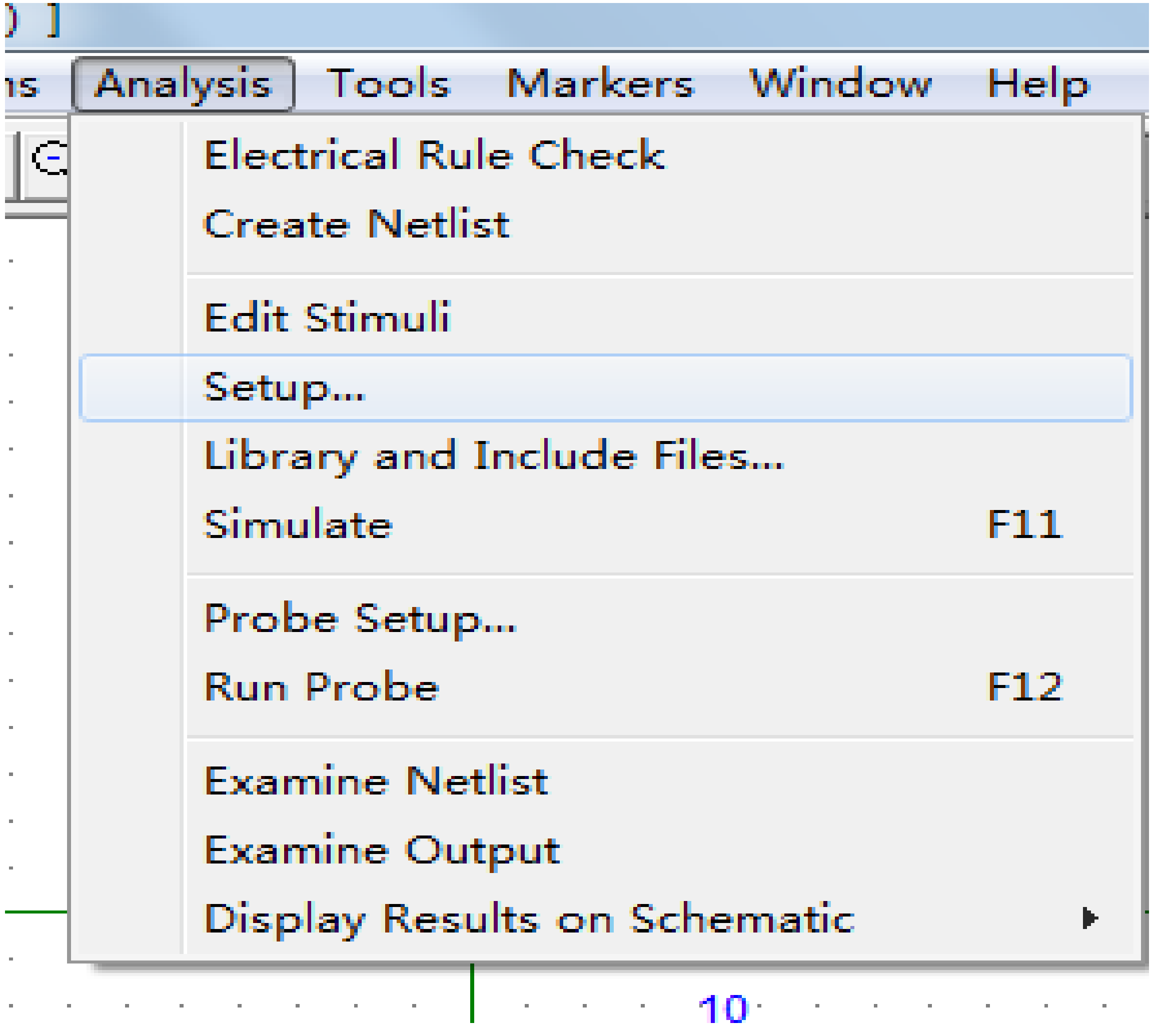


