PSpice Guidance (Additional)

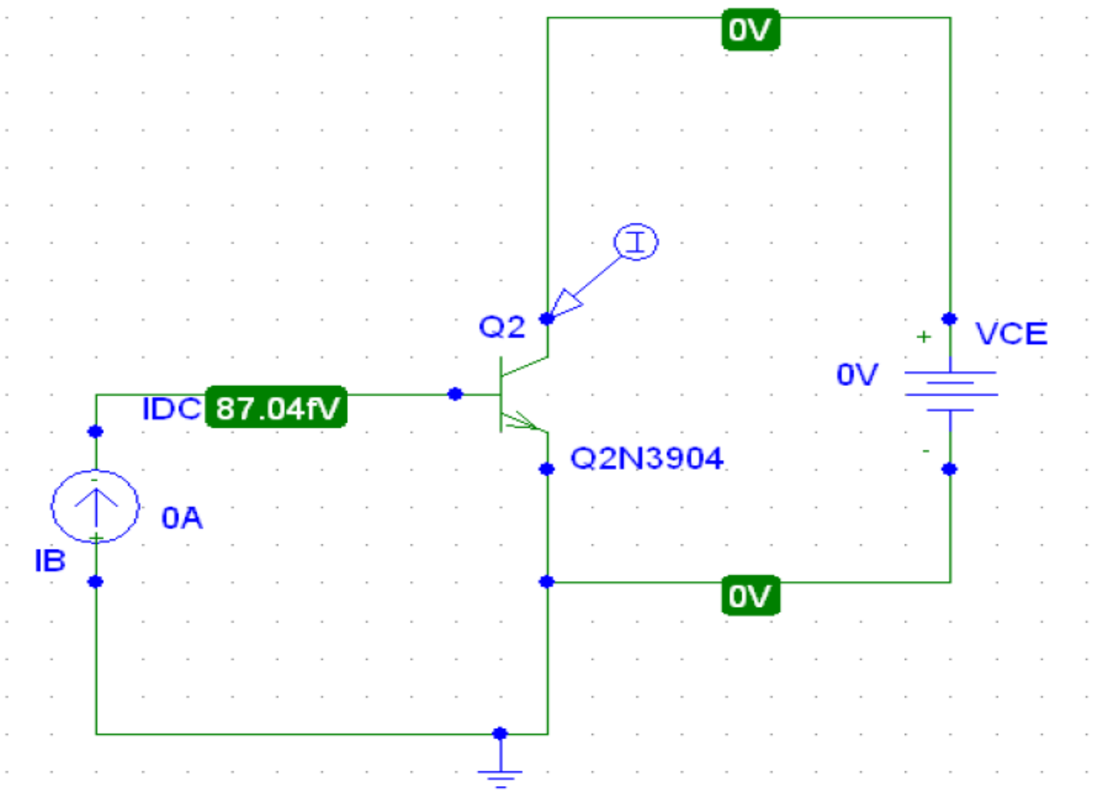
- PSpice Settings

Objectives

- Finding β of a Transistor
- Simulate Frequency Response of a CE Amplifier
- Simulate Frequency Response of a CC Amplifier

Finding β of the transistor (1)

- Input the circuit schematic below.



Finding β of the transistor (2)

