

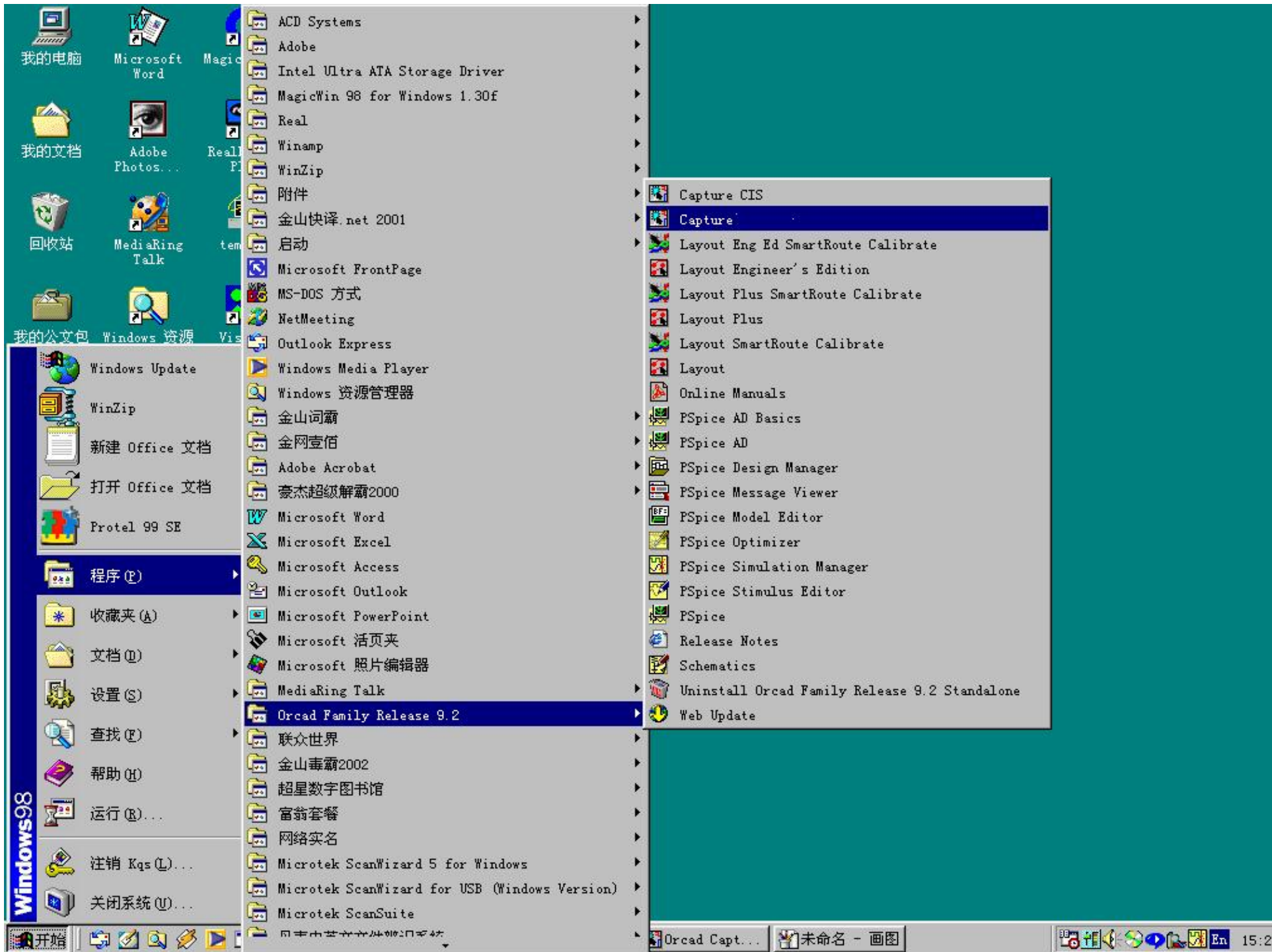
模拟电子学 基础实验

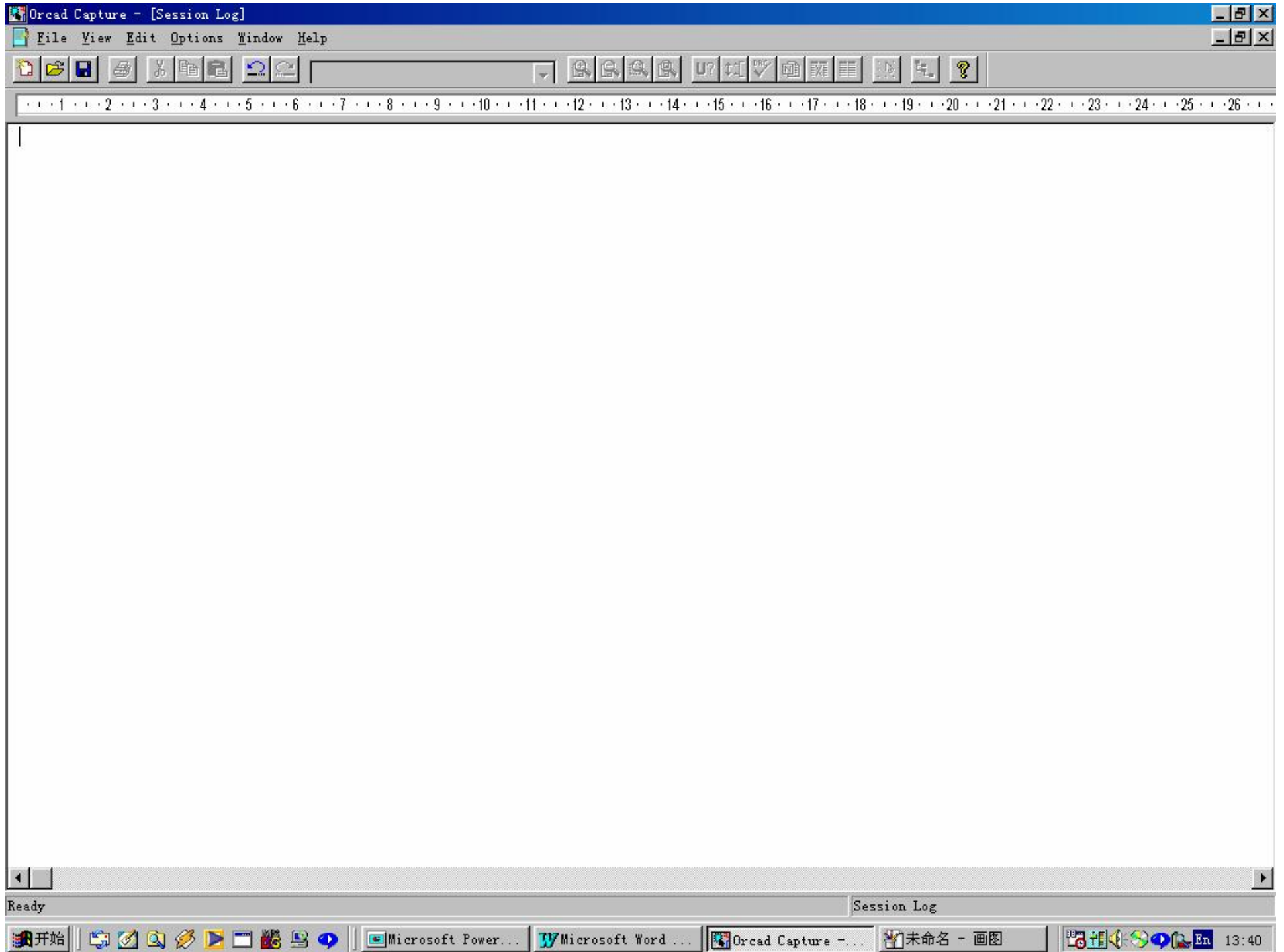
晶体管放大器的仿真 实验步骤

(一) 电路原理图输入

1、启动OrCAD/Capture

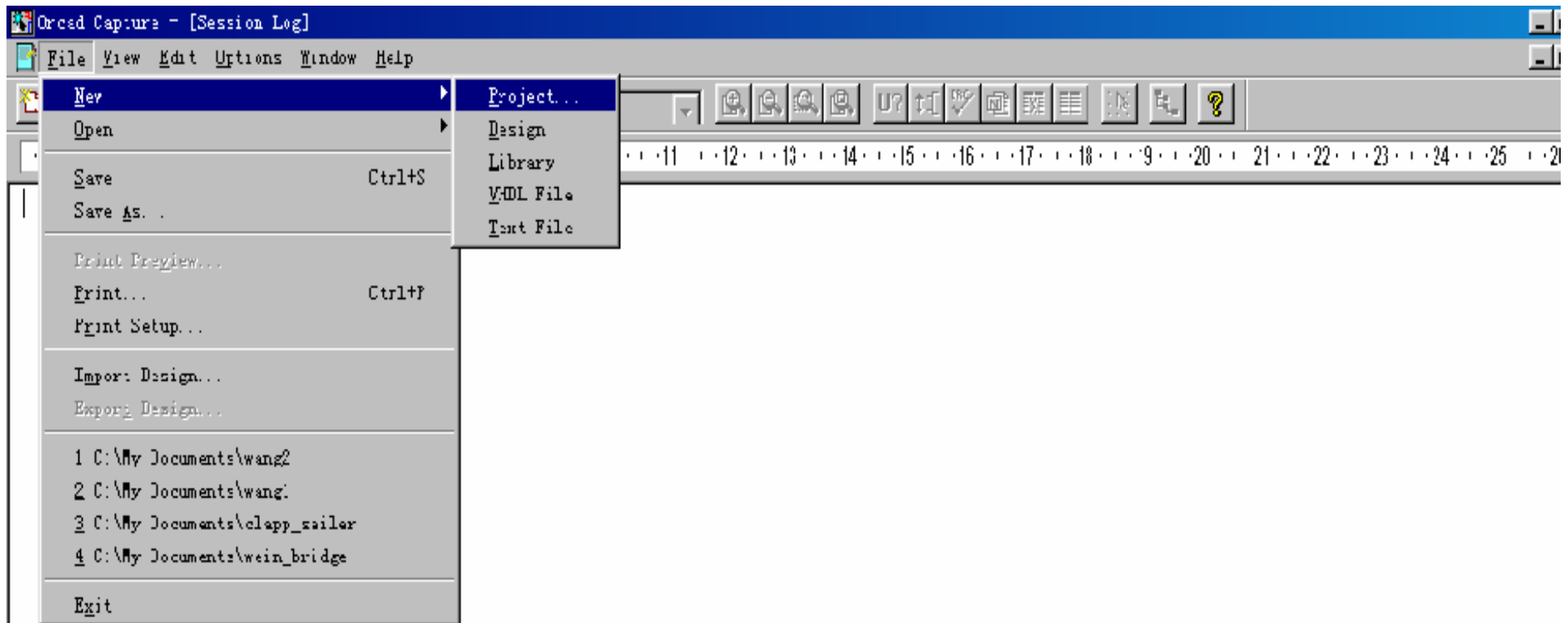
- 选择“开始”→“程序”→“OrCAD 9.2”→“Capture”，以进入 Capture 的工作环境





2、创建新项目

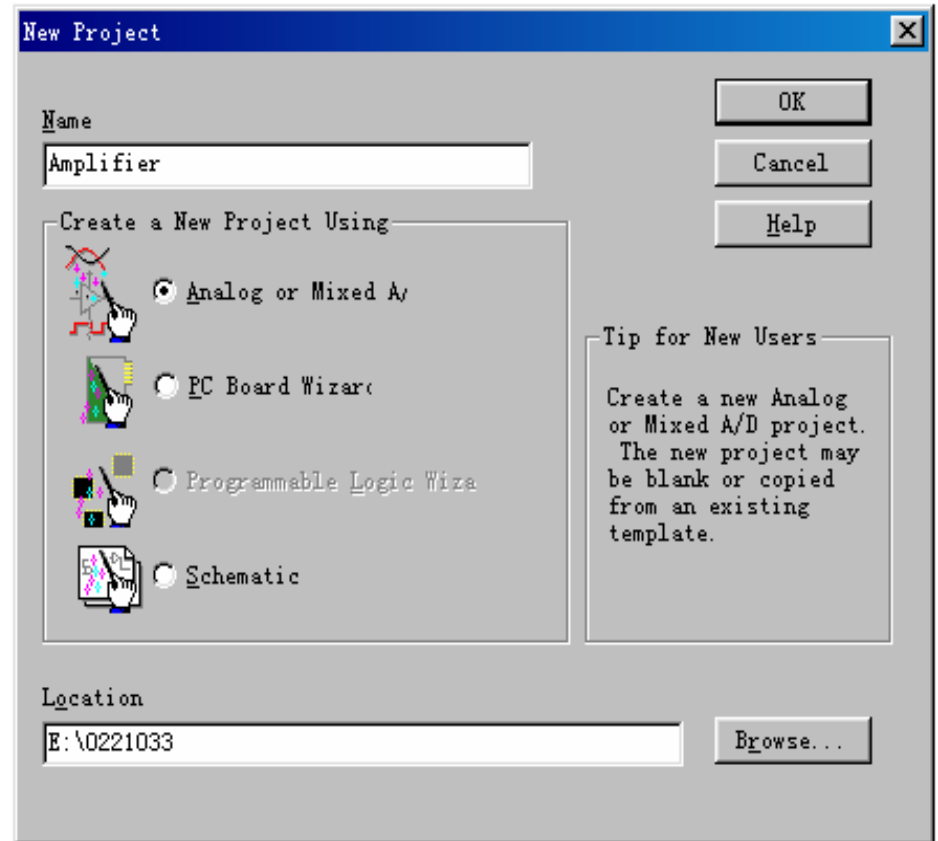
- 执行File/New/Project命令



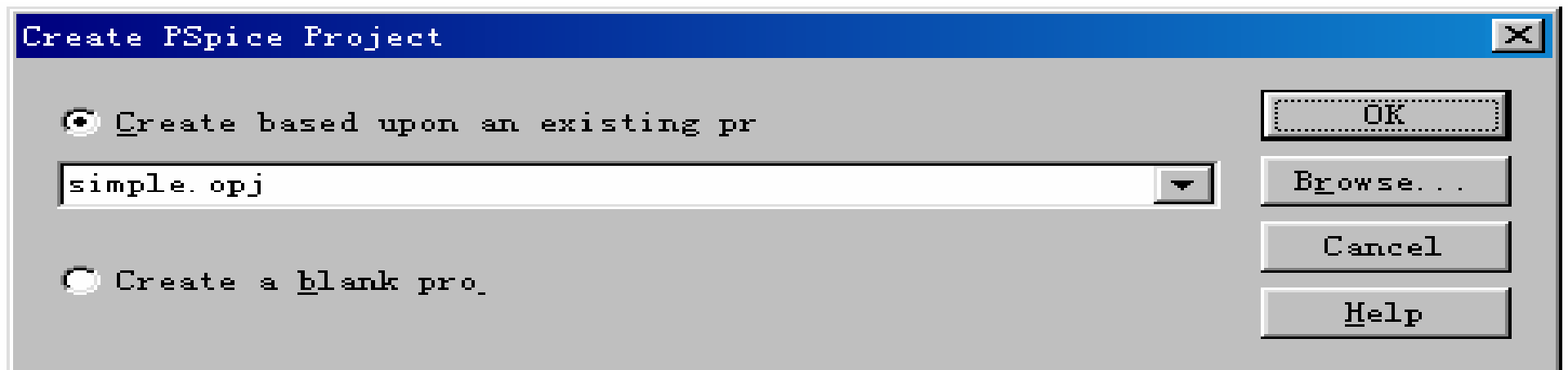
- 在Name框中键入欲建立项目的名称
(如: Amplifier)
- 在Location框中键入该项目的保存路径
(如: E:\0221033)
- 在Create a New Project Using复选框中选择Analog or Mixed-Signal Circuit
- 单击“OK”

中国电子爱好者园
地:www.cepark.com

专业电子技术论
坛:<http://bbs.cepark.com/>

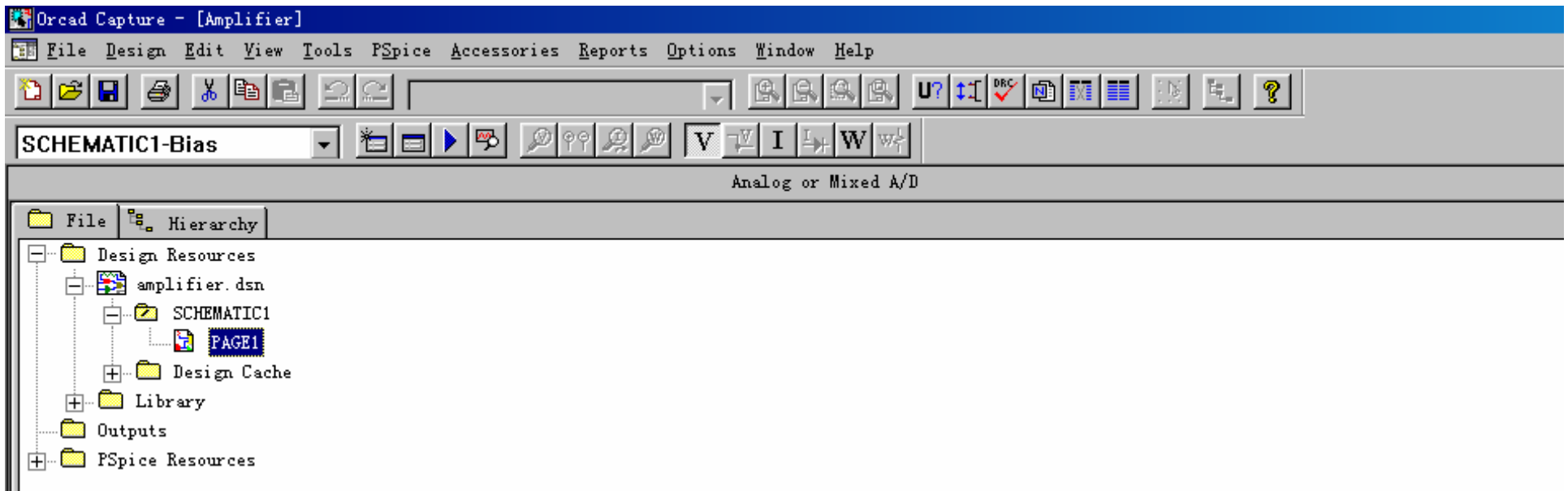


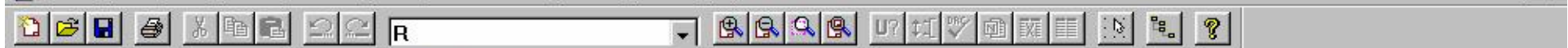
- 出现“Create Pspice Project”对话框
- 在Create base upon an existing project复选框中选择simple.opj
- 单击“OK”



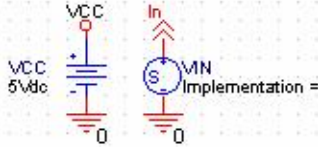
3、电路原理图编辑

- 在项目管理器中，依次双击“Design Resources”、“Amplifier.dsn”、“Schematic1”、“Page1”
- 自动进入原理图编辑器界面





SCHEMATIC1-Bias



Implementation =

Double click any label or value to change it.

Copy and paste the VCC circle or In port in order to connect the DC and signal sources to the appropriate wires in your circuit.

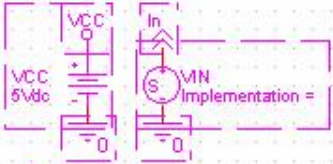
Select the part Vin, then the menu command Edit/PSpice Stimulus... to use the Stimulus Editor. (Basics users: replace this part with a supported voltage source like VSIN.)

- 删除原理图编辑区左边的多余符号，方法是：
- 用鼠标对准欲删区域左上角，按下鼠标左键不放。将鼠标移至欲删区域右下角，防开鼠标左键
- 按键盘Del键

Orcad Capture - [/ - (SCHEMATIC1 : PAGE1)]

File Edit View Place Macro PSpice Accessories Options Window Help

SCHEMATIC1-Bias



Double click any label or value to change it.

Copy and paste the VCC circle or In port in order to connect the DC and signal sources to the appropriate wires in your circuit.

Select the part Vin, then the menu command Edit/PSpice Stimulus... to use the Stimulus Editor. (Basics users: replace this part with supported voltage source like VSIN.)

7 items selected Scale=100% X=4.50 Y=2.90

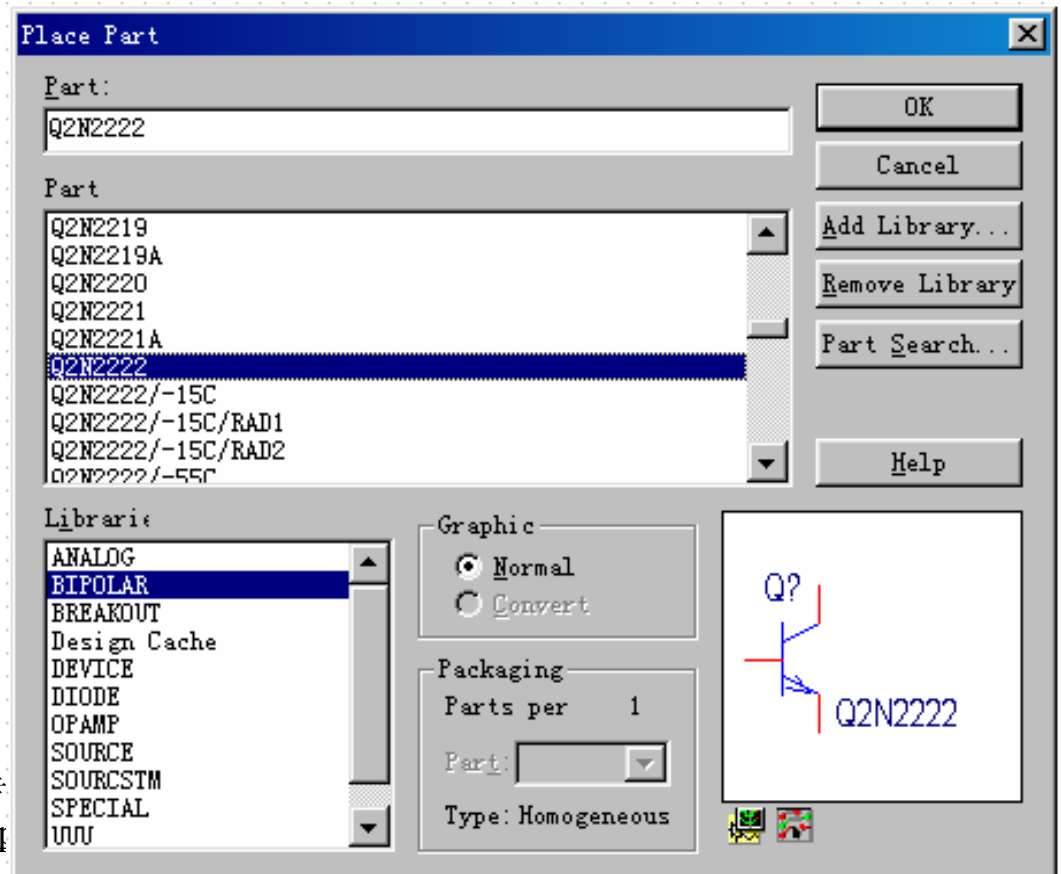
开始 Microsoft Power... Microsoft Word... Orcad Capture -... 未命名 - 画图 16:06

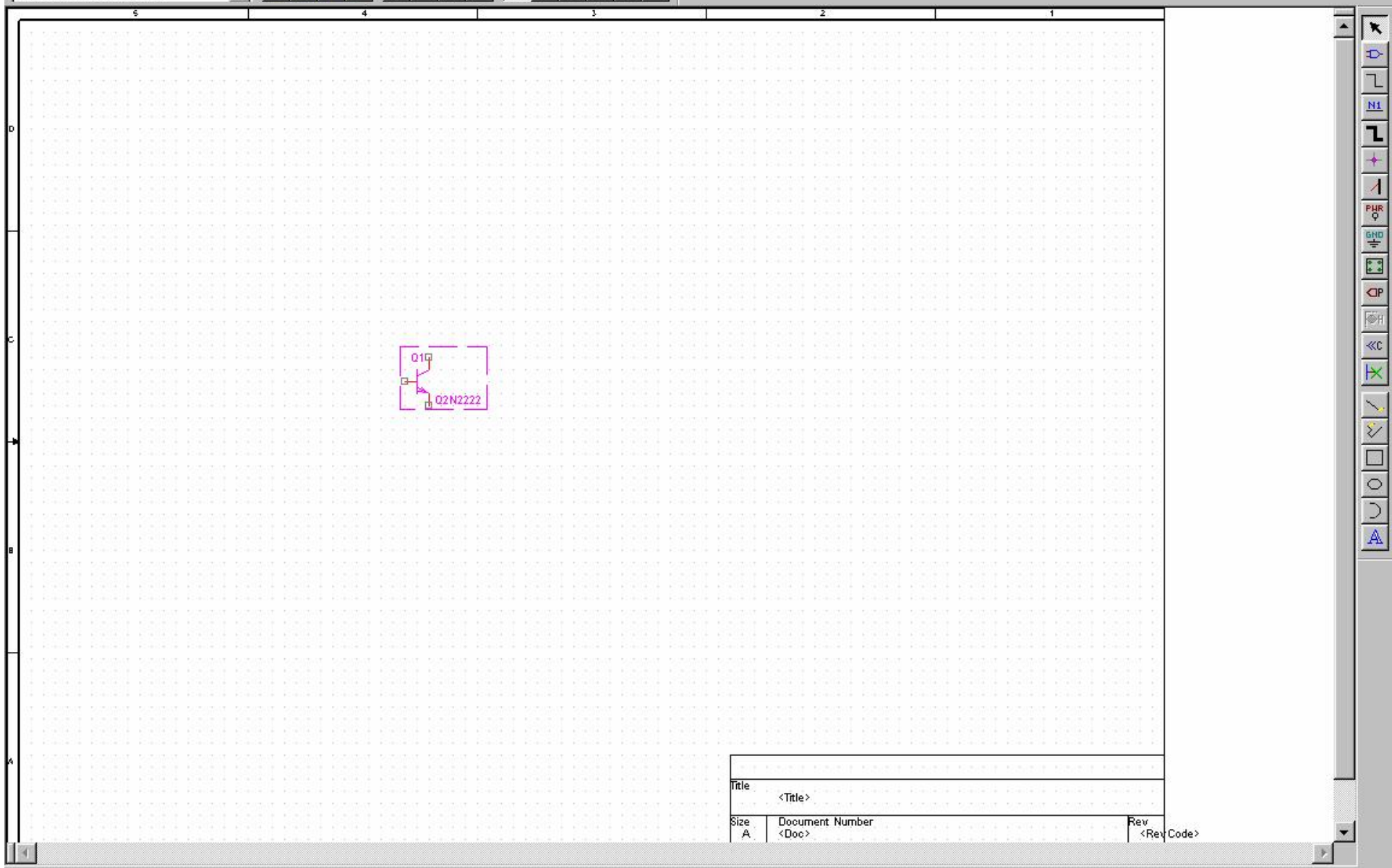
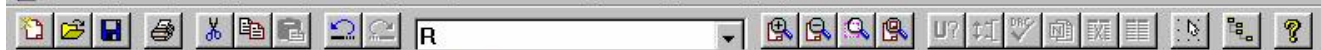
放置晶体管符号

- 执行Place/Part命令
- 在“Libraries”列表框中选择“BIPOLAR”
- 在“Part”列表框中选择“Q2N2222”
- 单击“OK”
- 将晶体管移至合适位置，按鼠标左键
- 按ESC键以结束绘制元器件状态

中国电子爱好者网
地: www.cepark.com

专
坛: <http://>

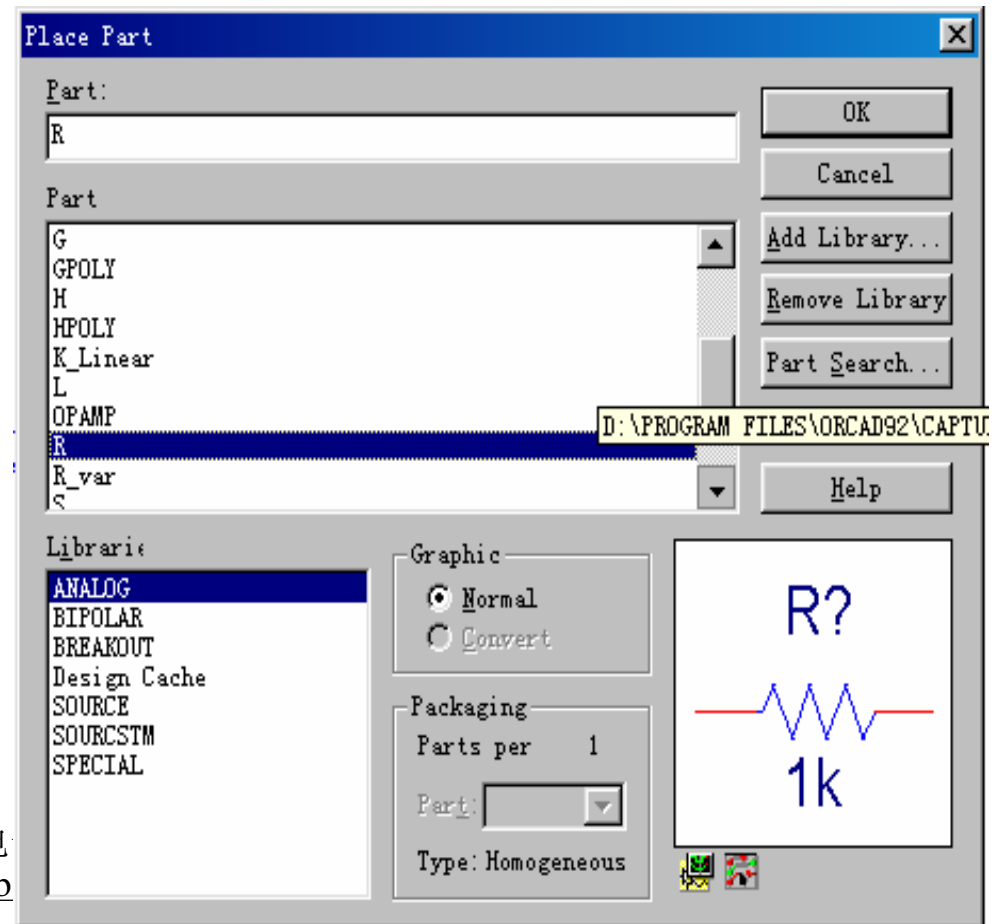


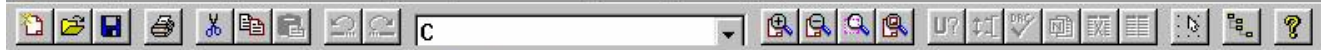


Title	<Title>
Size	Document Number
A	<Doc>
Rev	<Rev Code>

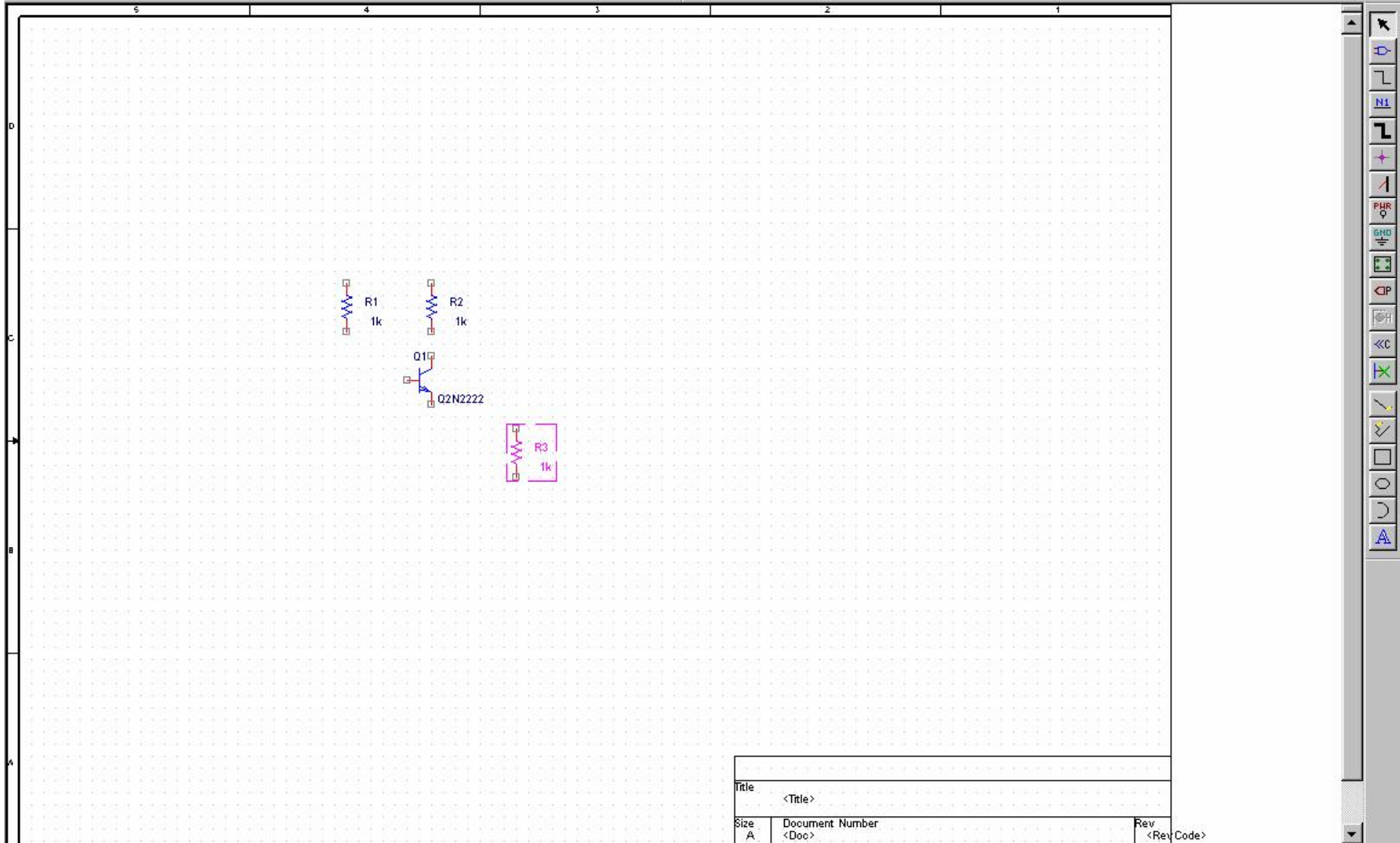
放置电阻符号

- 执行Place/Part命令
- 在“Libraries”列表框中选择“ANALOG”
- 在“Part”列表框中选择“R”
- 单击“OK”
- 将电阻R移至合适位置，按鼠标左键
- 按ESC键以结束绘制