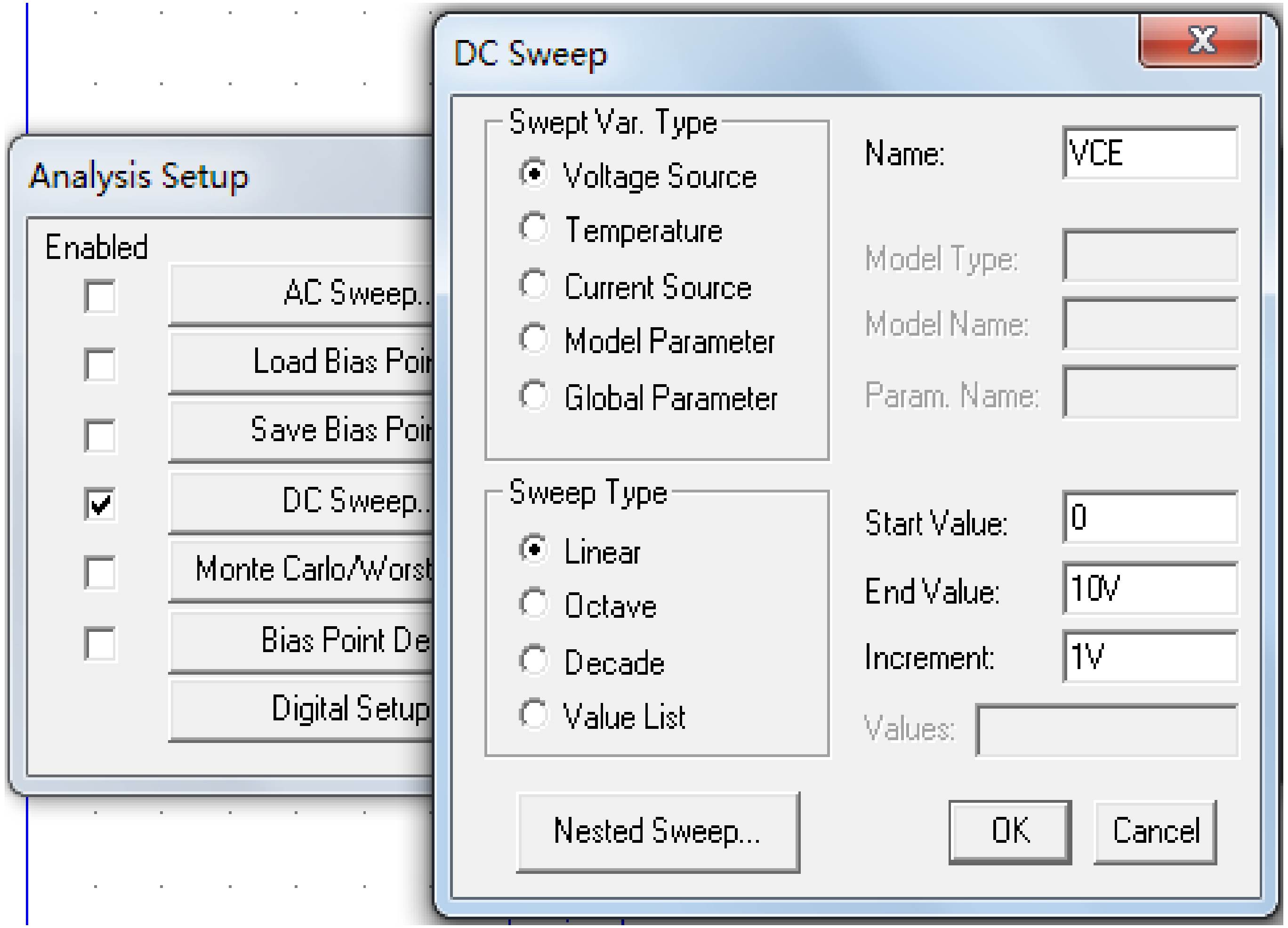
Transistor Output Characteristics (2)

