ADS Fundamentals - 2001

LAB 2: System Design Fundamentals

Overview - This chapter introduces the use of behavioral models to create a system such as a receiver. This lab will be the first step in the design process where the system level behavioral models are simulated to approximate the desired performance. By setting the desired specifications in the system components, you can later replace them with individual circuits and compare the results to the behavioral models.

OBJECTIVES

- ***Use the skills developed in the first lab exercise.***
- ***Create a system project for an RF receiver using behavioral models (filter, amplifier, mixer) where: RF = 1900 MHz and IF= 100 MHz.***
- ***Test the system: S-parameters, Budget, Spectrum, Noise, etc.***

Agilent Technologies

Table of Contents

Agilent Technologies

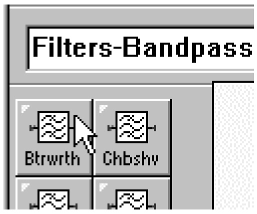
PROCEDURE

1. Create a New Project (system) and schematic.

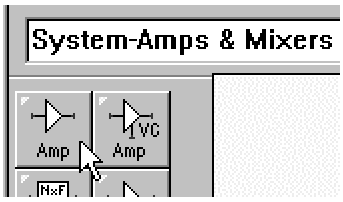
a. Use the File > New Project command and name the new project: system.

b. Open and save a new schematic with the name: rf_sys.

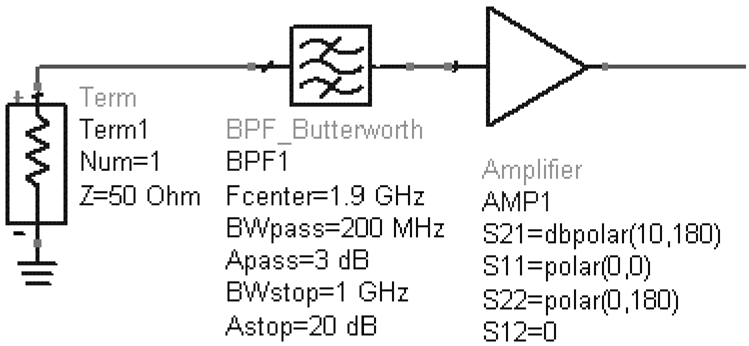
2. Build a behavioral RF receiver system.



a. <u>Butterworth filter</u>: Go to the palette and scroll down to Filters-Bandpass. Insert a Butterworth filter. Set it as shown: Fcenter = 1.9 GHz to represent the carrier frequency. Set BWpass = 200 MHz and BWstop = 1 GHz.



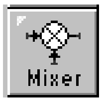
b. <u>Amplifier</u>: Go to the System-Amps & Mixers palette and insert the Amplifier. Set S21 = dbpolar (10,180).



c. <u>Term</u>: Insert a termination at the input for port 1. Terms are in the Simulation-S_Param palette or type in the name Term in the Component History and press Enter.

NOTE on Butterworth filter - The behavioral Butterworth response is ideal, therefore there is no ripple in the passband. Later on, when the filter and amplifier are replaced with circuit models, there will be ripple. For system filter modeling with ripple, use the behavioral Elliptical filter.

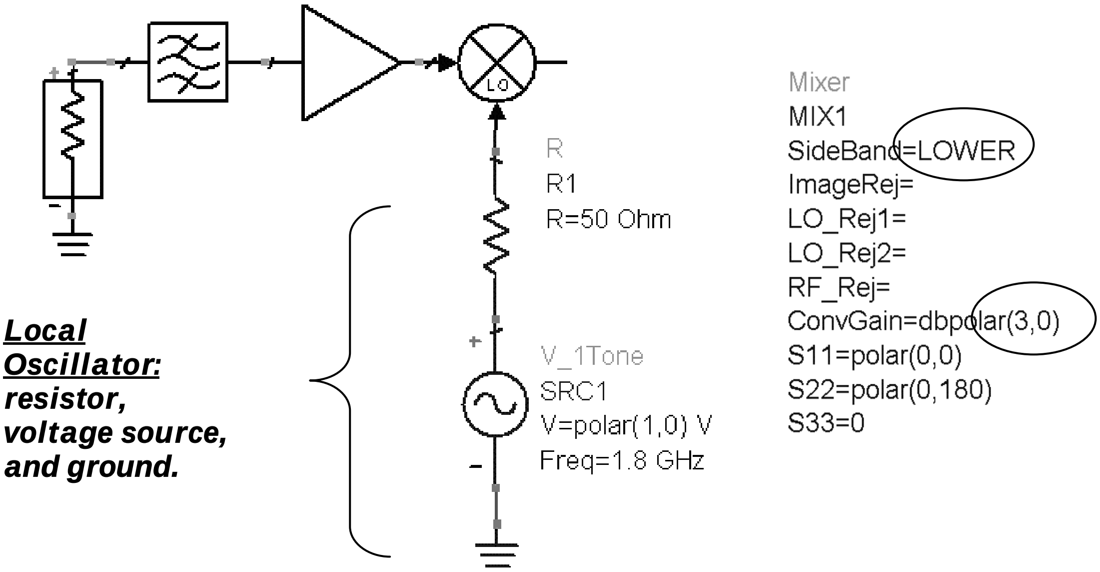
The next step is to add a behavioral mixer and LO with phase noise to the schematic.



d. ***From the System-Amps & Mixers palette, insert a behavioral Mixer at the amp output - be careful to insert the Mixer and not Mixer2. Mixer2 is for nonlinear analysis and Mixer works with the frequency conversion feature used here.***

e. ***Set the Mixer ConvGain = dbpolar (3,0). Also, set the Mixer SideBand = LOWER by inserting the cursor in front of the default (BOTH) and using the keyboard UP and DOWN arrow keys to toggle the setting to LOWER. Leave all other settings in the default condition.***