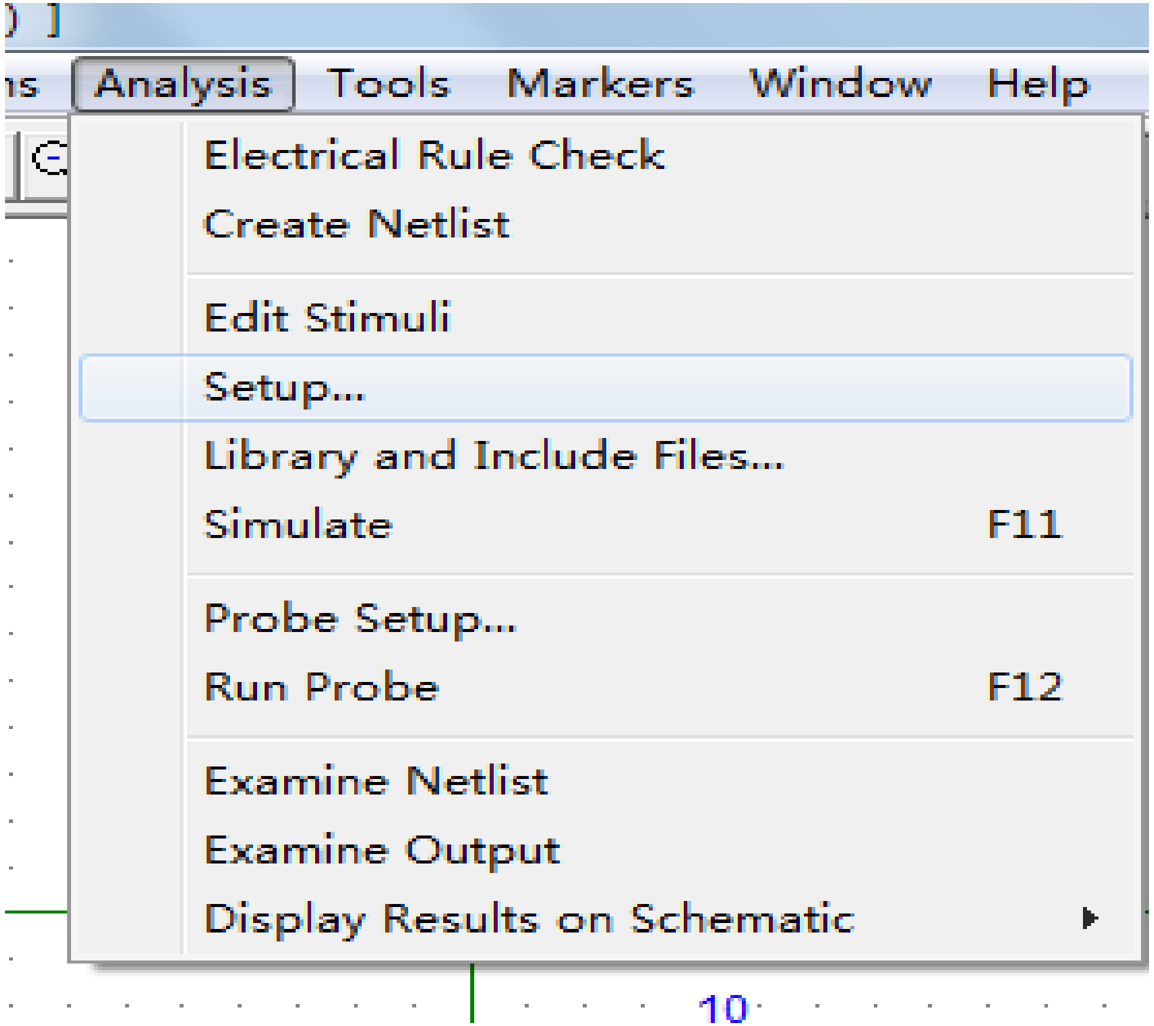
- To obtain the family of characteristics in one simulation, pull down the '**Analysis**' window, select '**Setup**..' check the '**DC sweep**' option and ***input the variable*** as 'Vce' with the desired range.

- 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***.

