SpectreRF Workshop

LNA Design Using SpectreRF

MMSIM6.0USR2

November 2005

Contents

Lower Noise Amplifier Design Measurements 3
Purpose 3
Audience 3
Overview 3
Introduction to LNAs 3
The Design Example: A Differential LNA 4
Testbench 5
Example Measurements Using SpectreRF 5
Lab1: Small Signal Gain (SP) 6
Lab2: Large Signal Noise Simulation (PSS and Pnoise) 17
Lab3: Gain Compression and Total harmonic Distortion (Swept PSS) 24
Lab4: IP3 Measurement---PSS Plus PAC analysis 34
Lab5: IP3 Measurement---QPSS Analysis with Shooting or Flexible Balance Engine 40
Lab6: IP3 Measurement---Rapid IP3 using AC analysis 49
Lab7: IP3 Measurement ---Rapid IP3 using PSS Plus PAC Analysis 53
Conclusion 58
Reference 58

Lower Noise Amplifier Design Measurements

The procedures described in this workshop are deliberately broad and generic. Your specific design might require procedures that are slightly different from those described here.

Purpose

This workshop describes how to use SpectreRF in the Analog Design Environment to measure parameters which are important in design verification of Low Noise Amplifier, or LNA. New features of MMSIM6.0USR2 are included.

Audience

Users of SpectreRF in the Analog Design Environment.

Overview

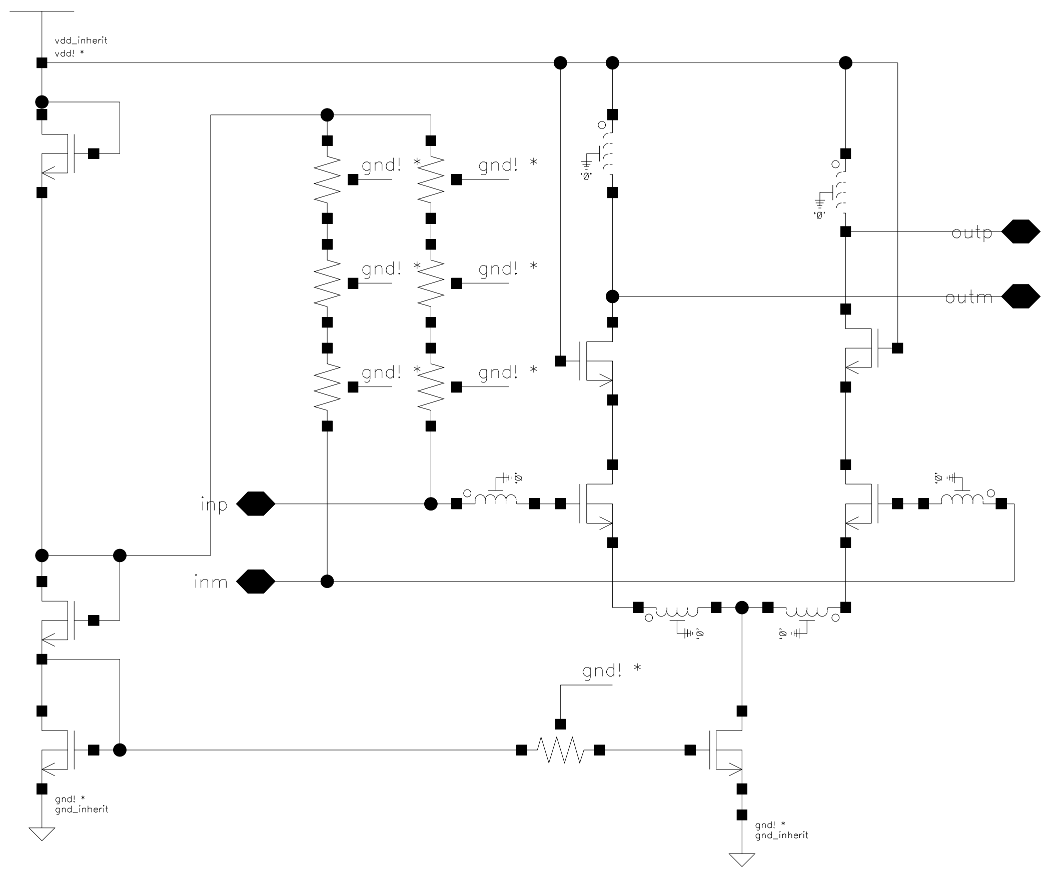
This application note describes a basic set of the most useful measurements for LNAs.