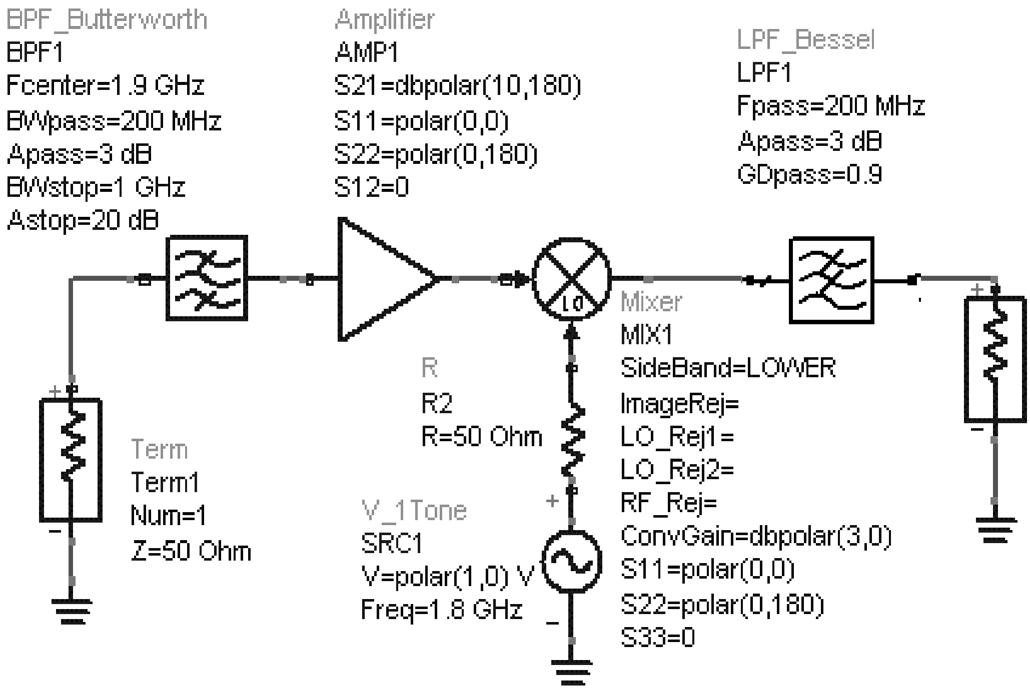
f. ***NOTE on moving component text - Click the F5 keyboard key and then click on a component to move its text.***



g. ***Add the LO by inserting a 50 ohm resistor in series with a V_1Tone source from the Sources-Freq Domain palette. Set the Freq to 1.8 GHz. This will provide an IF of 100 MHz at the output. Don't forget the ground.***

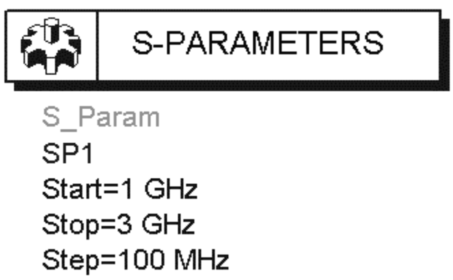
h. ***Add a low pass Bessel filter at the mixer output as shown here. The filter is in the Filters-Lowpass palette. Set Fpass = 200 MHz.***

i. ***Insert a Term for port 2. The final system circuit should look like the one shown here:***



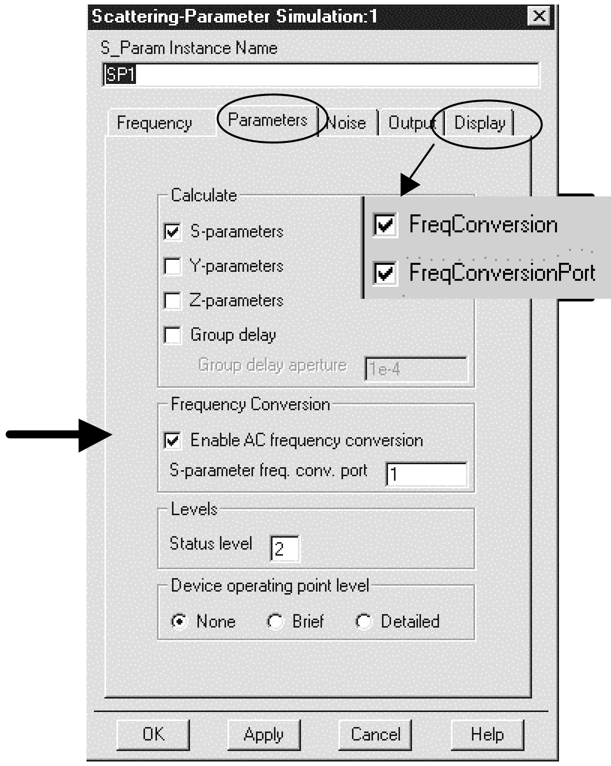
3. ***Set up an S-parameter simulation with frequency conversion.***

a. ***Insert the controller and setup the simulation: 1 GHz to 3 GHz in 100 MHz steps as shown here.***

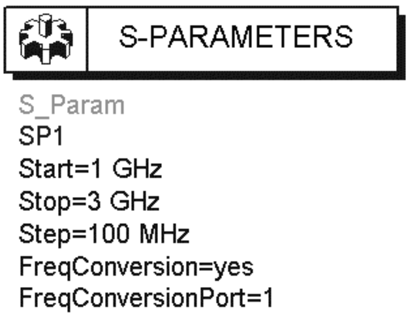


b. ***Edit the Simulation controller and, in the Parameters Tab, Enable AC frequency conversion by checking the box as shown here.***

c. ***Go to the Display tab and check the two boxes to display the settings shown here: FreqConversion and FreqConversionPort.***

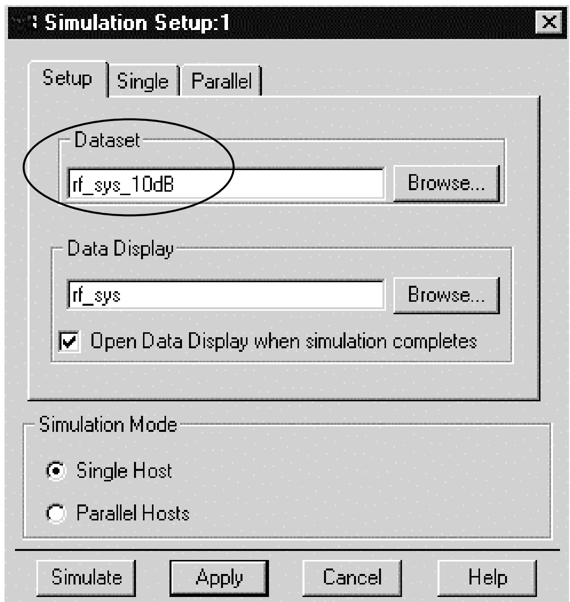


The S-parameter simulation controller should now look like the one shown here:

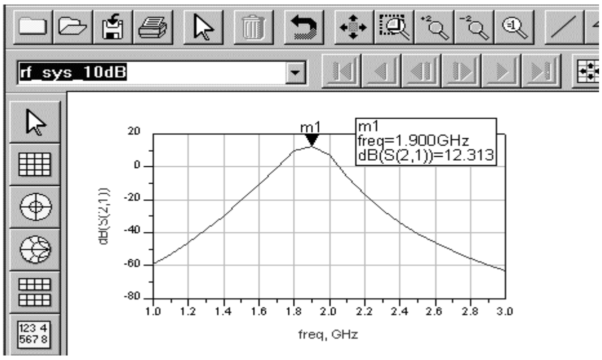


d. ***Click: Simulate > Simulation Setup. When the dialog appears, change the default dataset name to rf_sys_10dB to indicate that this simulation data represents the system with 10dB of amplifier gain.***

e. ***Click Apply and Simulate.***



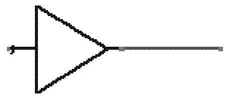
4. Plot the S-21 data.



f. In the Data Display window, insert a rectangular plot of S(2,1).

g. Put a marker on the trace at 1900 MHz. The gain includes mixer conversion gain minus some loss to due mismatches.