SCHEMATIC1-Bias



Title	<Title>	Rev	<Rev Code>
Size	Document Number		
A	<Doc>		

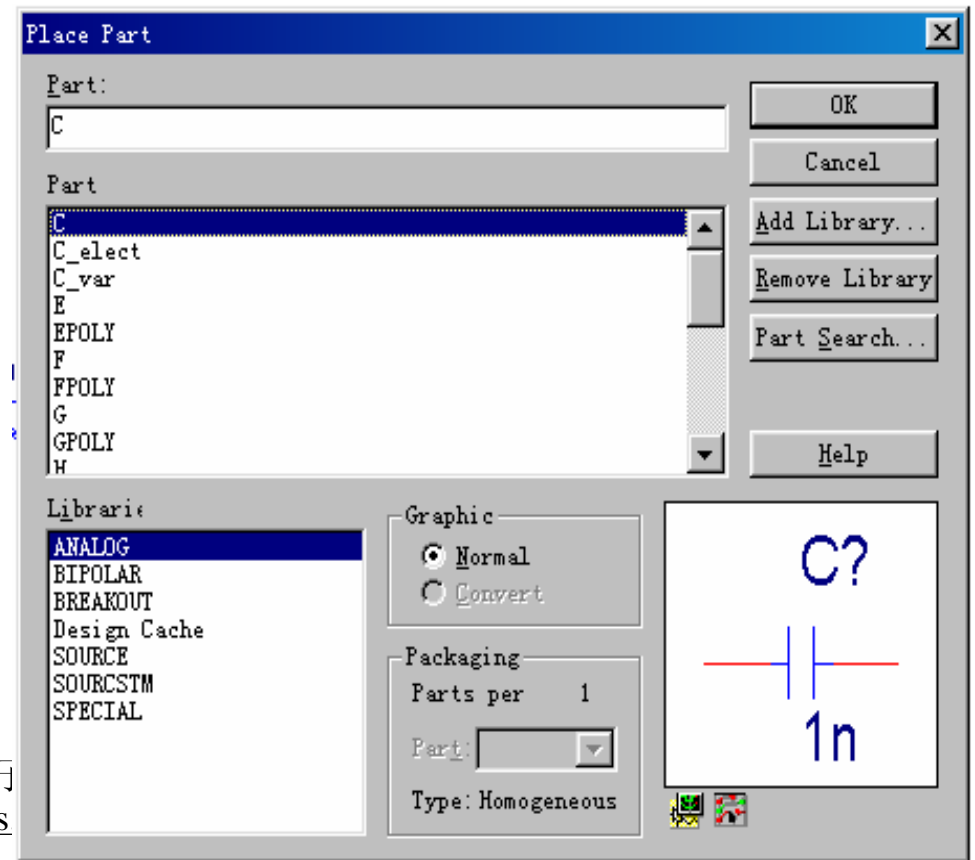
1 item selected Scale=100% X=3.90 Y=2.40

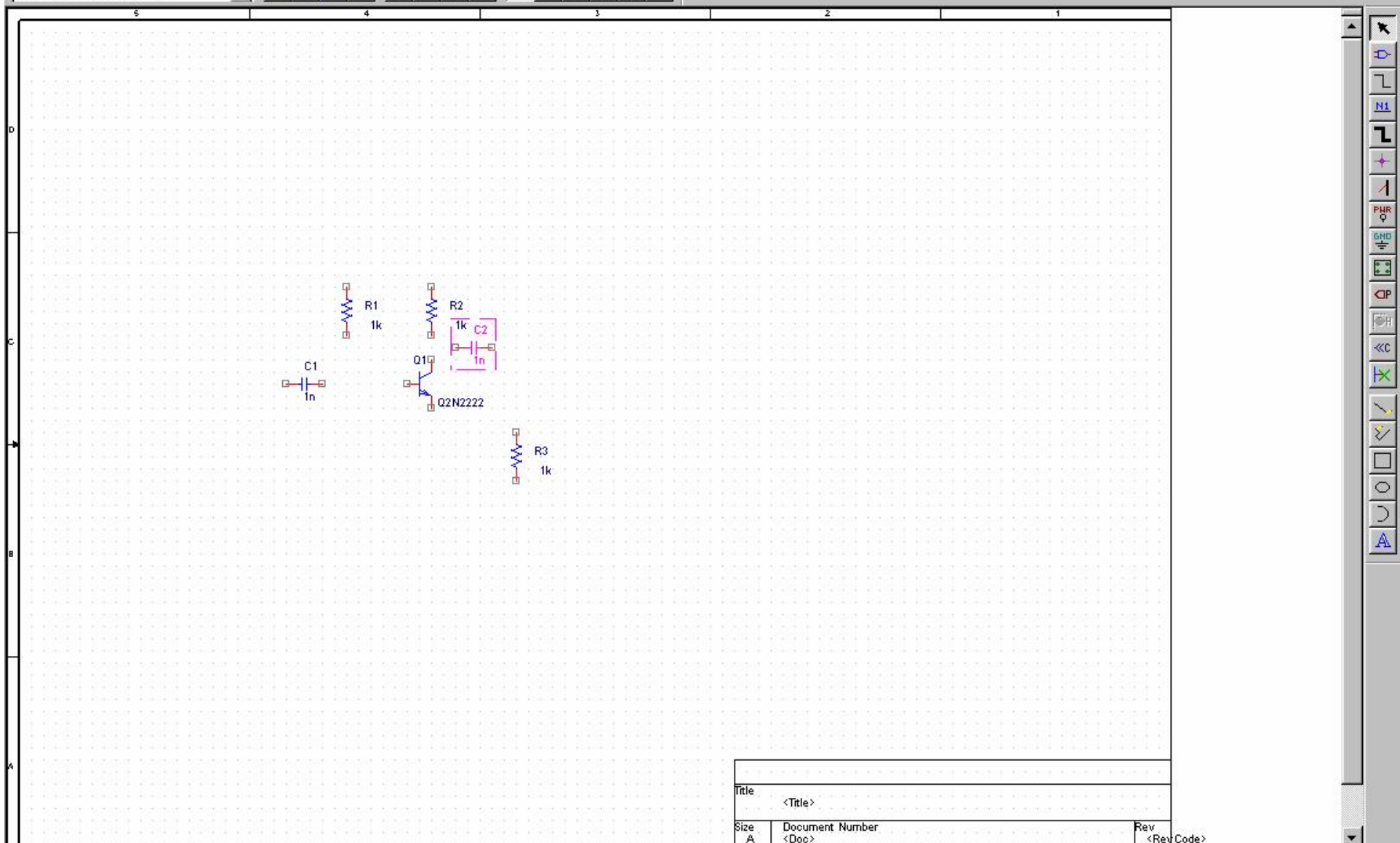
放置电容符号

- 执行Place/Part命令
- 在“Libraries”列表框中选择“ANALOG”
- 在“Part”列表框中选择“C”
- 单击“OK”
- 将电容C移至合适位置，按鼠标左键
- 按ESC键以结束绘制
元器件状态

电子爱好者家园
地: www.cepark.com

专业电子
坛: <http://bbs>





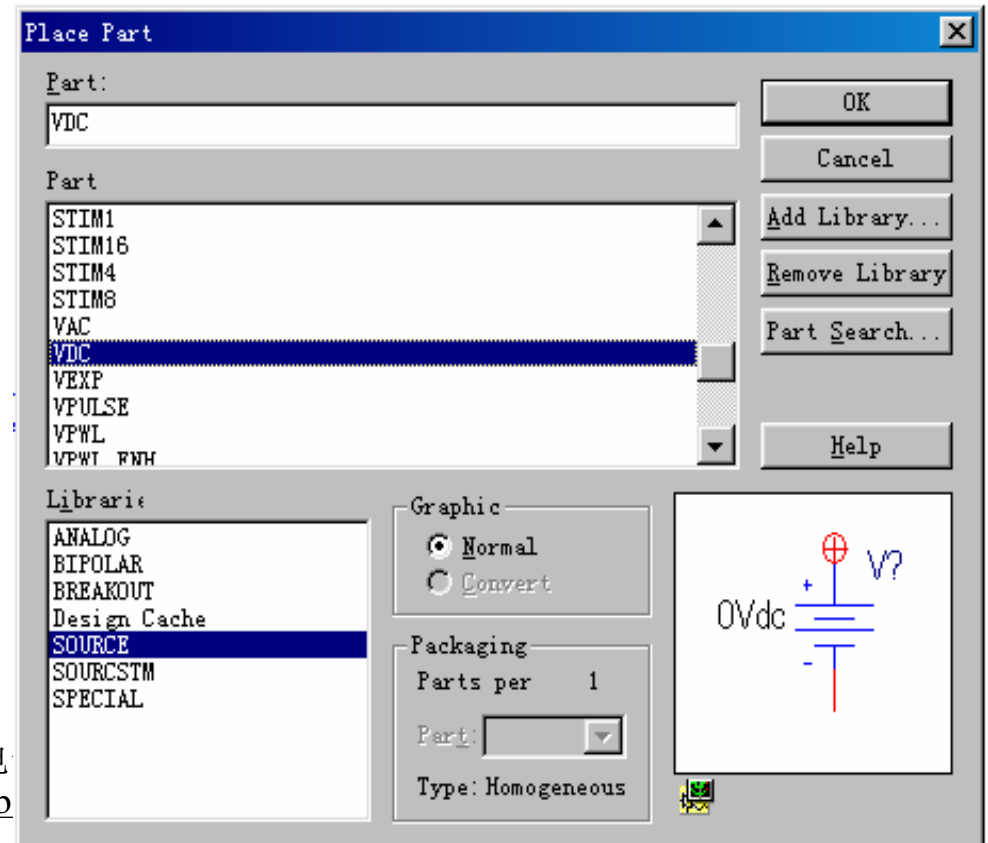
Title	<Title>	Rev	<Rev Code>
Size	A	Document Number	<Doc>

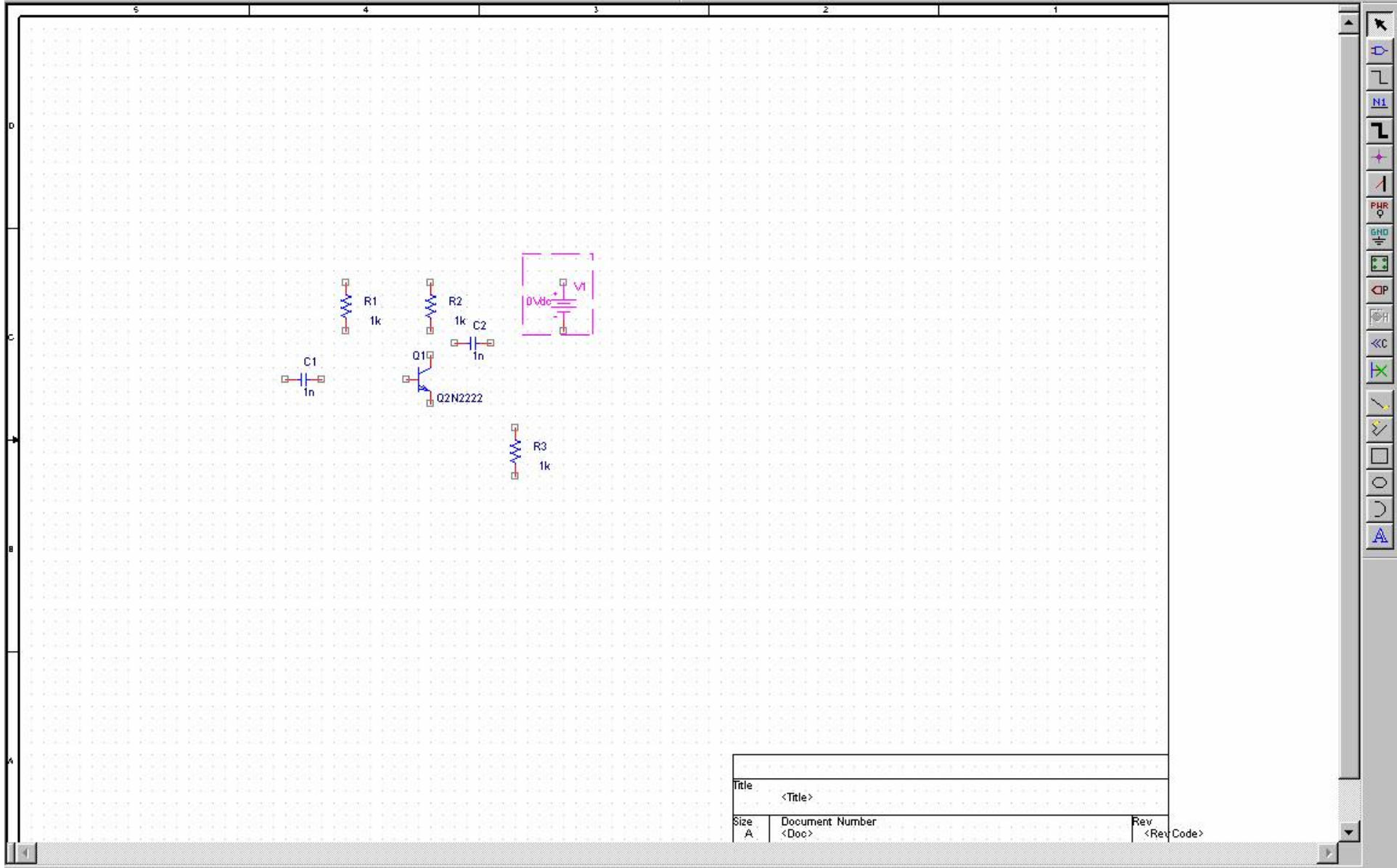
放置直流电源符号

- 执行Place/Part命令
- 在“Libraries”列表框中选择“SOURCE”
- 在“Part”列表框中选择“VDC”
- 单击“OK”
- 将直流源VDC移至合适位置，按鼠标左键
- 按ESC键以结束绘制

元器件爱好家园
地址: www.cepark.com

专业电
坛: <http://bb>





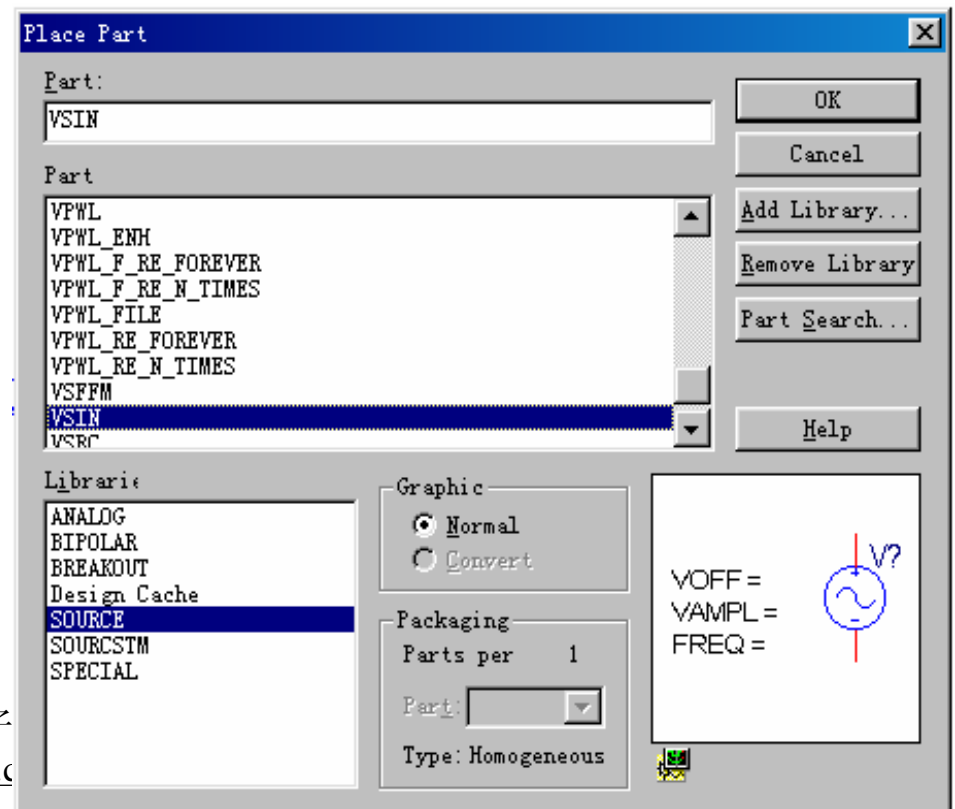
Title	<Title>	Rev	<Rev Code>
Size	Document Number		
A	<Doc>		

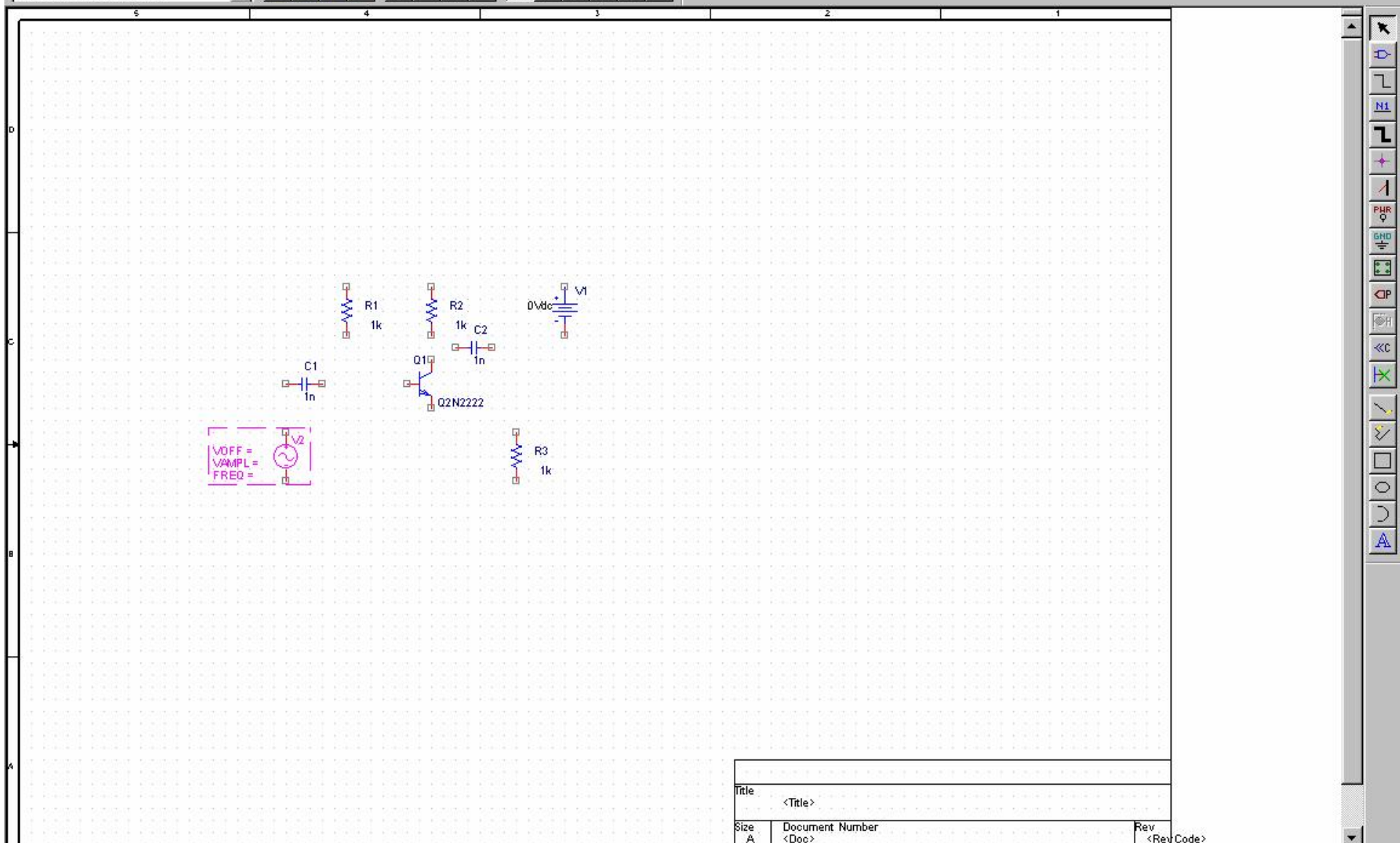
放置激励电源符号

- 执行Place/Part命令
- 在“Libraries”列表框中选择“SOURCE”
- 在“Part”列表框中选择“VSIN”
- 单击“OK”
- 将激励源VSIN移至合适位置，按鼠标左键
- 按ESC键以结束绘制

元器件爱好者家园
地址：www.cepark.com

专业电子
坛：<http://bbs.c>





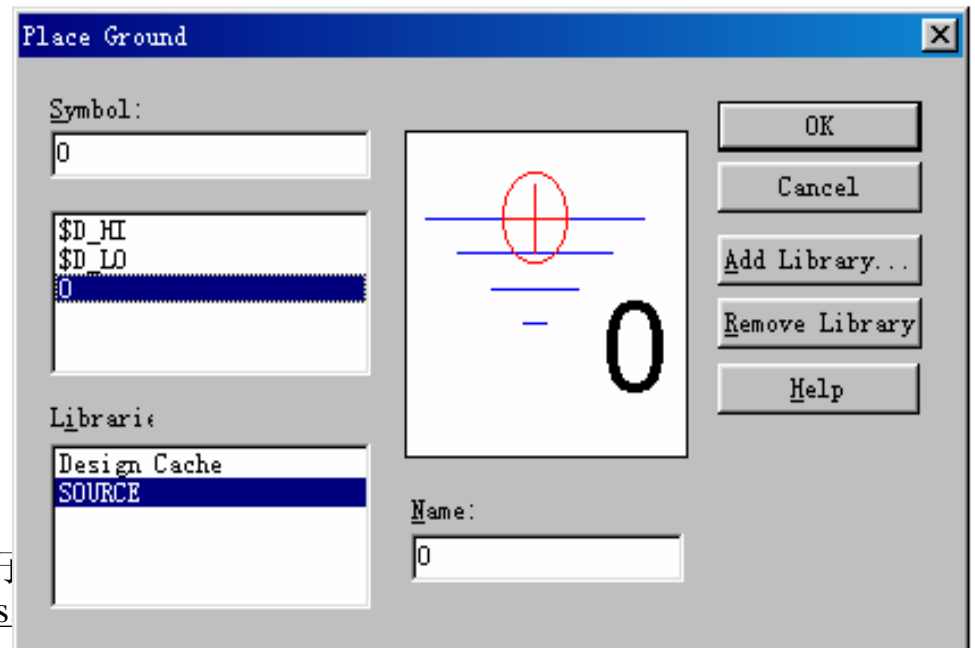
Title	<Title>	Rev	<Rev Code>
Size	Document Number		
A	<Doc>		

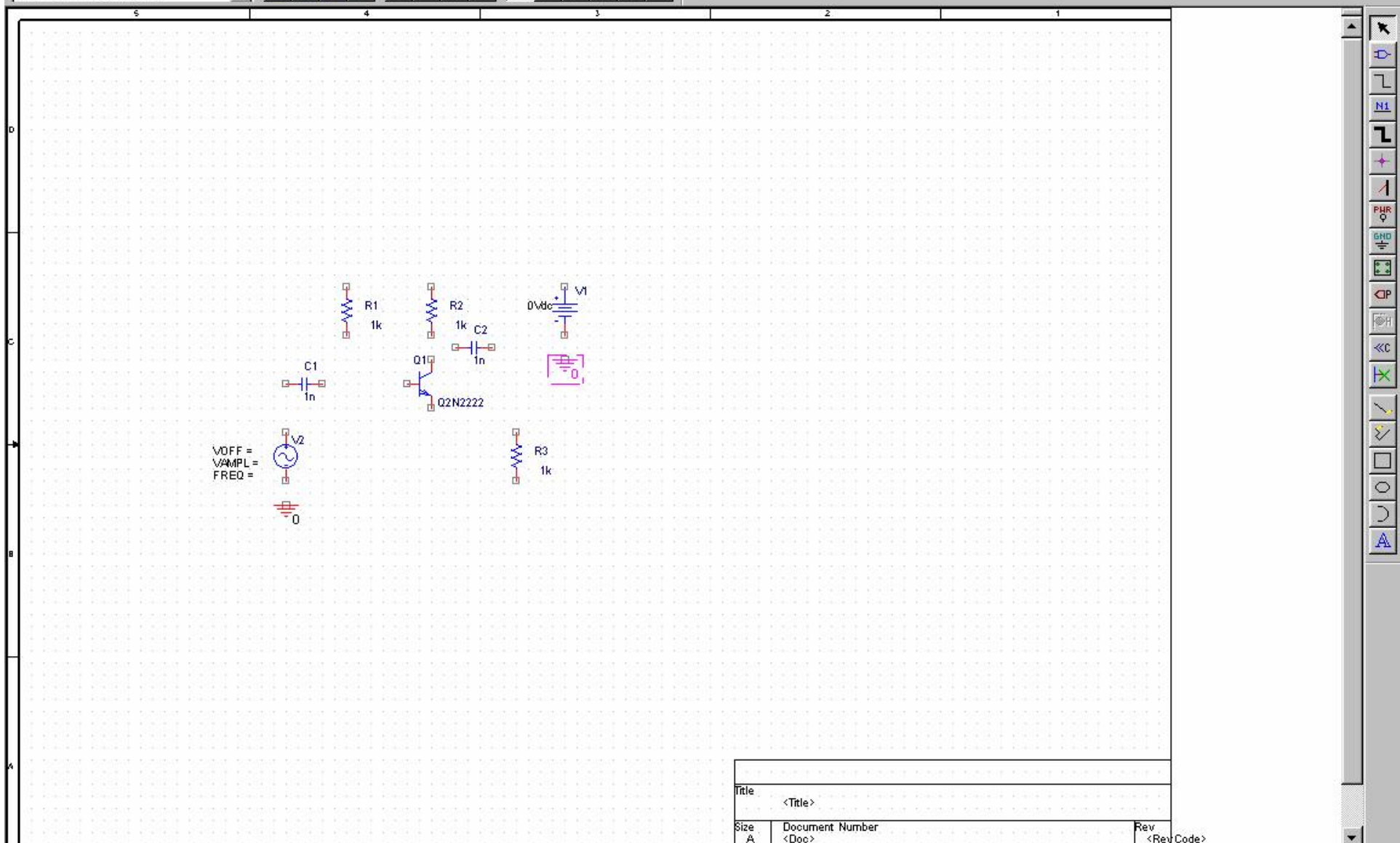
放置地符号

- 执行Place/Ground命令
- 在“Libraries”列表框中选择“SOURCE”
- 在“Symbol”列表框中选择“0”
- 单击“OK”
- 将地符号0 移至合适位置，按鼠标左键
- 按ESC键以结束绘制元

中国电子爱好者乐园
地址: www.cepark.com

专业电子
坛: <http://bbs>

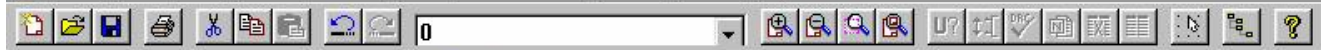




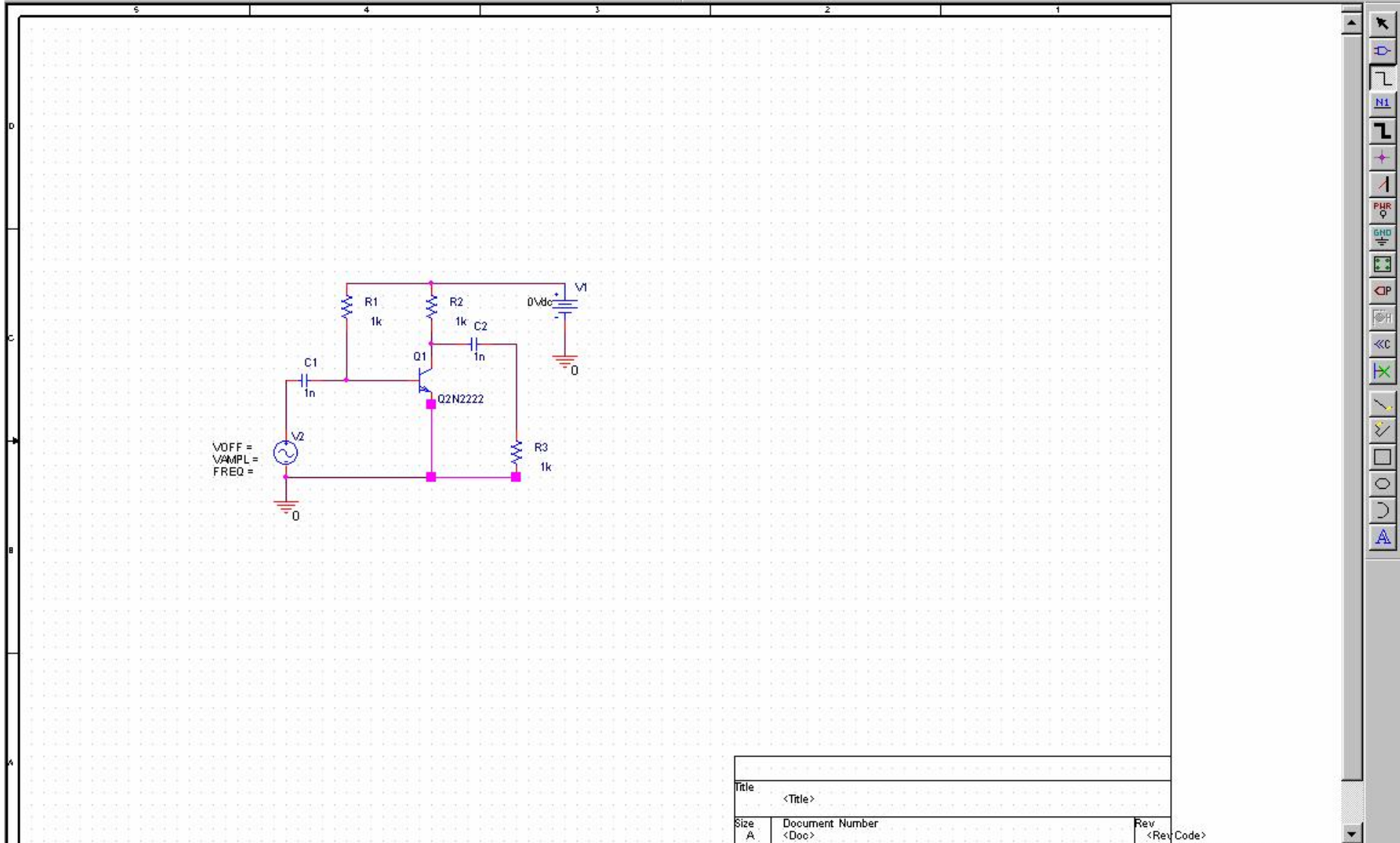
Title	<Title>
Size	Document Number
A	<Doc>
Rev	<Rev Code>

元器件间的电连接

- 执行Place/Wire命令
- 将光标移至互连线的起始位置处，点击鼠标左键
- 移动鼠标，互连线出现
- 在互连线终点，单击鼠标左键
- 继续移动鼠标，以绘制下一段互连线
- 单击鼠标右键，选择End Wire子命令，结束互连线绘制



SCHEMATIC1-Bias



Title	<Title>
Size	Document Number
A	<Doc>
Rev	<Rev Code>

将R1修改为Rb

- 将鼠标对准R1，双击鼠标左键，出现“Display Properties”窗口
- 在“Value”栏填入“Rb”
- 单击“OK”

Orcad Capture - [/ - (SCHEMATIC1 : PAGE1)]

File Edit View Place Macro PSpice Accessories Options Window Help

SCHEMATIC1-Bias

The circuit schematic includes a voltage source V2 with parameters: VOFF = 0, VAMPL = 1, and FREQ = 1. It is connected to a network of components: capacitor C1 (1nF), resistor R1 (1k), resistor R2 (1k), capacitor C2 (1nF), and a transistor Q1 (Q2N2222).

