3

This chapter shows the basics of AC simulations, including small signal gain and noise. It also shows many detailed features of the system.

Lab 3: AC Simulations

OBJECTIVES

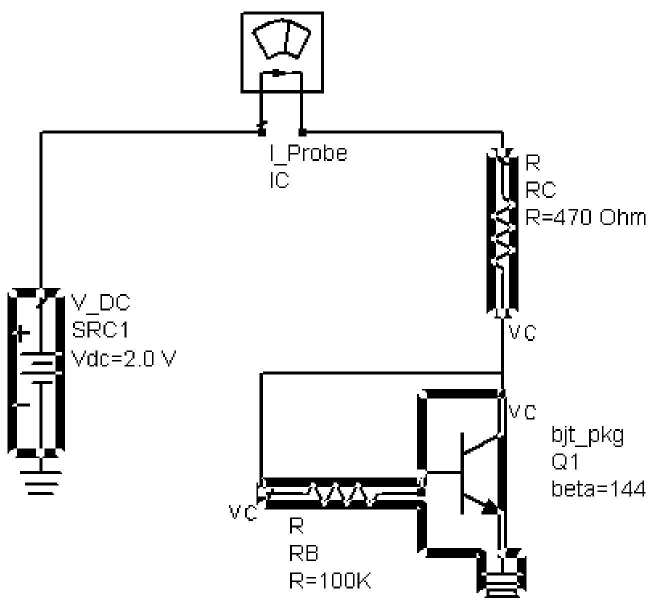
- Perform AC small-signal and noise simulations
- Sweep variables, tune parameters, write equations
- Control plots, traces, datasets, and AC sources

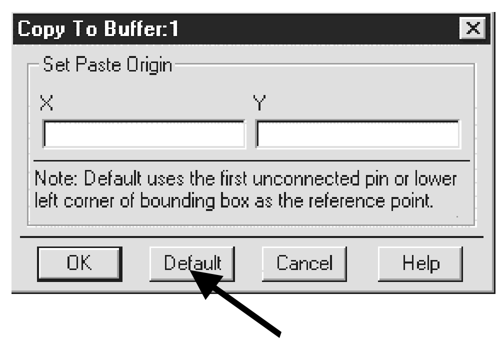
About this lab: This lab continues the mixer project and uses the same sub-circuit as the previous lab.

PROCEDURE

1. Use copy/paste to create a design

a. Open the last design from lab 2 **(dc_net)** and select (Shift click) the following items: **Vdc**, both bias **resistors**, the **bjt_pkg**, and the **ground**. Then click: **Edit > Copy / Paste > Copy to buffer**. Select the **Default** origin and then close the window.