Introduction to LNAs

The first stage of a receiver is typically a low-noise amplifier (LNA), whose main function is to set the noise boundary as well as to provide enough gain to overcome the noise of subsequent stages (for example, in the mixer or IF amplifier). Aside from providing enough gain while adding as little noise as possible, an LNA should accommodate large signals without distortion, offer a large dynamic range, and present good matching to its input and output, which is extremely important if a passive band-select filter and image-reject filter precedes and succeeds the LNA, since the transfer characteristics of many filters are quite sensitive to the quality of the termination.

The Design Example: A Differential LNA

The LNA measurements described in this workshop are calculated using SpectreRF in the Analog Design Environment. The design investigated is the differential Low Noise Amplifier shown in below:

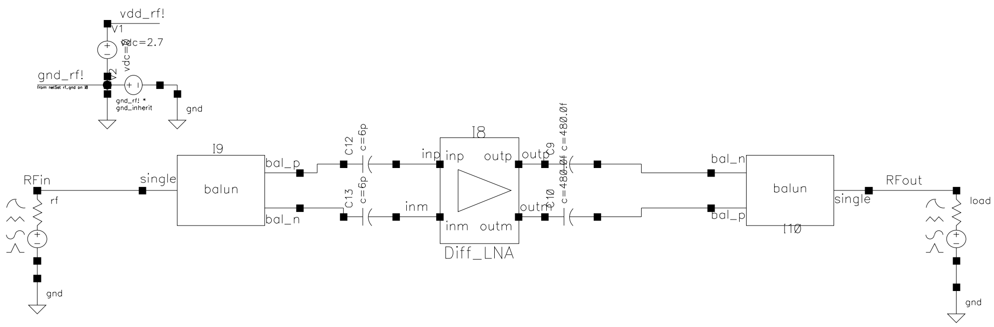


The following table lists typically acceptable values for the performance metrics of LNAs used in heterodyne architectures.

Measurement	Acceptable Value
NF	2 dB
IIP3	-10 dBm
Gain	15 dB
Input and Output Impedance	50 Ω
Input and Output Return Los	-15 dB
Reverse Isolation	20 dB
Stability Factor	>1

Testbench

The following figure shows the testbench for a differential LNA. The baluns used in the testbench are three-port devices. The baluns convert the input single-ended signals to the differential signals. Sometimes, they also perform the resistance transformation.



LNA design is a compromise among power, noise, linearity, gain, stability, input and output matching, and dynamic range. They are characterized by the design specifications in Table in page 4.

Example Measurements Using SpectreRF

The LNA measurements described in the following labs are calculated using SpectreRF in the Analog Design Environment.

We'll begin our examination of the flow by bringing up the Cadence Design Framework II environment and look at a full view of our reference design:

Change directory to…

Action: cd to **./lna** directory

Action: Invoke tool **icfb&**

Action: In the CIW window, select **Tools->Library Manager…**

Lab1: Small Signal Gain (SP)

The S Parameter (SP) analysis is the most useful linear small signal analysis for LNAs. Set up an SP analysis by specifying the input and output ports and the range of sweep frequencies.

Action1-1: In the Library Manager window, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action1-2: Select the **PORTrf** source by placing the mouse cursor over it and clicking the left mouse button. Then in the Virtuoso Schematic Editor select **Edit—Properties—Objects…** The Edit Object Properties window for the port cell should come up. Set up the port properties as follows:

Parameter	**Value**
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	dc

Action1-3: Set the source type of **PORT load** to DC.

Action1-4: check and save the schematic.

Action1-5: In the Virtuoso Schematic Editing window, select **Tools->Analog Environment**

Action1-6: You can choose **Session—Load State** in Virtuoso Analog Design Environment load state "**Lab1_sp**", then skip to Action1-10 or …