Display Properties

Name: Part Reference
 Value: R1

Display Format

- Do Not Display
- Value Only
- Name and Value
- Name Only
- Both if Value Ex

Font: Arial 7 (default)
 Change... Use Default

Color: Default

Rotation:

- 0°
- 90°
- 180°
- 270°

OK Cancel Help

Title: <Title>

Size: A	Document Number: <Doc>	Rev: <Rev Code>
---------	------------------------	-----------------

1 item selected Scale=100% X=2.90 Y=2.40

将R1值由1k修改为560k

- 将鼠标对准R1值1k，双击鼠标左键，出现“Display Properties”窗口
- 在“Value”栏填入“560k”
- 单击“OK”

Orcad Capture - [/ - (SCHEMATIC1 : PAGE1)]

File Edit View Place Macro PSpice Accessories Options Window Help

SCHEMATIC1-Bias

Display Properties

Name: Value
 Value:

Font: Arial 7 (default)
 Change... Use Default

Color: Default

Rotation: 0° 180°
 90° 270°

Display Format:
 Do Not Display
 Value Only
 Name and Value
 Name Only
 Both if Value Ex

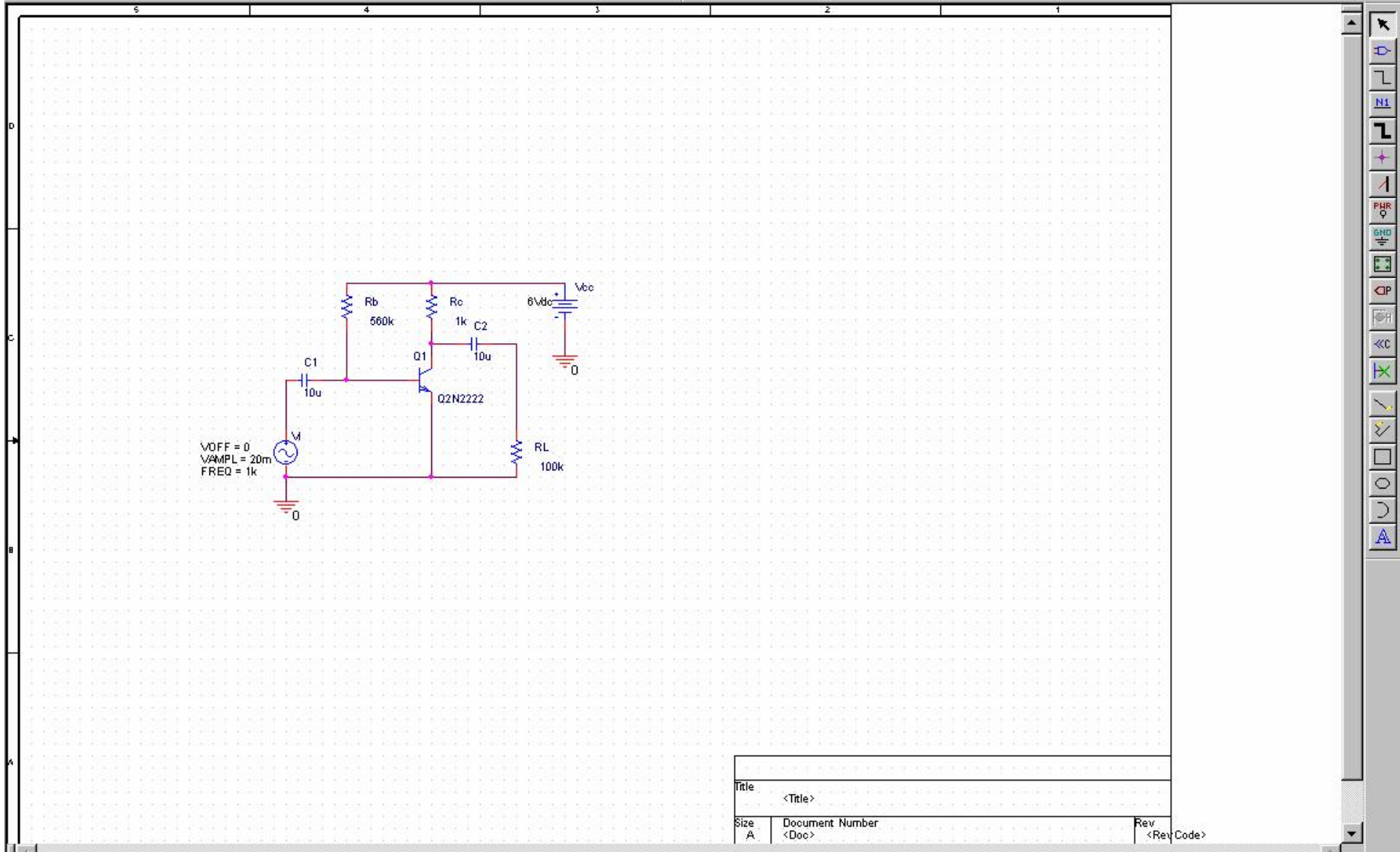
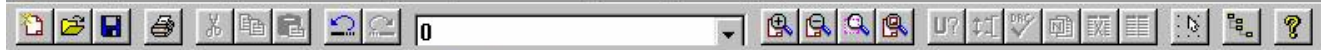
OK Cancel Help

Title: <Title>
 Size: A Document Number Rev: <Rev Code>

1 item selected Scale=100% X=3.00 Y=2.50

修改其余元件属性参数

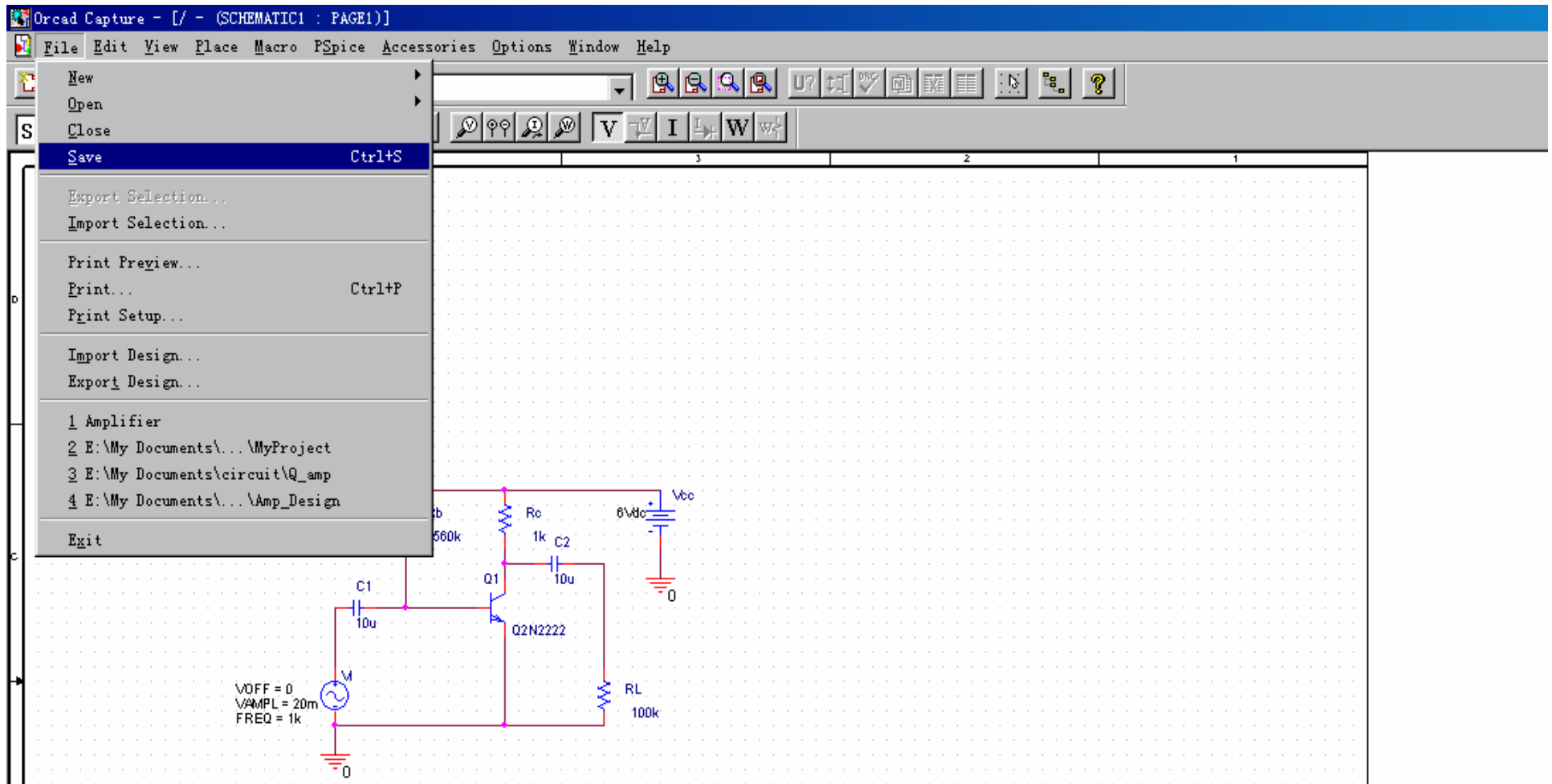
- 将R2修改为Rc
- 将R3修改为RL, 100k
- 将C1、C2值修改为10u
- 将V1修改为Vcc, 6V
- 将V2修改为Vi, VOFF=0,
VAMP=20m, FREQ=1k



Title	<Title>	Rev	<Rev Code>
Size	Document Number		
A	<Doc>		

电路原理图保存

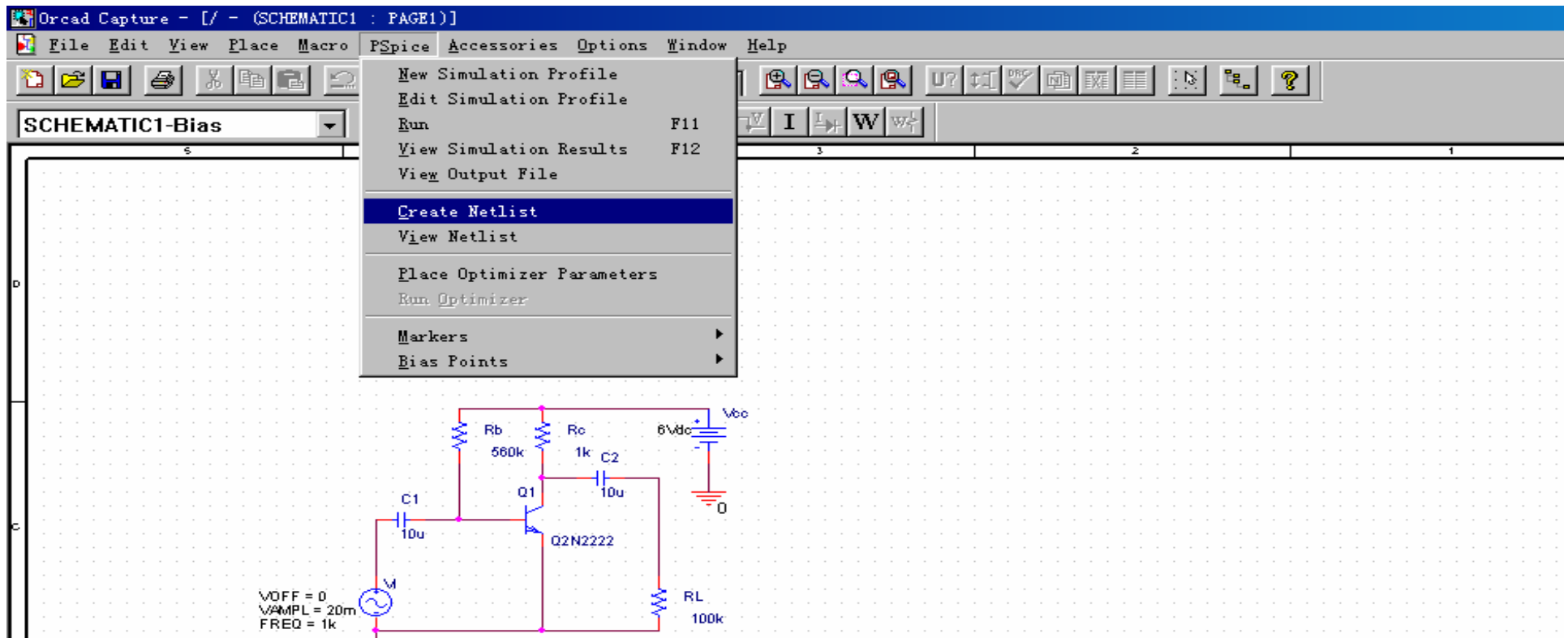
- 执行File/Save命令



(二) 电路的仿真(瞬态分析)

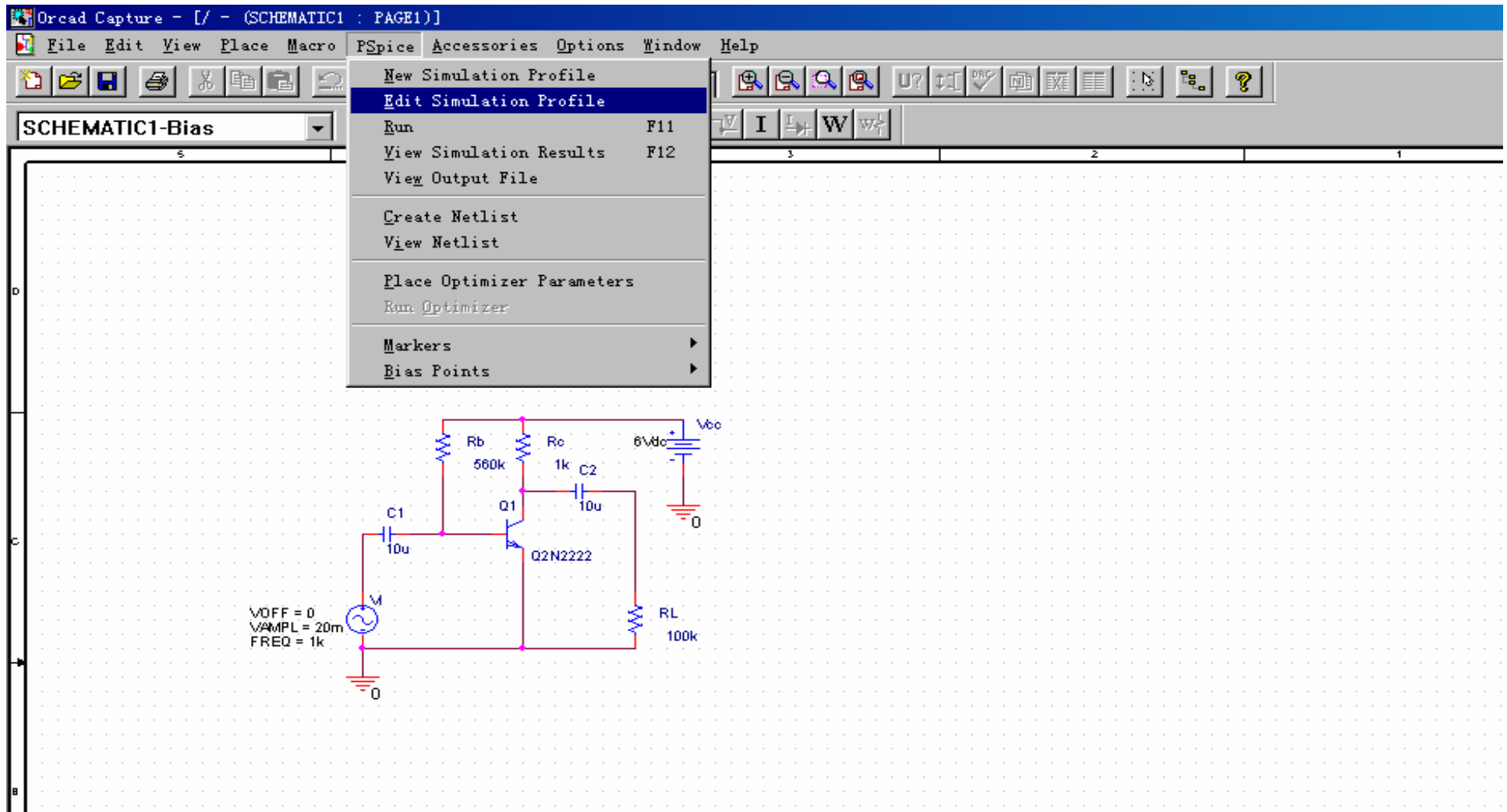
1、建立电路网表

- 执行PSpice/Create Netlist命令



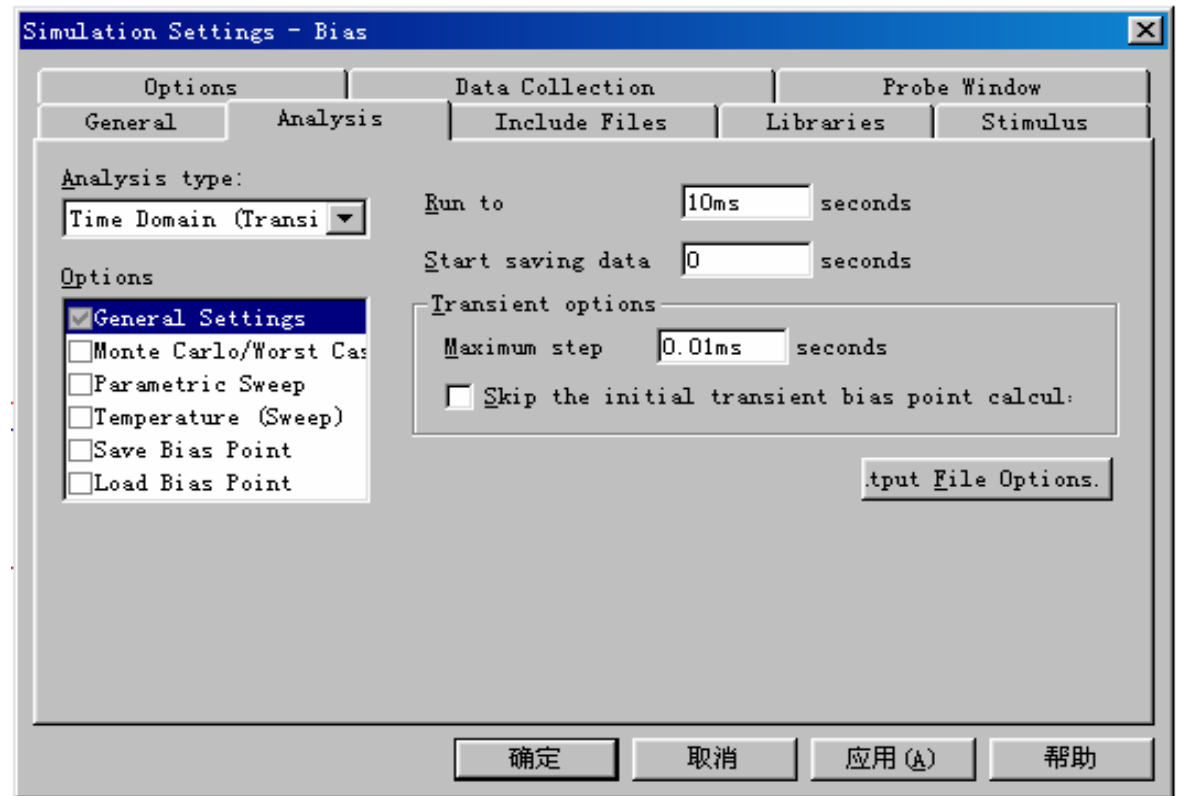
2、仿真参数类型设置

- 执行PSpice/Edit Simulation Profile命令



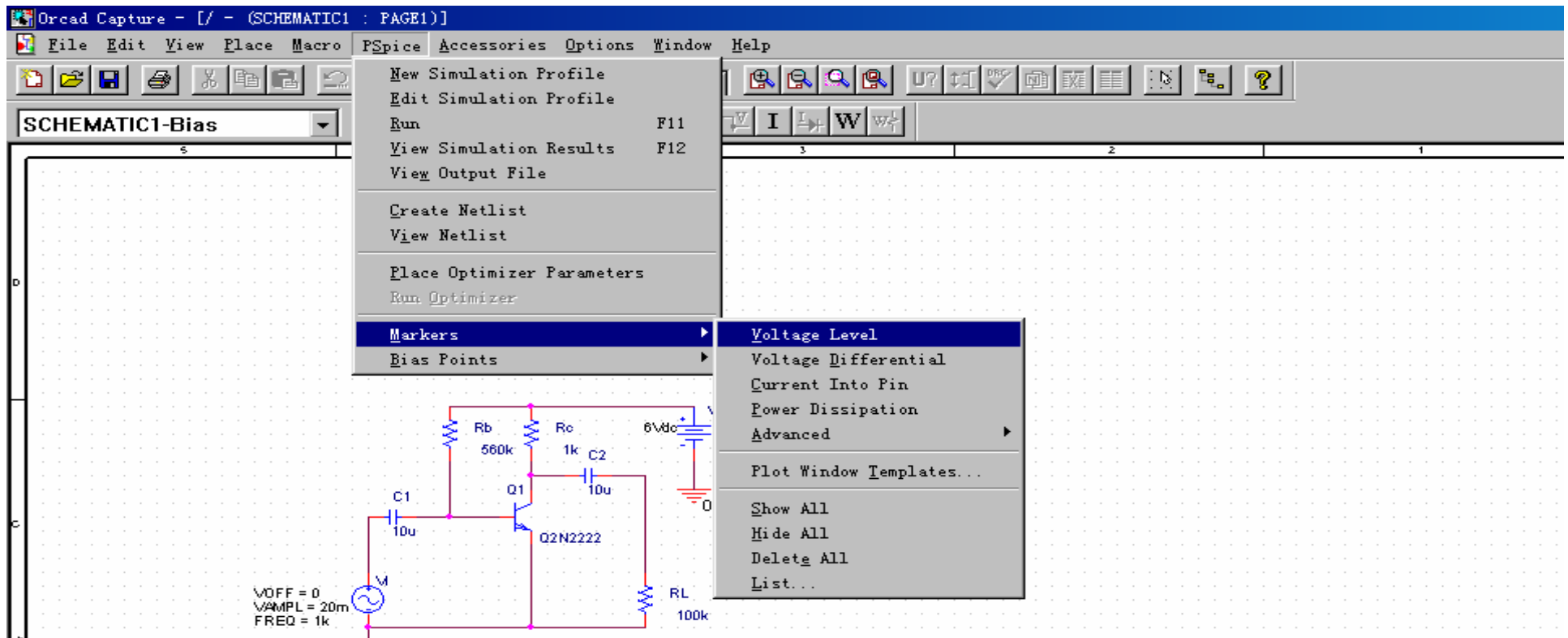
- 在Analysis Type栏，选择Time Domain(Transient)
- 在Start saving data栏填写0ms
- 在Run to栏填写10ms
- 在Maximum step栏，填写0.01ms
- 点击“确定”