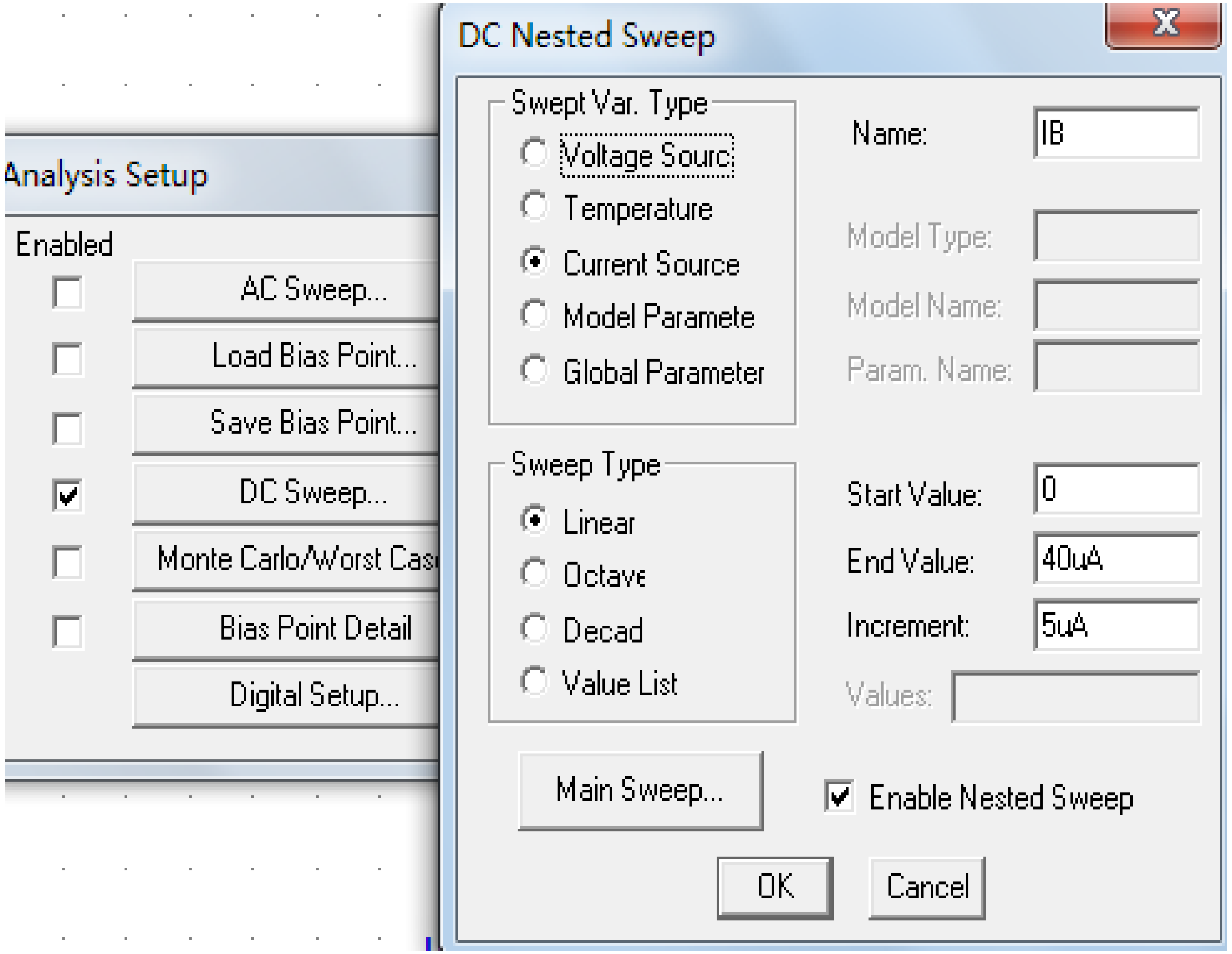
- To obtain the family of characteristics in one simulation, pull down the '**Analysis**' window, select '**Setup**..' check the '**DC sweep**' option and ***input the variable*** as 'V_{ce}' with the desired range.

- 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***.