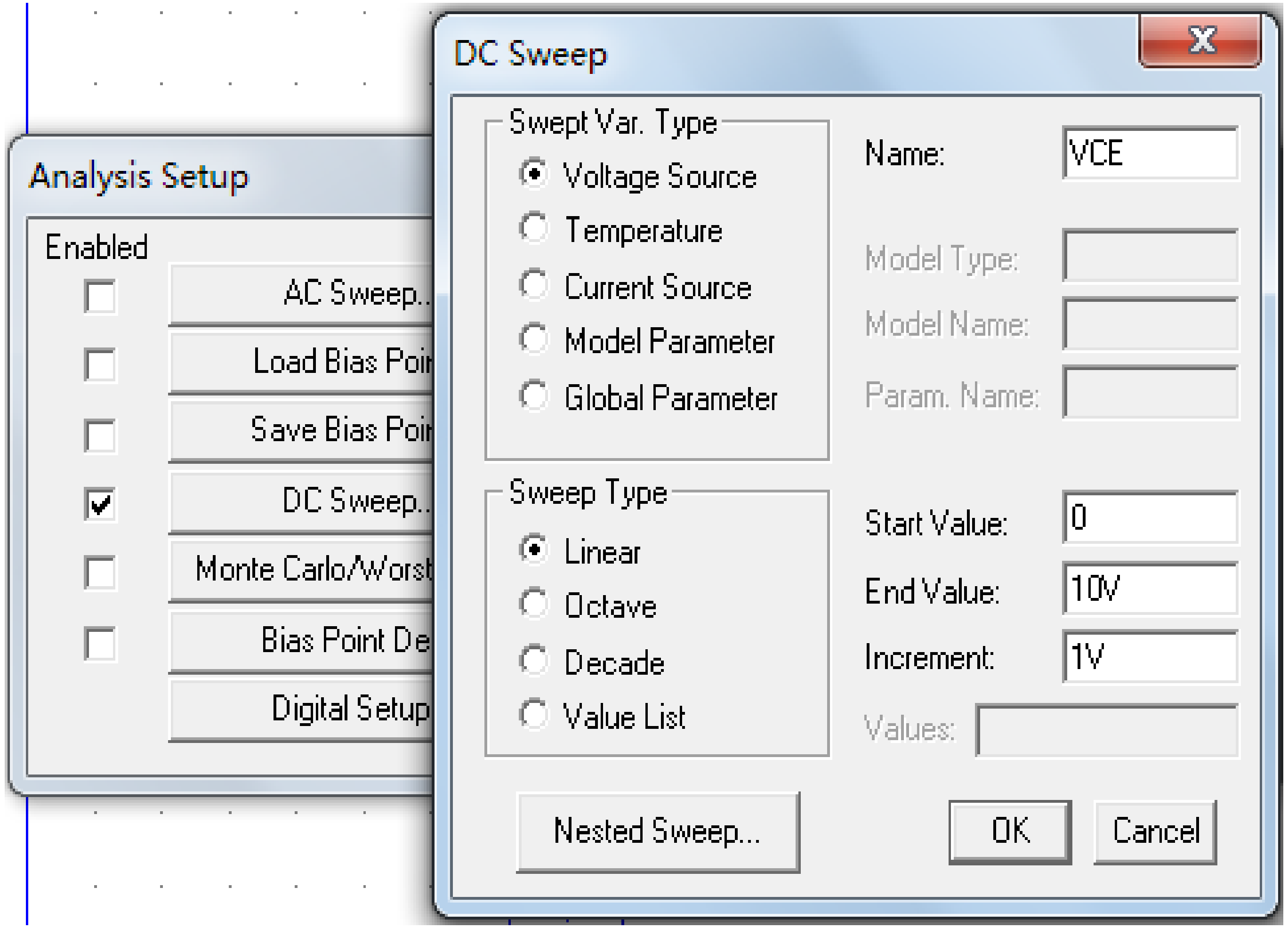
- Calculate the d.c. current gain, **beta** (also known as h_{FE}) at I_C ~ 5 mA.