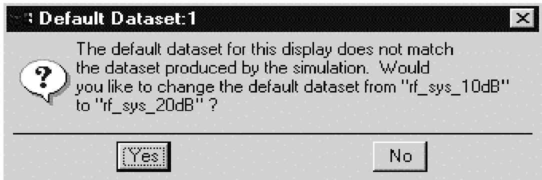
5. Increase gain, simulate, and add a sccond trace.



a. Go back to the schematic and chage the amplifier gain S21 from 10 to 20 dB as shown here.

b. In Simulate > Simulation Setup, change the dataset name to rf_sys_20dB. Click Apply and Simulate.

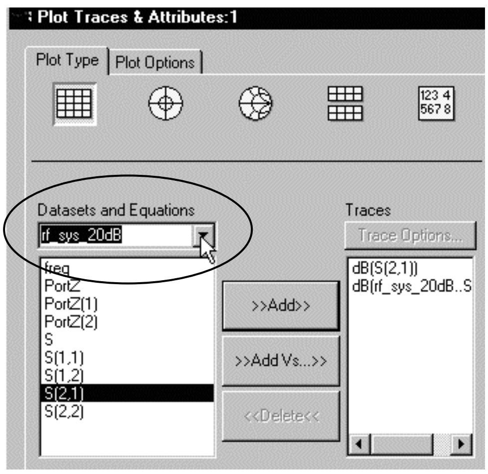
c. When the simulation finishes you will be prompted to change the default dataset – answer No.

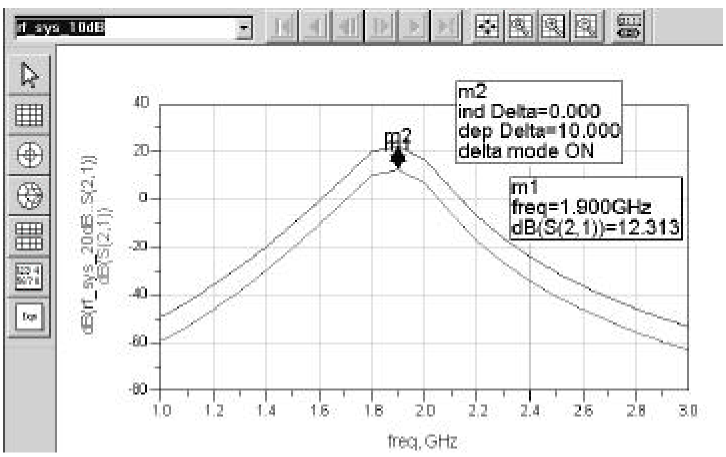


d. Edit the existing plot with the 10dB trace by double clicking on it. When the dialog appears, click the arrow to see the available Datasets and Equations (shown here) and select the rf_sys_20dB dataset.

e. Select the S(2,1) data and Add it in dB, clicking OK. Notice that the entire dataset pathname appears because it is not the default dataset.

f. Put a new marker on the new trace. Select both markers and click the command Marker > Delta Mode On to see the 10dB difference between the two simulations. Be sure to save the Data Display.

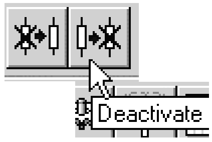


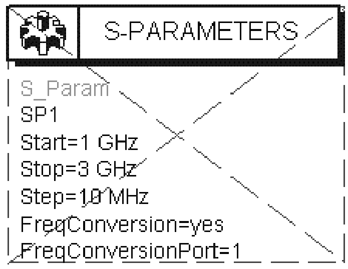


6. Set up an AC simulation with a P_1Tone source.

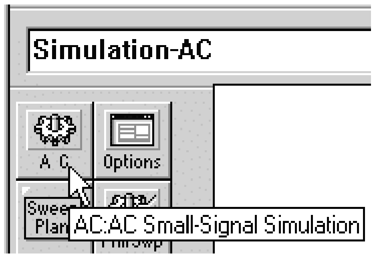
The budget gain simulation capability can be done with Harmonic Balance or with AC analysis if there is no transistor level mixer. In this lab, because the behavioral mixer's nonlinearity is known, the AC analysis can be used.

a. In the current rf_sys schematic, deactivate the S-parameter simulation controller by selecting it and using the icon shown here. You can always activate and deactivate components or controllers as necessary.



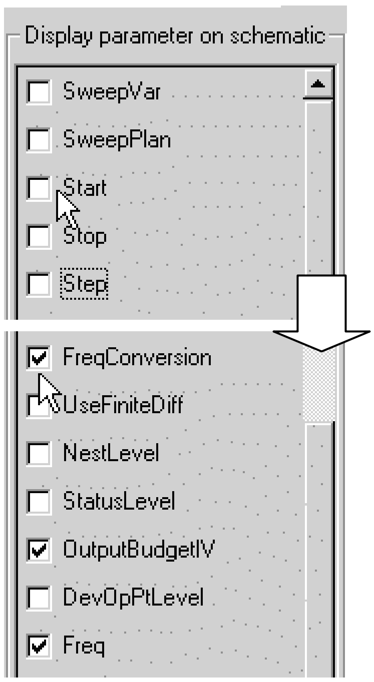


b. Go to the Simulation-AC palette and insert an AC simulation controller into the schematic.



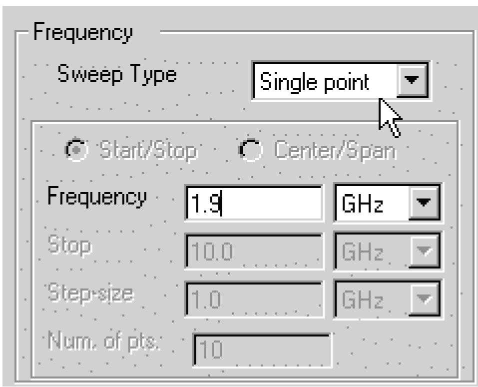
c. Edit the AC controller: go to the Frequency tab and set a Single point at 1.9 GHz which will be the RF input frequency to the system. Click Apply and the values will disappear from the screen.