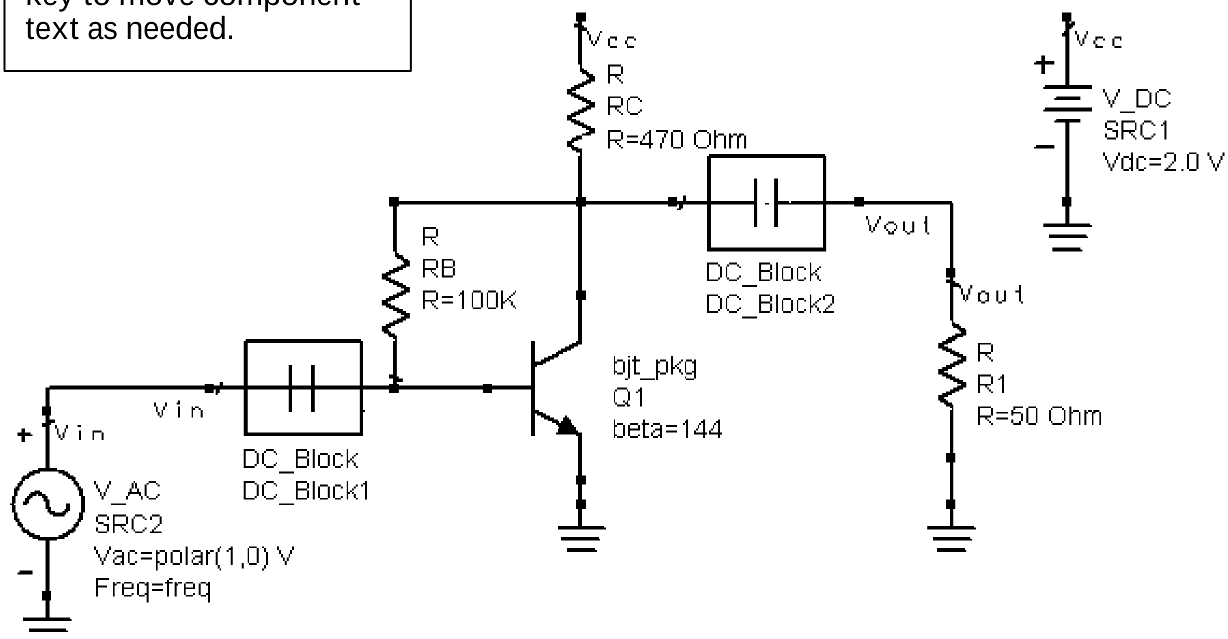


b. Use the File> New command to create a new schematic window and name it: **ac_sim**. Then click **Edit > Copy/Paste > Paste from buffer** and insert the ghost image on the schematic.

c. Save the new file. You must save it or it will not be written to the disk drive.

d. Continue building the circuit shown here using the following steps:

NOTE: After inserting a node name, use the F5 key to move component text as needed.

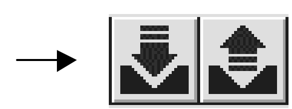


e. Insert the remaining components**: AC Simulation controller, dc blocking capacitors**, and the **V_AC voltage source**, **50 ohm load**, etc. Use the palettes to find the desired items.

f. Add **Vcc** as a **Node Name** instead of using a wire.

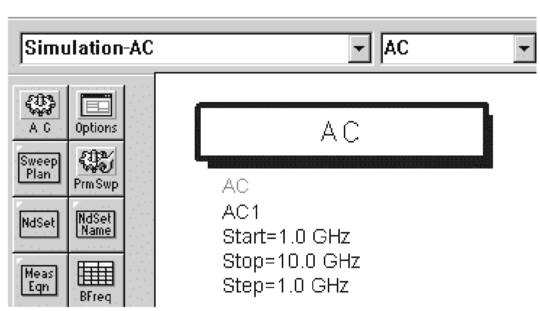
g. Add **Vin** and **Vout** as Node Names also.

h. Select the **bjt_pkg** and **push** into the sub-circuit (using the icon) to verify that it is your circuit, and then push out again.



2. Set up the AC Simulation

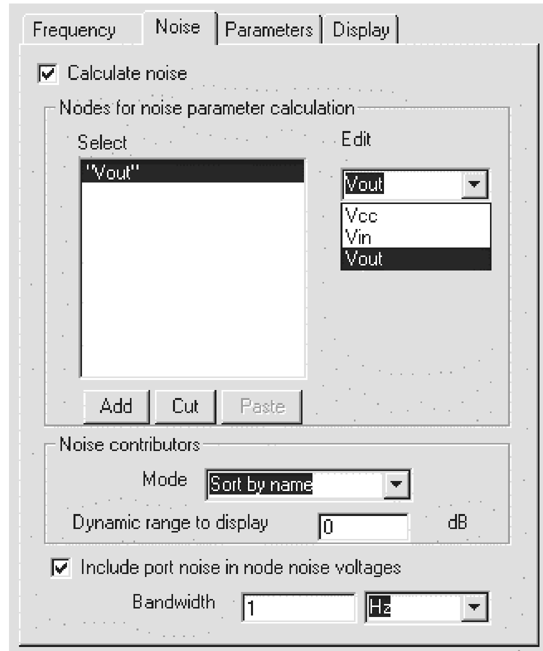
a. Insert an AC Simulation controller.

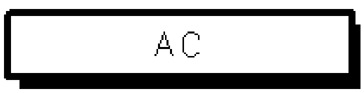


b. Edit the AC controller start, stop, and step as shown here.

c. Turn on the **Calculate noise** button and add the **Vout** node. Also, set the Mode to **Sort by Name**. You could sort by value to see the greatest contibutors listed first and then list the name in order to locate them on the schematic (good for large circuits)

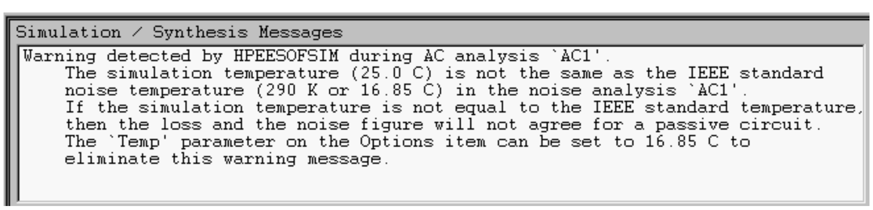
d. Turn on the **Display** for each of the parameters.