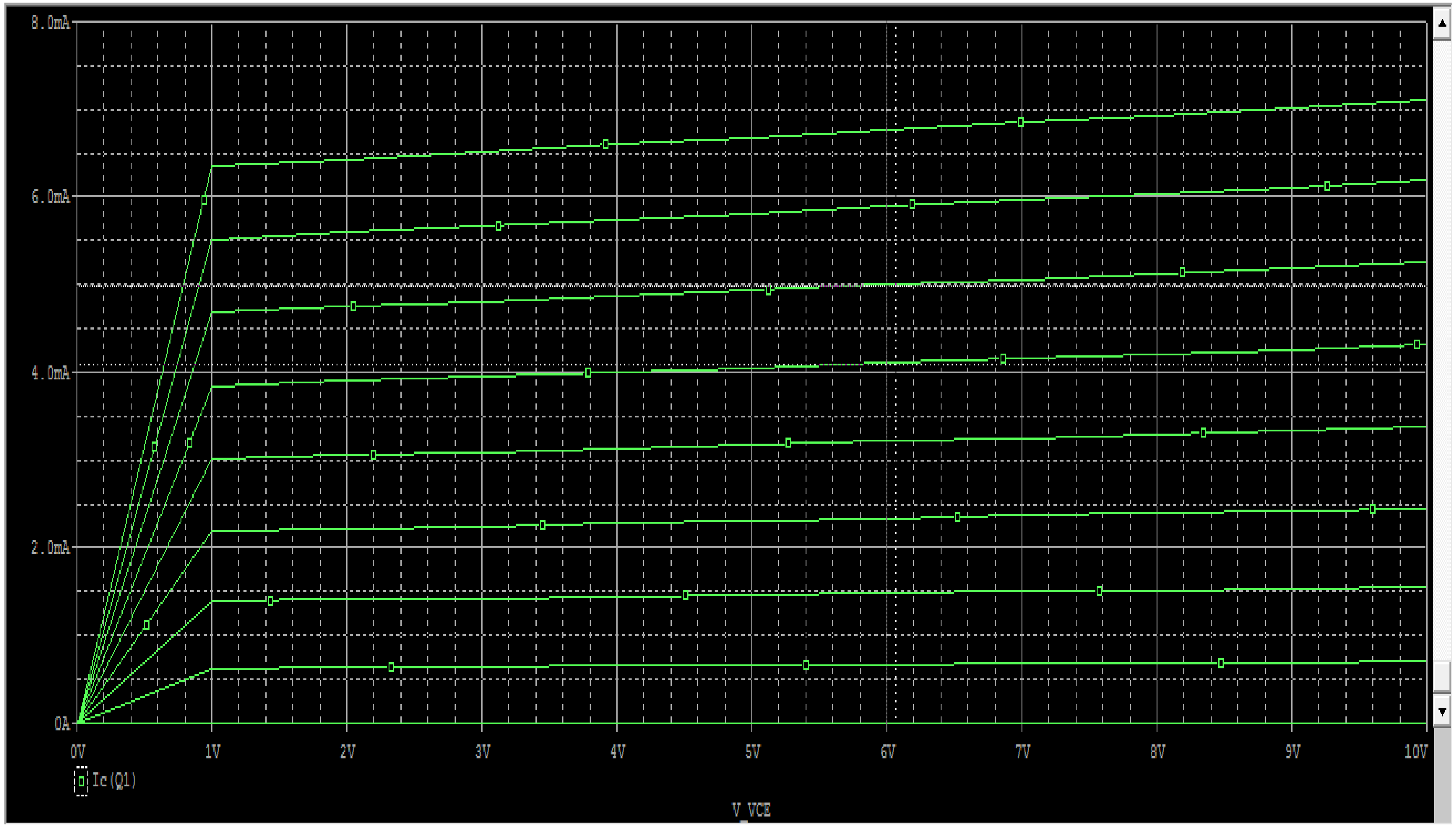
- Calculate the d.c. current gain, **beta** (also known as h_{FE}) at I_C ~ 5 mA.


Analysis
Tools
Markers
Window
Help
Electrical Rule Check
Create Netlist
Edit Stimuli
Setup...
Library and Include Files...
Simulate F11
Probe Setup...
Run Probe F12
Examine Netlist
Examine Output
Display Results on Schematic
10


Analysis Setup
Enabled
AC Sweep...
Load Bias Poi
Save Bias Poi
DC Sweep...
Monte Carlo/Worst
Bias Point De
Digital Setup
DC Sweep
Swept Var. Type
Voltage Source
Temperature
Current Source
Model Parameter
Global Parameter
Name:
VCE
Model Type:
Model Name:
Param. Name:
Sweep Type
Linear
Octave
Decade
Value List
Start Value:
0
End Value:
10V
Increment:
1V
Values:
Nested Sweep...
OK
Cancel


Analysis Setup
Enabled
AC Sweep...
Load Bias Point...
Save Bias Point...
DC Sweep...
Monte Carlo/Worst Cas
Bias Point Detail
Digital Setup...
DC Nested Sweep
Swept Var. Type
Voltage Sourc
Temperature
Current Source
Model Paramete
Global Parameter
Name:
IB
Model Type:
Model Name:
Param. Name:
Sweep Type
Linear
Octave
Decad
Value List
Start Value:
0
End Value:
40uA
Increment:
5uA
Values:
Main Sweep...
Enable Nested Sweep
OK
Cancel

Example of Output IV Characteristics



PSPICE Simulations – Common-Emitter Amplifier (2)

1.2. For the circuit in Figure 1 calculate the values of Rc, Re, R1 and R2. 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.