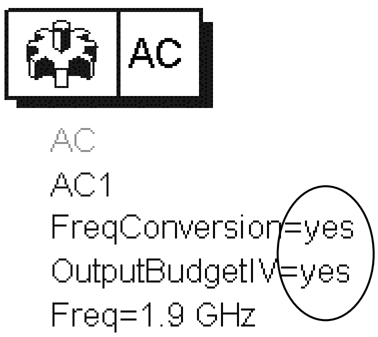
d. Display tab - Also in the AC controller, go to the Display tab and uncheck the Start, Stop, and Step settings (shown here). Then scroll down to the bottom and check the boxes for: FreqConversion, OutputBudgetIV, and Freq as shown here and click Apply and OK.



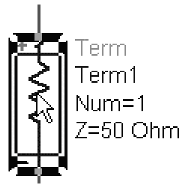
e. Change the two settings from no to yes.

The AC controller should look like the one shown here:

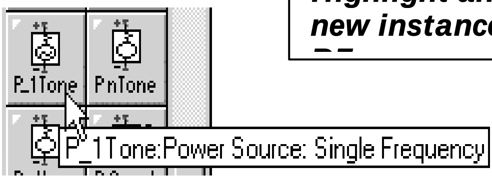




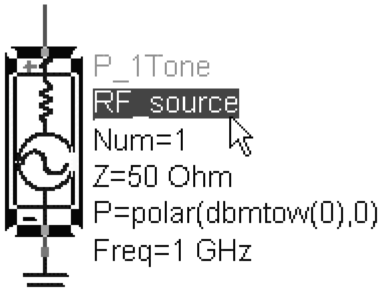
f. ***Select the Port 1 termination and delete it using the keyboard Delete key or the ADS delete icon (trash can).***



g. ***Go to the Sources-Freq Domain palette and insert a P_1Tone source where the Term was located. Next, rename the instance as RF_source. Although this is a Harmonic Balance source, you can use it for AC.***

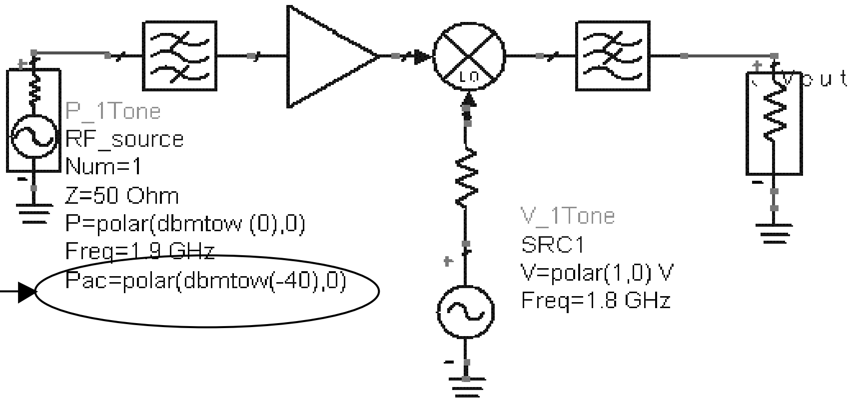


Highlight and type the new instance name:



h. ***Edit the P_1Tone RF_source (double click it). Check the box to Display Parameter on Schematic for Pac and Apply it. Next, set the Pac= –40 dBm and set Freq = 1.9 GHz as shown here. For many sources, the Power default uses the polar function and the dbmtow function. The dbmtow function passes the value (-40 dBm) to the simulator because the simulator operates***

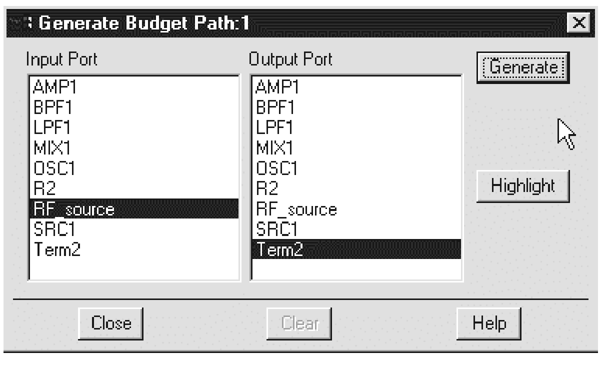
AC simulation requires AC source power settings. Check the Display box so see this setting.



in units of <u>watts</u>.

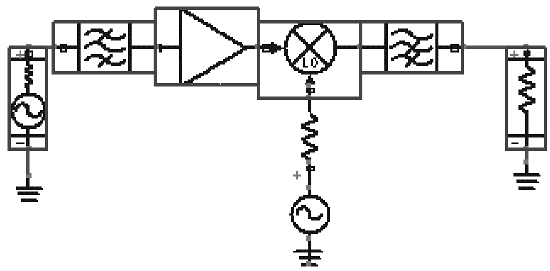
7. ***Set up the Budget Gain***

a. ***Two items are required to do the budget analysis: a budget path and a budget gain equation. Click: Simulate > Generate Budget Path. Then select input port RF_source and output port Term2. Click:***



Generate and then click: Highlight.

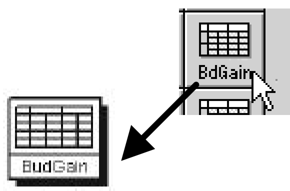
b. ***Highlighted path - Notice that all the components in the path from RF_source to Term2 are highlighted. However, the components in the LO path are not highlighted. Click: Close to dismiss the Generate Budget dialog box.***



c. ***BudPath equation - When you clicked the Generate button, a MeasEqn (measurement equation) BudPath was automatically created and inserted on your schematic. Locate it (shown here) and move it closer to the design. This path equation can be passed into (used for) various budget measurements.***

MeasEqn
BudPath
budget_path = ["RF_source.t1","BPF1.t2","AMP1.t2","MIX1.t2","LPF1.t2","Term2.t1"]

d. ***Insert a BudGain component from the Simulation-AC palette. Modify the equation so that only the budget_path argument is in parentheses. In this case the default system impedance of 50 ohms will be used.***



BudGain
BudGain1
BudGain1=bud_gain (budget_path)

8. ***Simulate with a new dataset name.***

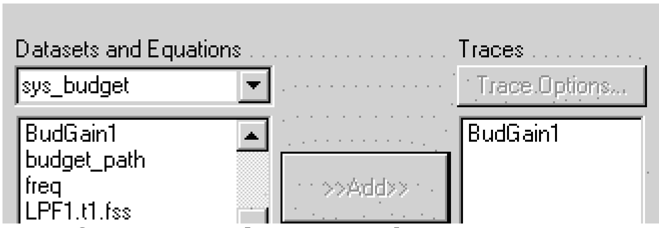
   a. ***Click: Simulate > Simulation Setup. Change the Dataset name to sys_budget. Keep the same data display window (rf_sys.dds). Now, you should begin to recognize that a dataset has the extension .ds and the data display window has the extension .dds.***