AC1
Start=100 MHz
Stop=2 GHz
Step=100 MHz
CalcNoise=yes
NoiseNode[1]="Vout"

e. **Simulate** the circuit: press the F7 key (default dataset name is the same as the schematic: ac_sim). Look at the status window. It should give a warning message like the one here because the default simulation-temperature is room temperature (25^{o} C) and not at the IEEE standard for noise measurements (290^{o} Kelvin).



Simulation / Synthesis Messages

Warning detected by HPEESOFSIM during AC analysis `AC1'.
The simulation temperature (25.0 C) is not the same as the IEEE standard noise temperature (290 K or 16.85 C) in the noise analysis `AC1'.
If the simulation temperature is not equal to the IEEE standard temperature, then the loss and the noise figure will not agree for a passive circuit.
The `Temp' parameter on the Options item can be set to 16.85 C to eliminate this warning message.

3. Set the *Options* card and Simulate

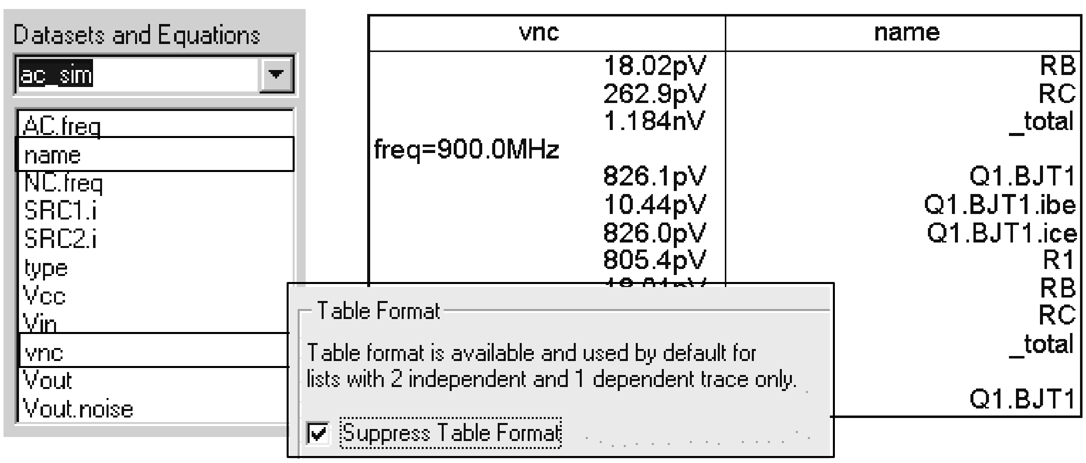
From the simulation palette, insert the **Options** card. This is a global used for temperature. Set **Temp** to **16.85**. **Simulate again** and there should be **no warning message**. The Options card also sets the tolerance for DC solutions.

Options
Options1
Temp=16.85
TopologyCheck=yes
V_RelTol=1e-6
I_RelTol=1e-6
GiveAllWarnings=yes
MaxWarnings=10

4. Display the noise data

a. Open a new data display and save it as **ac_data**.

b. Insert a list of **name** and **vnc** (voltage noise contributors) and click **Plot Options** and **Suppress Table Format**. As shown here, Q1.BJT1 is the total noise voltage for the device. It is composed of two pieces: Q1.BJT1.ibe and Q1.BJT1.ice. This means that the total BJT noise comes from both the base-emitter current (ibe) and collector-emitter current (ice).
However, these are two uncorrelated noise voltages that have been added as noise <u>powers</u>: $(V_{total})^2 = (V_{ibe})^2 + (V_{ice})^2$.
Also, note that the total vnc is the same as Vout noise. If you have time, insert a separate list of Vout.noise and verify this.



c. Save the data display.

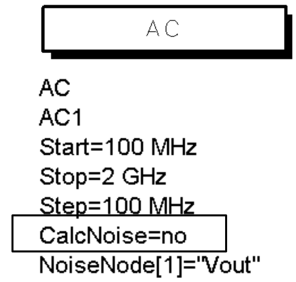
5. Write a Measurement Equation to calculate gain

a. Insert a **MeasEqn** from the AC simulation palette. Or, you can type in **MeasEqn** in the component history list.