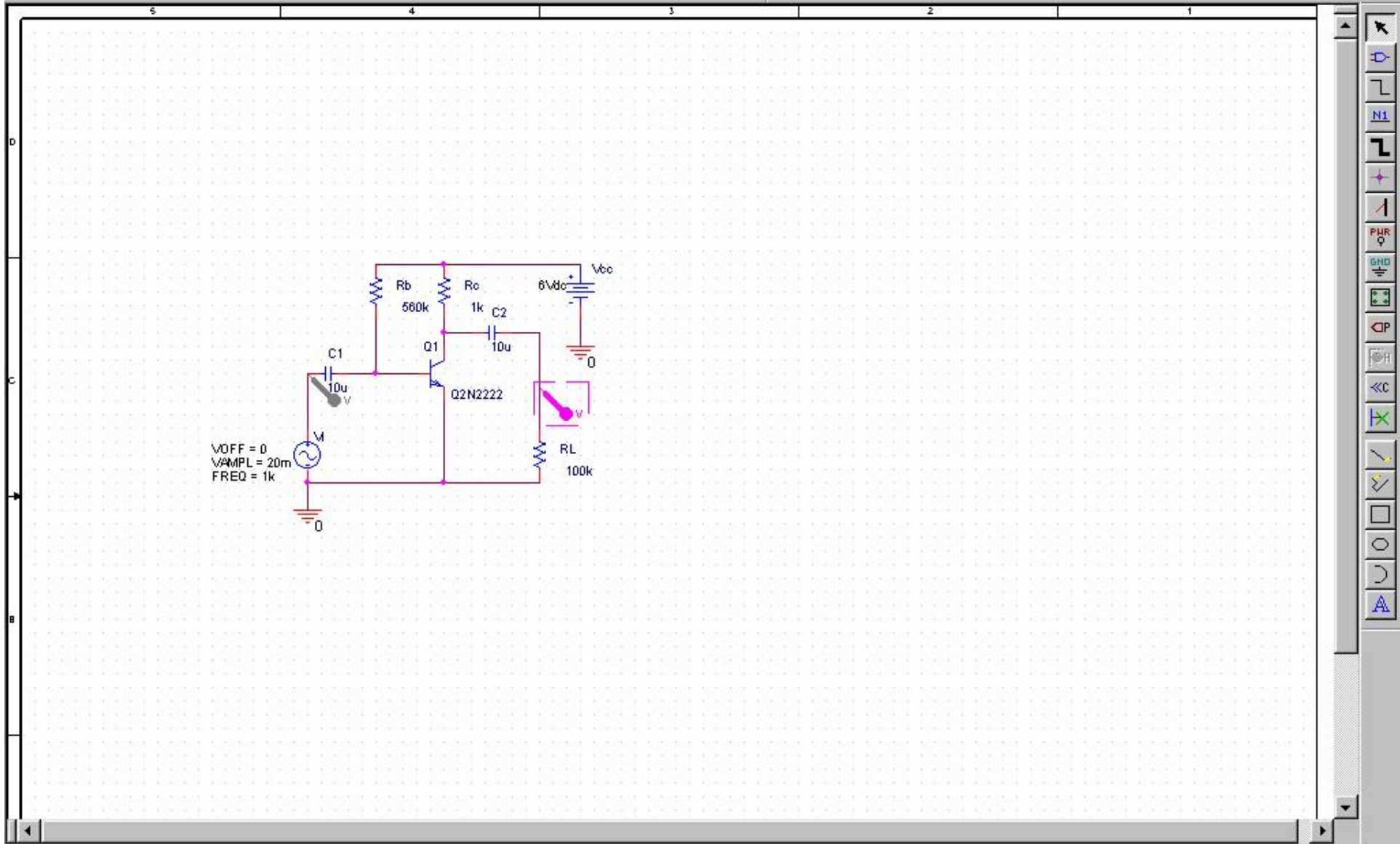
中国电子爱好者园
地址: www.cepark.com



3、放置仪器探头

- 执行PSpice/Markers/Voltage Level命令
- 将电压探头拖至输入端Vi、输出端RL处
- 按ESC键，以结束仪器探头放置

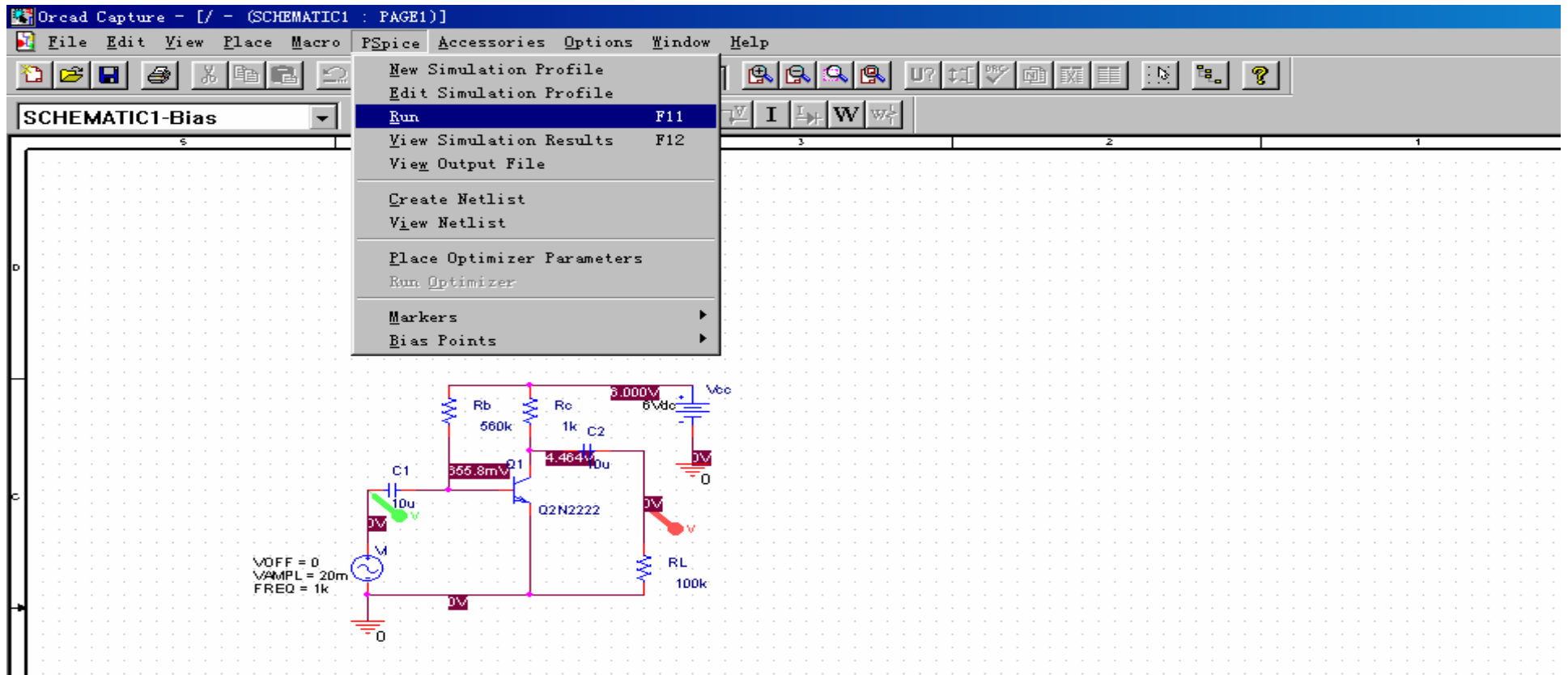


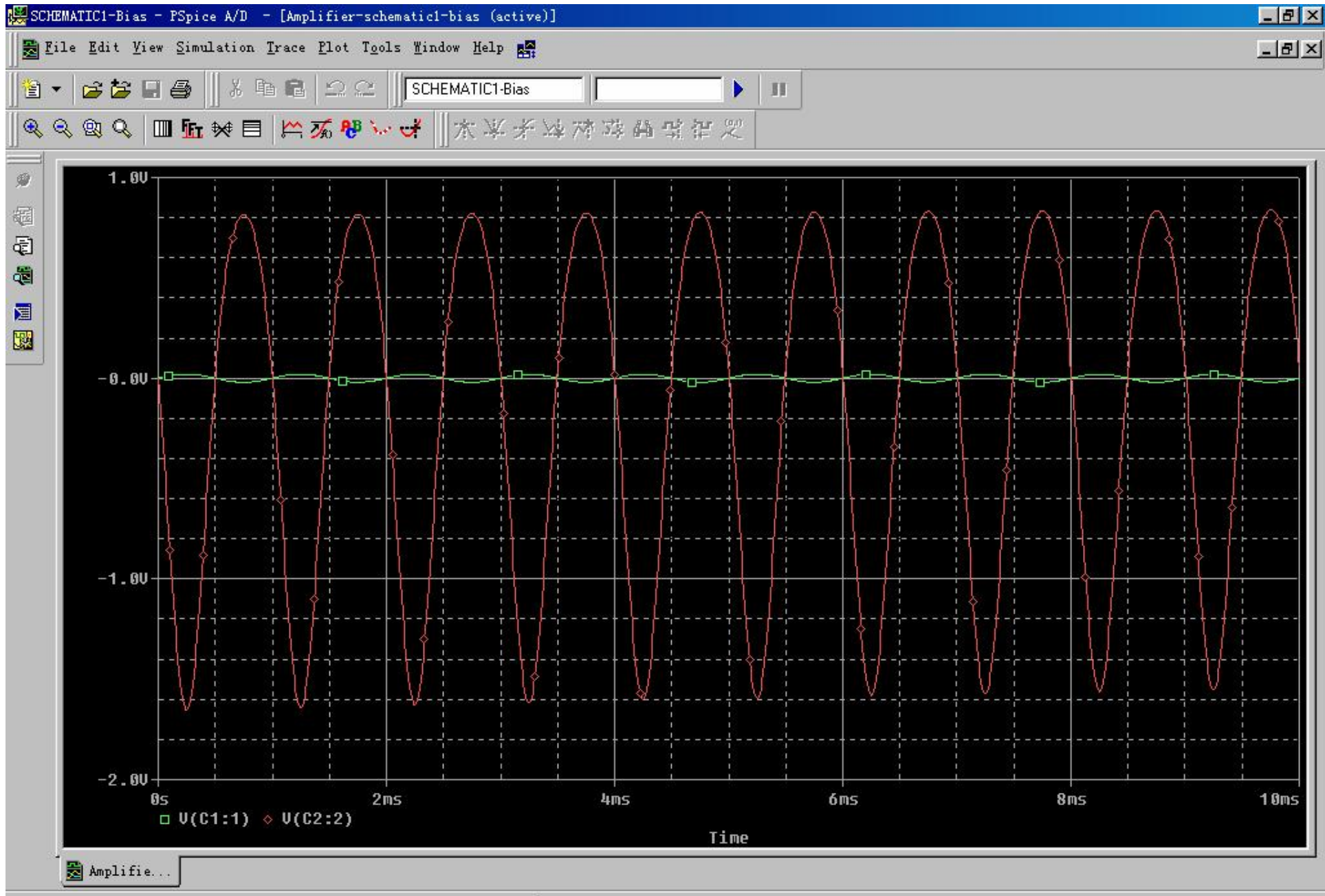


1 item selected Scale=100% X=4.80 Y=2.80

4、运行仿真分析程序

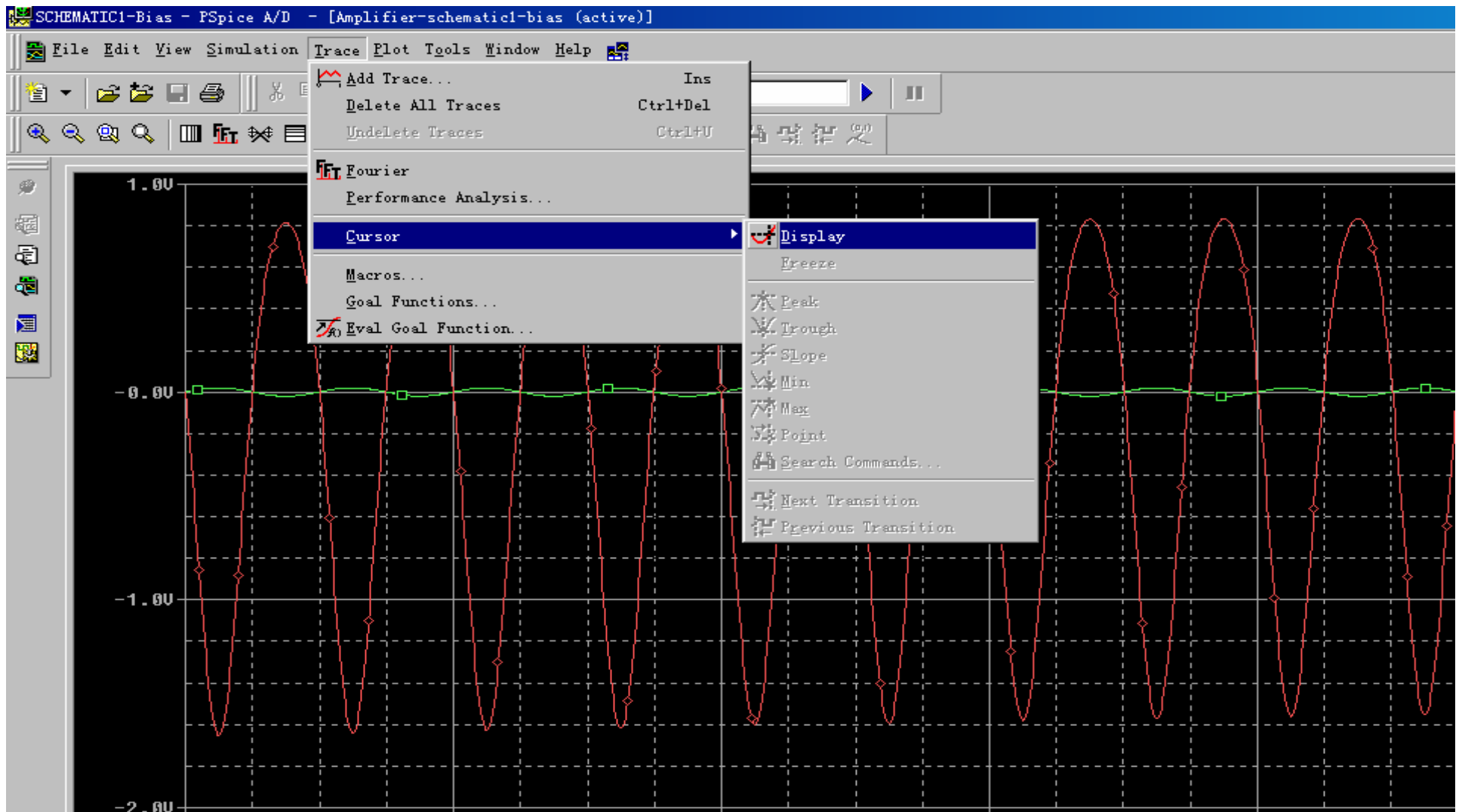
- 执行PSpice/Run命令
- 屏幕上出现PSpice仿真分析窗口



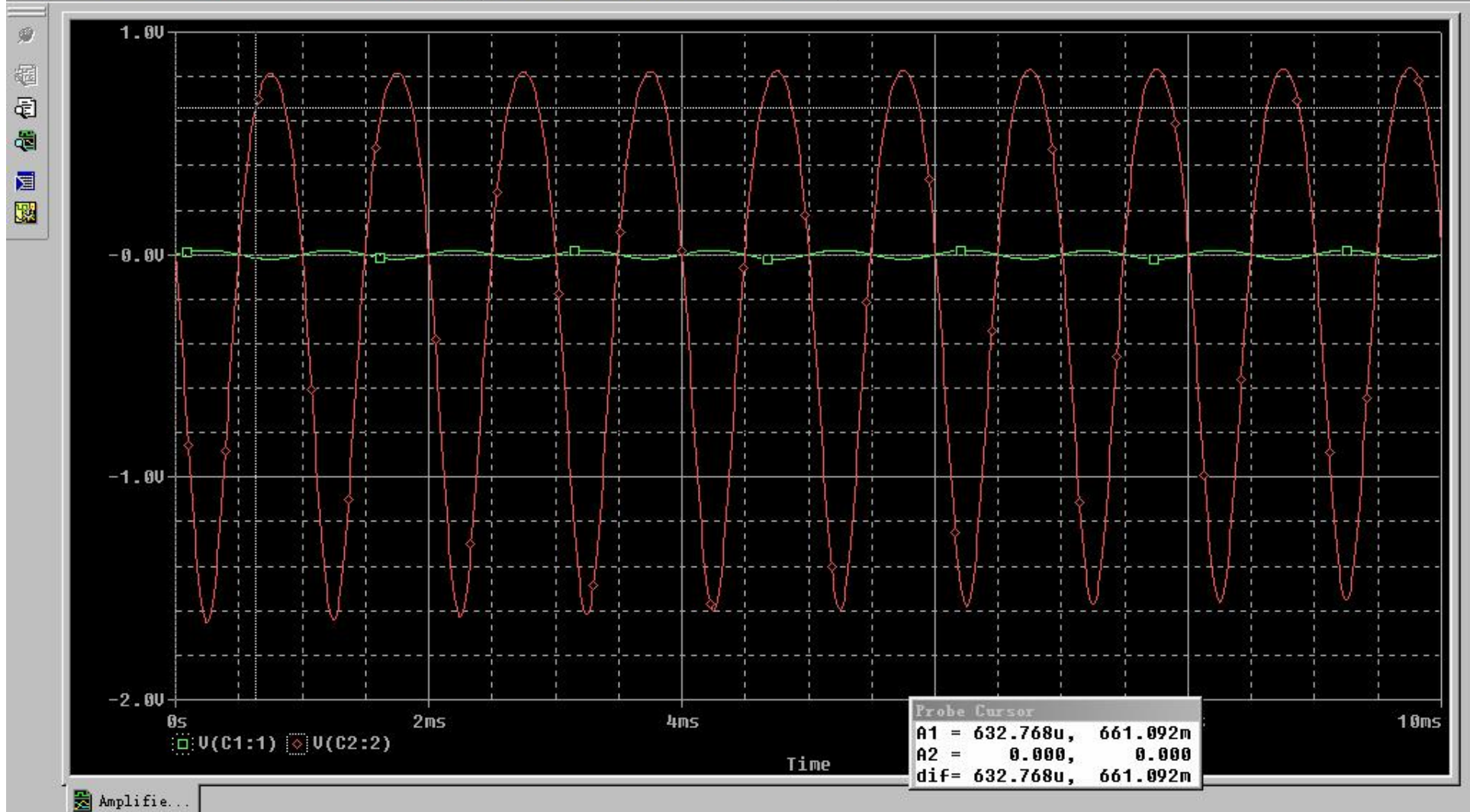


5、波形测量

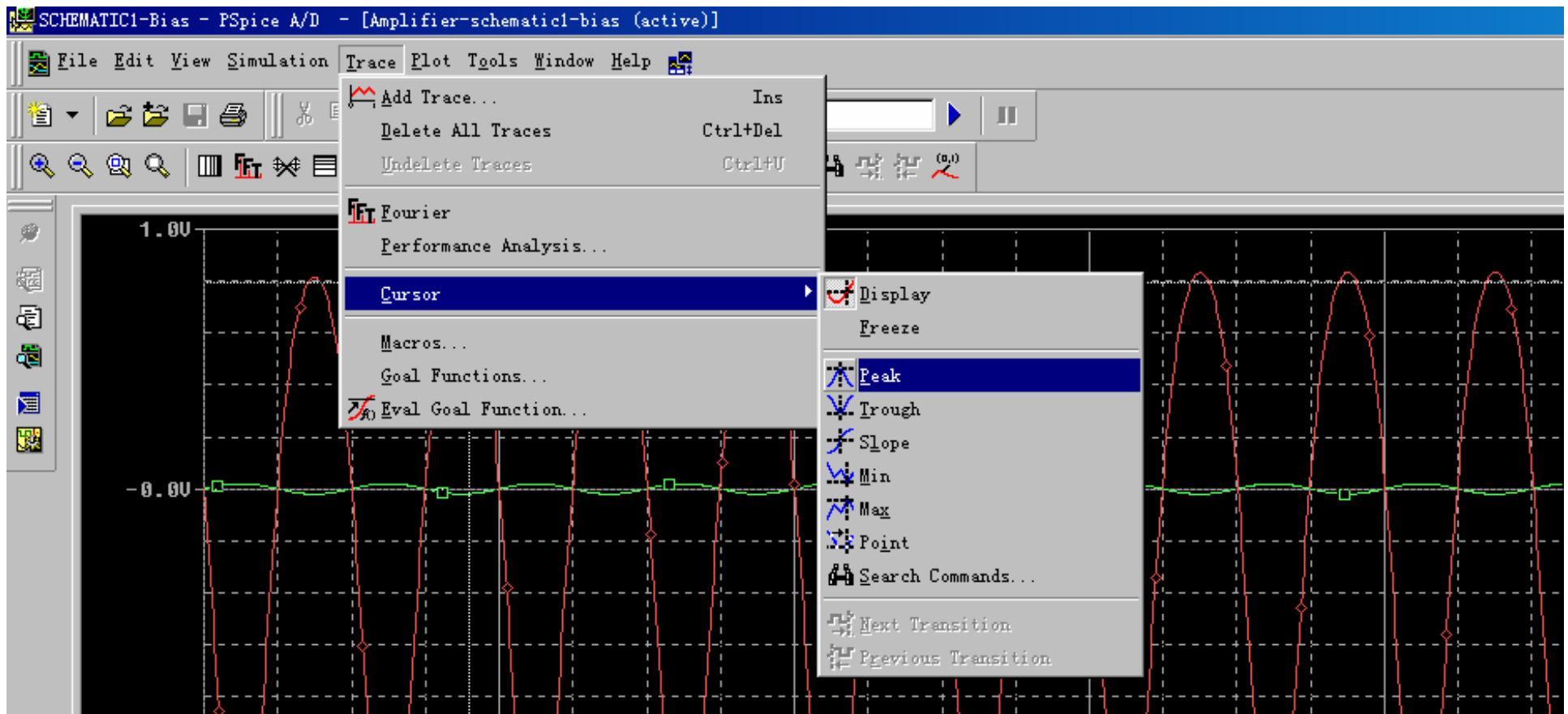
- 执行Trace/Cursor/Display命令



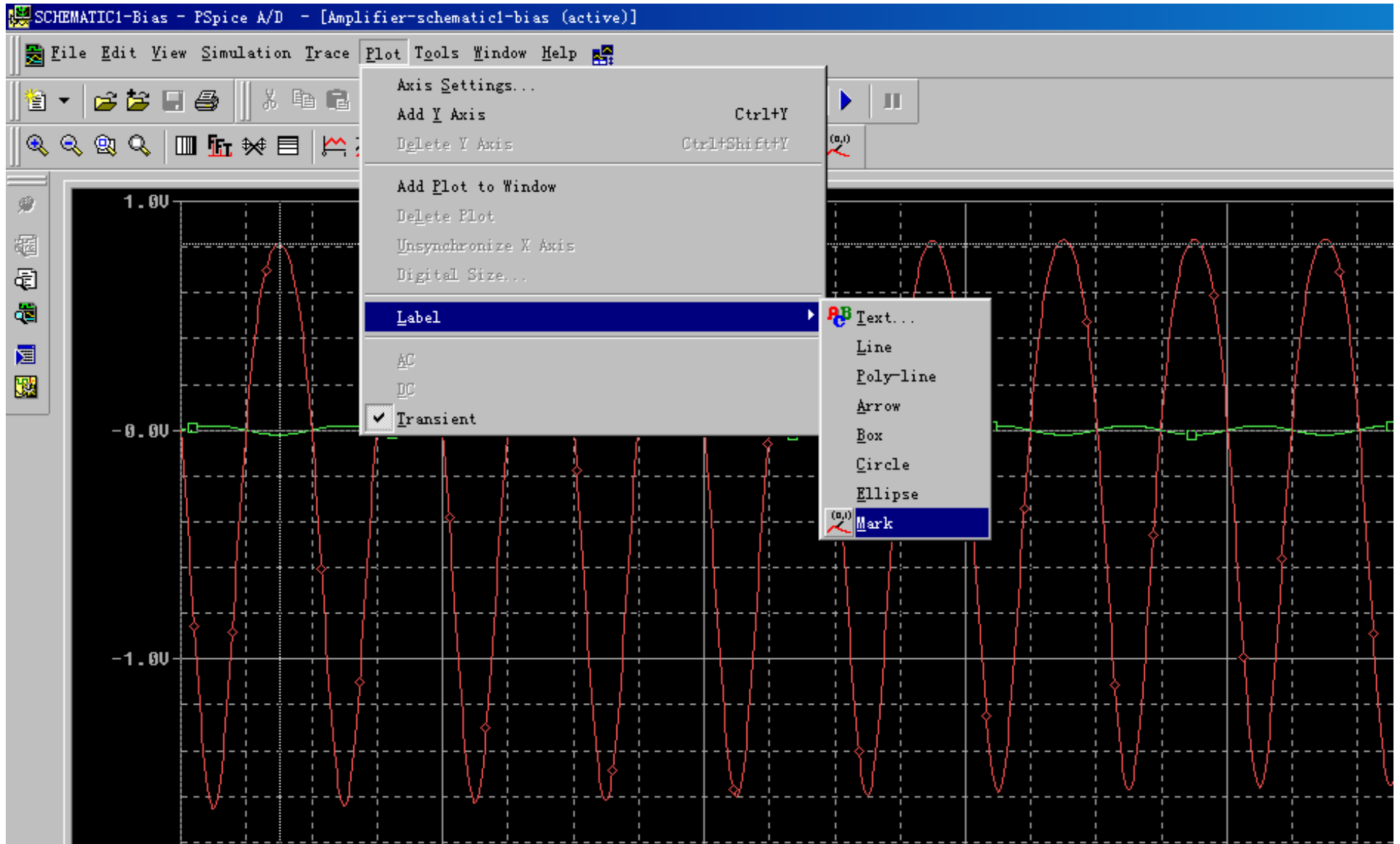
SCHEMATIC1-Bias



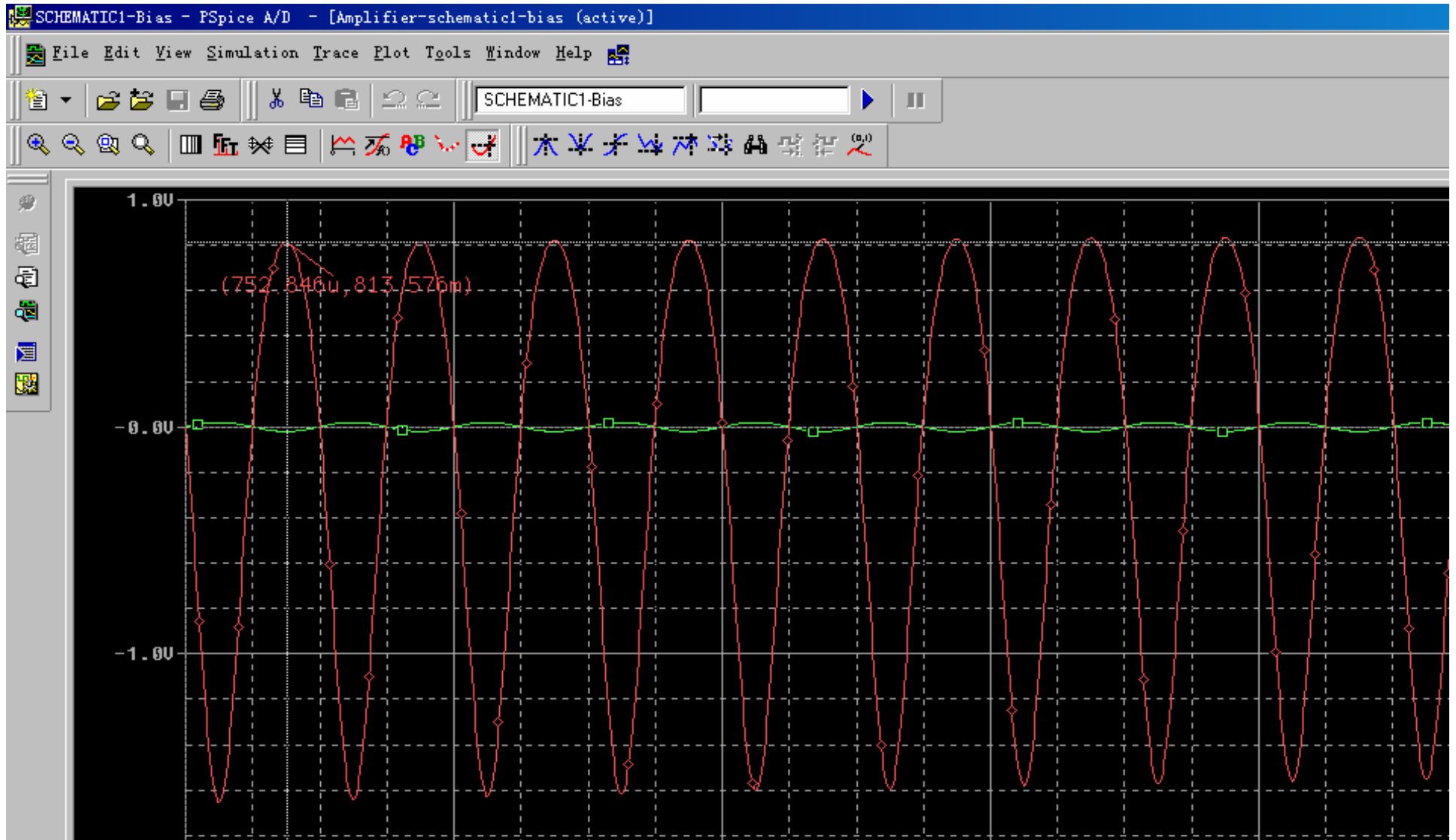
- 点击分析窗口左下角“V(C2:2)”，选择输出端C2与RL连接处的输出电压波形
- 执行Trace/Cursor/Peak命令，测量标尺定位于输出波形顶峰



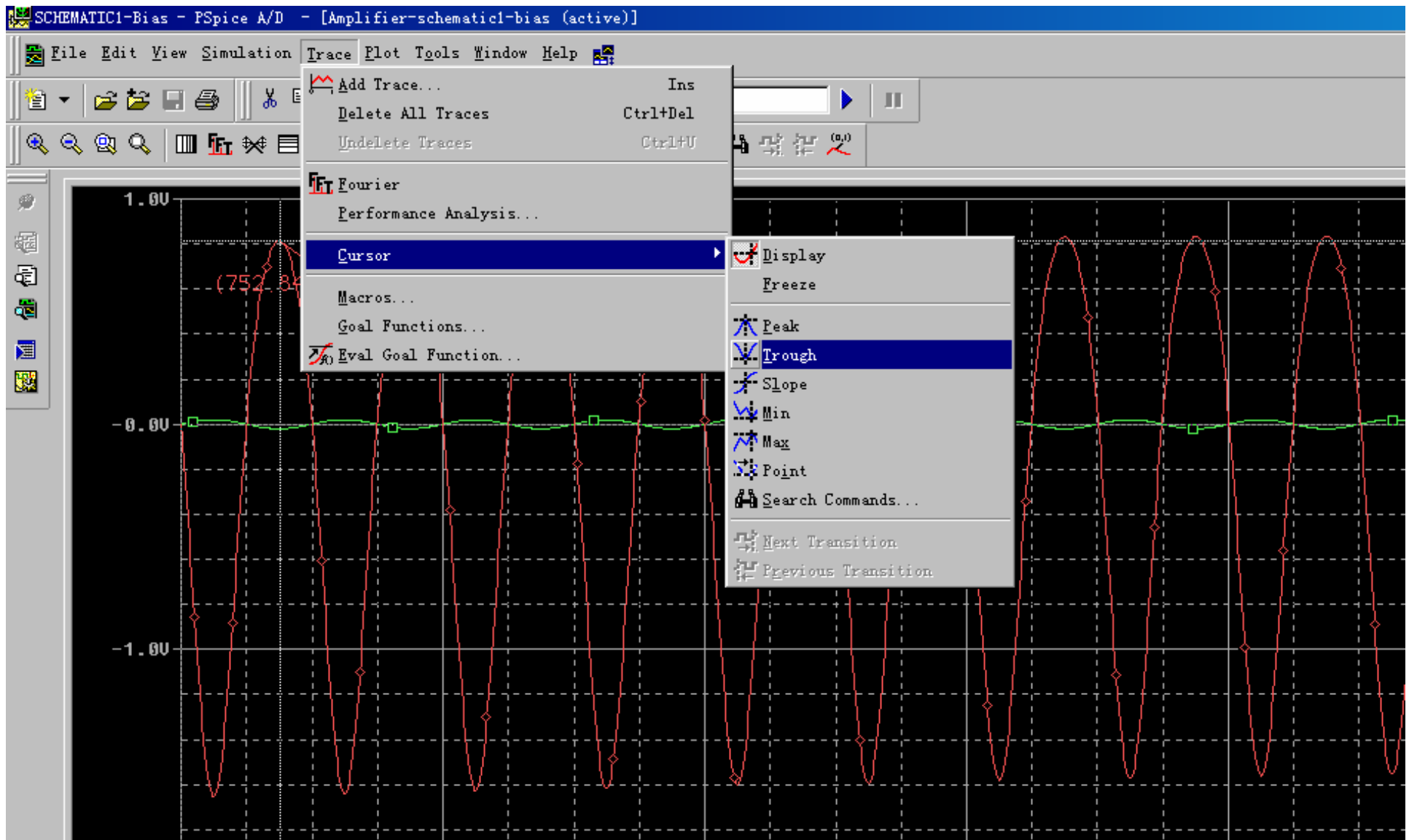
- 执行Plot/Label/Mark命令，显示输出波形顶峰标尺坐标



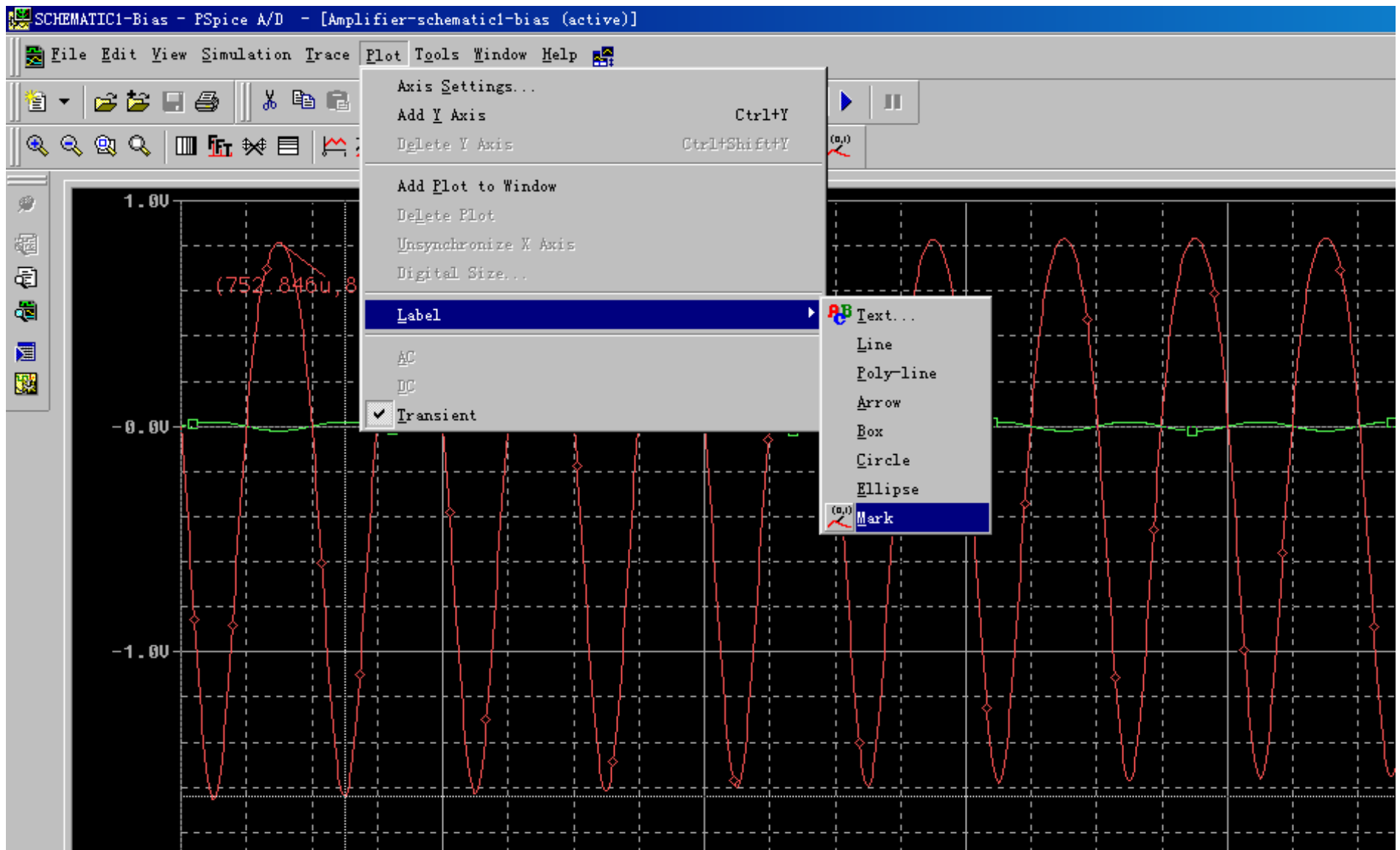
- 第一位置坐标为顶峰处时间数值
- 第二位置坐标为顶峰处电压数值



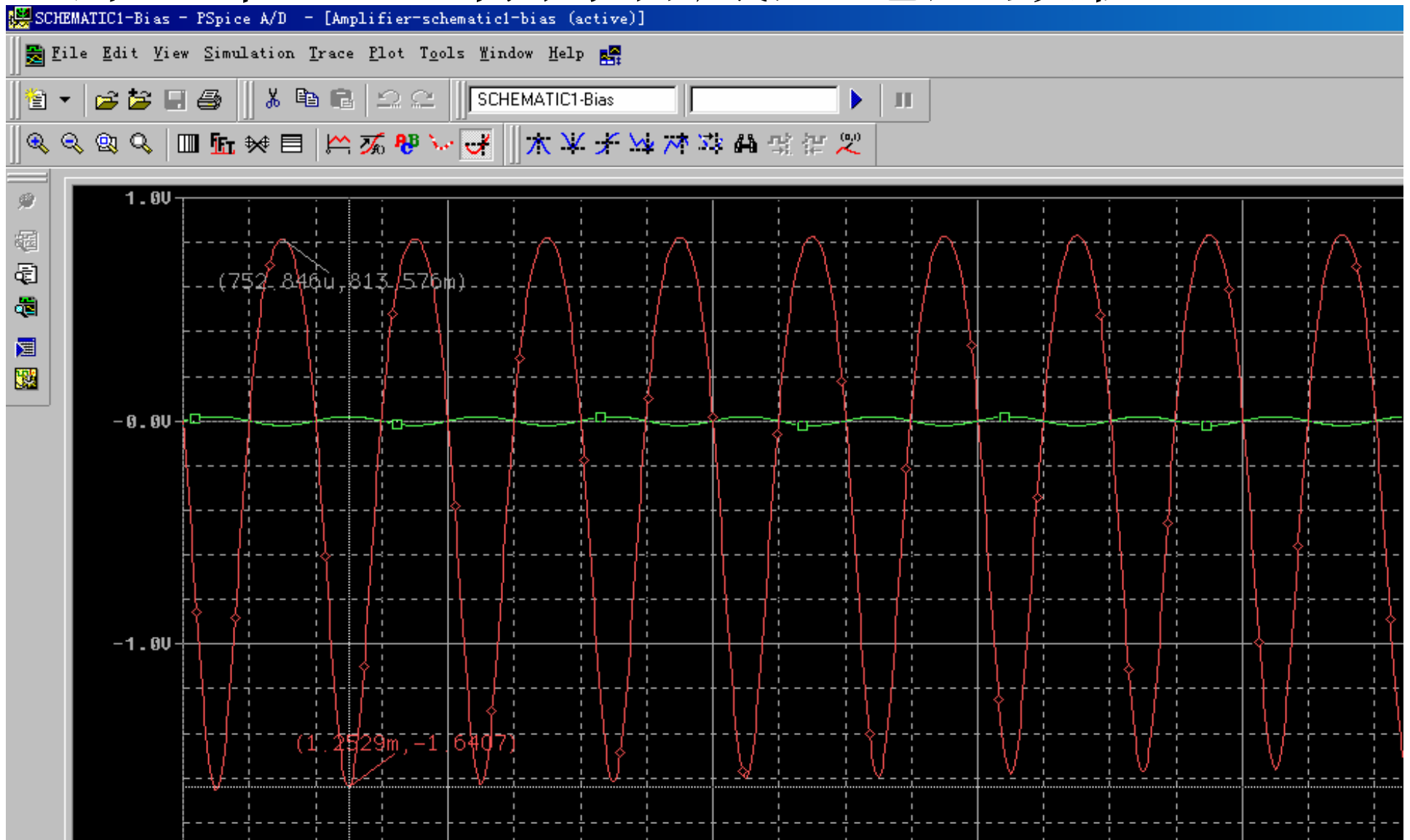
- 执行Trace/Cursor/Trough命令，测量标尺定位于输出波形谷底



- 执行Plot/Label/Mark命令，显示输出波形谷底标尺坐标



- 第一位置坐标为谷底处时间数值
- 第二位置坐标为谷底处电压数值



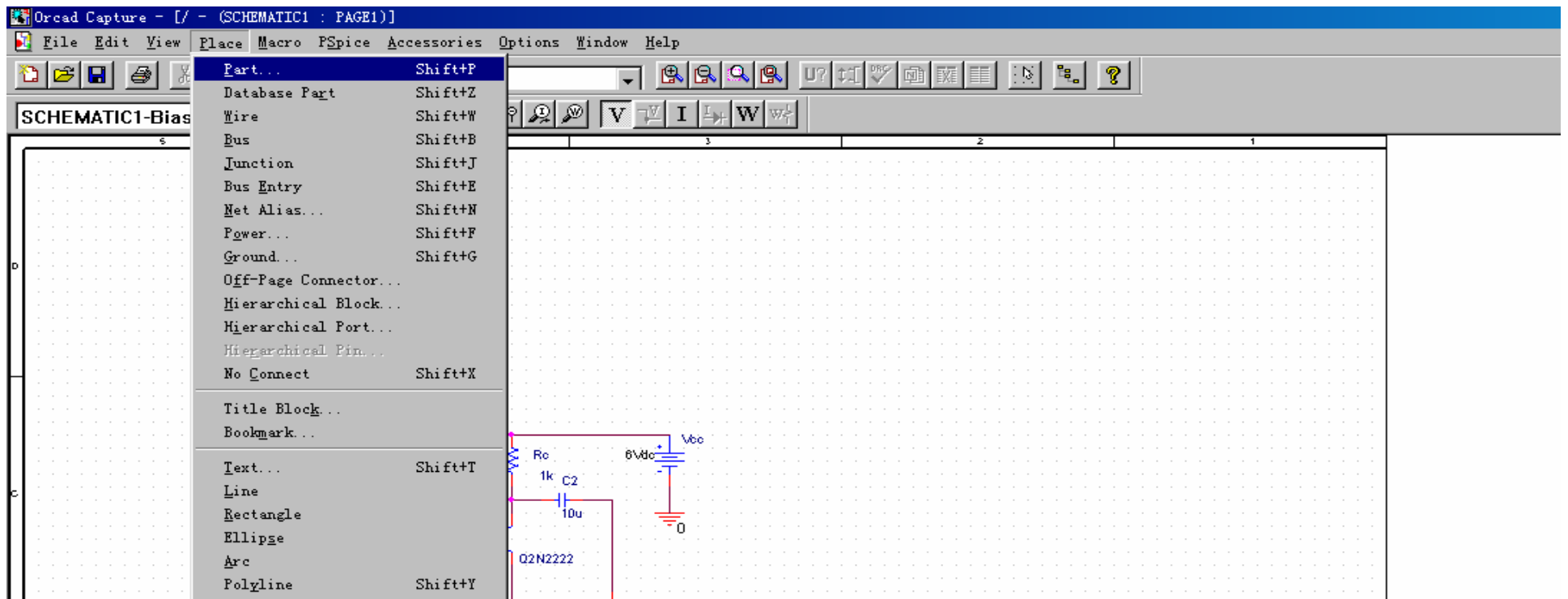
输出峰峰值以及系统增益

- 将顶峰处电压数值与谷底处电压数值相减，得到输出波形峰峰值
 $V_{opp}=0.813576-(-1.6407)=2.4543V$
- 输出波形峰峰值 V_{opp} 与输入波形峰峰值 V_{ipp} 相除，得到系统放大增益
 $A_v=V_{opp}/V_{ipp}=2.4543V/40mV=61$

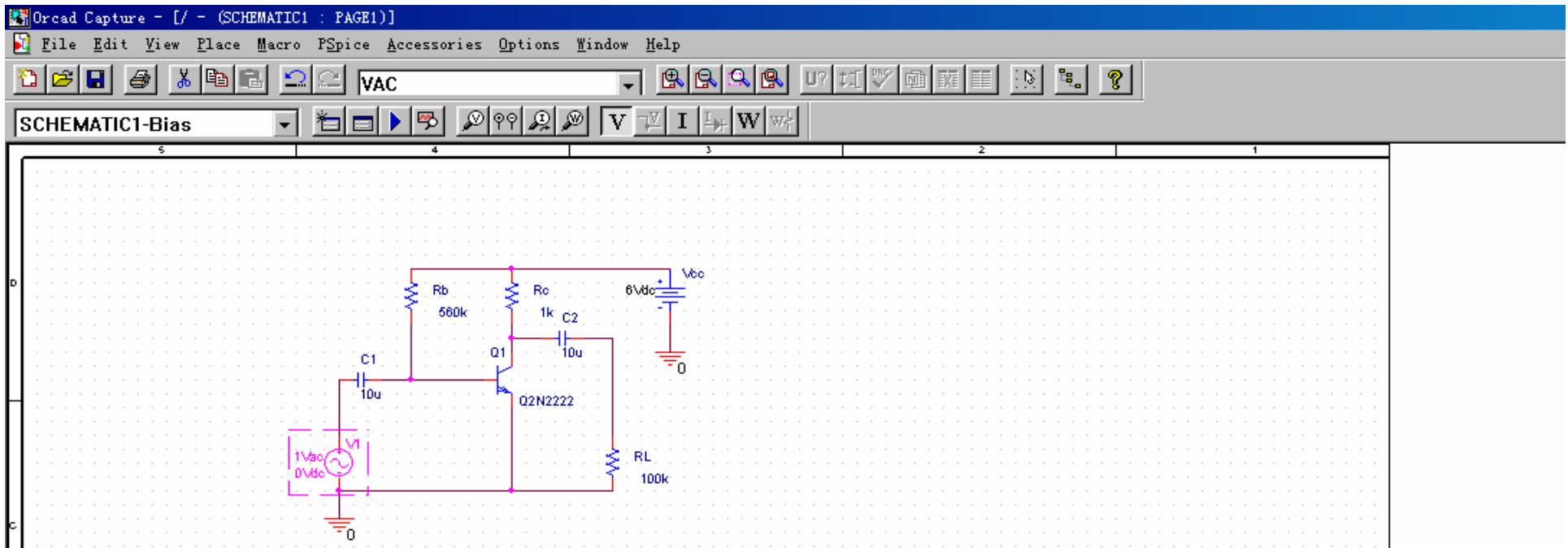
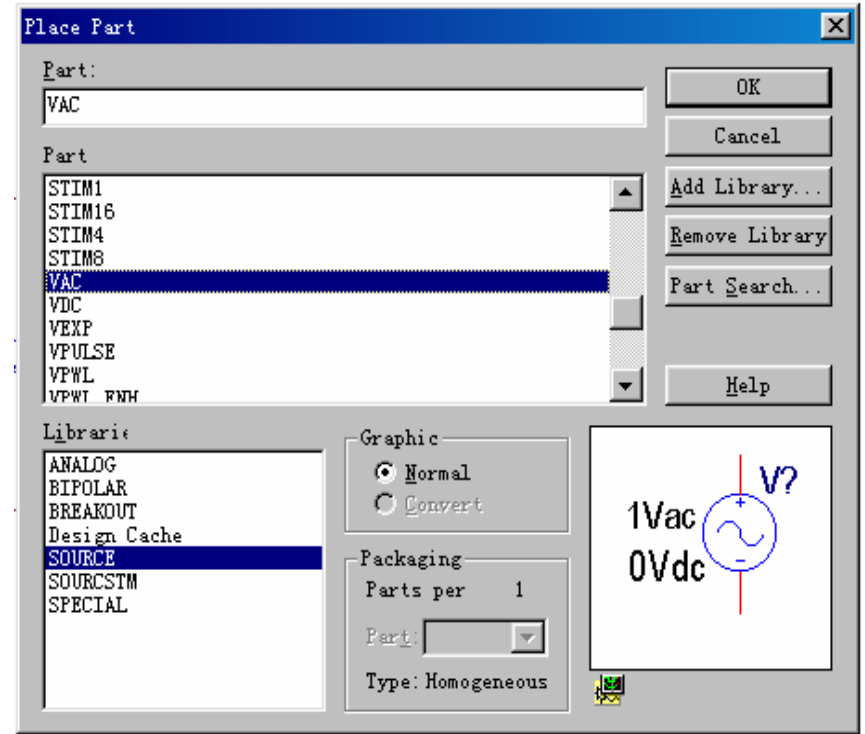
(三) 电路的仿真(交流分析)