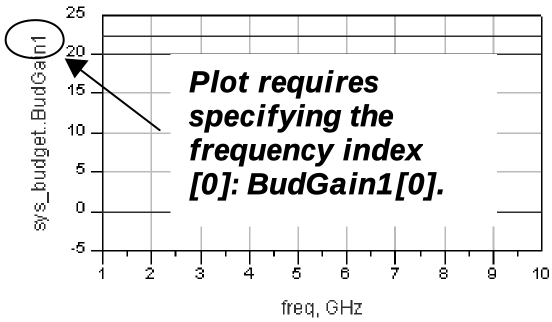
   b. ***Click Apply and Simulate. The data display (rf_sys.dds) should still be opened from the last simulation.***

9. *Set up Data Display pages and plot budget data.*

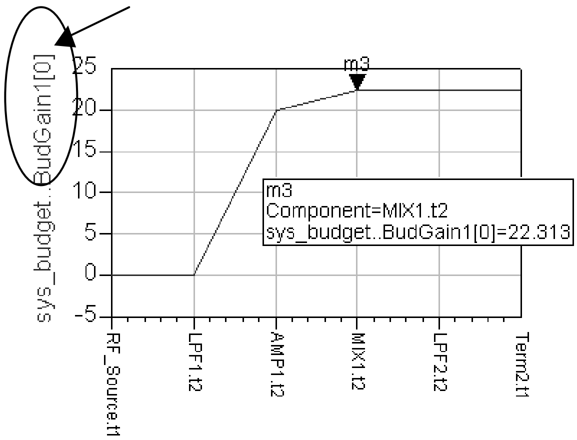
a. ***In the current data display window, rename the existing page, click Page > Rename Page and type in a name: s_params. Then add a new page for the budget analysis. Click: Page > New Page. Type in a name for the new page: budget. Now you know how to add pages in the data display.***

b. ***Insert a rectangular plot and then select the dataset: sys_budget and add the BudGain1 data as shown. This data needs to be specified for the frequency (next step).***

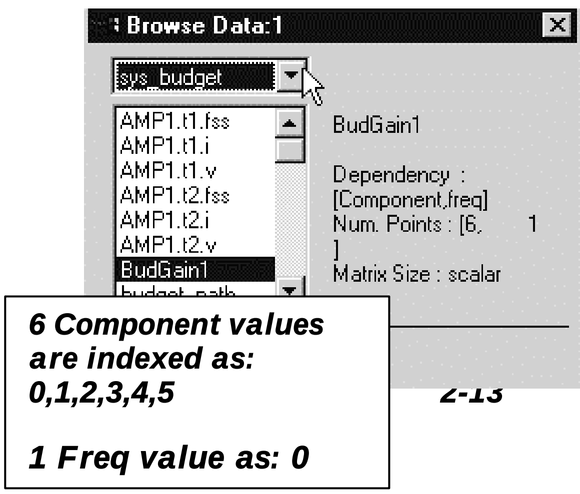




c. ***Edit the Y-axis label BudGain1 by inserting the cursor at the end of BudGain1 and typing: [0]. This is needed, because <u>the frequency index value must be specified</u>. In this case, there is only one frequency point (1.9 GHz) in the data matrix, and its index value is zero. The Y-axis should now read: BudGain1[0]. Also, put a marker on the trace to read the gain.***



d. ***To learn more about how data is handled in ADS, double click the trace (not the plot). When the dialog appears, click the button labeled Variable Info. Then select the dataset (sys_budget) and the data item BudGain1 as shown here. Now you can see that there are 6***



components and 1 frequency point in the matrix. This will be useful later.

e. Save the schematic and data display.

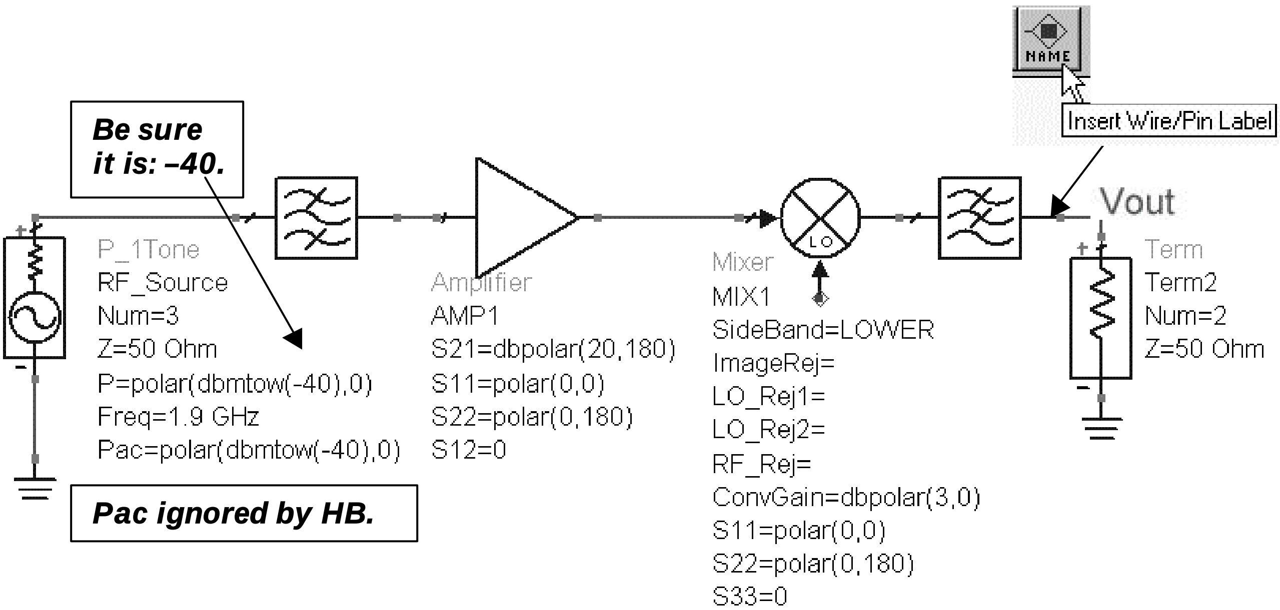
10. Set up the circuit using an LO with Phase Noise.

This next step shows how to simulate phase noise, contributed by a behavioral oscillator, using the Harmonic Balance simulator. At this point in the course, it is not required that you understand all the Harmonic Balance settings (covered later).