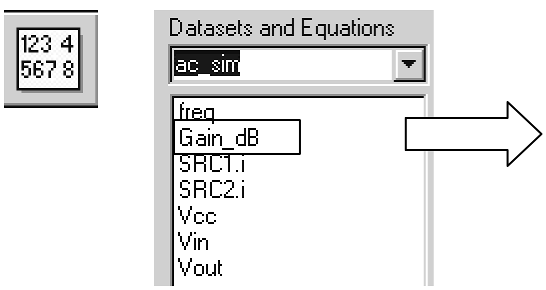
b. Now, edit the equation so it looks like the one shown. It computes the gain in dB using voltages at name nodes Vin and Vout:

6. Simulate without noise and display the results

a. In the schematic, **turn off the noise** calculation by editing the simulation controller setting on-screen. Turning off the noise calculation will save simulation time and data, especially for large circuits. Of course, this will make the list you inserted (name and vnc) become invalid.



b. Save the schematic. **Simulate** again. When the simulation is finished, insert the **Gain_db** equation in a list.



freq	Gain_dB
100.0MHz	5.572
200.0MHz	5.572
300.0MHz	5.571
400.0MHz	5.571
500.0MHz	5.570
600.0MHz	5.569
700.0MHz	5.567
800.0MHz	5.566
900.0MHz	5.564
1.000GHz	5.563

c. Now, in the data display, **insert an equation** to calculate the same gain. However, this time give it a different name, such as **dB_Gain**:

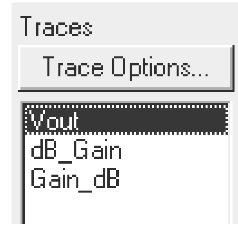
Eqn dB_Gain=20*log(mag(Vout) / mag(Vin))

d. **Edit the list** and add the data display equation: **dB_Gain**. Now you have two results (they are the same) from two equations – one written before simulation and one after simulation.

freq	Gain_dB	dB_Gain
100.0MHz	5.572	5.572
200.0MHz	5.572	5.572
300.0MHz	5.571	5.571
400.0MHz	5.571	5.571
700.0MHz	5.567	5.567
800.0MHz	5.566	5.566
900.0MHz	5.564	5.564
1.000GHz	5.563	5.563

Equation from schematic

Equation from data display



e. **Edit the list** one more time and add **Vout.** With the Vout data selected, click the **Trace Options** button.

f. In the **Trace Expression** field, change Vout to read: **dB (Vout)** as shown and click **OK**. You are using the built-in dB function on the Vout data. Because the AC signal at Vin is 1 volt, the dB value of Vout will have the same value as the dB gain equations you wrote.

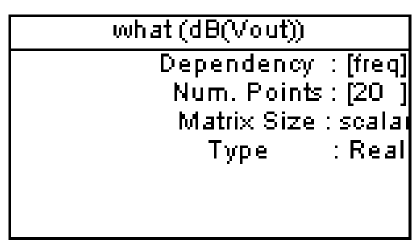
freq	dB_Gain	Gain_dB	dB(Vout)
100.0MHz	5.572	5.572	5.572
200.0MHz	5.572	5.572	5.572
300.0MHz	5.571	5.571	5.571
400.0MHz	5.571	5.571	5.571

NOTE on equations: The point of these last steps was to show the similarity and difference between equations you write in schematic and those you write in the data display. In addition, you should remember that variable equations in schematic (VarEqn) are primarily used to initialize (declare) variables sweeping, scaling, etc.

7. Use the *what* function on the Vout data

a. Insert a new **list** (dataset is still **ac_sim**). Add the **Vout** data again, select it, and click on the **Trace Options** button.

b. When the dialog box appears, insert the cursor in front of the trace expression and type the **what** function in front of the dB of Vout as shown here, using parentheses on each side. Click **OK** and you get the similar information as clicking *Variable Info* but you get it for the explicit expression: dB(Vout). Of course, the dependency is the same for dB(Vout) and Vout: freq. Try clicking **Variable Info** and see. Later on, you will use this function to determine how to index into dataset values, especially S-parameters and harmonic balance simulations where there is mixing.



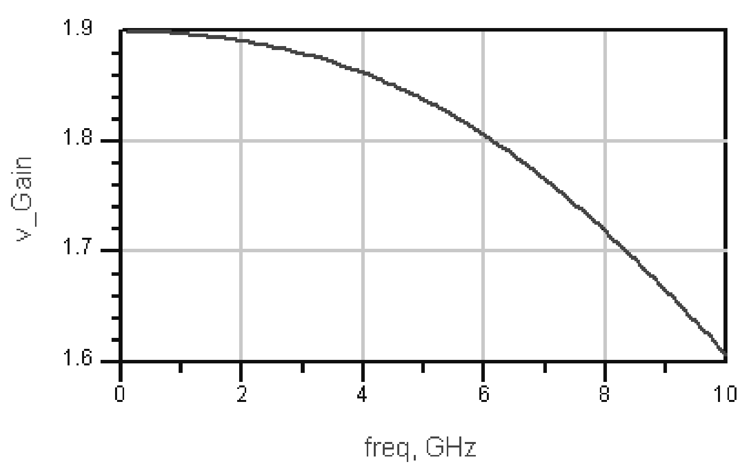
8. Copy the data display equation using Ctrl C Ctrl V

a. Select the **dB_Gain equation** and then press: **Ctrl C** and **Ctrl V**. Move the cursor and click nearby. The highlighted copy of the equation will appear with “1’ appended to the equation name (dB_Gain1).