Action1-7: In Analog Design Environment window, select **Analyses->Choose…**

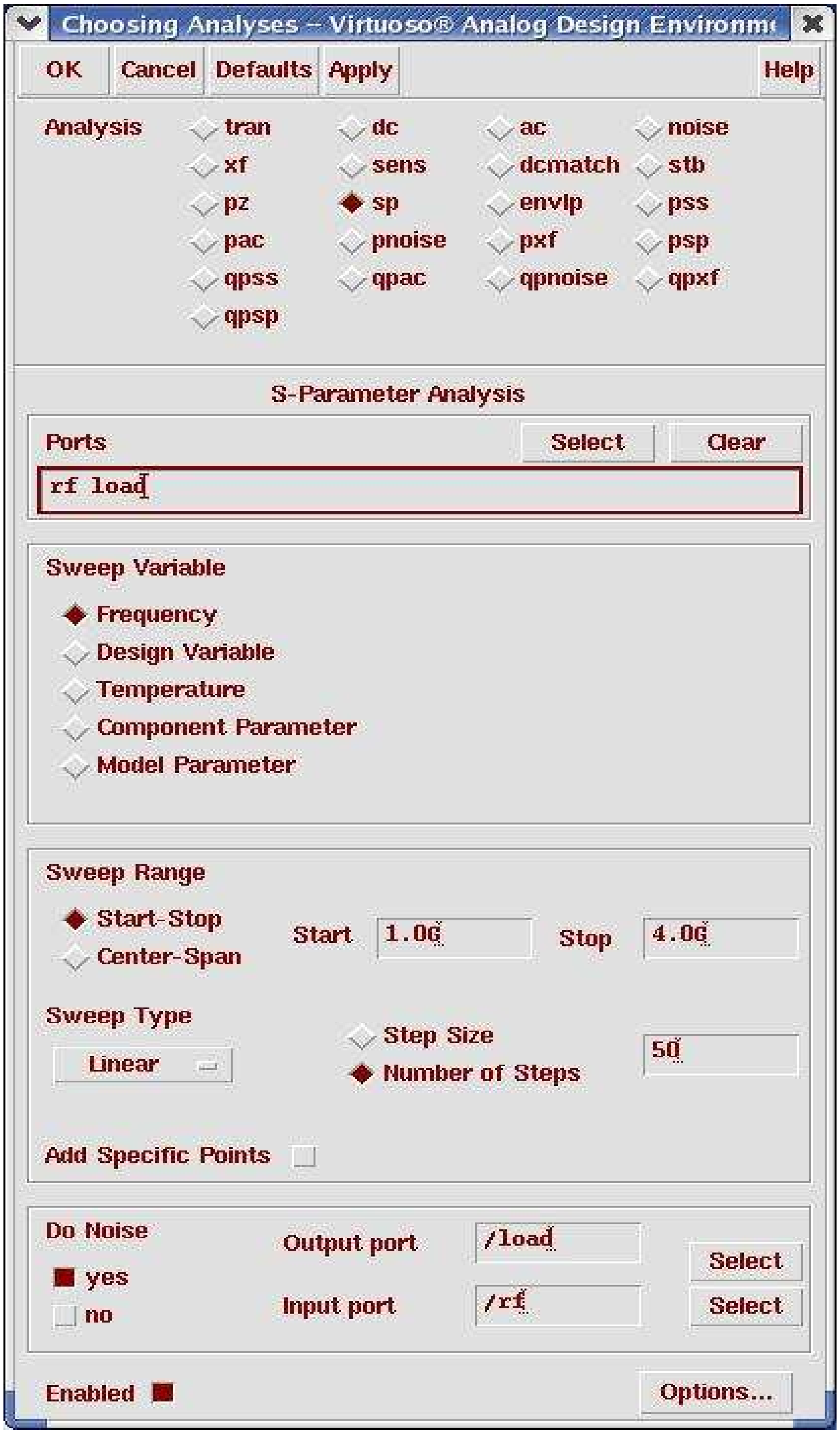
Action1-8: In the Choosing Analyses window, select the **sp** button in the **Analysis** field of the window.

Action1-9: In S-Parameter Analysis window, in the **Ports** field, click the **Select** button. Then, in the Virtuoso Schematic Editing window, in order, select the port cells, **rf** (input) and **load** (output). Then, while the cursor is in the schematic window, hit the **ESC** key.