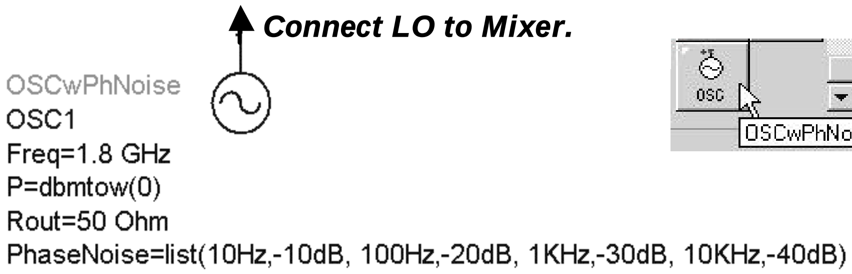
a. Save the current schematic with a new name. Click: File > Save Design As and type in the name: rf_sys_phnoise.

b. In the saved schematic, <u>delete the following components</u>: simulation controllers, both budget equations, the V_1Tone LO and the LO impedance 50 ohm resistor R1.

c. Insert a wire label Vout (node) and set RF_source power to –40 so the schematic looks like the one shown here:

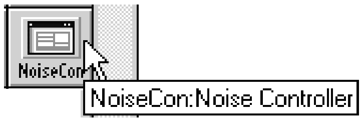


d. Go to the Sources-Freq Domain palette, scroll to the bottom, select and insert the OSCwPhNoise connected to the mixer. Set Freq = 1.8 GHz and change the PhaseNoise list as shown. Notice that the default value of P is the power in dBm and it has 50 ohms Z in (Rout).


Connect LO to Mixer.
OSCwPhNoise
OSC1
Freq=1.8 GHz
P=dbmtow(0)
Rout=50 Ohm
PhaseNoise=list(10Hz,-10dB, 100Hz,-20dB, 1KHz,-30dB, 10KHz,-40dB)


OSC
OSCwPhNoise:Oscillator with Phase Noise

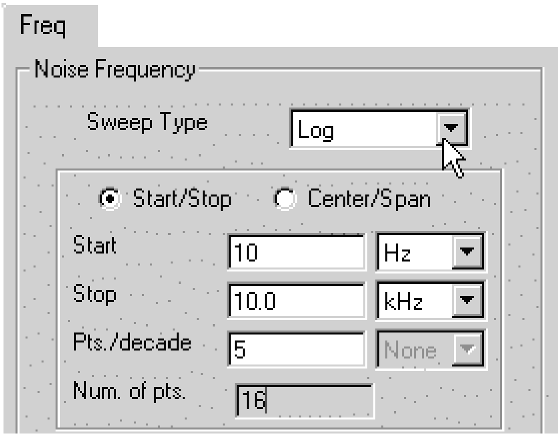
11. Set up a HB Noise Controller.

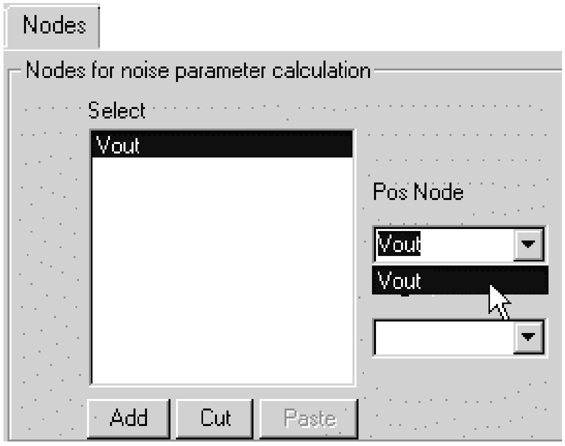


a. Insert the HB noise controller - Go to the Simulation-HB palette and insert a Noise Con (Noise Controller) on the schematic.

NOTE on NoiseCon: This component is used with the HB simulation controller. It allows you to conveniently keep all noise measurements separate from the simulation controller. Also, you can setup and use multiple noise cons for different noise measurements while only using only one HB controller.

b. Freq tab - Edit the Noise Con – go to the Freq tab and set the Sweep Type to Log from 10 Hz to 10KHz with 5 points per

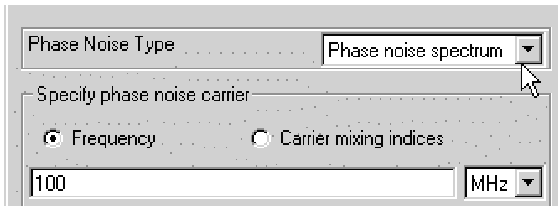




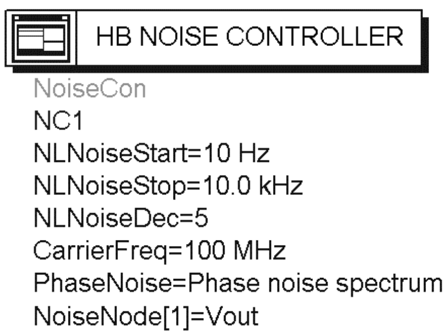
decade.

c. Nodes tab – Click the Pos Node arrow, select the Vout node, and click the Add button. The noise controller, like other ADS componets, can read and identify node names in the schematic.

d. PhaseNoise tab – Select Type as Phase Noise spectrum and set the carrier Frequency to 100 MHz. This is the IF frequency which has phase noisedue to the LO.



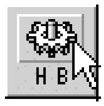
e. Display tab – Go to the Display tab and check the boxes for the settings you made (shown here). In the future, you may prefer to display the desired settings



Display these settings:

first and then edit them on the schematic.

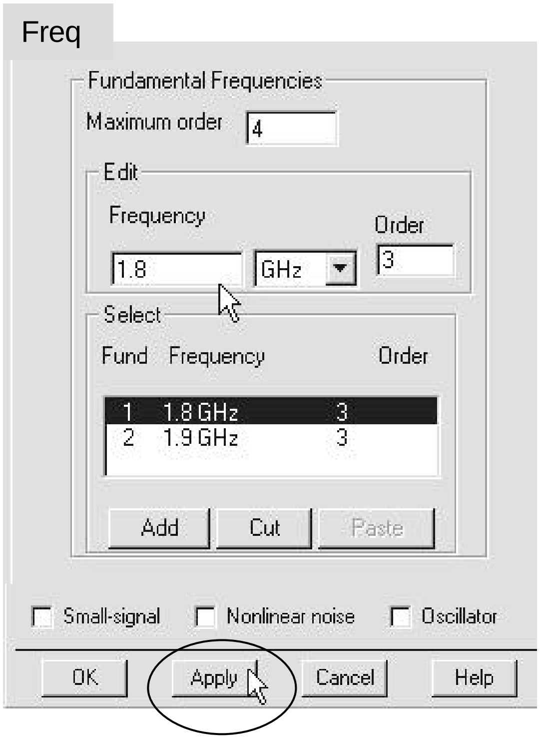
12. Set up the HB controller.



a. Go to the Simulation-HB palette and insert a HB simulation controller on the schematic

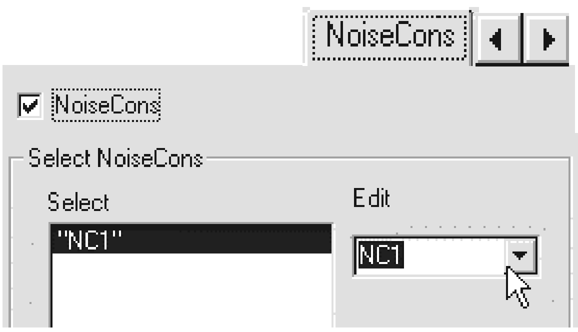
b. Edit the HB controller (double click). Change the default freq setting to 1.8 GHz using the Apply button. Then add the RF frequency 1.9 GHz and click Apply again.