b. Edit the copied equation to become a voltage gain equation:

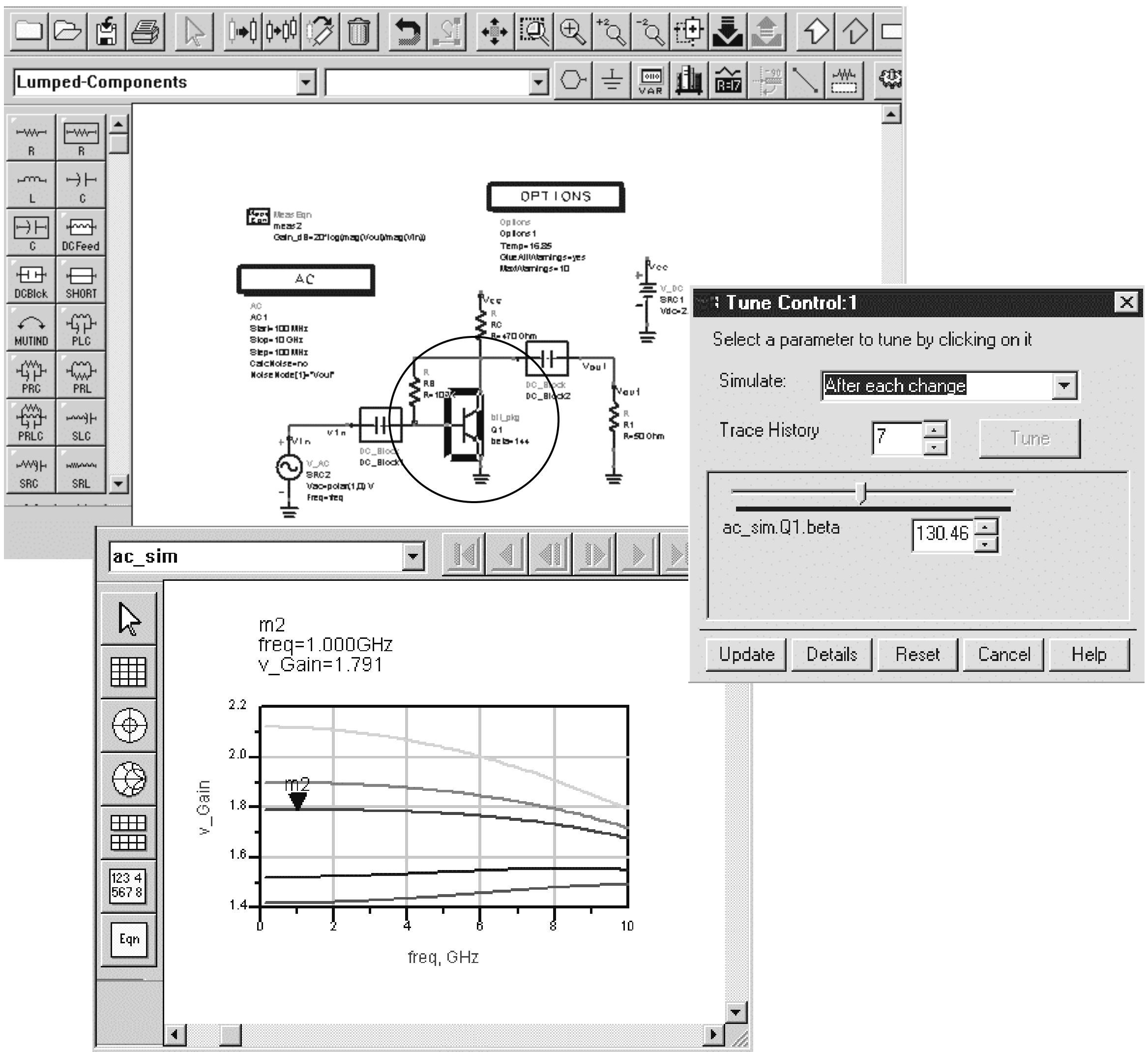
c. Return to the schematic, change the simulation stop frequency **to 10 GHz**, and simulate.

d. In the data display, insert a plot and add the **v_Gain** equation. You should see a plot similar to the one here showing the voltage gain.



9. Tune the beta parameter:

a. Position the schematic window and the data display so you can see them both. The select the bjt_pkg and start the tune mode (Simulate >Tuning). Put a marker on the trace – as you tune the parameter, the marker will move to the most recent trace.