PSPICE Simulations – Common-Emitter Amplifier (3)

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

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

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

$$V_b = V_e + 0.7$$

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

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

PSPICE Simulations – **Common-Emitter Amplifier (4)**

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

1.4. 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.*

PSPICE Simulations – Common-Emitter Amplifier (5)

- 1.5. Using the PSPICE ***AC analysis*** function 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!

PSPICE Simulations – Common-Emitter Amplifier (6)

PSPICE simulations tips:

- * 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)

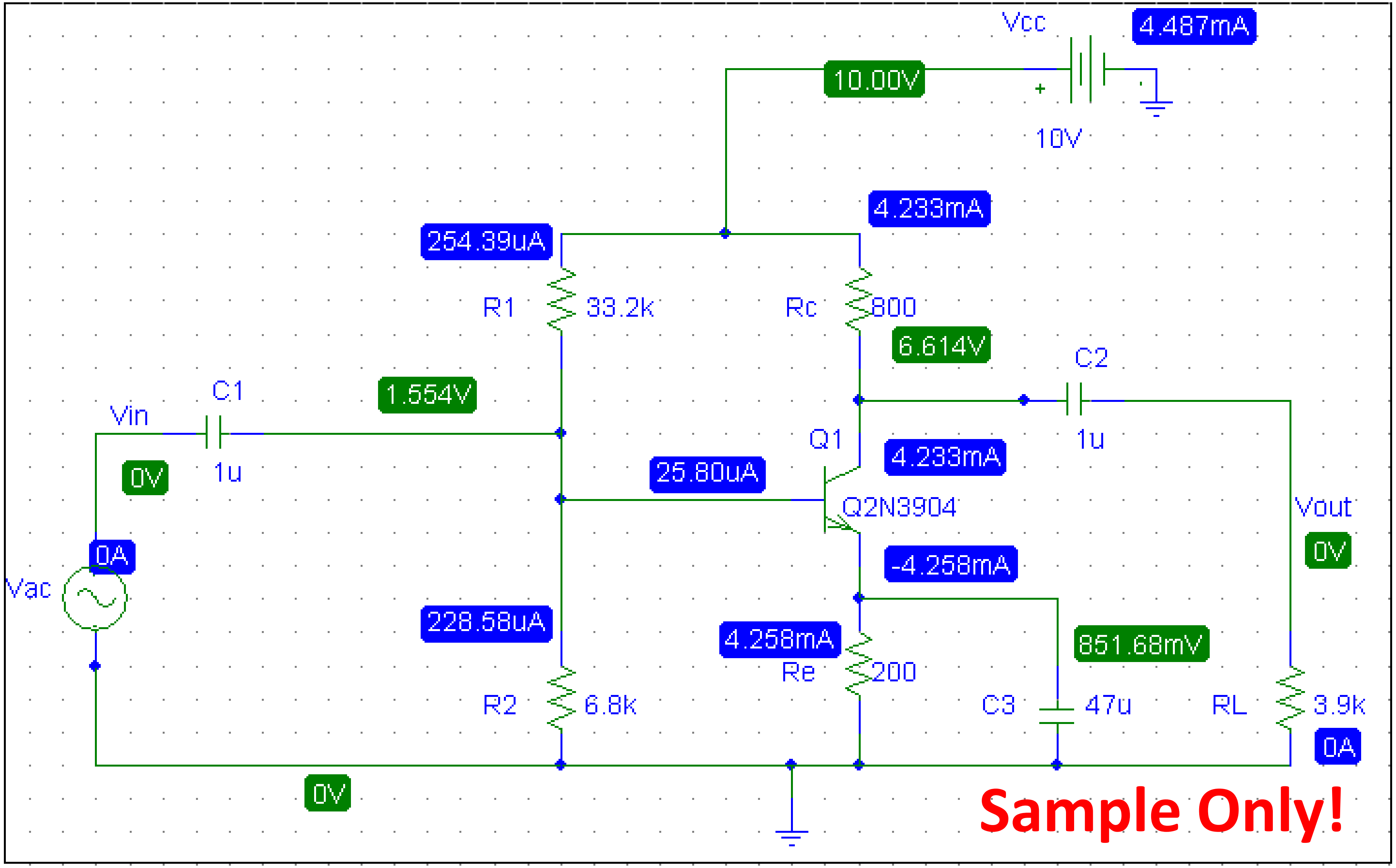
- **To obtain the gain and phase frequency response plots for this circuit you must run "AC ANALYSIS". To get best results for your plots set the AC Analysis Limits as follows:

PSPICE Simulations – Common-Emitter Amplifier (7)

f [Hz]	V_{in} [V]	V_{out} [V]	Θ[deg]	A_v[dB]
50				
75				
100				
200				
500				
1k				
2k				
5k				
10k				
20k				
50k				
100k				
200k				
500k				
1M				

Expand the table if necessary.

PSPICE Simulations – Common-Emitter Amplifier (8)



Sample Only!

PSPICE Simulations – Common-Emitter Amplifier (9)

f[Hz]	Vin[V]	Vout[V]	θ[deg]	Av [dB]
50	0.020	0.115	-51.599	15.198
75	0.020	0.198	-65.933	19.977
100	0.020	0.273	-75.002	22.712
200	0.020	0.557	-94.706	29.033
500	0.020	1.181	-122.115	35.415
1K	0.020	1.640	-144.034	38.275
2K	0.020	1.866	-160.478	39.411
5K	0.020	1.951	-172.057	39.707
10K	0.020	1.964	-175.979	39.843
20K	0.020	1.967	-178.010	39.857
50K	0.020	1.968	-179.268	39.861
100K	0.020	1.968	-179.729	39.862
200K	0.020	1.968	-180.065	39.862
500K	0.020	1.968	-180.583	39.861
1M	0.020	1.968	-181.289	39.860

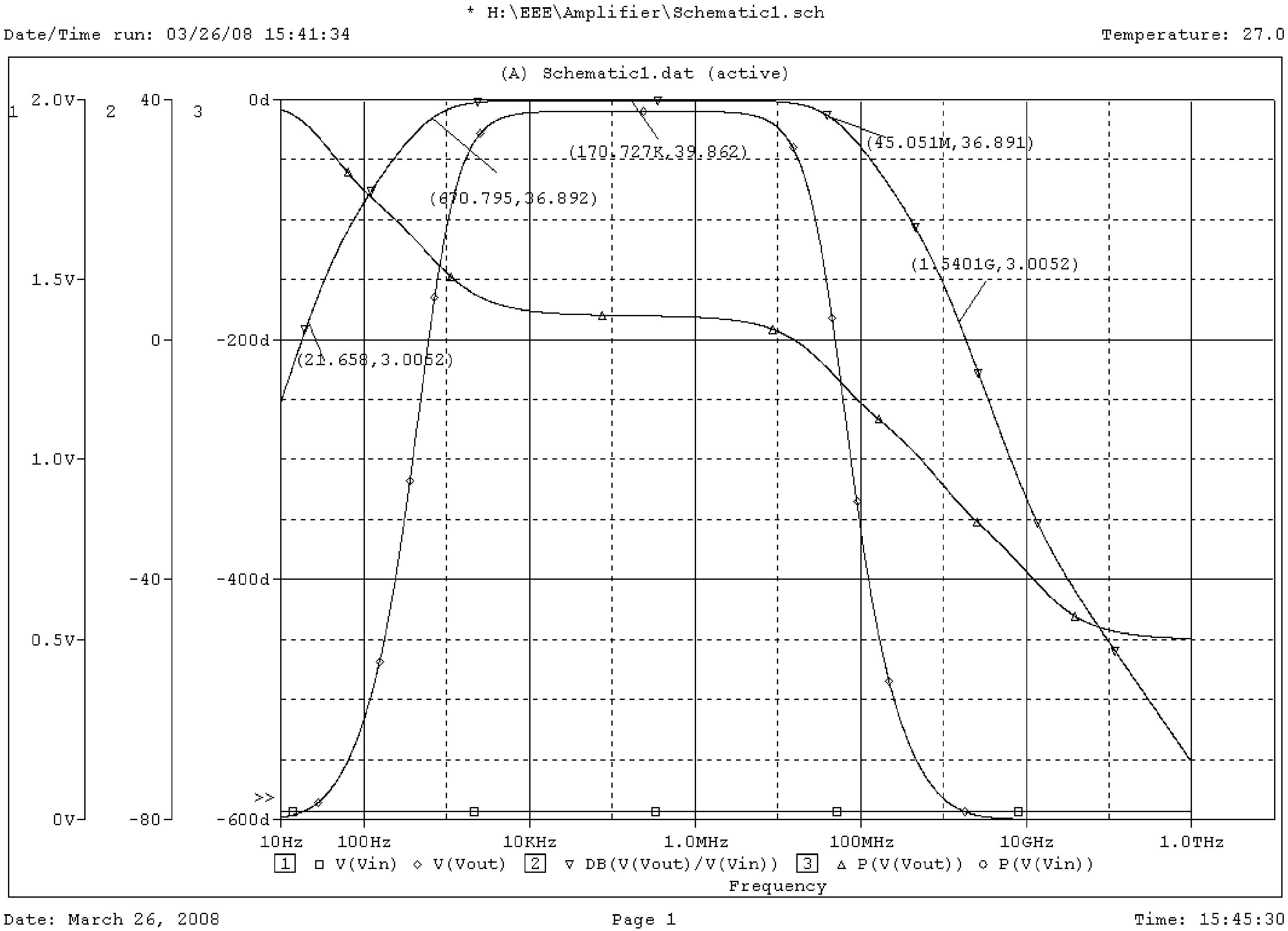
Sample Only!