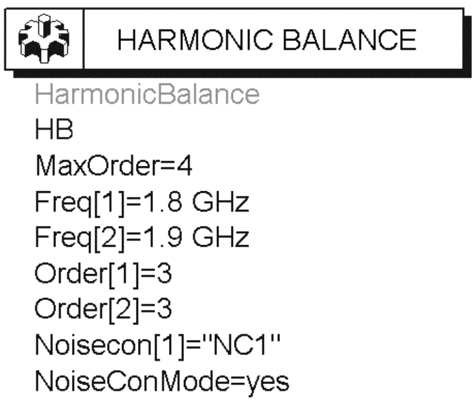
c. In the Display tab, check the box to display MaxOrder and click Apply.



NOTE on HB freq settings - You only need to specify the LO freq (1.8 GHz) and the RF freq (1.9 GHz) in the controller. There is no need to specify any other frequencies because the defaults for Order (harmonics) and Maximum order (mixing products) will calculate the the other tones in the circuit, including the 100 MHz IF.

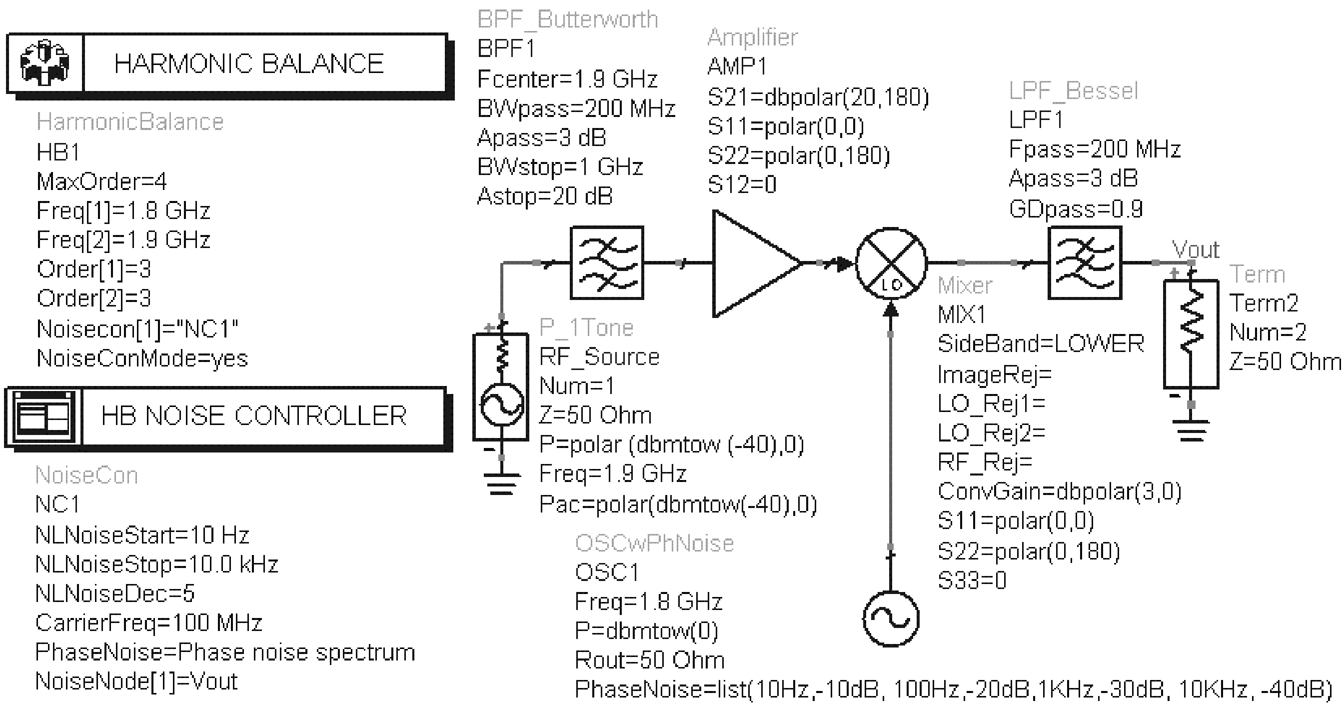
d. Go to the NoiseCons tab and check the NoiseCons box as shown. Then use the Edit button to select NC1 which is the instance name of the Noise Con you set up. Click Add and Apply.





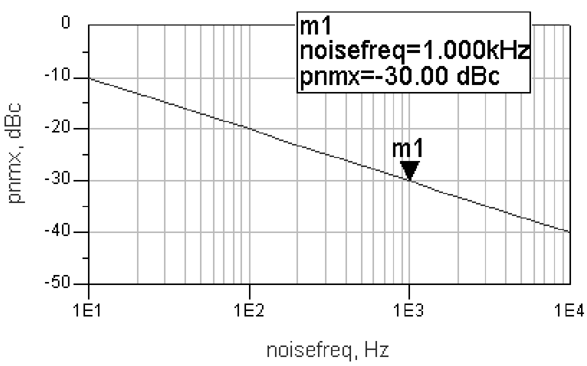
e. ***Display tab – Go to the HB Display tab and check the boxes for the settings shown here. The noise con settings are near the bottom of the list as you scroll down.***

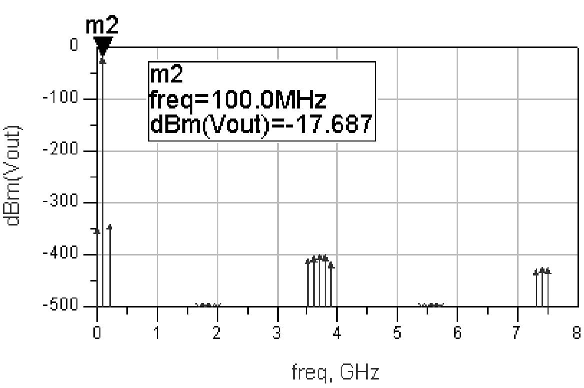
The <u>complete schematic</u> for simulating LO Phase Noise at the IF is shown here. Check your schematic before simulating:



13. ***Simulate and plot the response: pnmx and Vout.***

a. ***Insert a rectangular plot of pnmx. Use Plot Options to set the X-axis to Log scale. Insert a marker to see the frequency offset as shown. Also, insert a rectangular plot of Vout in dBm with a marker on the 100 MHz IF signal. At -40 dBm input, plus about 23 dB of amp and conversion gain, the output should be***





almost –17dBm as shown.