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

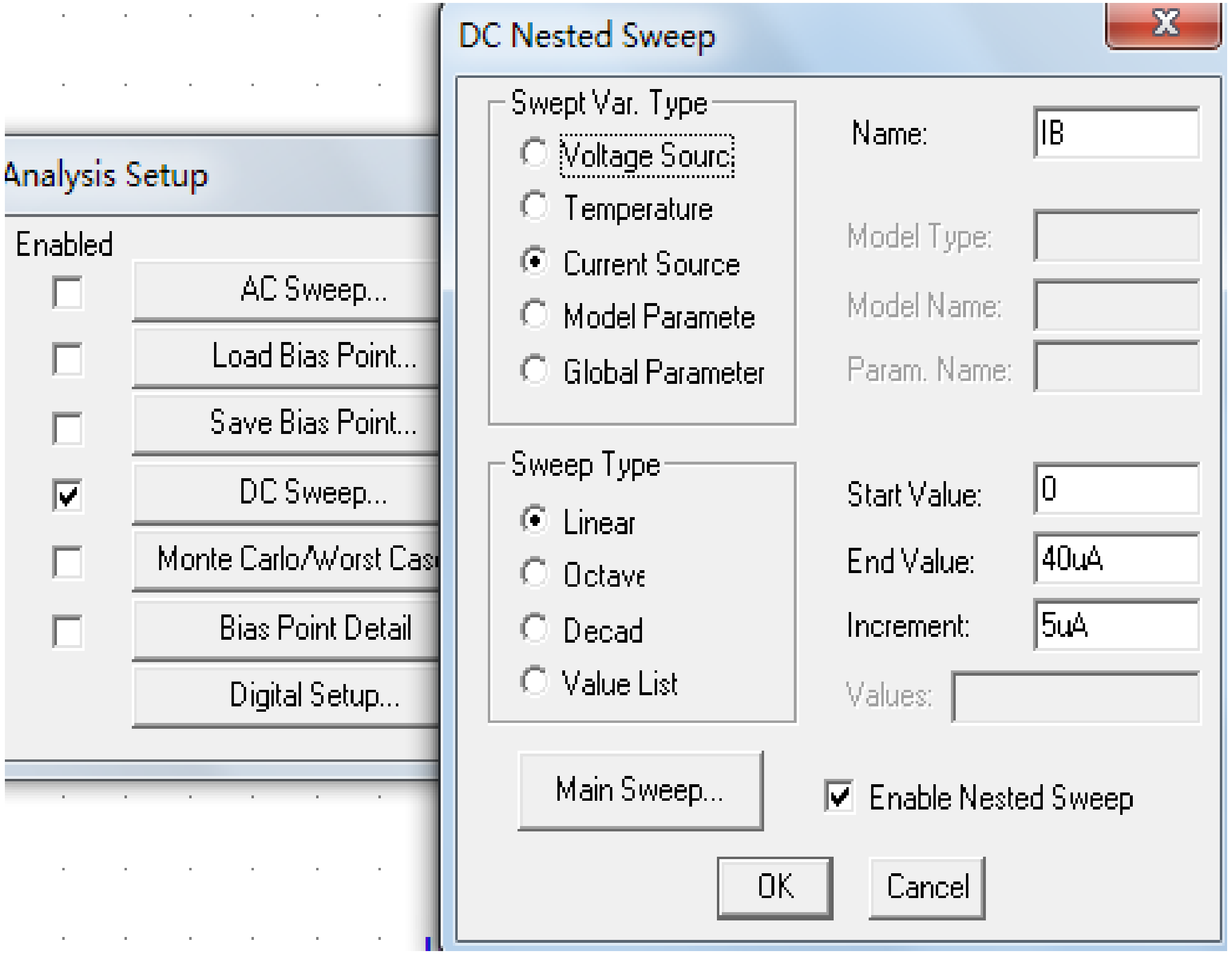
PSpice Settings



PSpice Settings



PSpice Settings

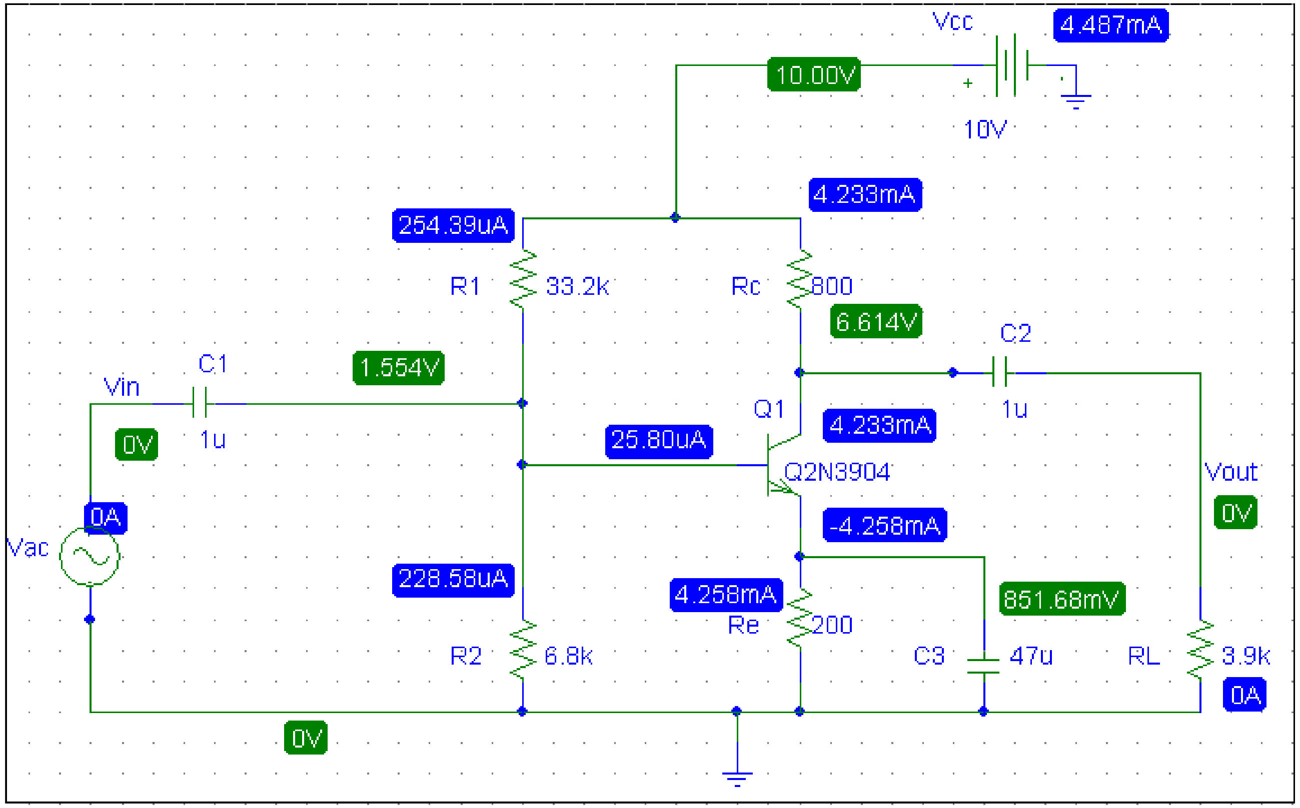


Simulate Frequency Response of a CE Amplifier (1)