PSPICE Simulations – Common-Emitter Amplifier (10)

- Through PSpice simulation, you need to obtain four curves:

1. V_{in}
2. V_{out}
3. dB (V(Vout/Vin)) = 20log10(V_{out}/V_{in}) = A_v (assume V_{in} = 0.02v)
4. P(V(V_{out})) = phase of V_{out}
5. P(V(V_{in})) = phase of V_{in}

PSPICE Simulations – Example of simulation result (C-E)



PSPICE Simulations - Common-Collector amplifier (1)

2.1. 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.**

PSPICE Simulations - Common-Collector amplifier (2)

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.

$$V_{cc} = V_{ce} + I_e R_e$$

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

$$V_b = V_e + 0.7$$

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

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

PSPICE Simulations - Common-Collector amplifier (3)

2.2. Calculate the **voltage gain**, the **current gai**n, the **input resistance**, and the **output resistance** for this amplifier. All the calculations must be shown.

2.3. 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.*

PSPICE Simulations - Common-Collector amplifier (4)

- 2.4. 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 - Common-Collector amplifier (5)

PSPICE simulations tips:

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

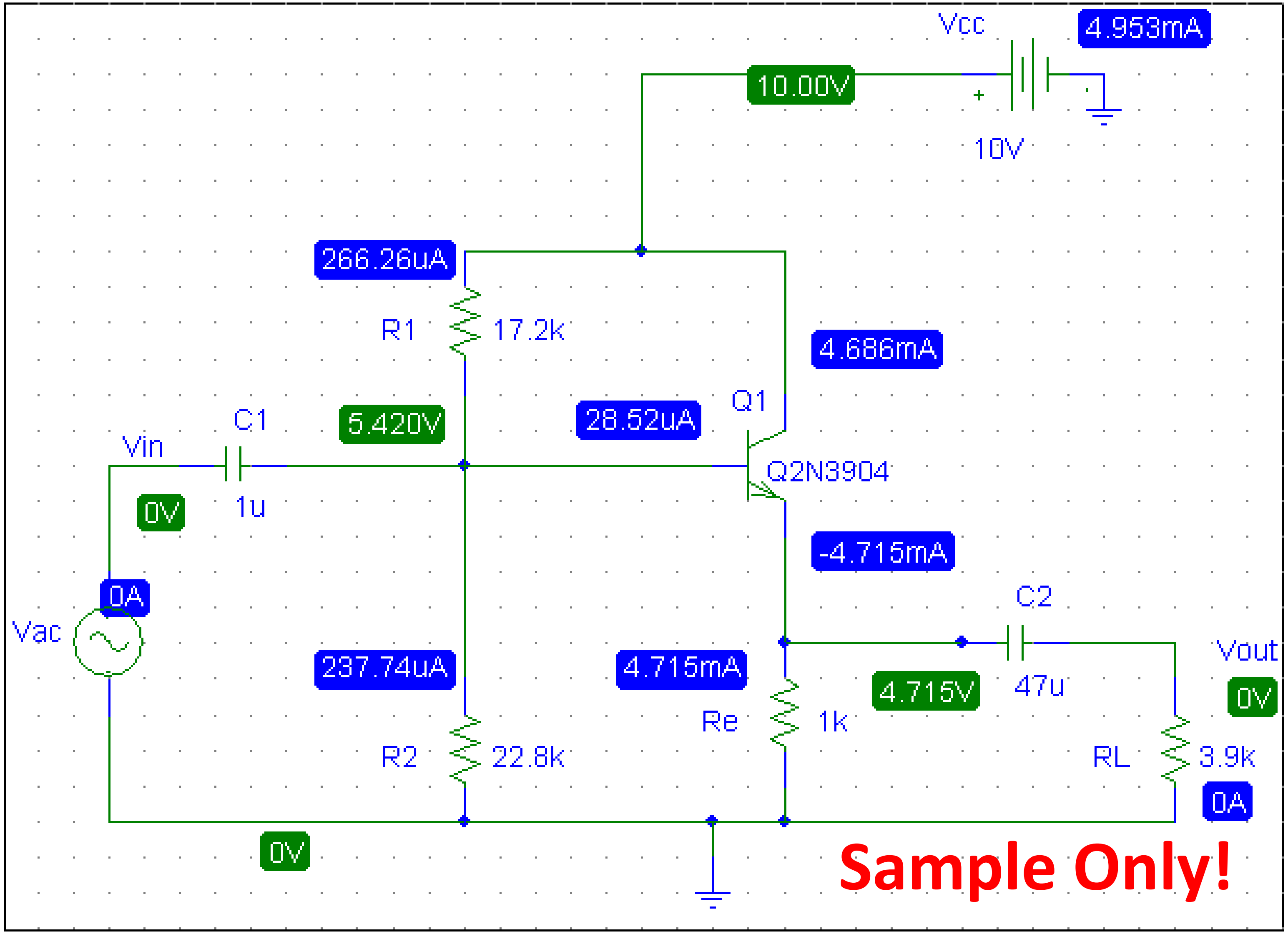