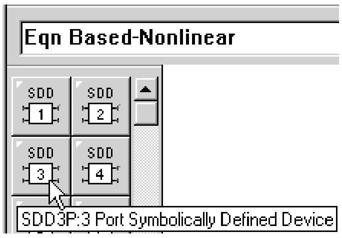
b. Save all your work. You have now completed the first step in the design process for the rf_receiver. In the next labs, you will build the circuits that will replace the system components.

14. OPTIONAL - SDD (Symbolically Defined Device) simulation

SDD's allow you to write an equation to describe the behavior at the nodes of a component, either linear or nonlinear. For this step, you will write a simple linear equation describing sums and differences that appear at the output of a 3 port SDD.

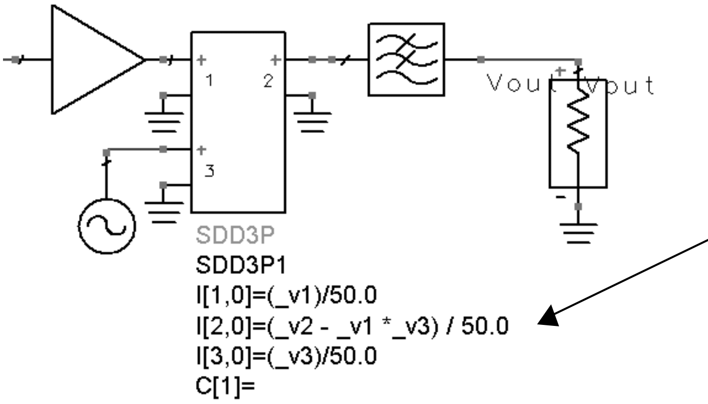
*a. **Use Save Design As to give the current design (rf_sys_phnoise) the name: rf_sys_sdd.***

*b. **Delete the behavioral mixer in the circuit.***



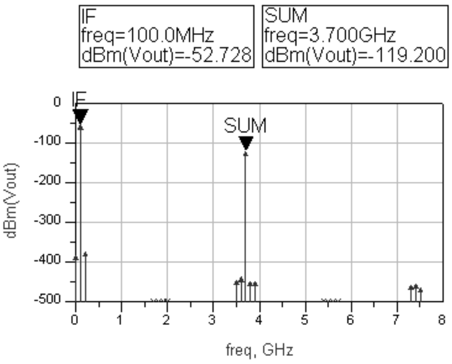
*c. **Go to the palette Eqn Based-Nonlinear and insert the 3 port SDD on the schematic, in place of the mixer. Connect grounds on the negative terminals as shown here.***

*d. **Edit the I[2,0] value by inserting the cursor directly on the text and adding the values shown: - _ v1 * _v3. By subtracting the voltage of the mixing terms of the RF (_v1) and LO (_v3), the IF (_v2) voltage remains. The SDD is now a mixer with no conversion gain, and both the sum and the difference frequencies will appear at the output.***

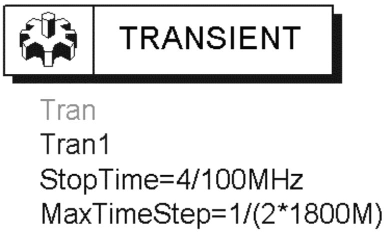


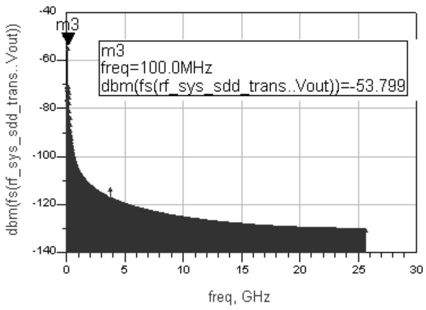
NOTE: SDDs perform numerical operations. This means v1*v3 is a multiplier and does not represent

e. Simulate and plot the spectrum of Vout in dBm. As you can see, without conversion gain, the IF signal is much lower. Also, both the difference and the SUM (RF+LO) appear. Although SDDs can be useful to describe behavior, writing the proper equations can be complicated (requires advanced course).



f. Run a Transient simulation (setup is shown here) and compare the results to HB results using the fs function: dBm(fs(Vout)).



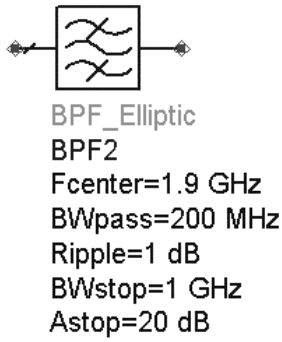


g. Save the design and data.

<u>EXTRA EXERCISES</u>:

1. ***Try running a Transient simulation for the system (not SDD) and comparing results with the fs function.***

2. ***Go back and replace the Butterworth filter with an elliptical filter model shown here and simulate. Try setting different ranges for the Ripple value or try using the tuner to adjust the ripple parameter. Then display the results and look at the ripple in the passband. To do this, you will have to use the zoom commands on the data display.***



3. ***Try tuning various parameters in the design.***

4. ***Enter values of LO and RF rejection to the behavioral mixer and look at the simulation results.***

5. ***Experiment with writing I[2,0] equations for the SDD mixer for conversion gain.***

THIS PAGE IS 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

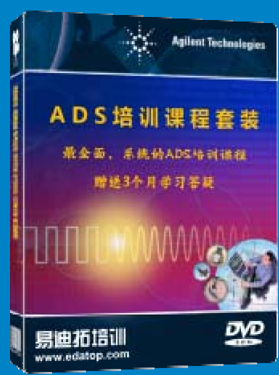
射频工程师养成培训课程套装

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...



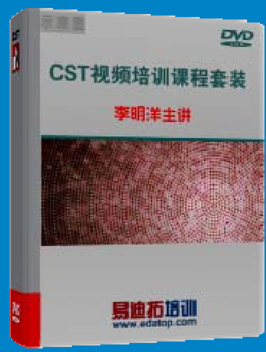
课程网址： http://www.edatop.com/peixun/ads/13.html

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