- Use 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**.

Simulate Frequency Response of a CE Amplifier (2)

- Input the circuit schematic below.



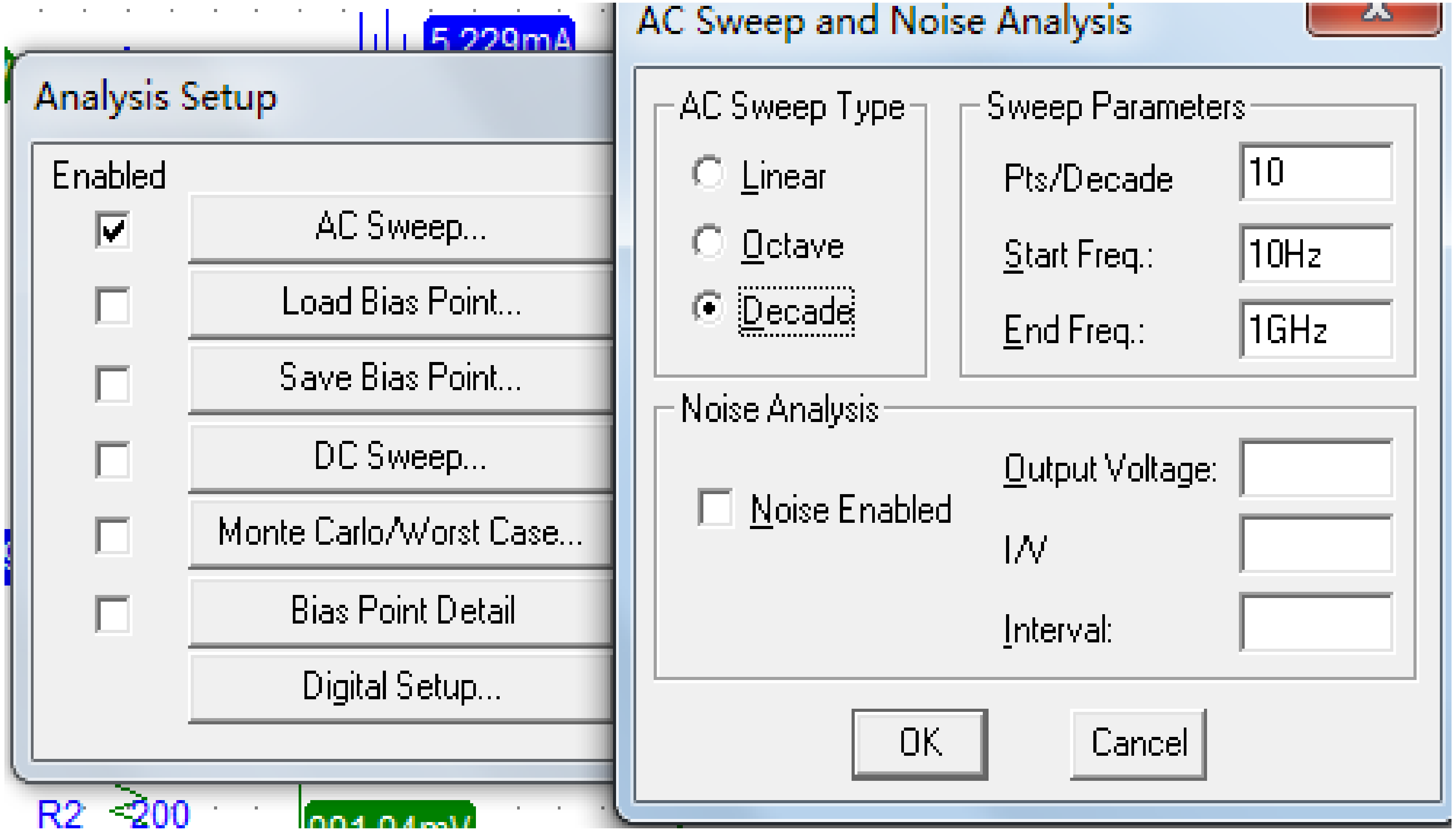
Simulate Frequency Response of a CE Amplifier (3)