1、更换激励信号源

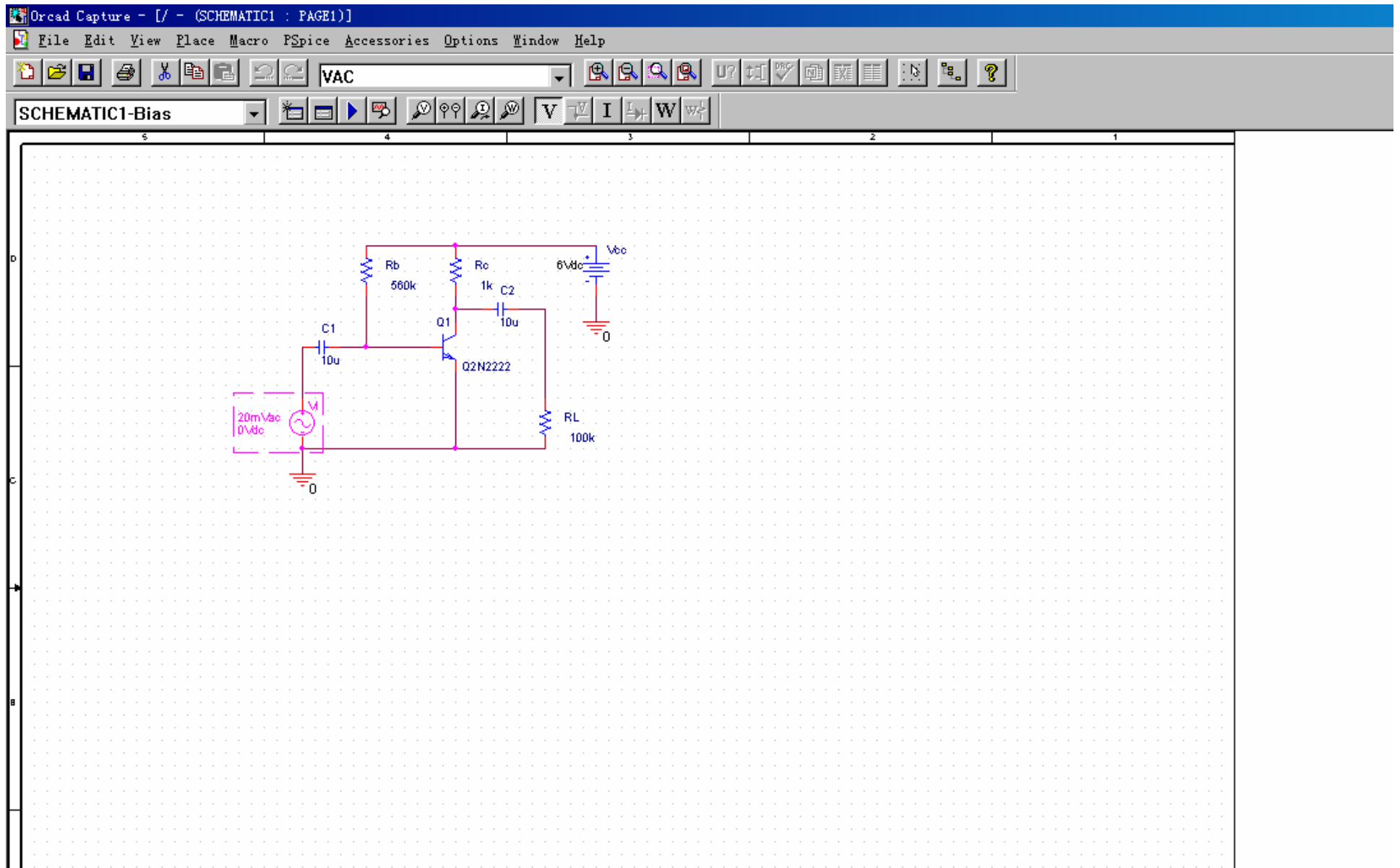
- 删除电压仪探头、信号源VSIN
- 执行Place/Part命令，放置信号源VAC



- 在“Libraries”列表中选择SOURCE
- 在“PART”列表中选择VAC
- 单击“OK”

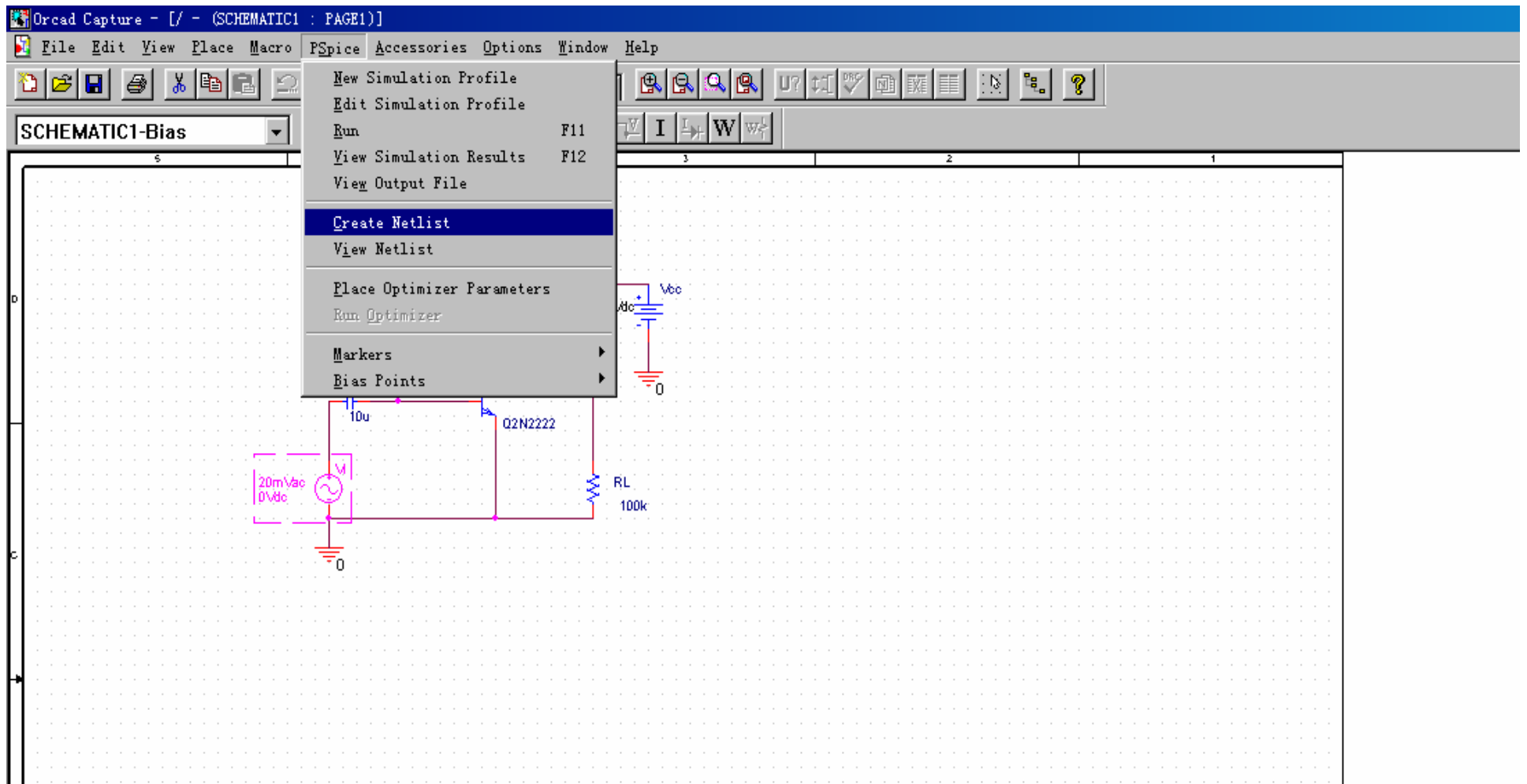


- 将V1修改为Vi、20mVac、0Vdc



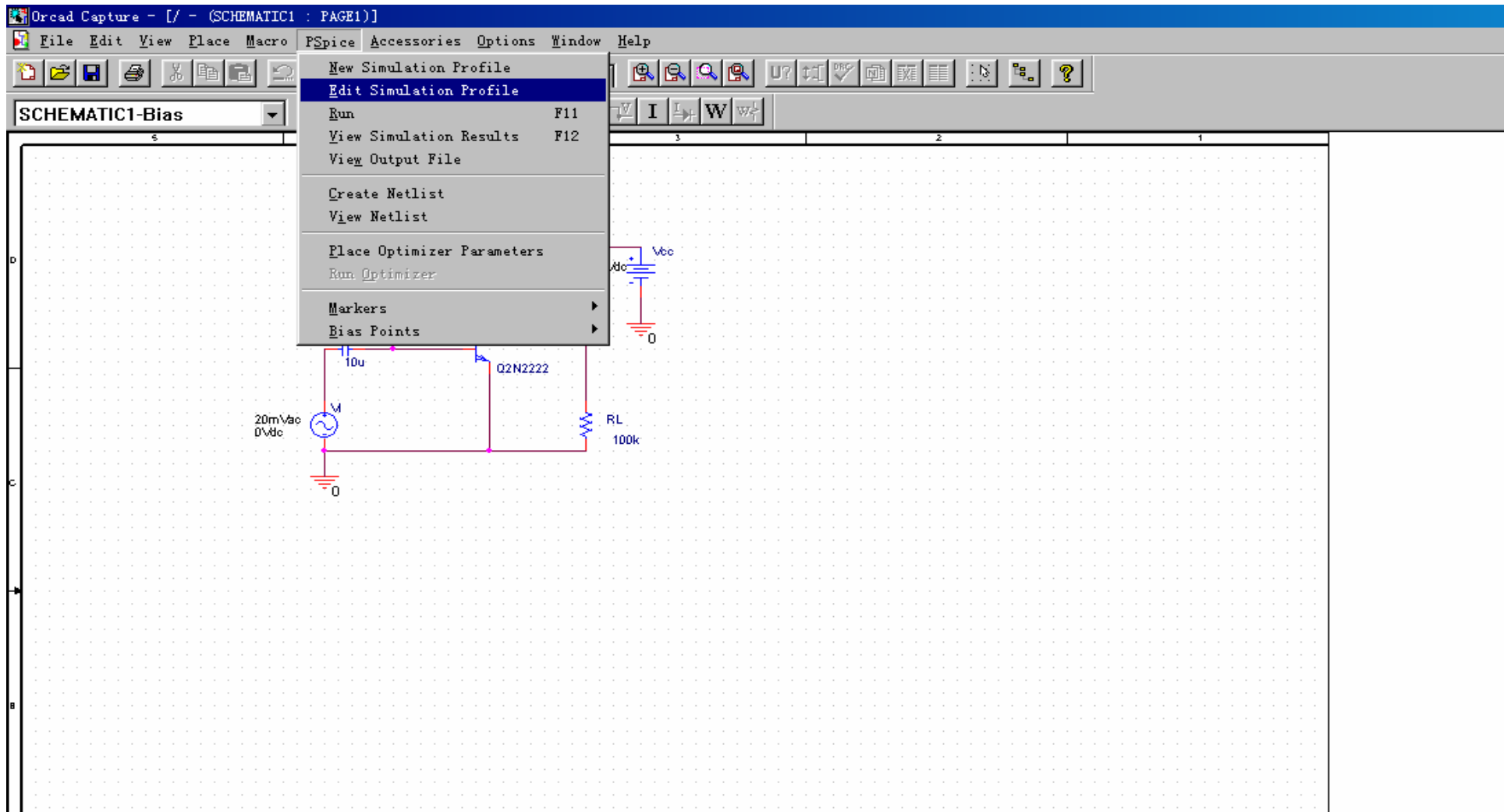
2、建立电路网表

- 执行PSpice/Create Netlist命令



3、仿真参数类型设置

- 执行PSpice/Edit Simulation Profile命令

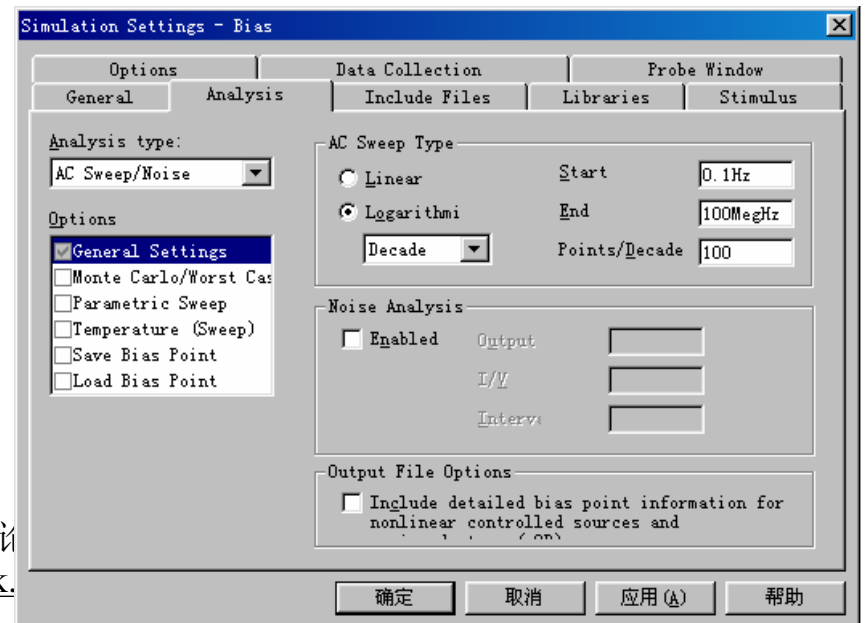


- Analysis Type 栏选择 AC Sweep/Noise
- AC Sweep Type 栏选择 Logarithmic 及 Decade
- Start 栏填写 0.1Hz
- End 栏填写 100MegHz
- Points/Decade 填写 100

- 点击“确定”按钮

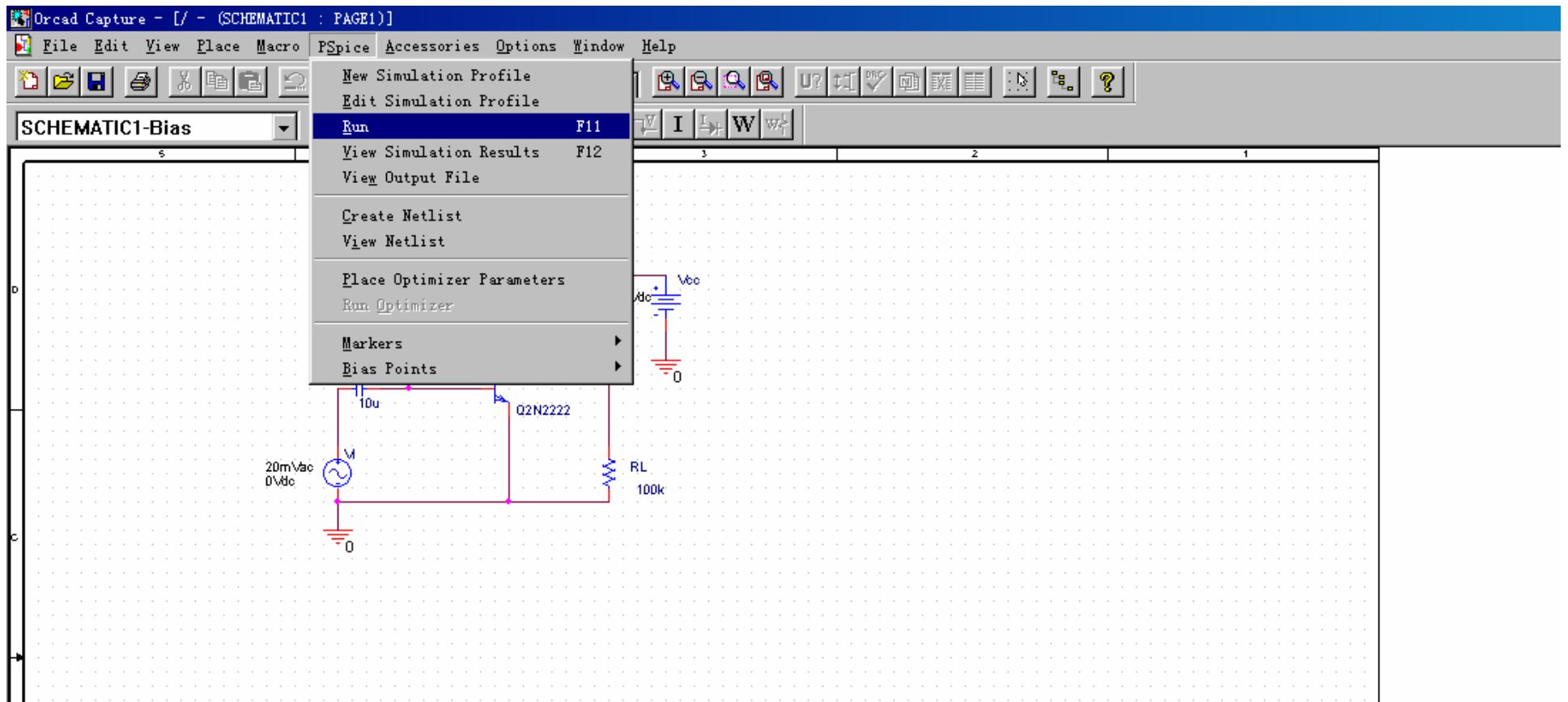
中国电子爱好者网
地: www.cepark.com

专业电子技术论坛
坛: <http://bbs.cepark.com>



4、运行仿真分析程序

- 执行PSpice/Run命令
- 屏幕上出现PSpice仿真分析窗口

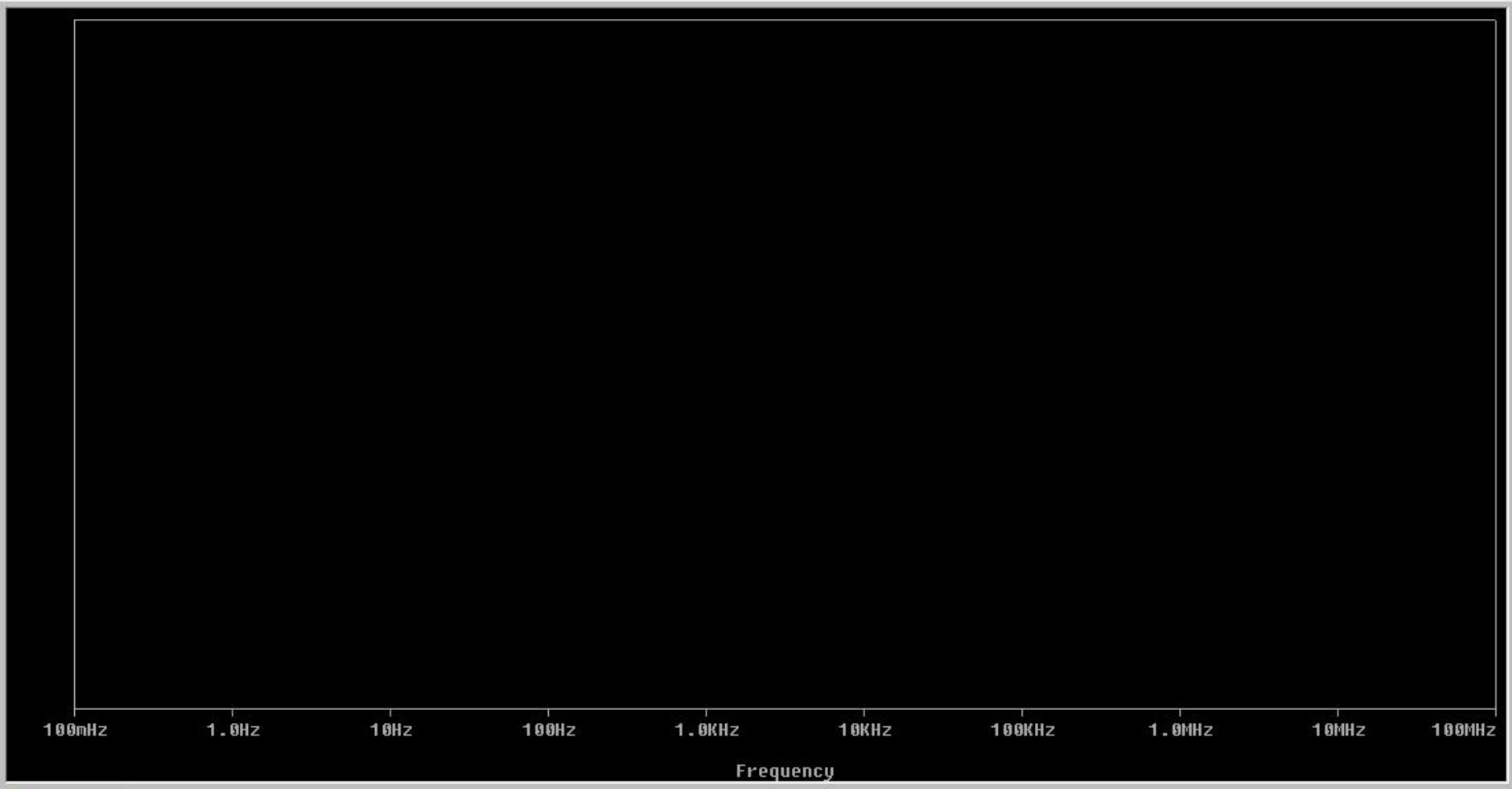


SCHEMATIC1-Bias - PSpice A/D - [Amplifier-schematic1-bias (active)]

File Edit View Simulation Trace Plot Tools Window Help

SCHEMATIC1-Bias

本单才... (Chinese characters)



Amplifie...

Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

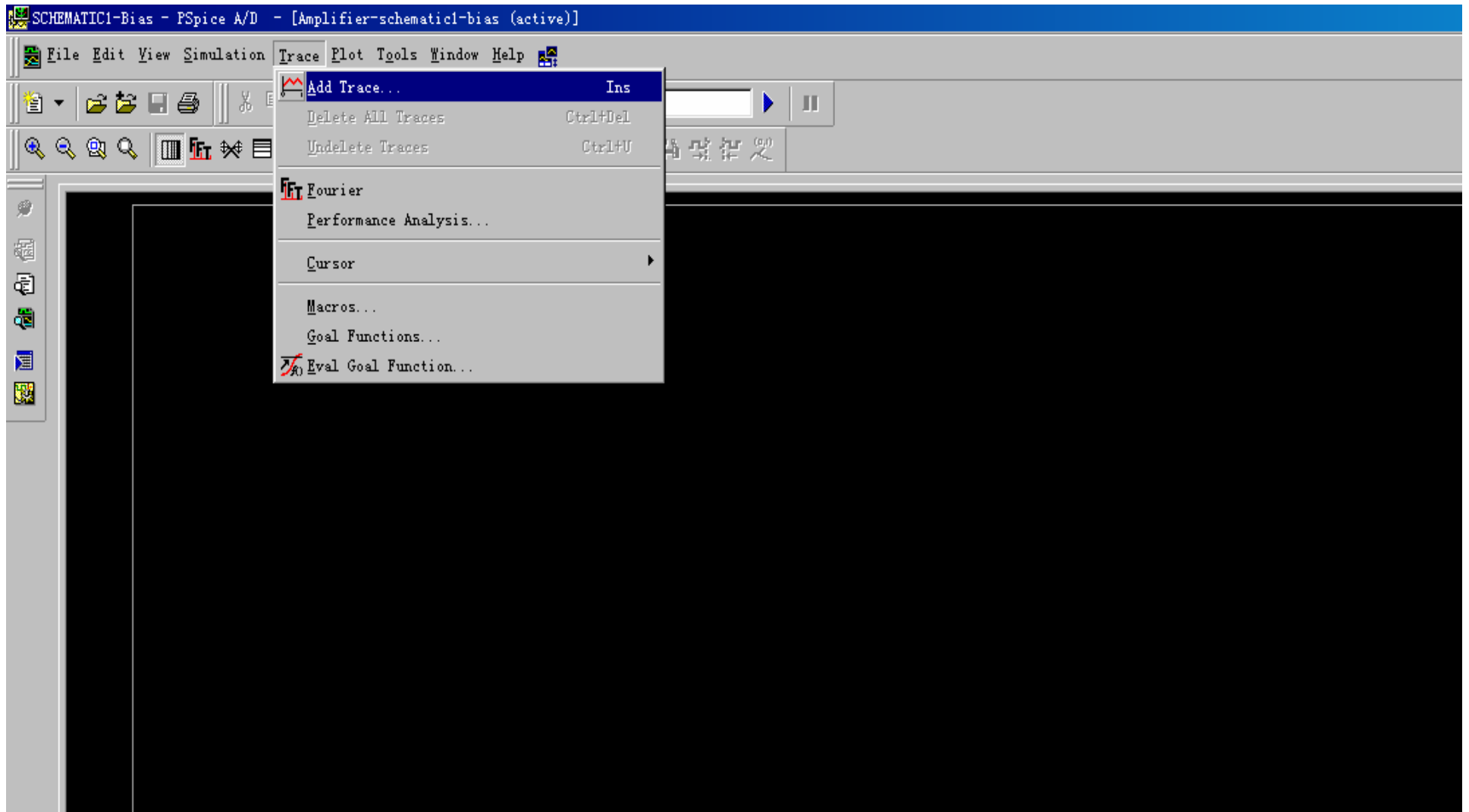
Analysis / Watch / Devices /

For Help, press F1 Freq = 100.0E+06 100%

开始 Microsoft Powe... Microsoft Word... 未命名 - 画图 Orcad Capture ... SCHEMATIC1-Bia... 21:19

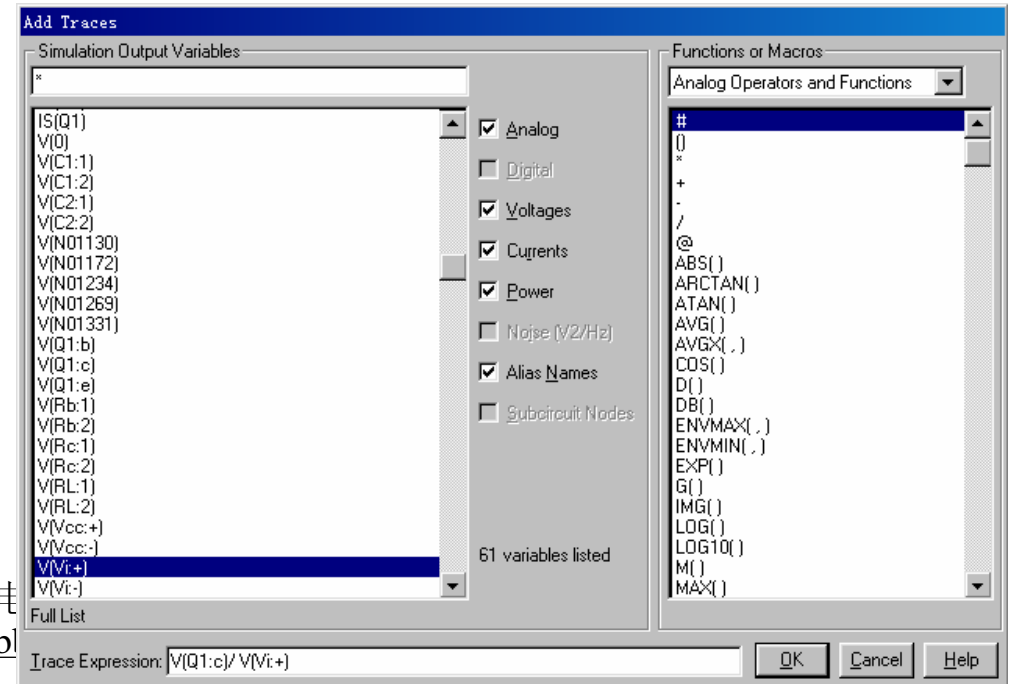
5、系统增益频率特性分析

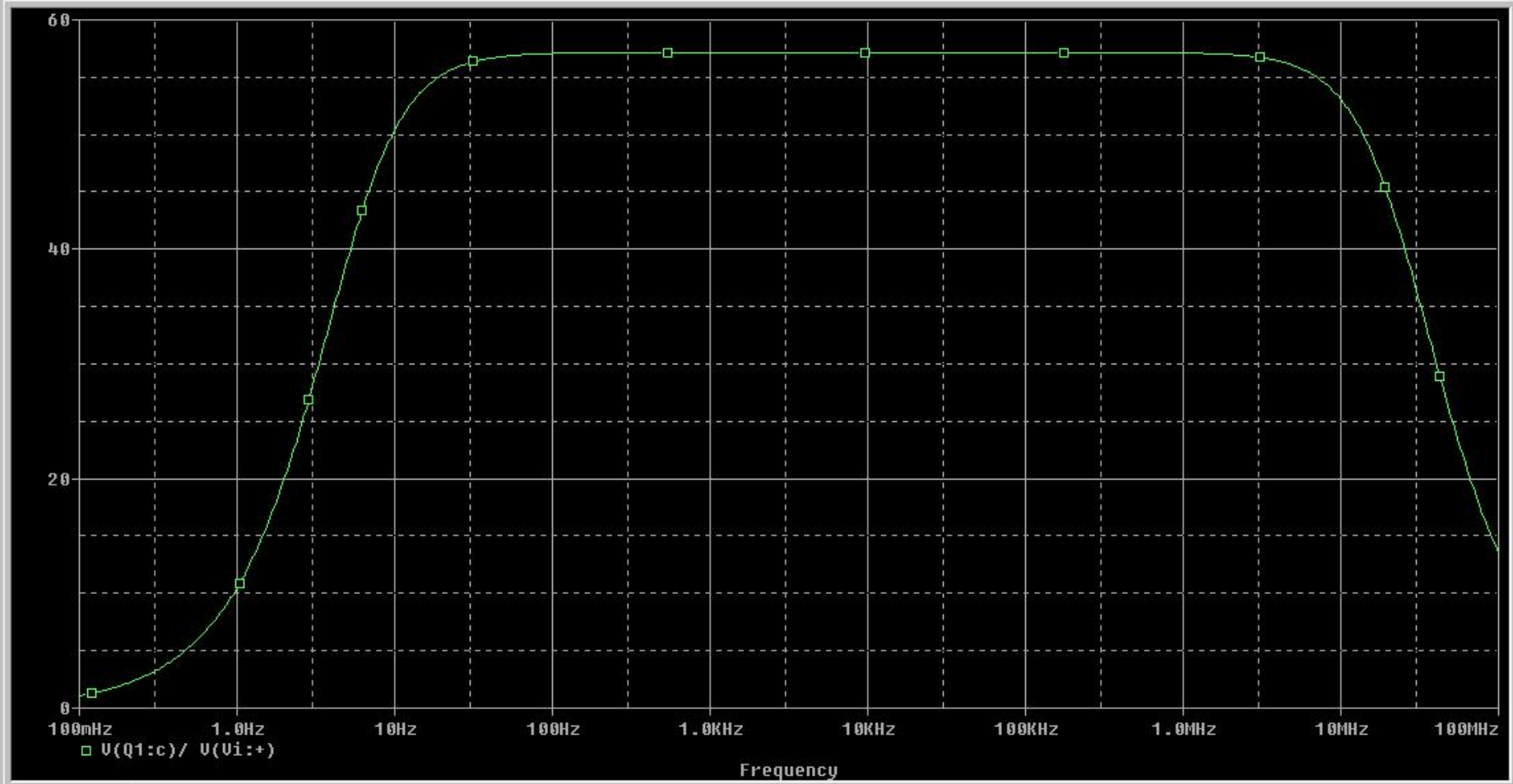
- 执行Trace/Add Trace命令



在Add Traces对话框

- Trace Expression栏填写
 $V[Q1:c]/V[Vi:+]$ ，输出幅度与输入幅度之比即为增益 A_v 随信号频率变化的关系
- 单击“OK”按钮