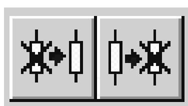


b. Try clicking **Update** to see the updated value of beta on the schematic. Note that the Reset button only resets the initial value on the Tune Control dialog. Be sure that beta is 144 when you Cancel the tuning or simply edit beta on the schematic to be 144.

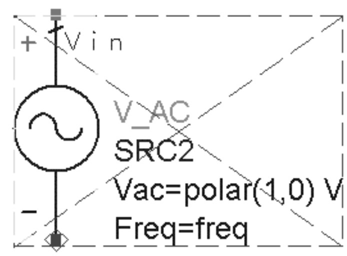
10. Use another source for the analysis

This step shows how sources are related to simulation controllers. By substituting a different source in the design, you will see the relationship. The V_AC, I_AC and P_AC sources are specifically designed for use with the AC Simulation controller. However, almost any frequency domain source can be used for an AC simulation if it has the Vac, Iac or Pac variable.

a. In the circuit schematic, select the V_AC source and move it (**Edit > Move > Move and Disconnect)** to the side of the schematic and **deactivate** it.

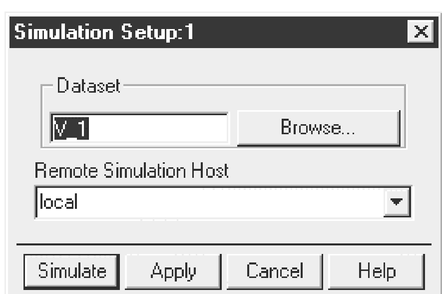


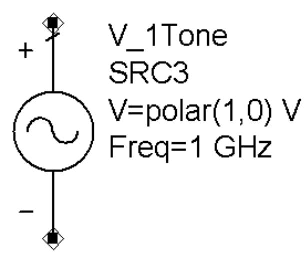
Activate and Deactivate icons



b. Insert a **V_1Tone** source (Frequency Domain sources). This source is designed to be used with the harmonic balance simulator but can be used here also. Note the difference in the default for **freq** (= freq or 1 Ghz).

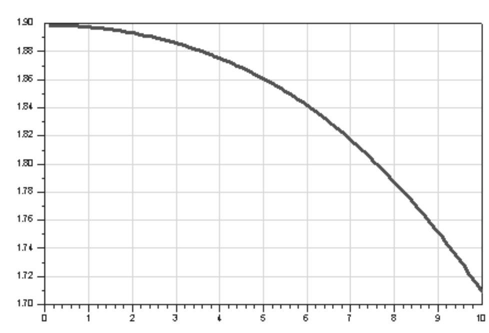
c. **Simulate** with the **dataset name = V_1**.



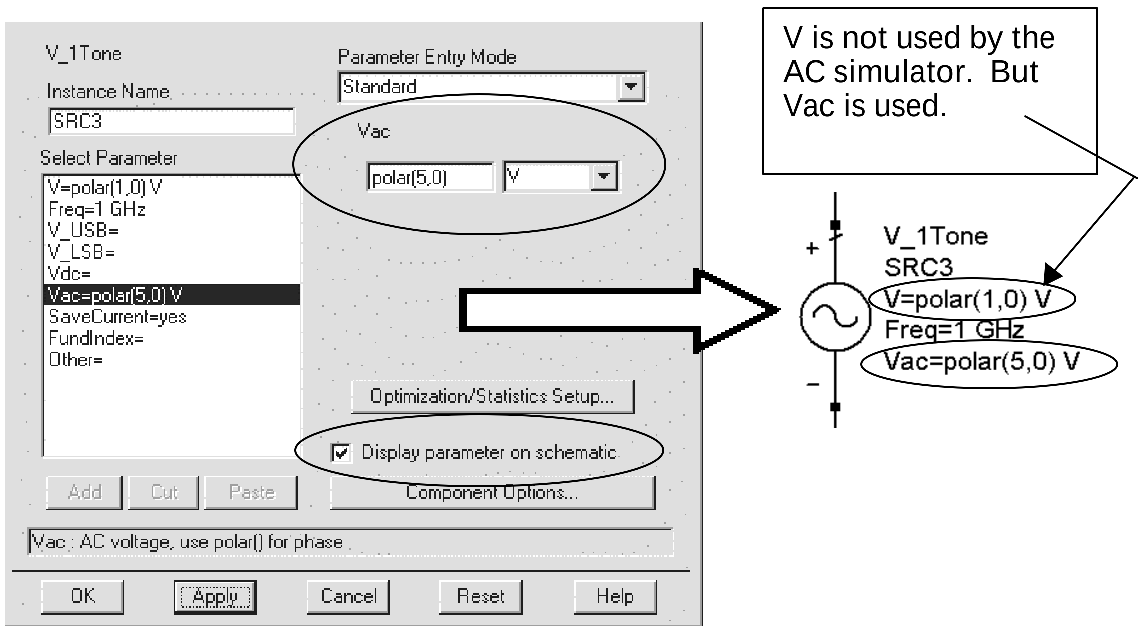


d. In the data display, insert a plot of the magnitude of **Vout**.