- 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 Settings



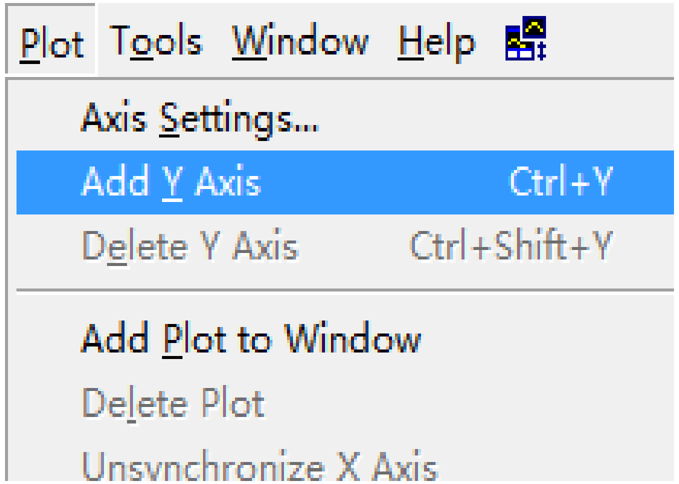
Simulate Frequency Response of a CE Amplifier (4)

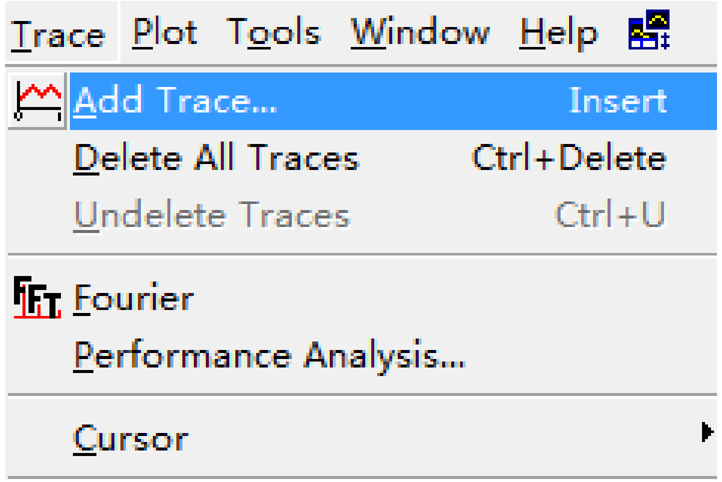
- Through PSpice simulation, you need to obtain Five curves:

1. V_{in}
2. V_{out}
3. dB (V(Vout/Vin)) = $20\log10(V_{out}/V_{in})$ = 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}