- To get better results for your plots set the AC Analysis Limits as follows:

PSPICE Simulations - Common-Collector amplifier (6)

f [Hz]	V_{in} [V]	V_{out} [V]	Θ[deg]	A_v[dB]
50				
75				
100				
200				
500				
1k				
2k				
5k				
10k				
20k				
50k				
100k				
200k				
500k				
1M				

Expand the table if necessary.

PSPICE Simulations - Common-Collector amplifier (7)



PSPICE Simulations - Common-Collector amplifier (8)

f[Hz]	Vin[V]	Vout[V]	θ[deg]	Av [dB]
50	1.000	0.937	20.055	-0.566
75	1.000	0.967	13.705	-0.295
100	1.000	0.978	10.369	-0.197
200	1.000	0.989	5.234	-0.099
500	1.000	0.991	2.088	-0.072
1K	1.000	0.992159	1.047	-0.068
2K	1.000	0.992272	0.518	-0.067384
5K	1.000	0.992304	0.209	-0.067107
10K	1.000	0.992308	0.105	-0.067068
20K	1.000	0.992309	0.052	-0.067059
50K	1.000	0.992310	0.021	-0.067056
100K	1.000	0.992310	0.009	-0.067055
200K	1.000	0.992310	0.004	-0.067055
500K	1.000	0.992310	-0.002	-0.067055
1M	1.000	0.992310	-0.007	-0.067055

PSPICE Simulations - Common-Collector amplifier (9)

- Through PSpice simulation, you need to obtain four curves:

1. V_{in}
2. V_{out}
3. dB $(V(V_{out})) = 20\log10(V_{out}/V_{in}) = A_v$ (assume V_{in} = 1v)
4. $P(V(V_{out}))$ = phase of V_{out}
5. $P(V(V_{in}))$ = phase of V_{in}

PSPICE Simulations – Example of simulation result (C-C)