e. Go back and change the voltage from 1 V to 100 volts: **V=polar (100,0) V**. Now, set the dataset name as: **V_100**. **Simulate** and add the trace to the plot. You will see the exact same value. The next step will explain this…

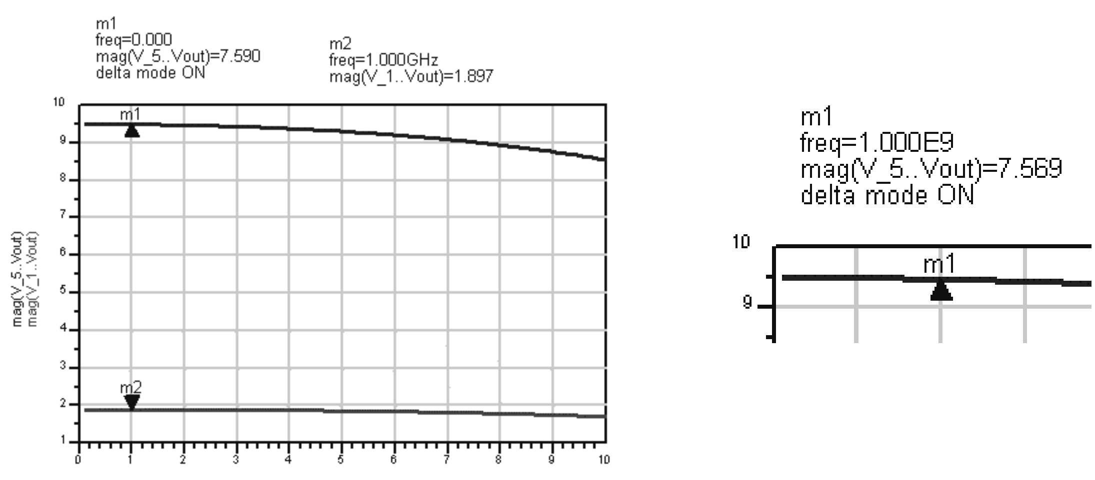


f. The AC simulation controller only reads the Vac setting. Because the source can be used with different simulation controllers, the setting of V is necessary. Therefore, edit the source so that the **Vac=5** volts and that the **Vac setting is displayed**.



g. Simulate again with the same **dataset name: V_5**. When the simulation is finished, edit the plot and add the magnitude of V_5 to the plot and you should now see the two traces**.**

h. **Delta Marker Mode**: Insert a marker on each trace at 1 GHz. Then select the two markers (shift click) and then click**: Marker > Delta Mode On** and choose M2 as the reference. The text will show the difference between the two markers on the Y axis. Move m1 to 2 GHz and see the change in the displayed marker text.

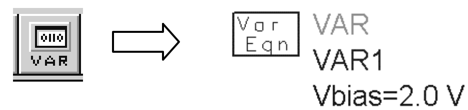


11. Sweep Vcc (as if the battery voltage dropped below 2 volts)

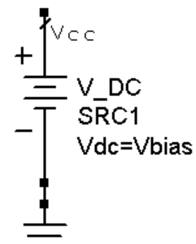
This step will require you to use the skills you already learned in this lab and in lab 2. You will set up a parameter sweep for **Vdc** from **1.8** to **2** volts in **0.05** volt steps.

a. Replace the V_1Tone source with a **V_AC** source and deactivate the V_1Tone – use the command **Edit > Move > Move and Disconnect and Edit > Component > Deactivate.**

b. Insert a **VAR** (variable equation) initializing **Vbias = 2 volts**.

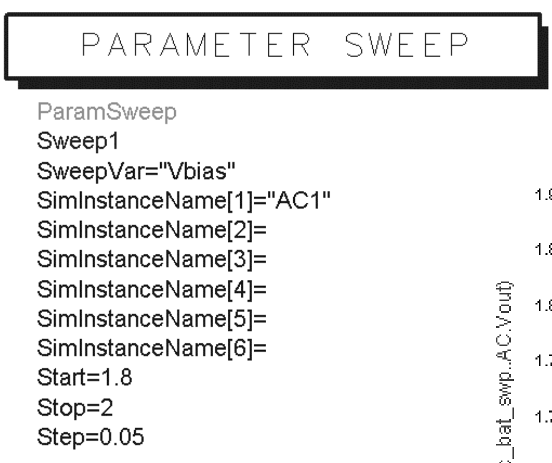


c. Redefine Vcc: **Vdc = Vbias**.