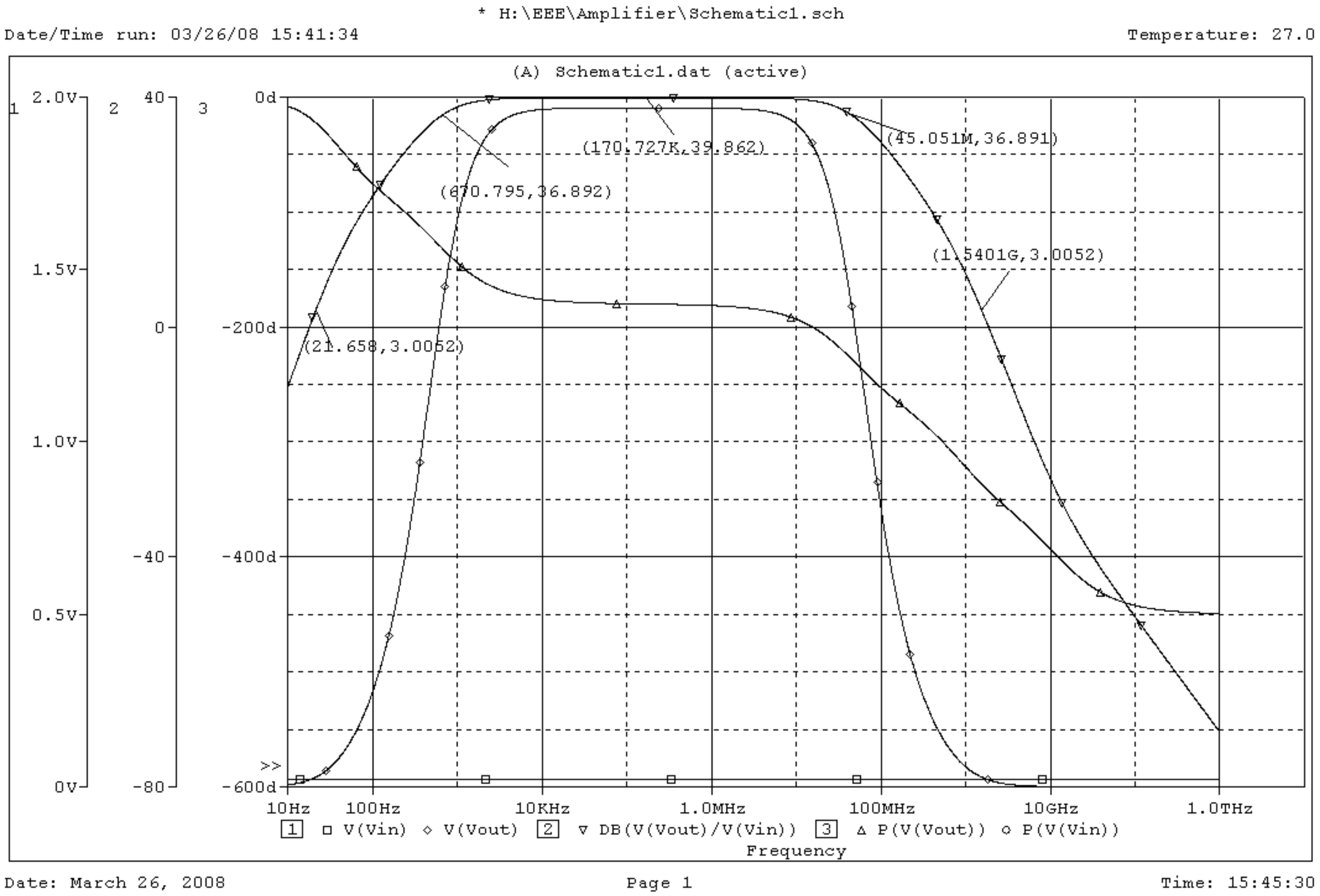
PSpice Settings

- Add two markers of ‘vphase’ to measure the phase of input voltage and output voltage.
- Click ‘Simulate’. The plots of phases of input voltage and output voltage are shown at first. Chose ‘Add Y Axis’ to add Y Axis and chose ‘Add Trace’ to add other plots.





Simulate Frequency Response of a CE Amplifier (5)



Simulate Frequency Response of a CE Amplifier (6)

- Use ‘toggle cursor’ to obtain the 3dB points at the curve of voltage gain A_V in dB.