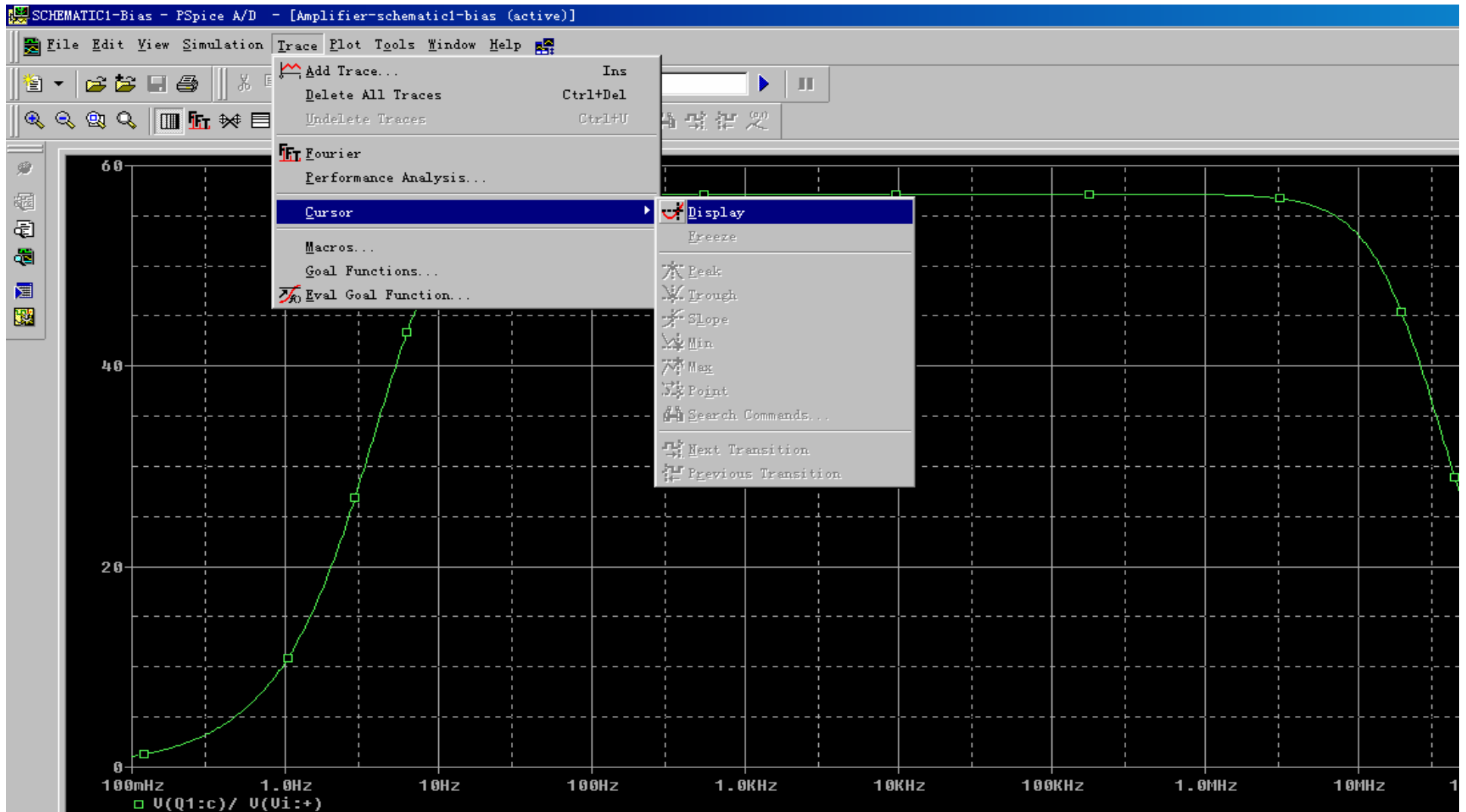


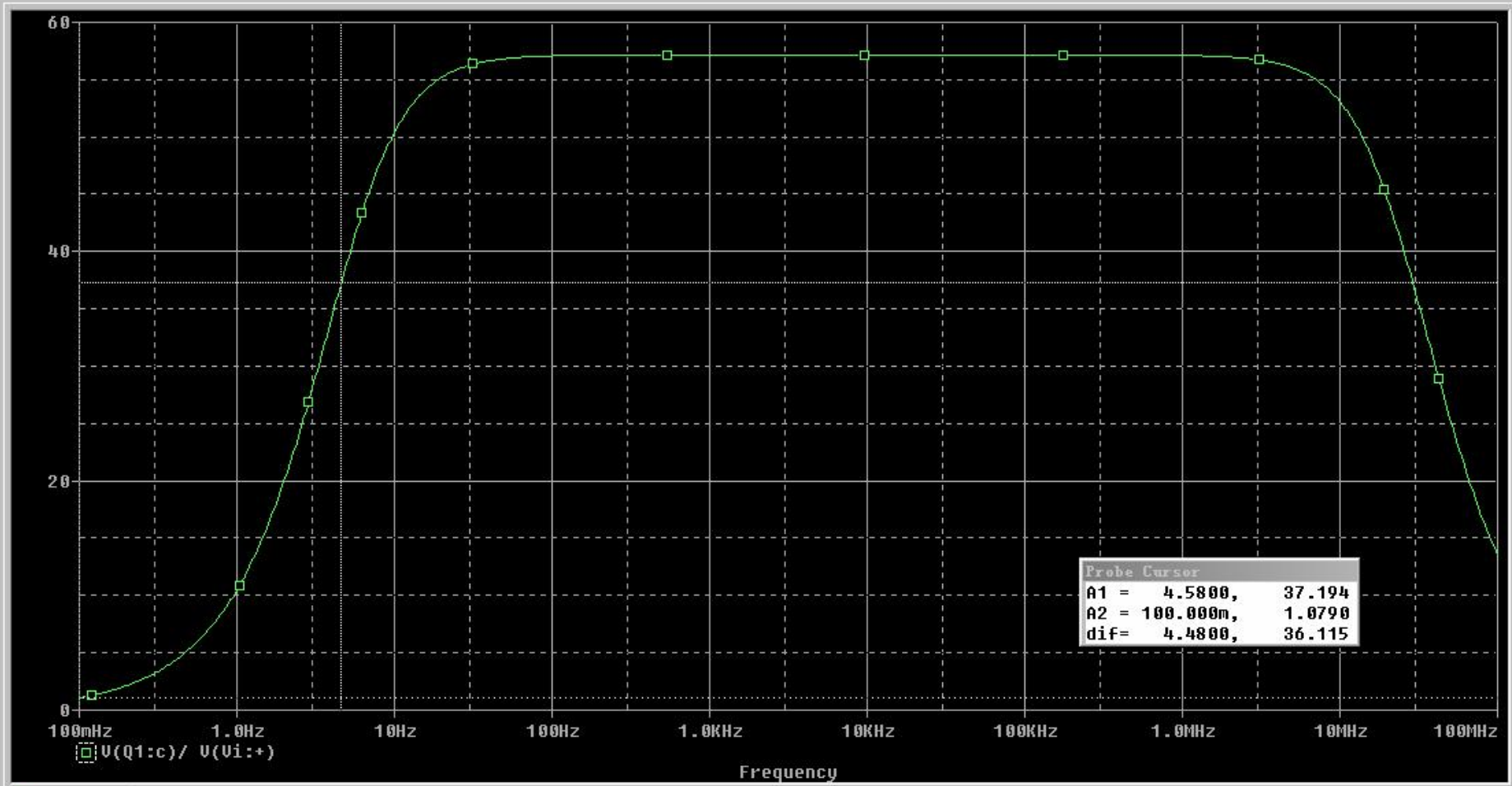
Amplifie...
Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

Analysis / Watch / Devices /

标尺工具

- 执行Trace/Cursor/Display命令





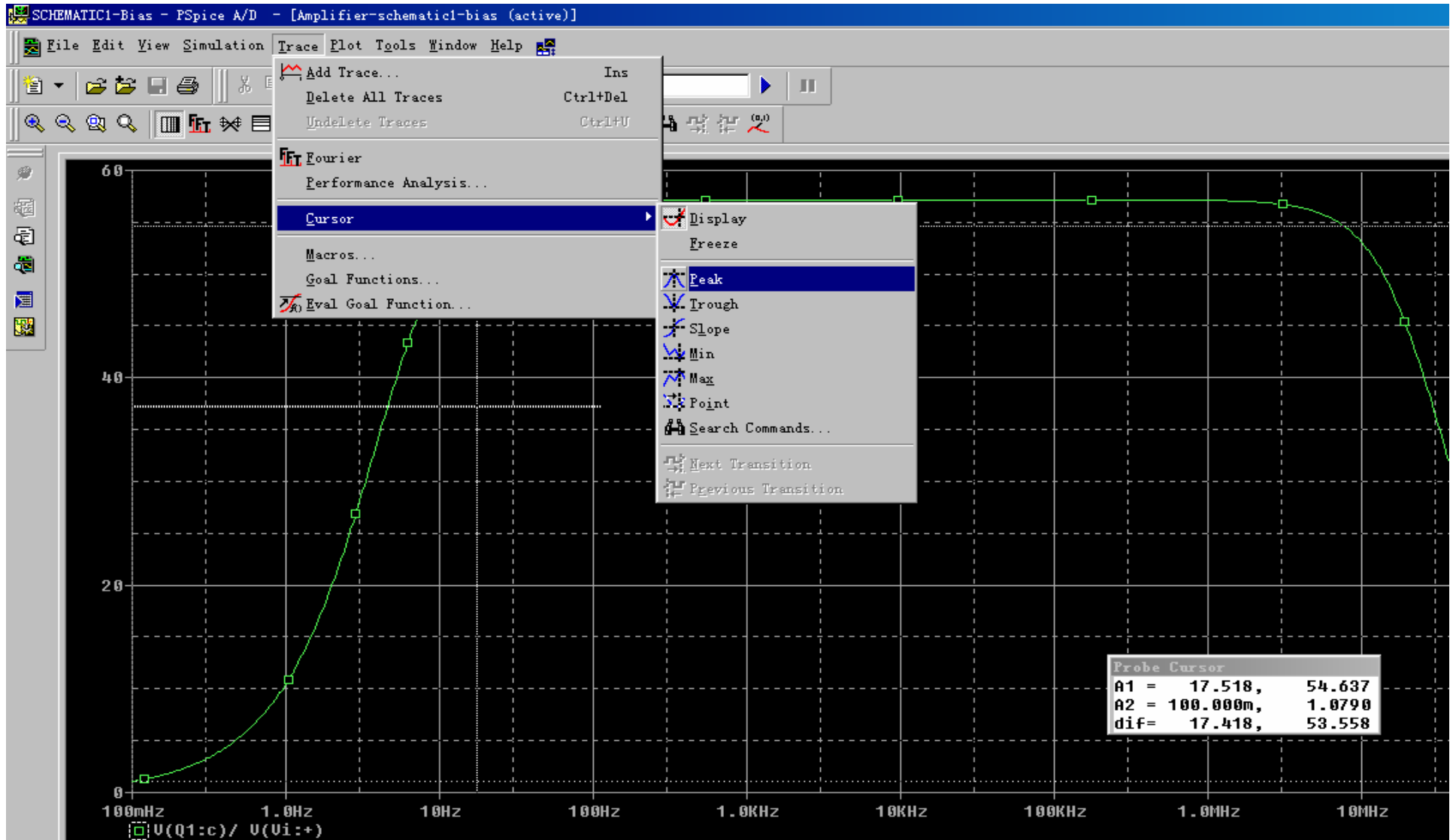
Amplifie...

Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

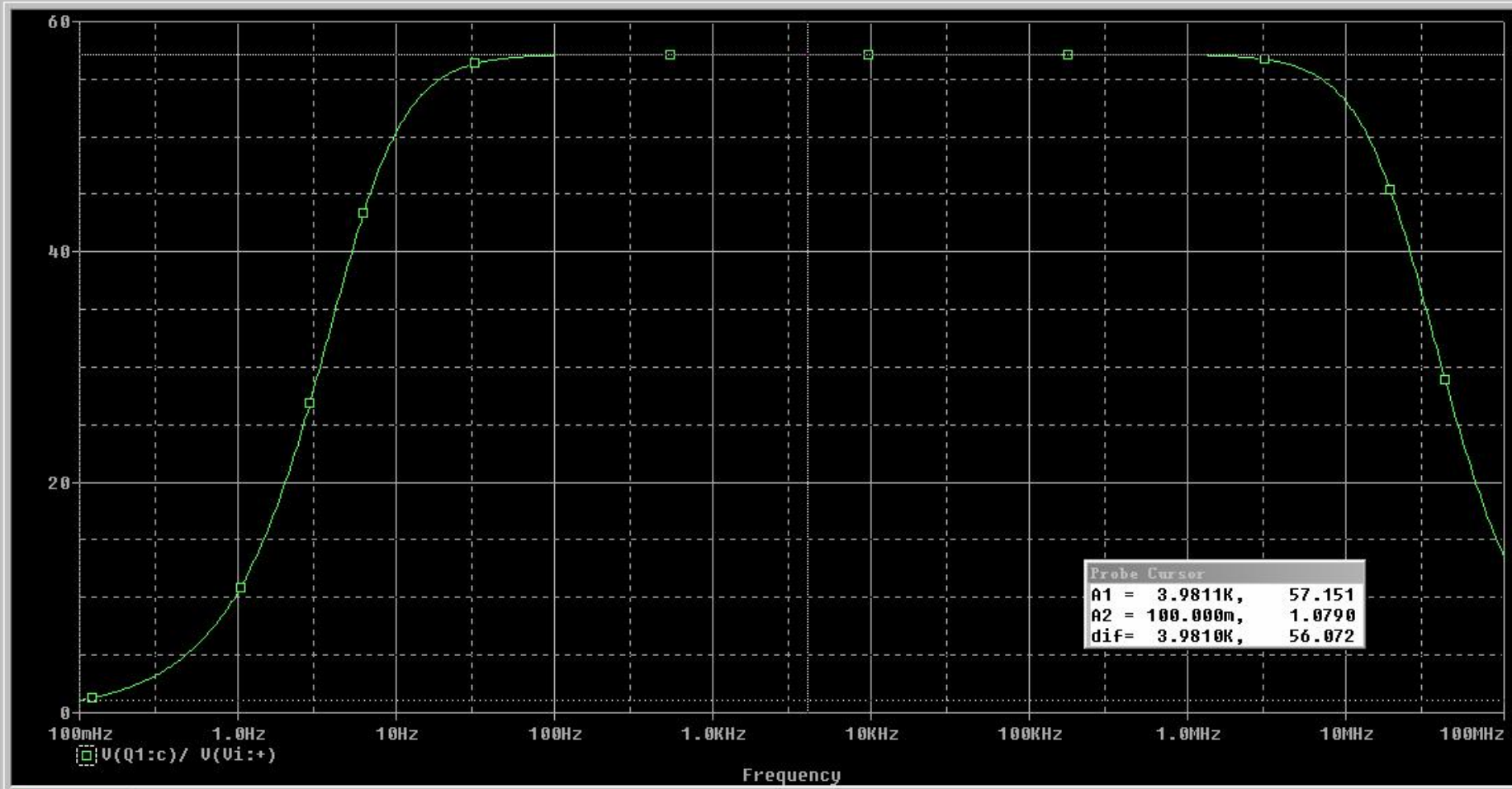
Analysis Watch Devices

标尺对准Av曲线中频点

- 执行Trace/Cursor/Peak命令



SCHEMATIC1-Bias



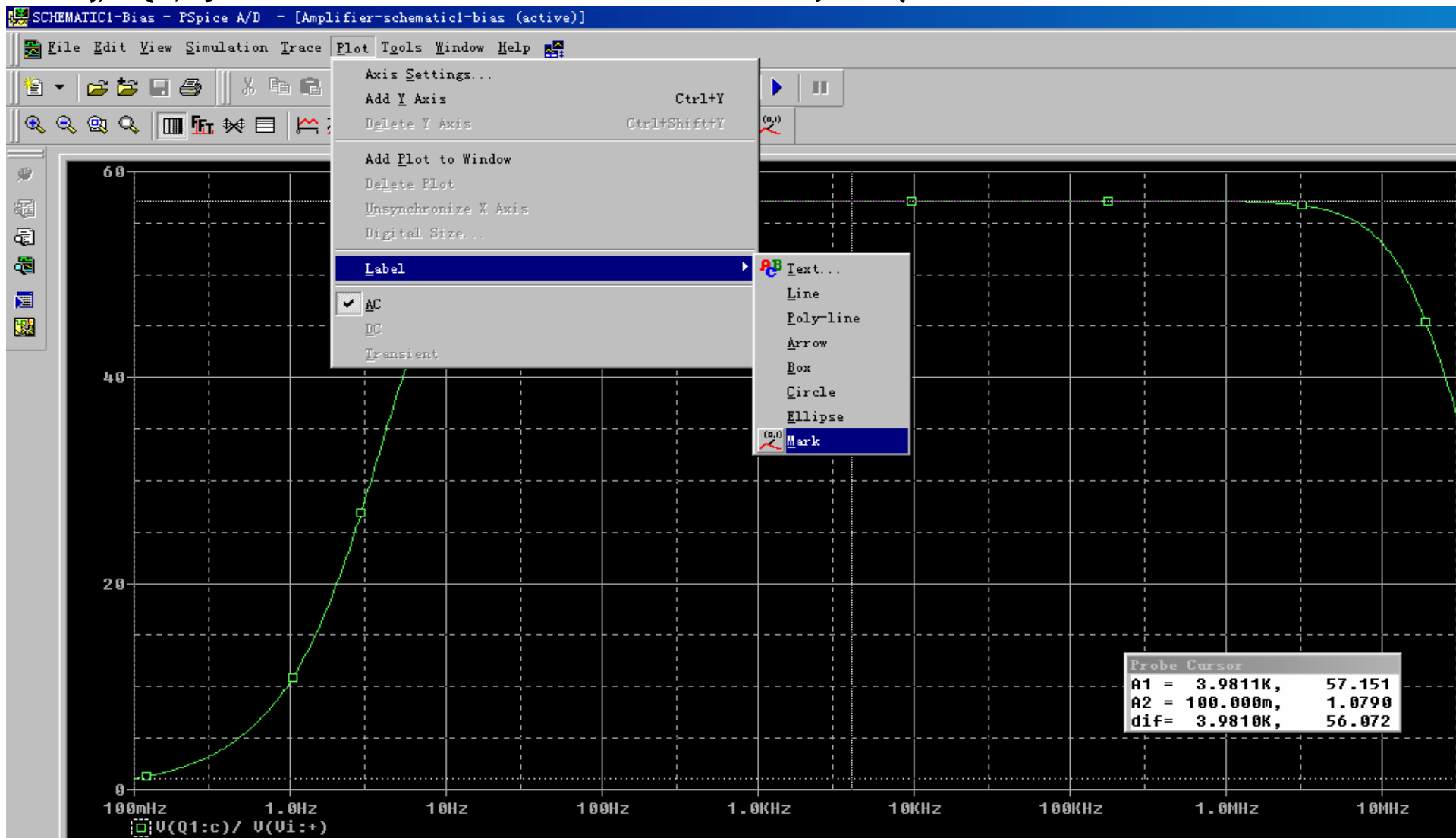
Amplifie...

Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

Analysis Watch Devices

在中频点处标记位置坐标

- 执行Plot/Label/Mark命令

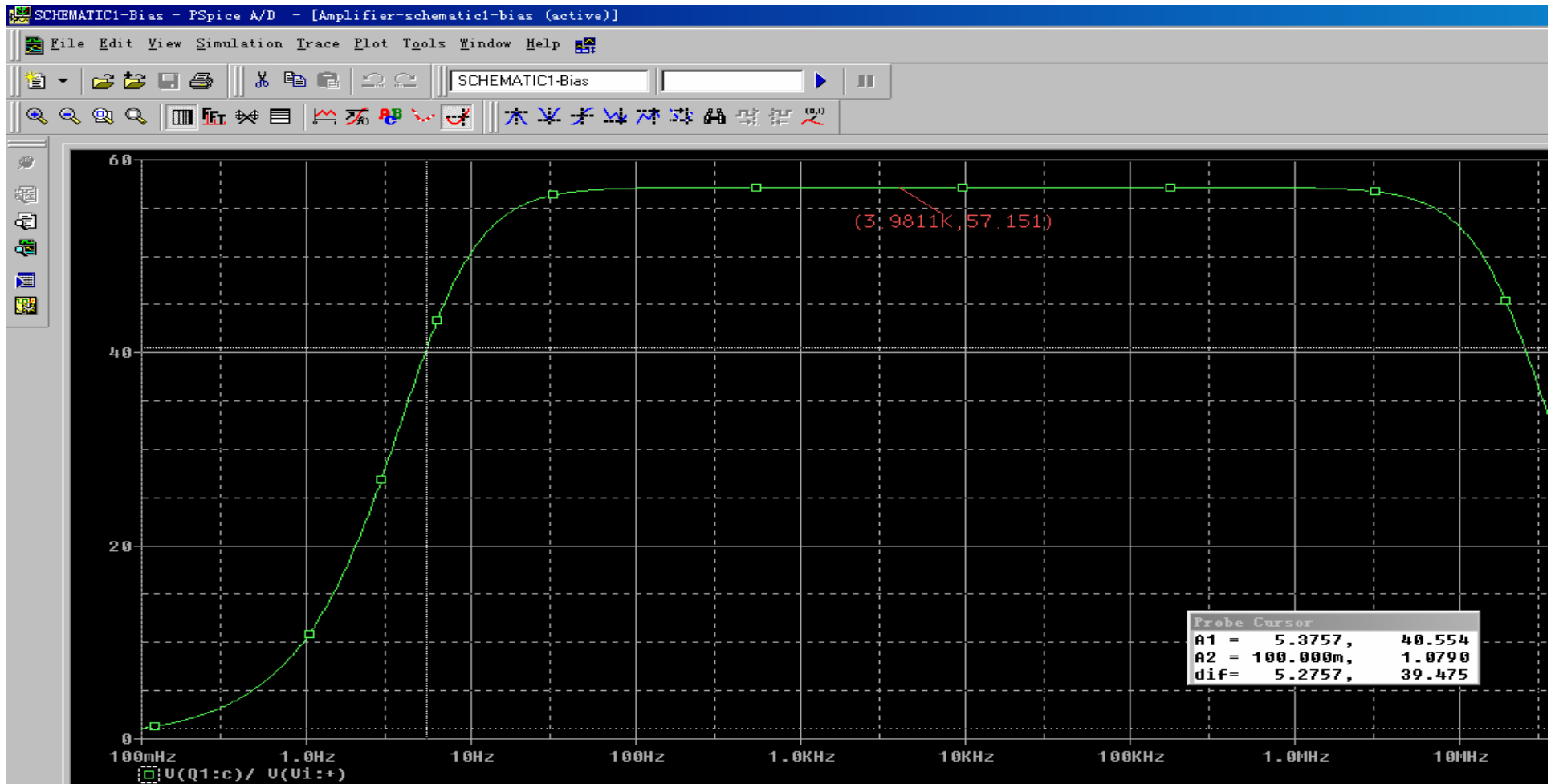


- 第一为中频处频率 $f_o = 3.9811\text{kHz}$
- 第二为中频处放大倍数 $A_{vo} = 57.151$



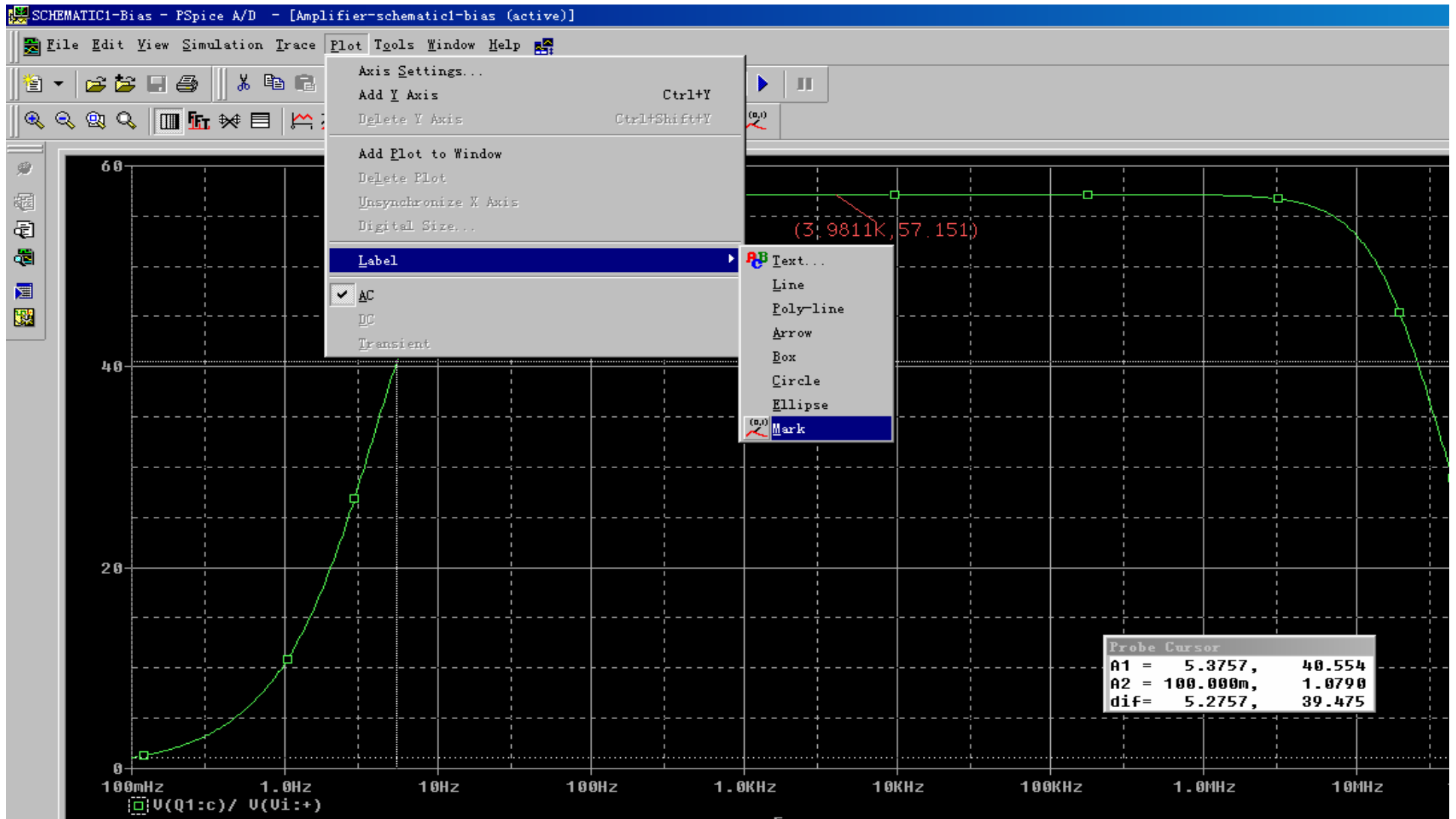
测量低半功率点频率 f_L

- 向左拖动十字标尺，对准0.707倍中频放大倍数处(约40.554)，即低半功率点

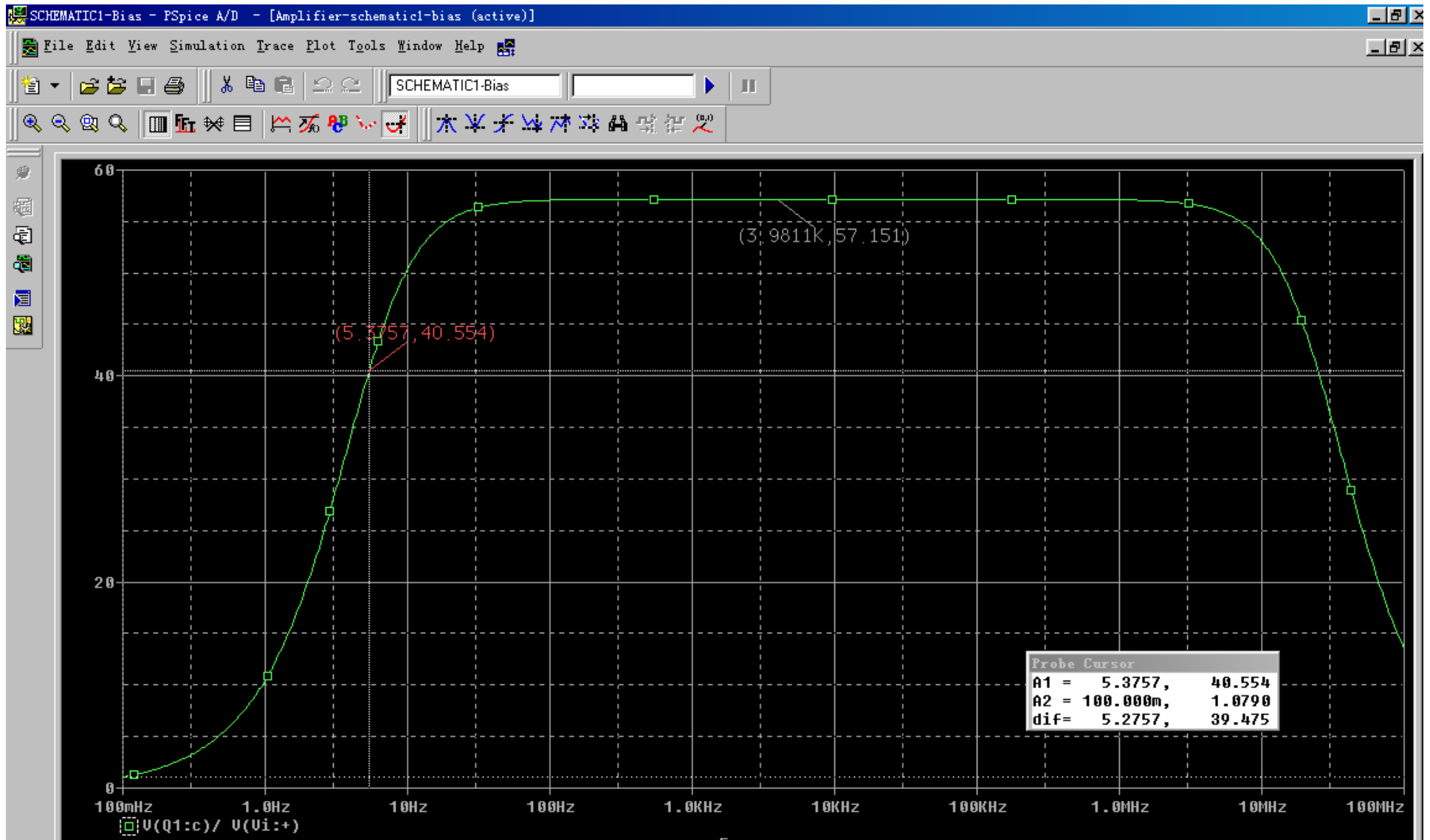


标记低半功率点处坐标

- 执行Plot/Label/Mark命令

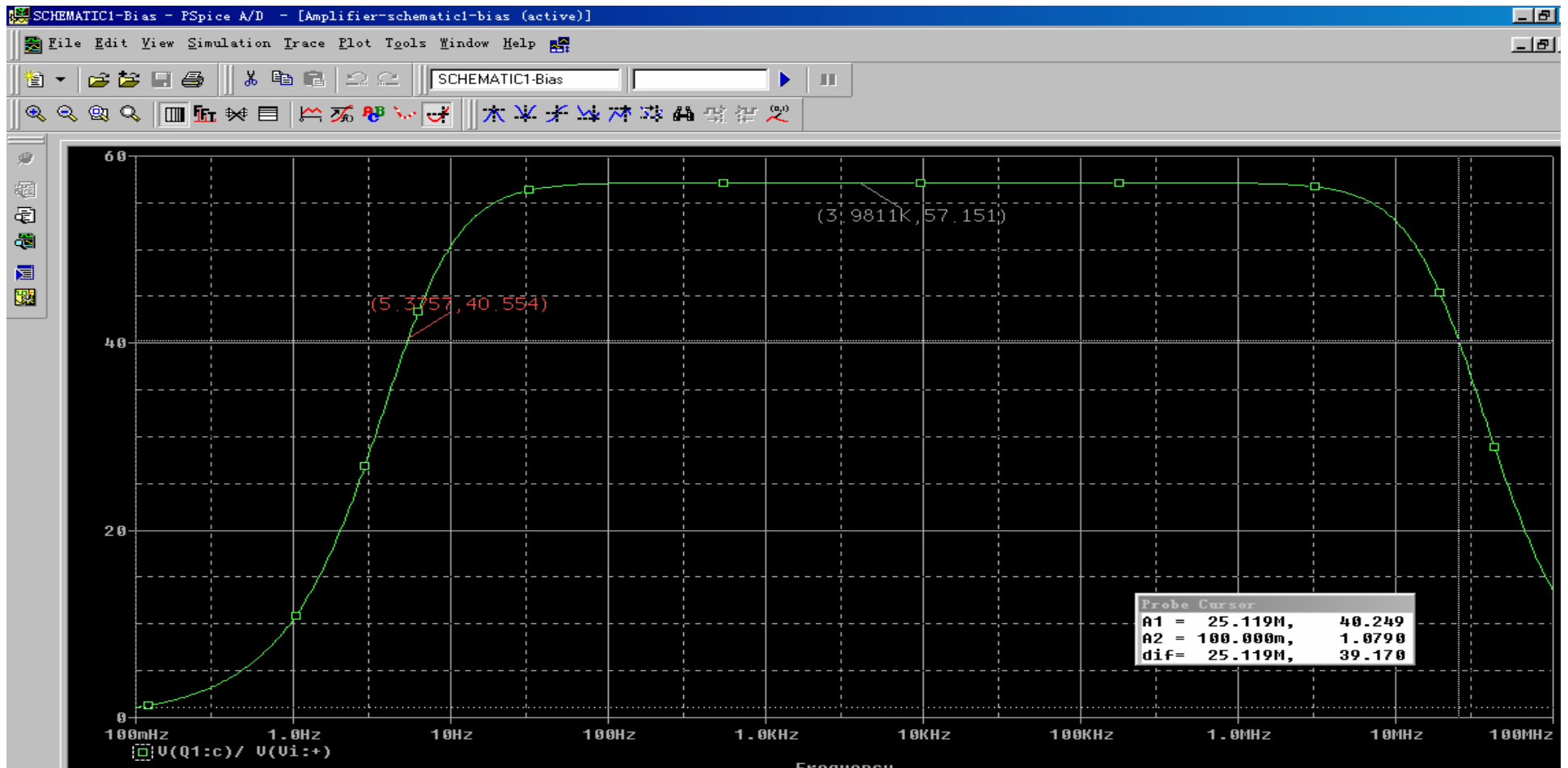


- 第一为低半功率点频率 $f_L=5.3757\text{Hz}$
- 第二为 f_L 处放大倍数 $A_v=40.554$



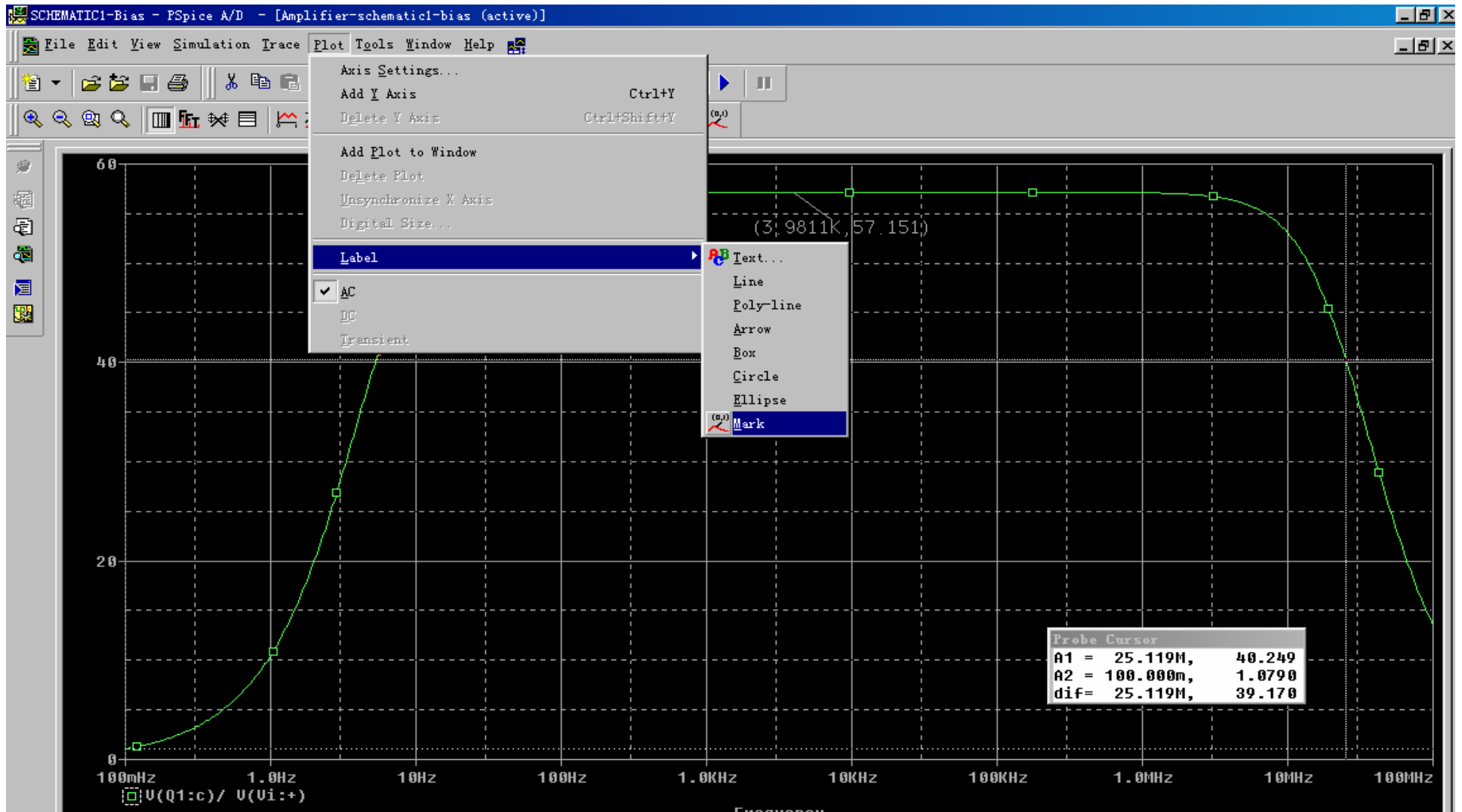
测量高半功率点频率 f_H

- 向右拖动十字标尺，对准0.707倍中频放大倍数处(约40.249)，即高半功率点

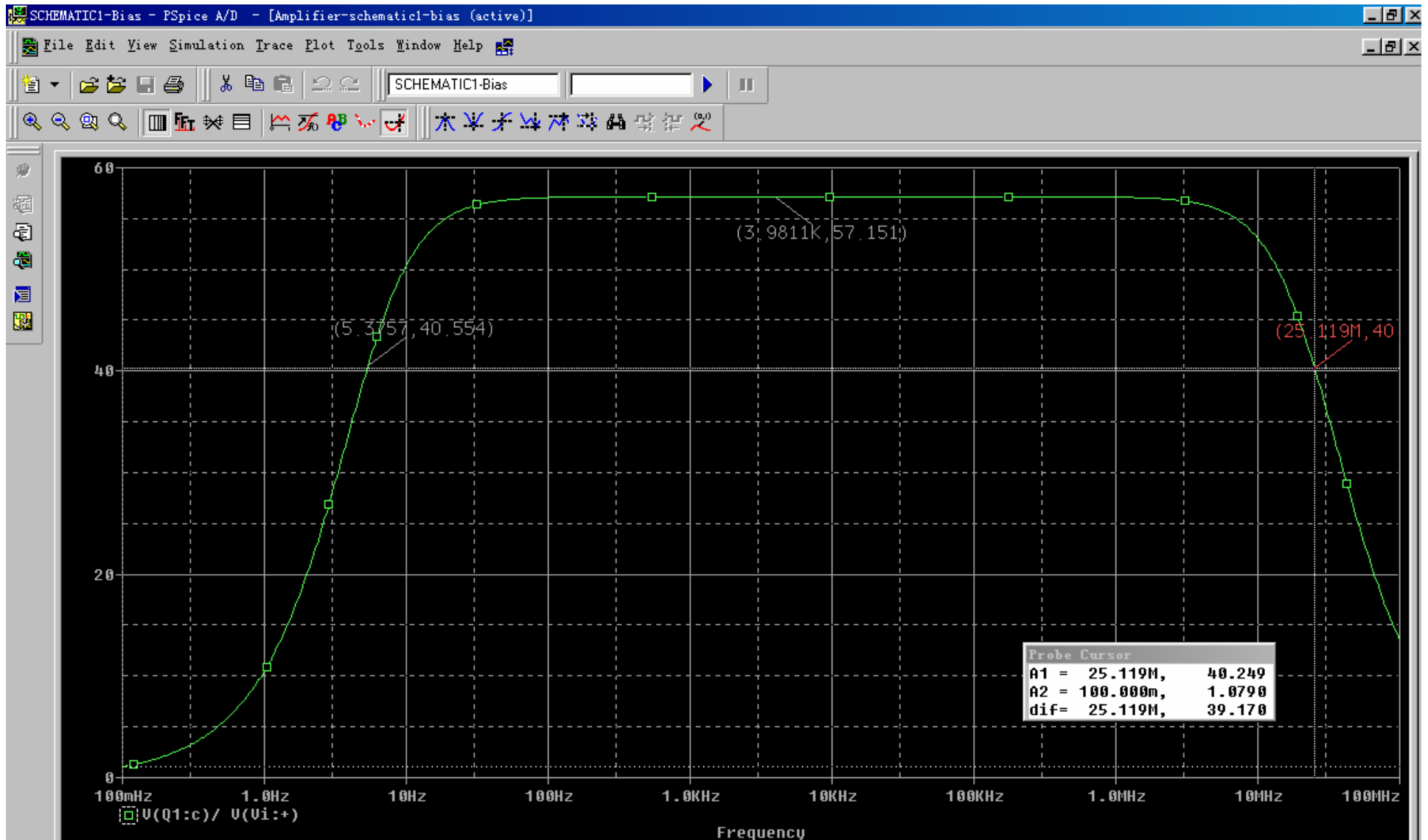


标记高半功率点处坐标

- 执行Plot/Label/Mark命令



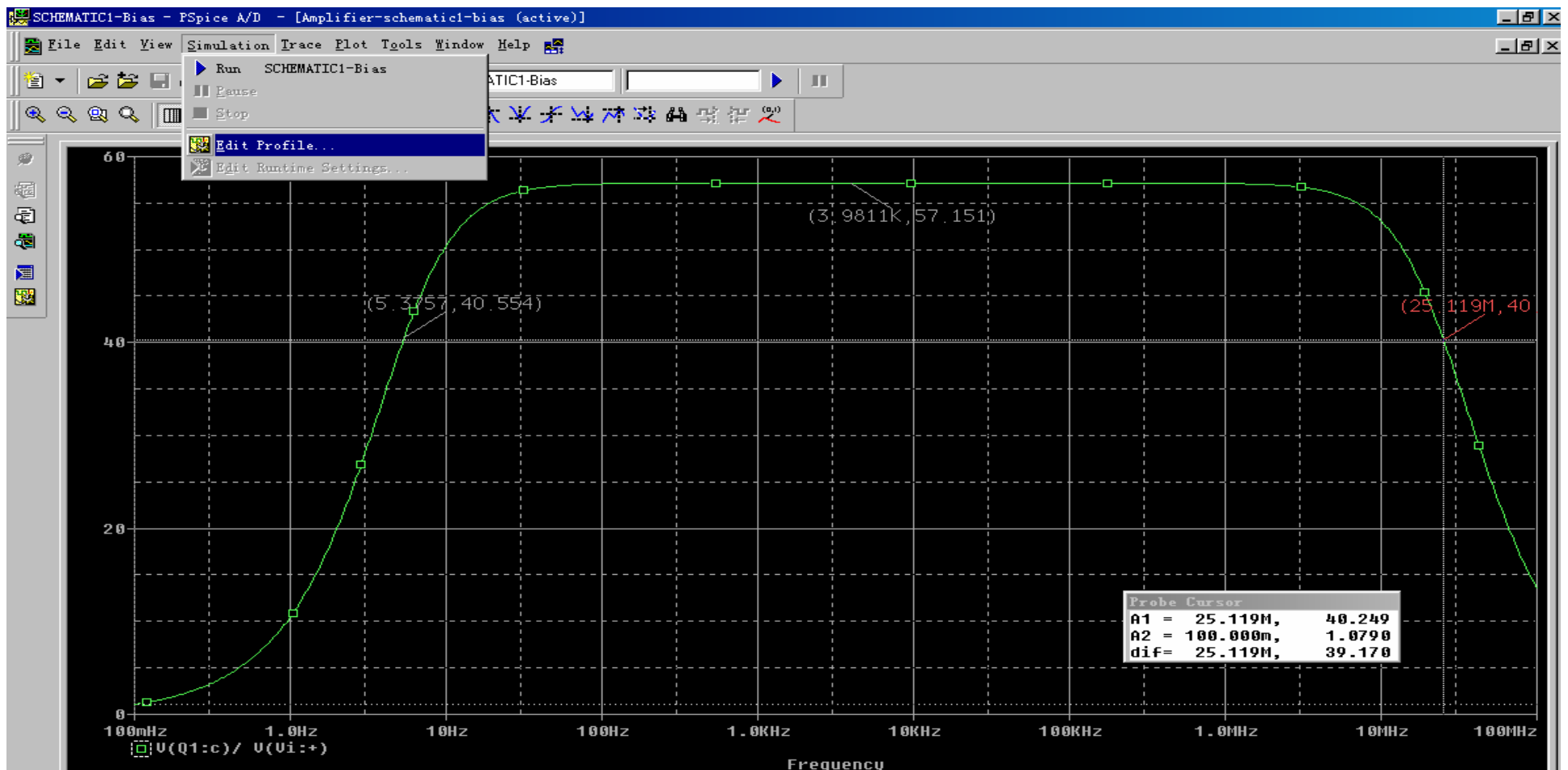
- 第一为高半功率点频率 $f_H=25.119\text{MHz}$
- 第二为 f_H 处放大倍数 $A_v=40.249$



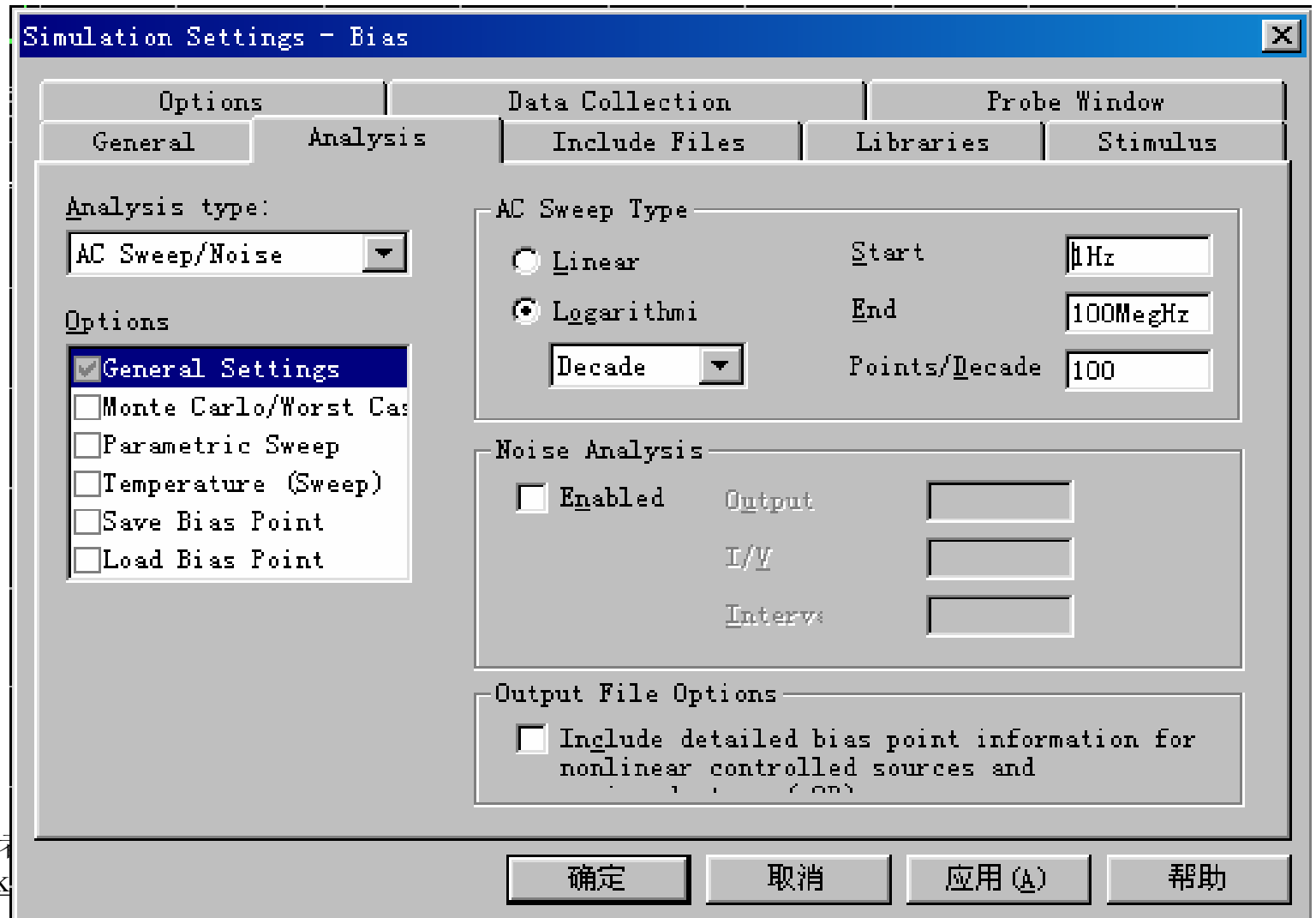
6、输入阻抗频率特性分析

仿真起始频率重新设定

- 执行Simulation/Edit Profile命令

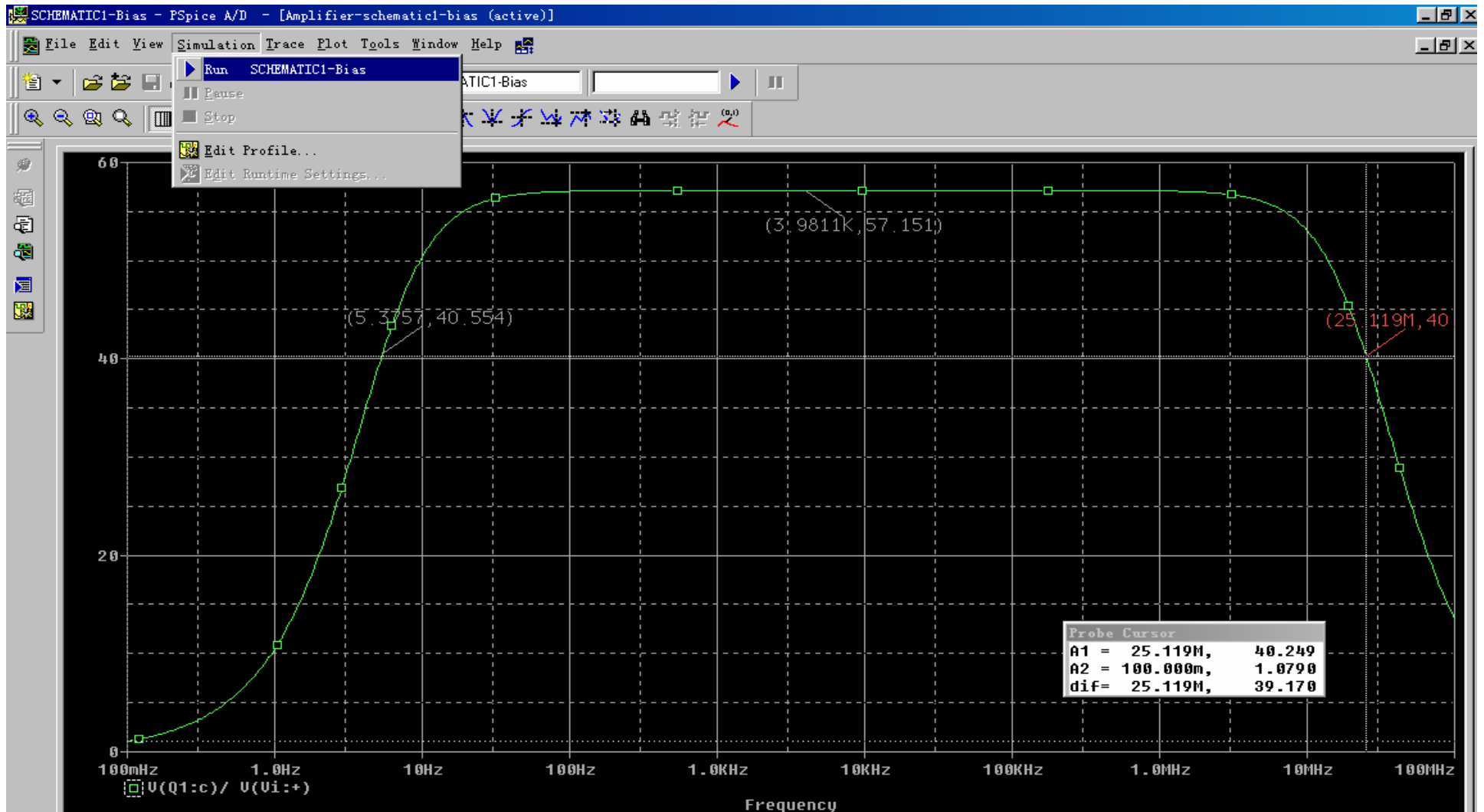


- 将Start栏由0.1Hz改为1Hz
- 单击“确定”按钮



运行仿真程序

- 执行Simulation/Run命令

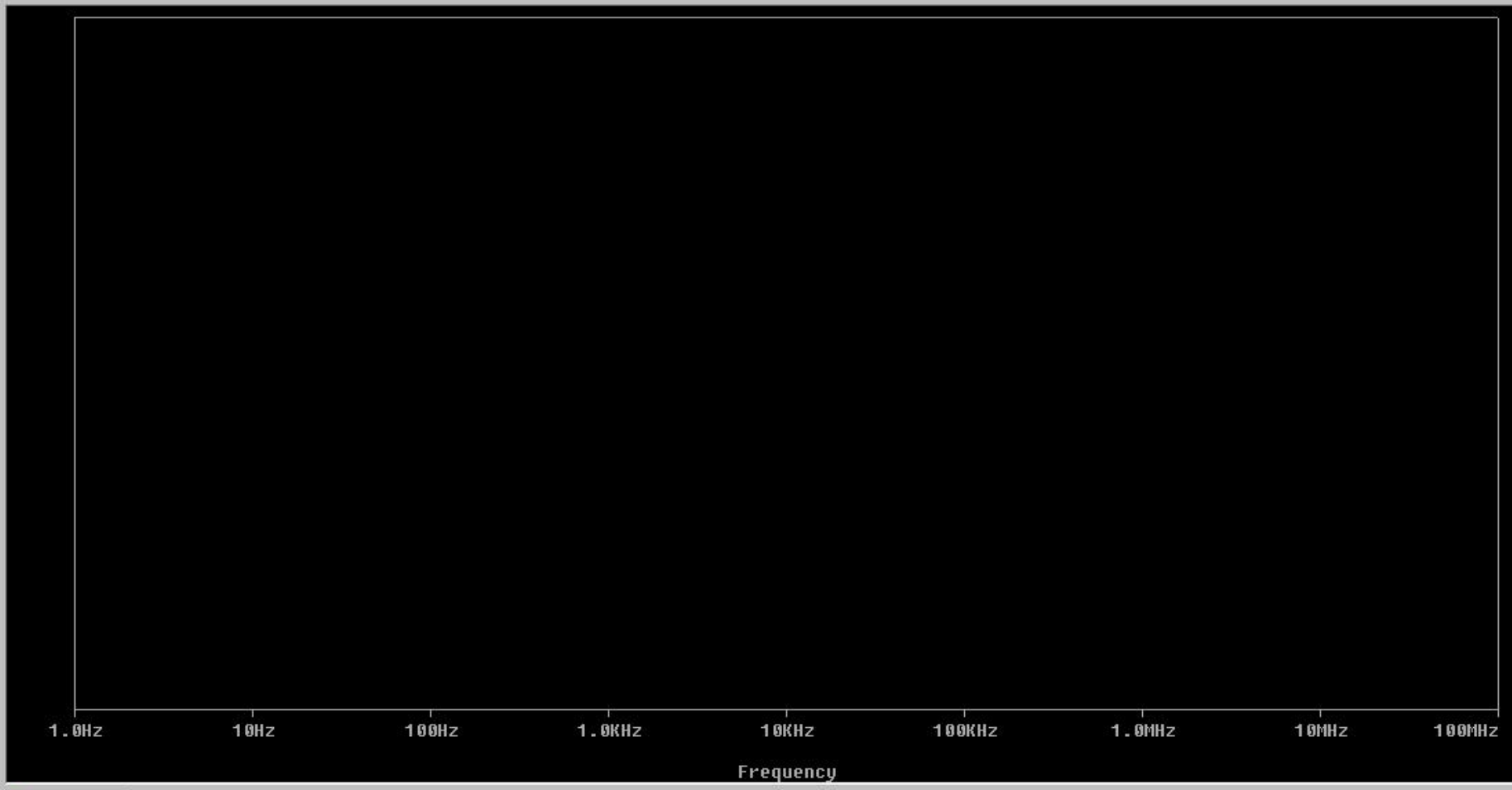


SCHEMATIC1-Bias - PSpice A/D - [Amplifier-schematic1-bias (active)]

File Edit View Simulation Trace Plot Tools Window Help

SCHEMATIC1-Bias

本单才... (Chinese characters)



Amplifie...

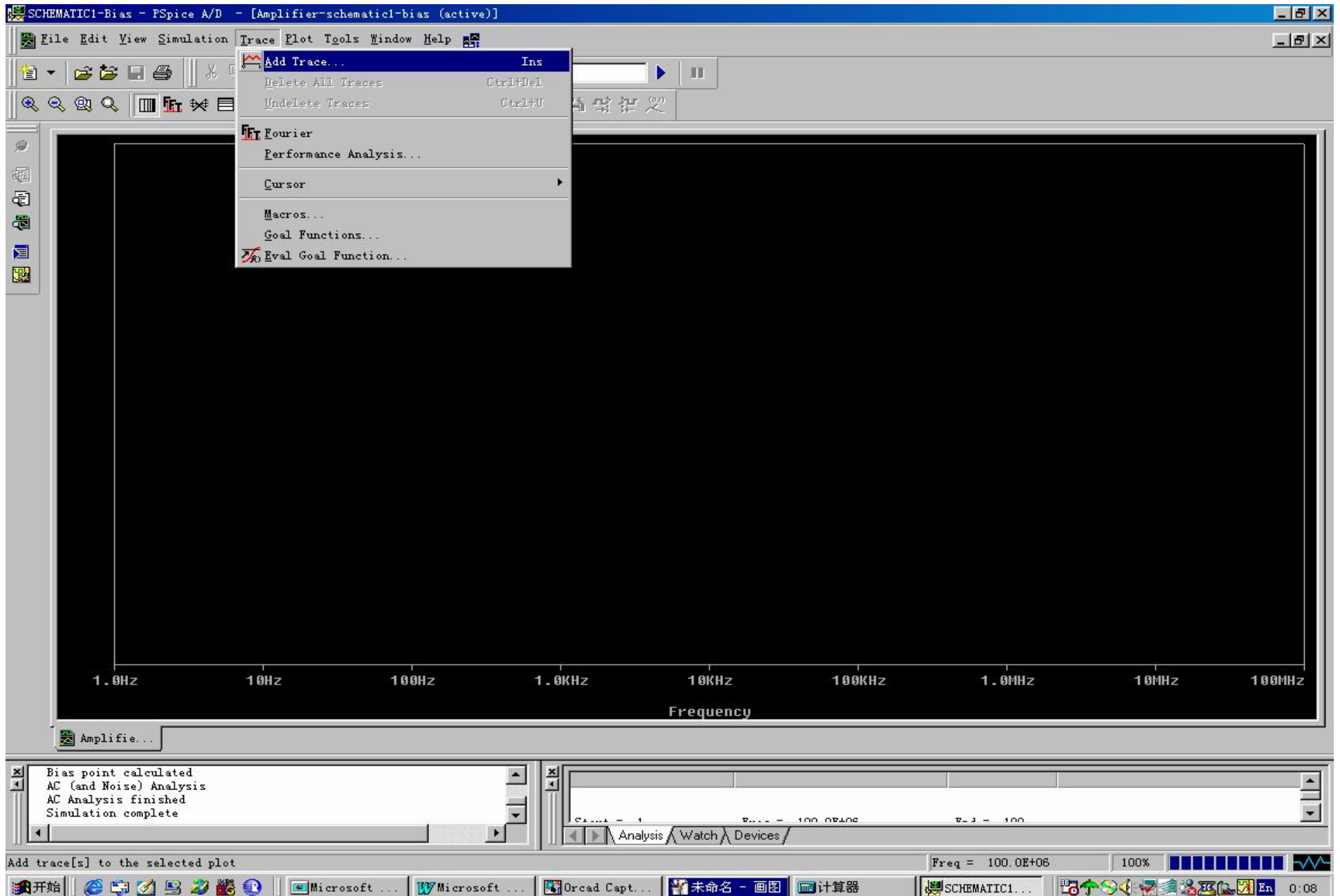
Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

Analysis / Watch / Devices /

For Help, press F1 Freq = 100.0E+06 100%

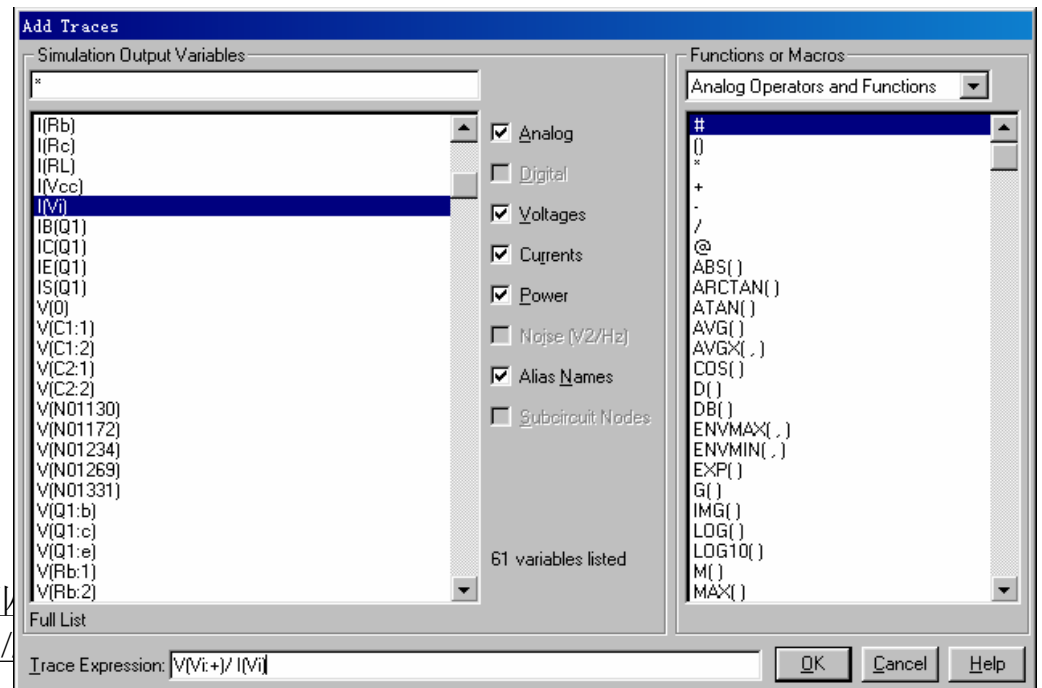
开始 Microsoft ... Microsoft ... Orcad Capt... 未命名 - 画图 计算器 SCHEMATIC1... 0:04

• 执行Trace/Add Trace命令



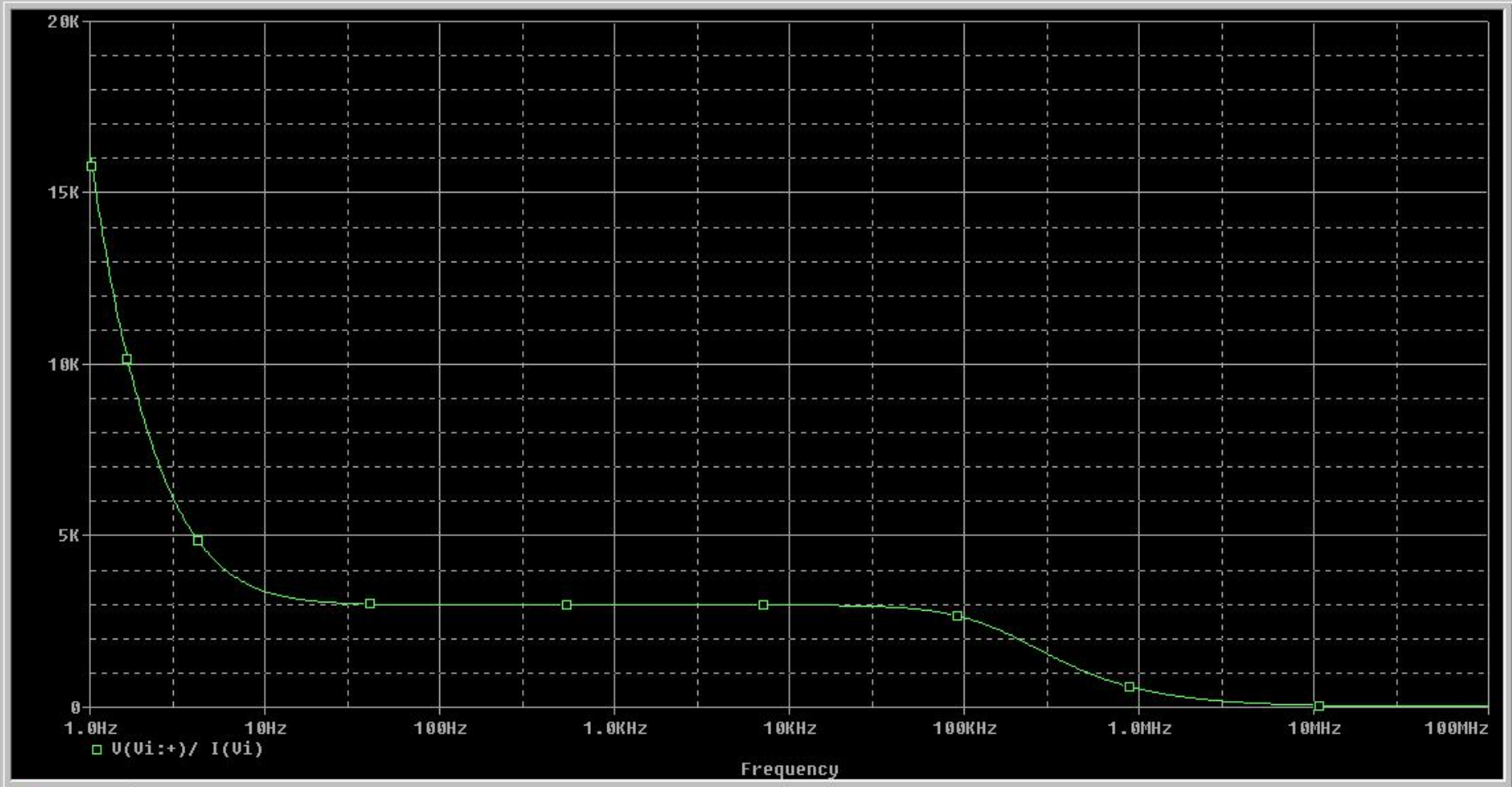
在Add Traces对话框

- Trace Expression栏填写 $V[V_i: +] / I[V_i]$, 激励源输出电压与电流之比即为放大器系统输入阻抗 R_i
- 单击“OK”按钮



SCHEMATIC1-Bias

本单才... (Chinese characters)