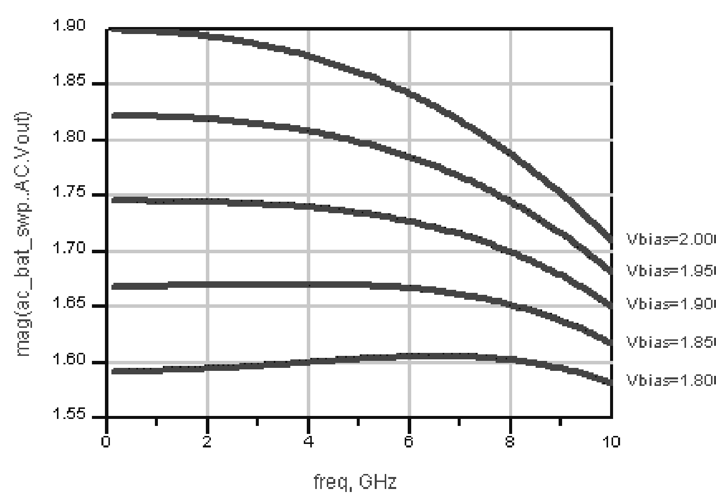


d. Insert a **Parameter Sweep**. Then set the **SweepVar** (sweep variable) to be **Vbias**, and be sure the Simulation Instance Name of the AC simulation controller is also set.

e. **Simulate** as **ac_bat_swp** (dataset name) and then display the magnitude of the **Vout** data as shown.



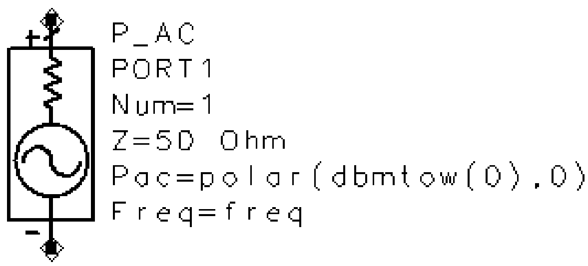
Simulated battery drain over broad frequency range. Also, Trace Options used to thicken the trace lines.

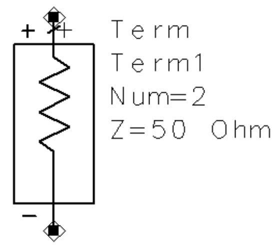


12. Save all your work and close the windows

EXTRA EXERCISES:

1. Simulate with port noise and ports. To do this, use a P_AC source as the input port (Num=1) and place a Term on the output as port 2 (Num=2). These two components are shown here with the port numbers.

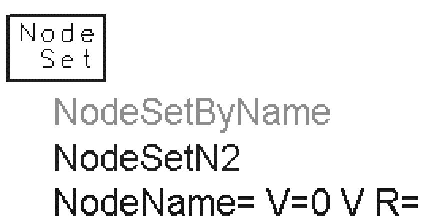


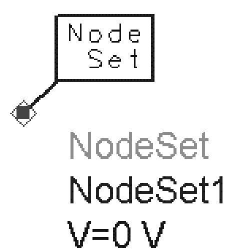


2. Insert the I_AC constant current source and simulate. To do this, you need to put a large resistance in parallel with the source because the simulator needs to verify a dc path to ground and the current sources are open circuits.

3. Insert the P_AC source and look at the power gain. Also, sweep another parameter and plot the results.

4. Try using the node settings in the AC simulation palette. You can set initial voltages at nodes using the Node Set or by referring to name nodes using the NodeSetByName component.





THIS PAGE LEFT INTENTIONALLY BLANK

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

射频工程师养成培训课程套装

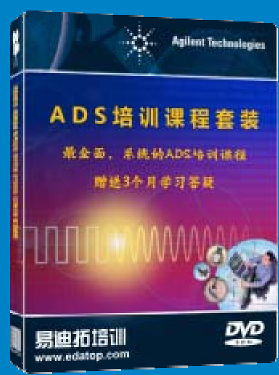
该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

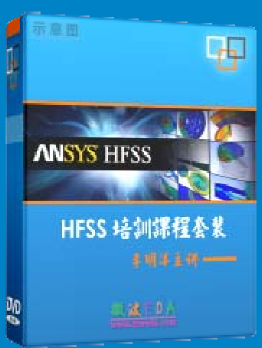
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html





HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com