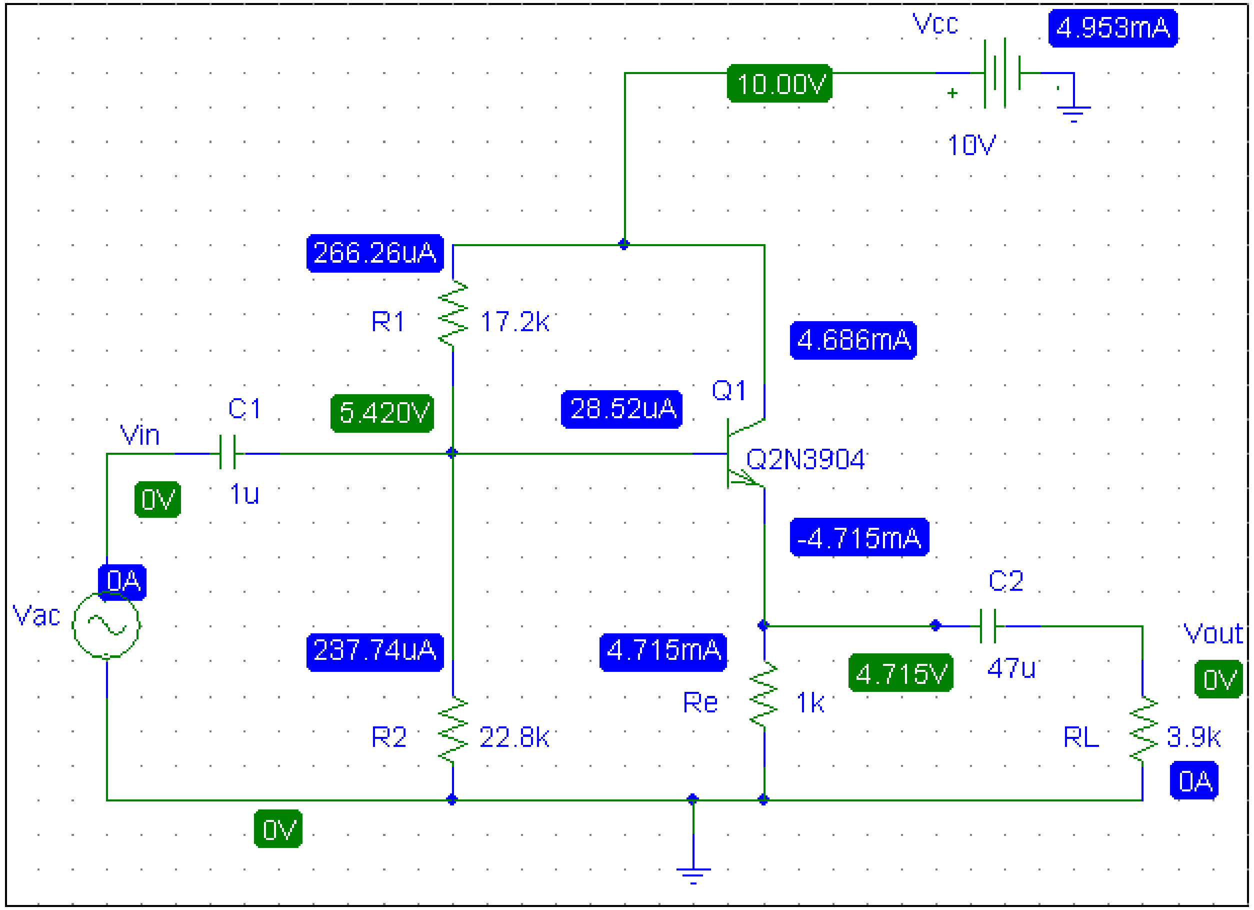
Simulate Frequency Response of a CC Amplifier (1)

- Use 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**.

- Follow the same procedure as the CE amplifier in the simulation.

Simulate Frequency Response of a CC Amplifier (2)

- Input the circuit schematic below.



Simulate Frequency Response of a CC Amplifier (3)

- For a sine wave signal source (used for simulating the Vin), use a Vac= **1** (V)
- Follow the same procedure like CE amplifier to check ‘AC Sweep’ and set the ‘Analysis’ limits.
- Chose ‘Add Y Axis’ to add Y Axis and chose ‘Add Trace’ to add other plots.
- Use ‘toggle cursor’ to obtain the 3dB points at the curve of voltage gain A_V in dB.

Simulate Frequency Response of a CC Amplifier (4)

- Through PSpice simulation, you need to obtain Five 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}

Simulate Frequency Response of a CC Amplifier (5)