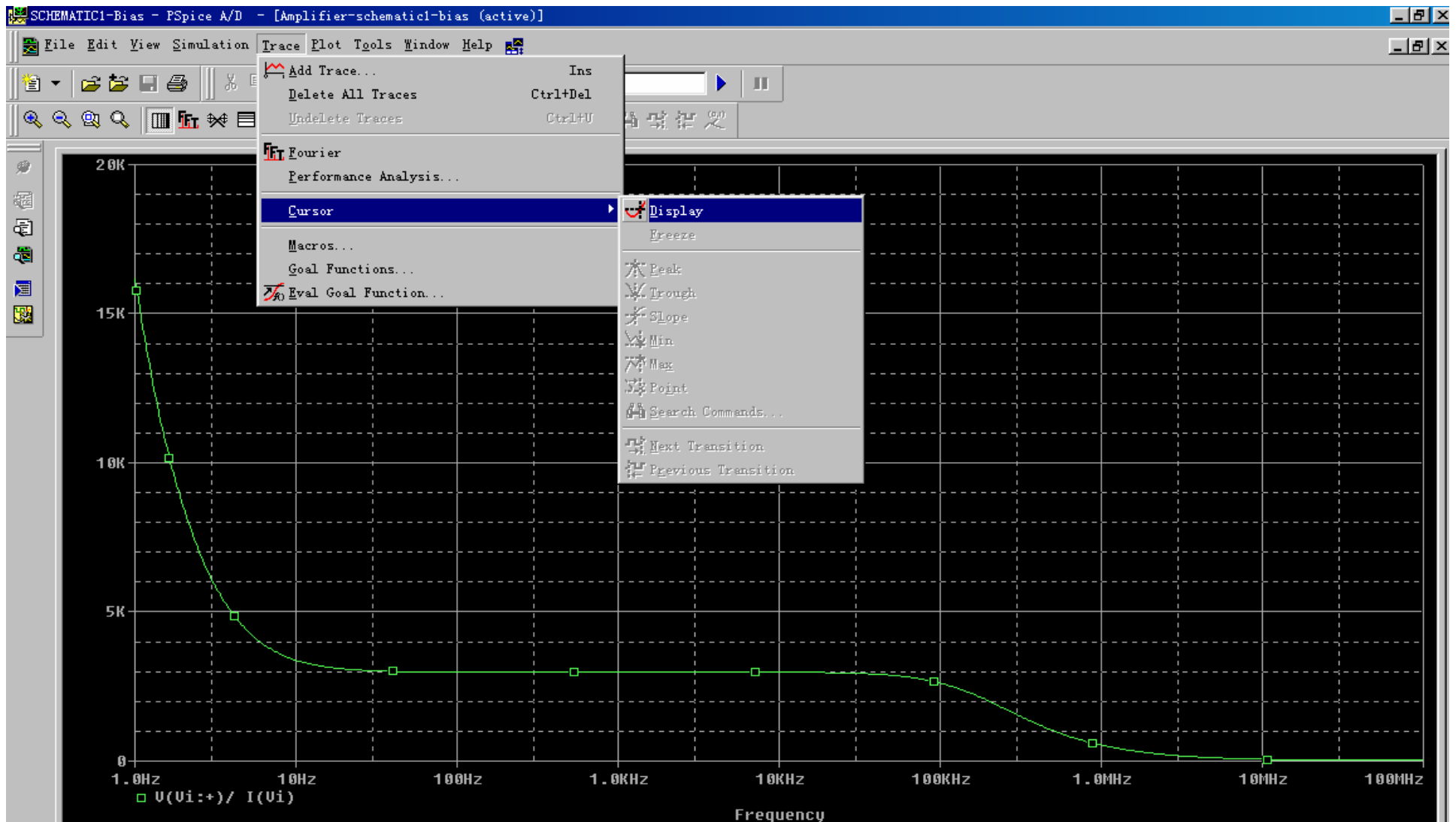


Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

Analysis / Watch / Devices /

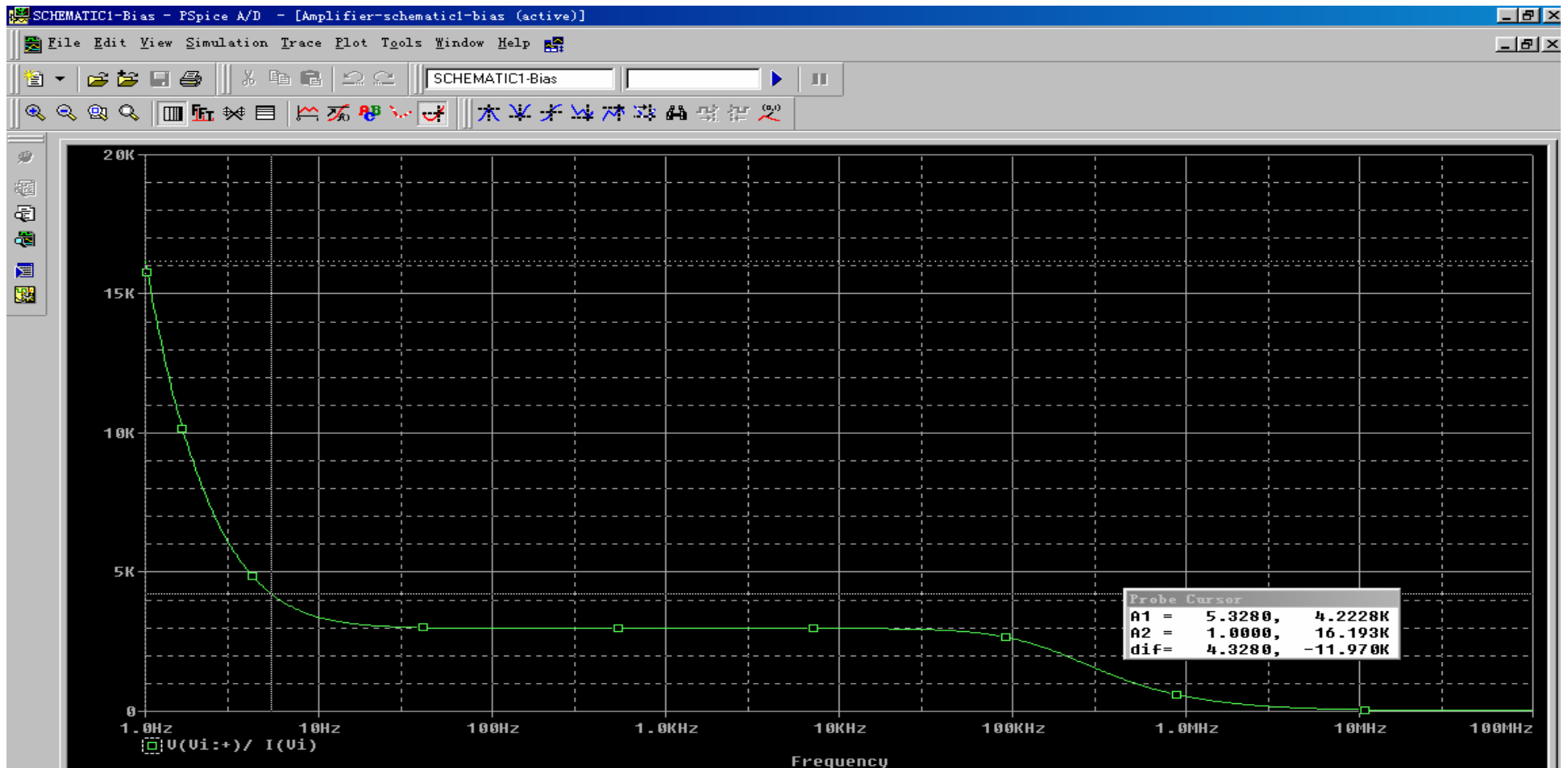
标尺工具

- 执行Trace/Cursor/Display命令



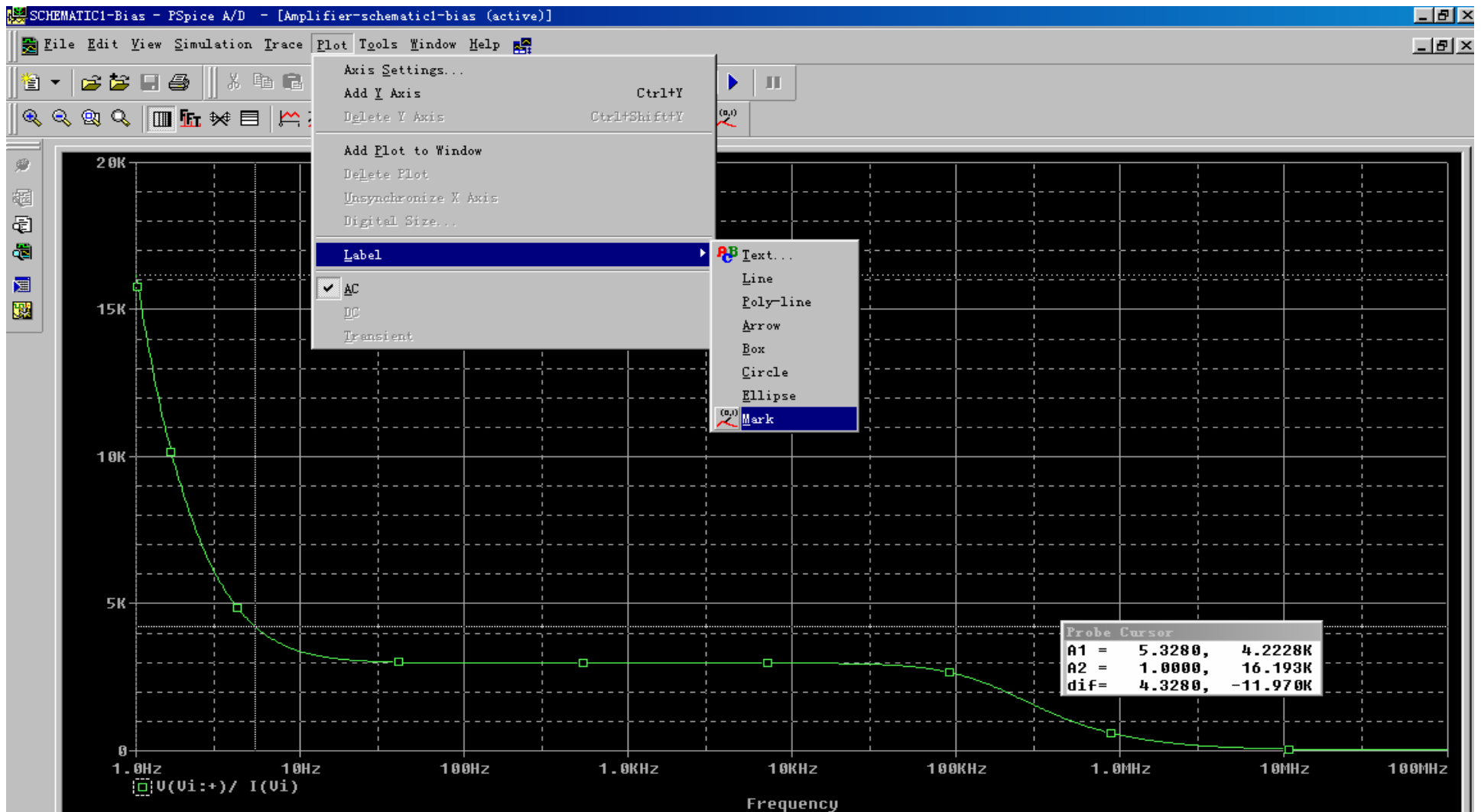
测量低半功率点 f_L 处 R_i

- 向右拖动十字标尺，对准低半功率点 f_L 处(约5.3280Hz)

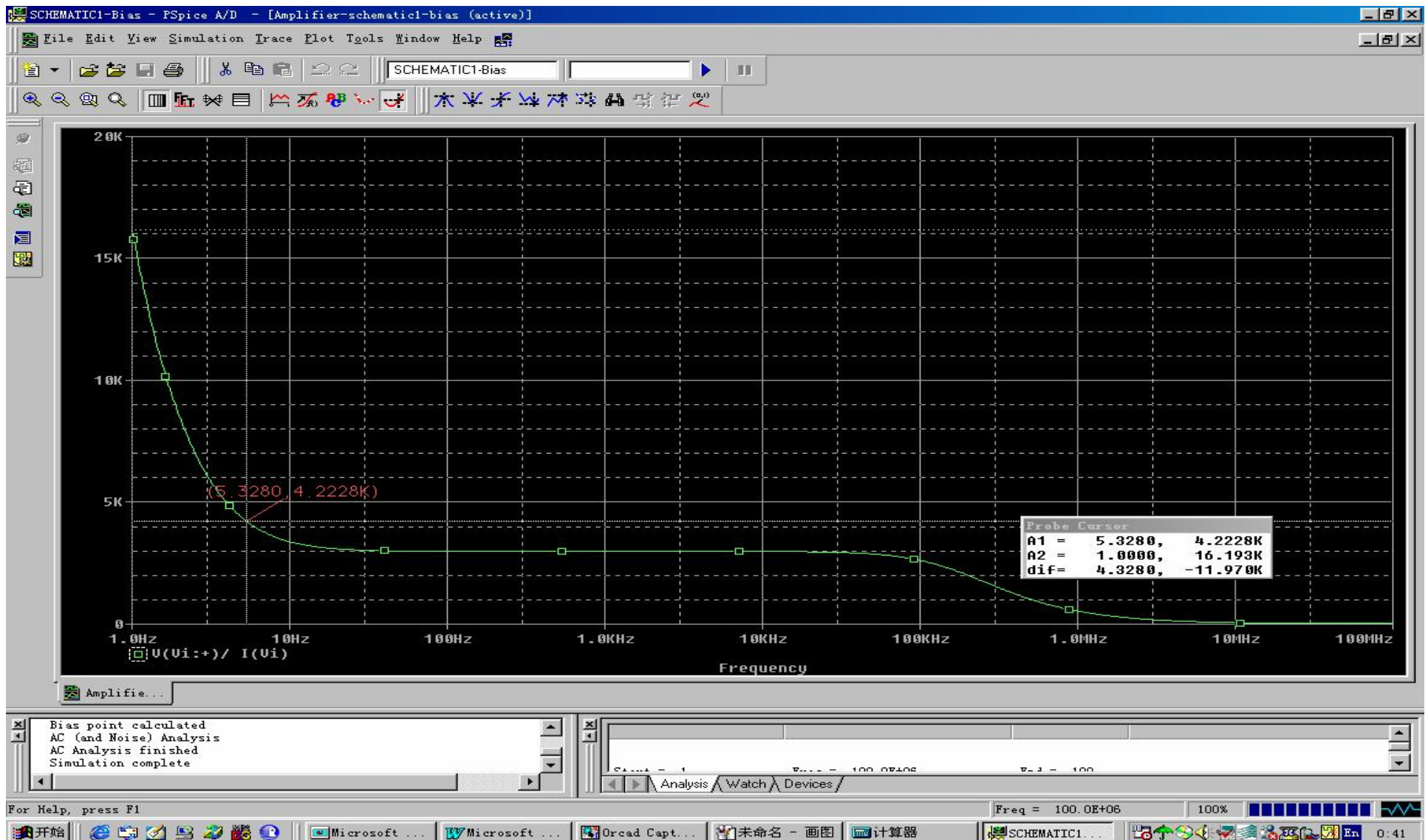


标记低半功率点处坐标

- 执行Plot/Label/Mark命令

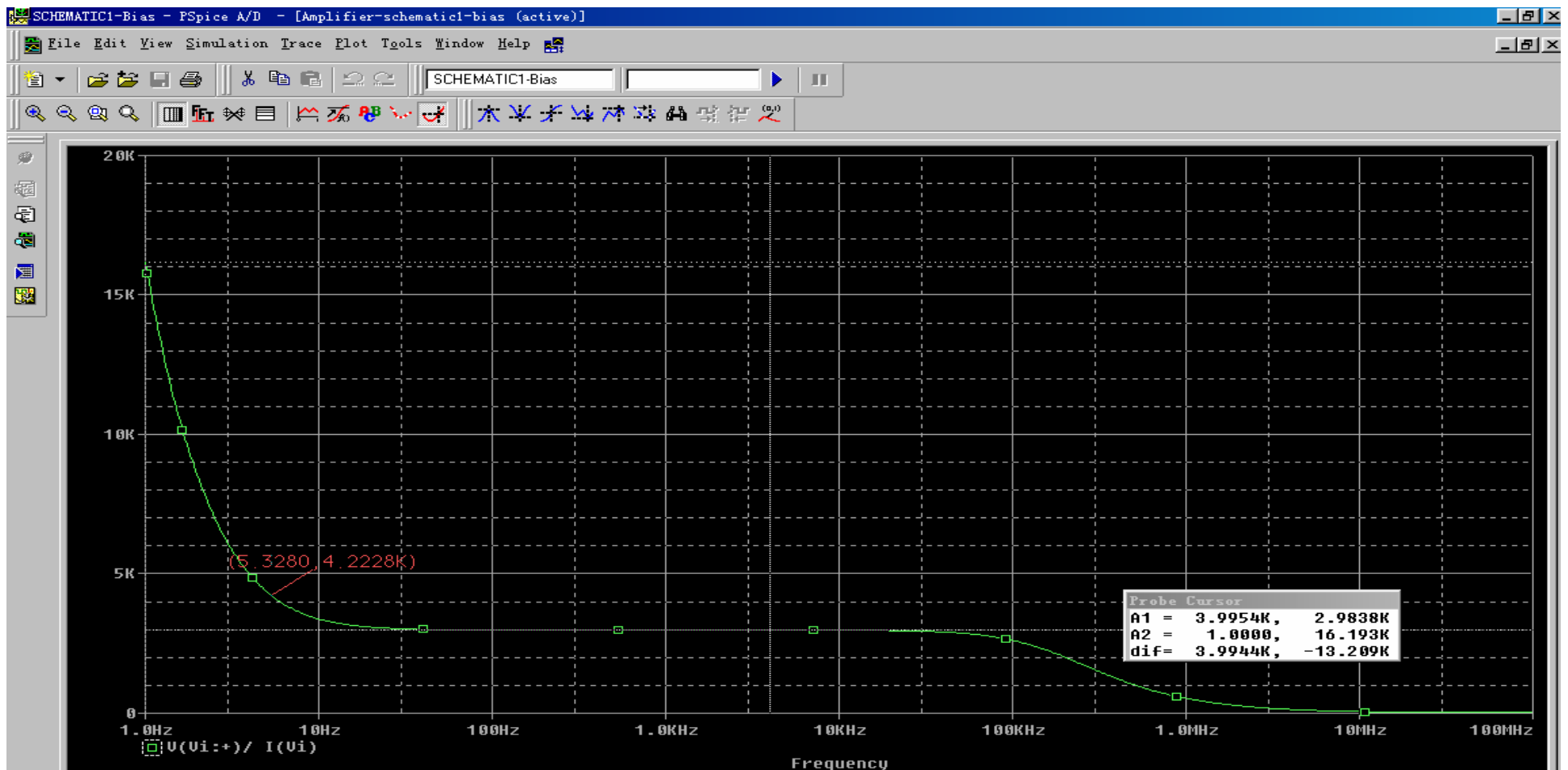


- 第一为低半功率点频率 $f_L=5.3280\text{Hz}$
- 第二为 f_L 处输入阻抗 $R_i=4.2228\text{k}\Omega$



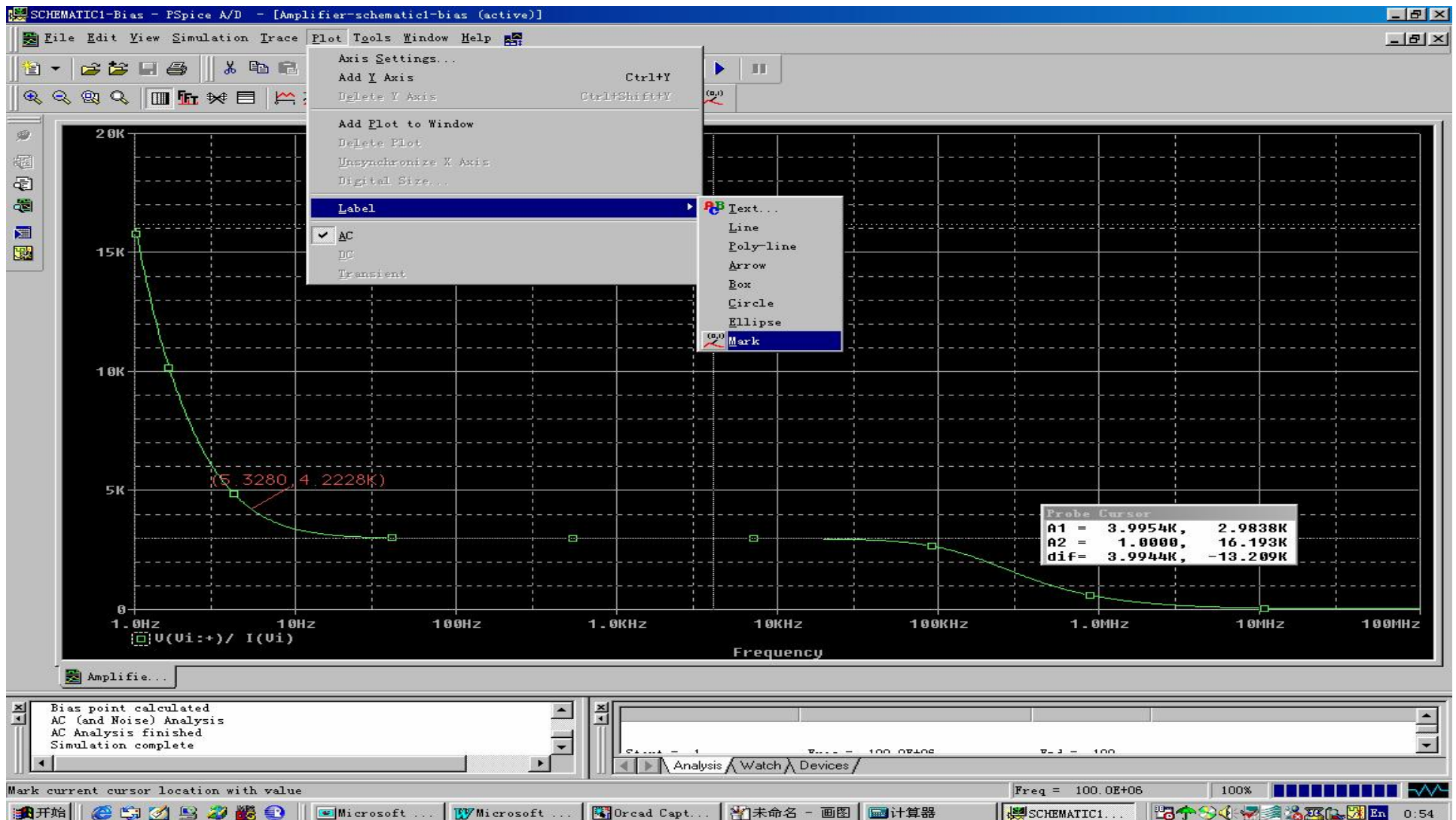
测量中频点 f_0 处 R_i

- 向右拖动十字标尺，对准测量中频点 f_0 处(约3.9954kHz)

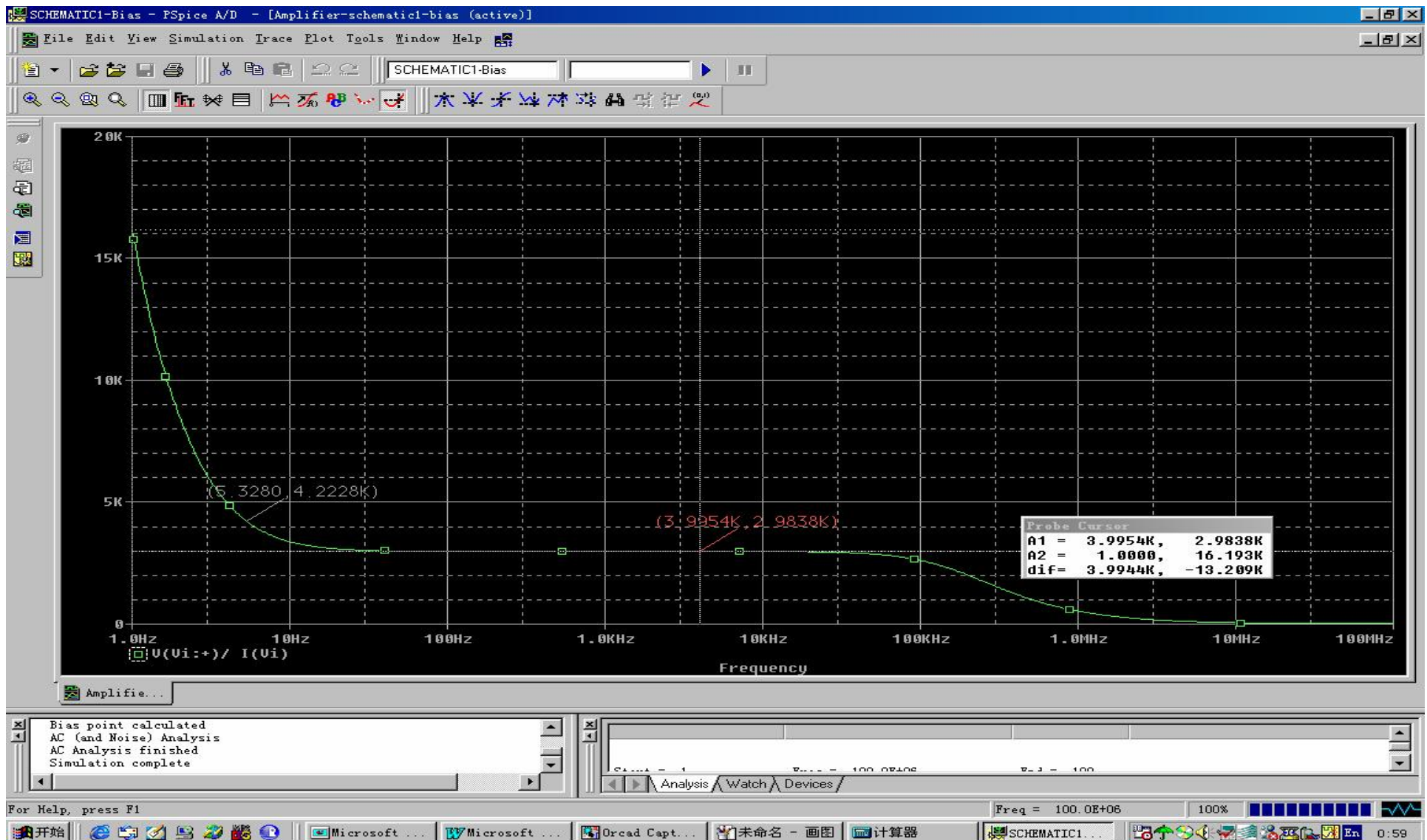


标记中频点处坐标

- 执行Plot/Label/Mark命令

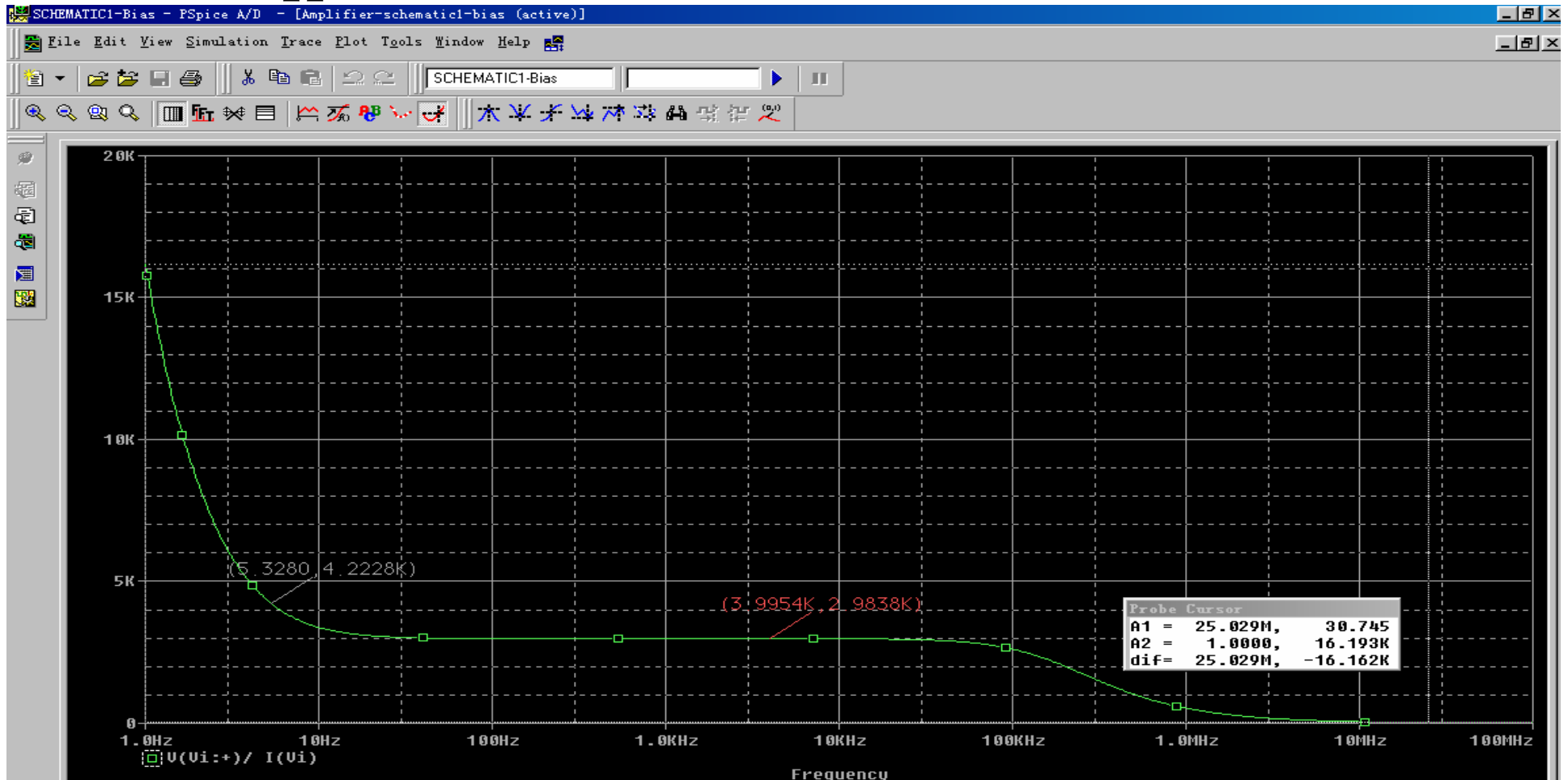


- 第一为中频点频率 $f_o=3.9954\text{kHz}$
- 第二为 f_o 处输入阻抗 $R_i=2.9838\text{k}\Omega$



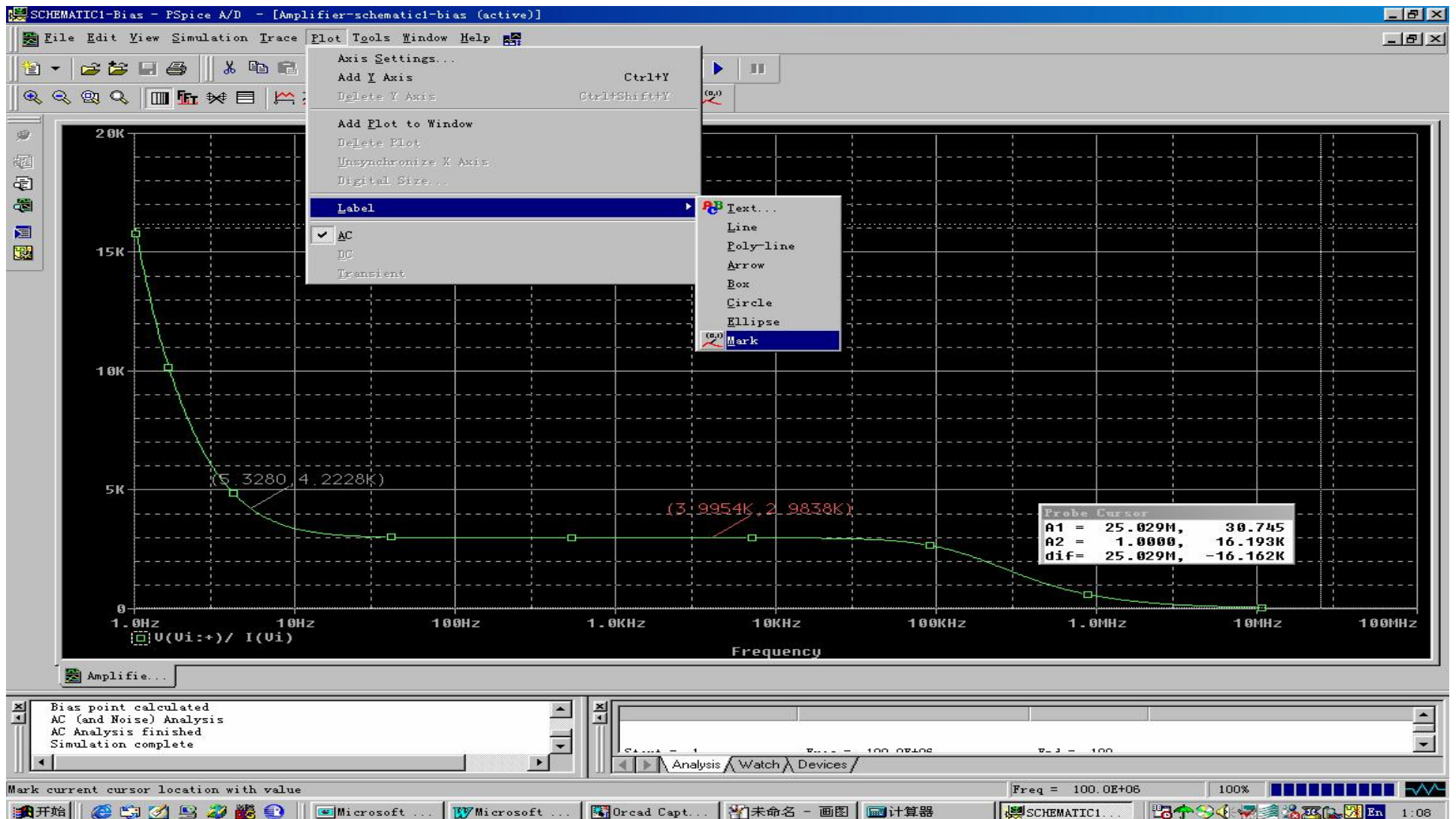
测量高半功率点 f_H 处 R_i

- 向右拖动十字标尺，对准测量高半功率点 f_H 处(约25.029MHz)

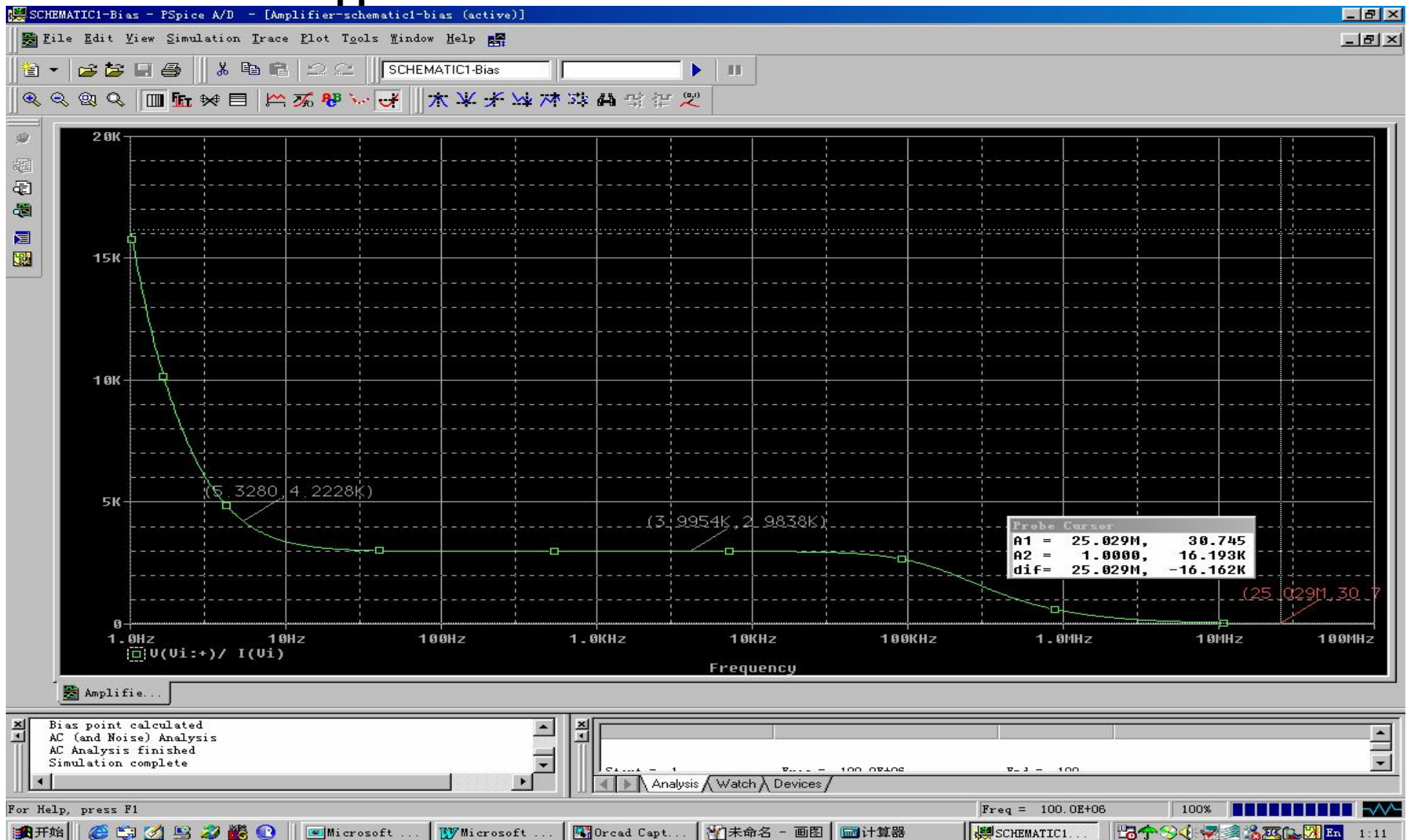


标记高半功率点处坐标

- 执行Plot/Label/Mark命令



- 第一为高半功率点频率 $f_H=25.029\text{MHz}$
- 第二为 f_H 处输入阻抗 $R_i=30.745\Omega$



(四)数据记录与处理

- 制定数据记录表格
- 记录与处理放大器瞬态分析数据
- 记录与处理放大器交流分析数据
- 至此，实验内容全部完成