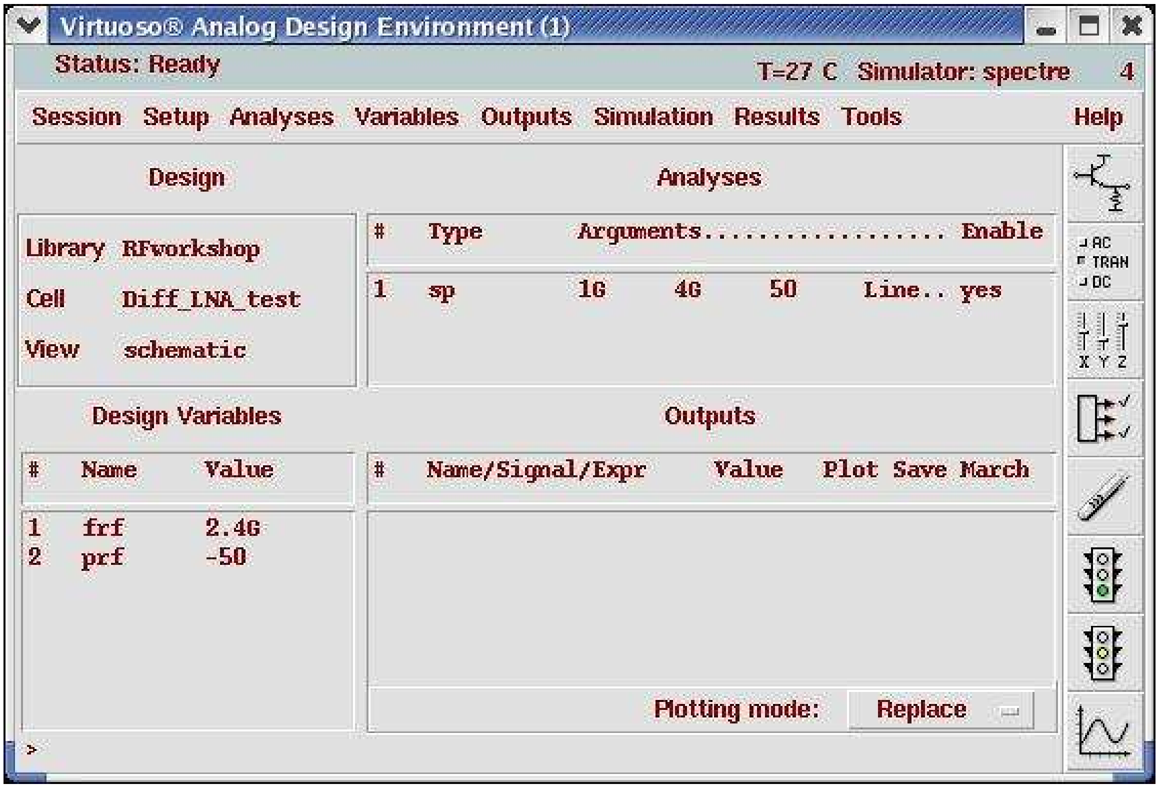
In the **Sweep Variable** field, select **Frequency.**

In the **Sweep Range** field, select **Start-Stop**, set **Start** to 1.0 G and **Stop** to 4.0G, set **Sweep Type** to Linear, select **Number of Steps** and set that to 50. In the **Do Noise** field, select **yes**, set **Output port** to /load and **Input port** to /rf.

The form should look like this:



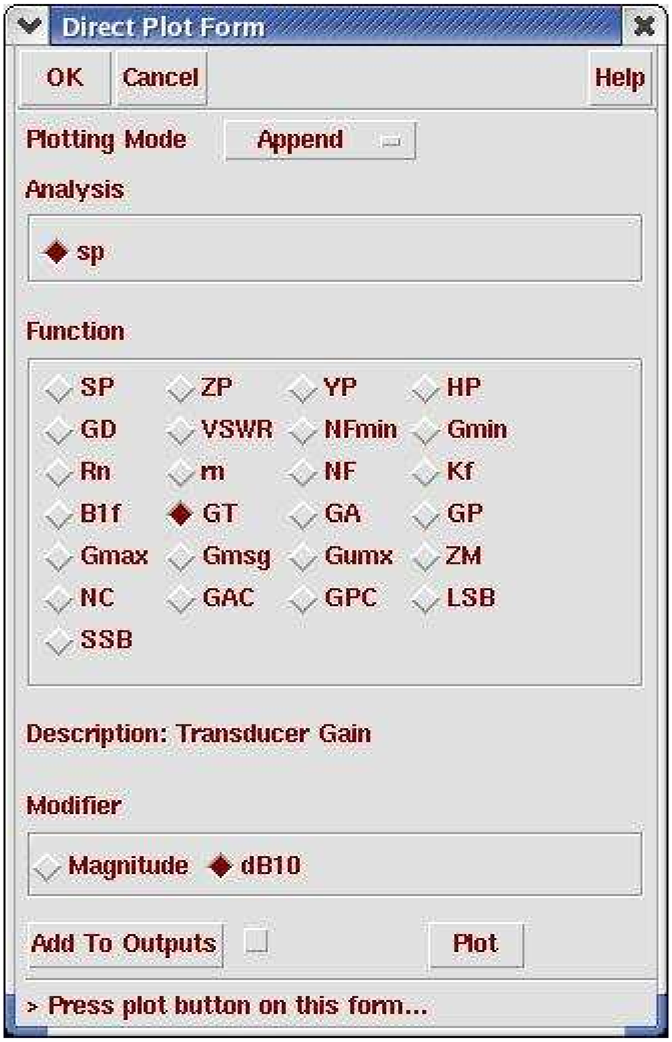
Now your Virtuoso Analog Design Environment looks like this:



Action1-10: Choose **Simulation—Netlist and Run** to start the simulation or click on the **netlist and Run** icon in the Virtuoso Analog Design Environment window.

Action1-11: In Analog Design Environment window, select **Results->Direct Plot->Main Form…**
A Waveform window and a Direct Plot Form window should open.

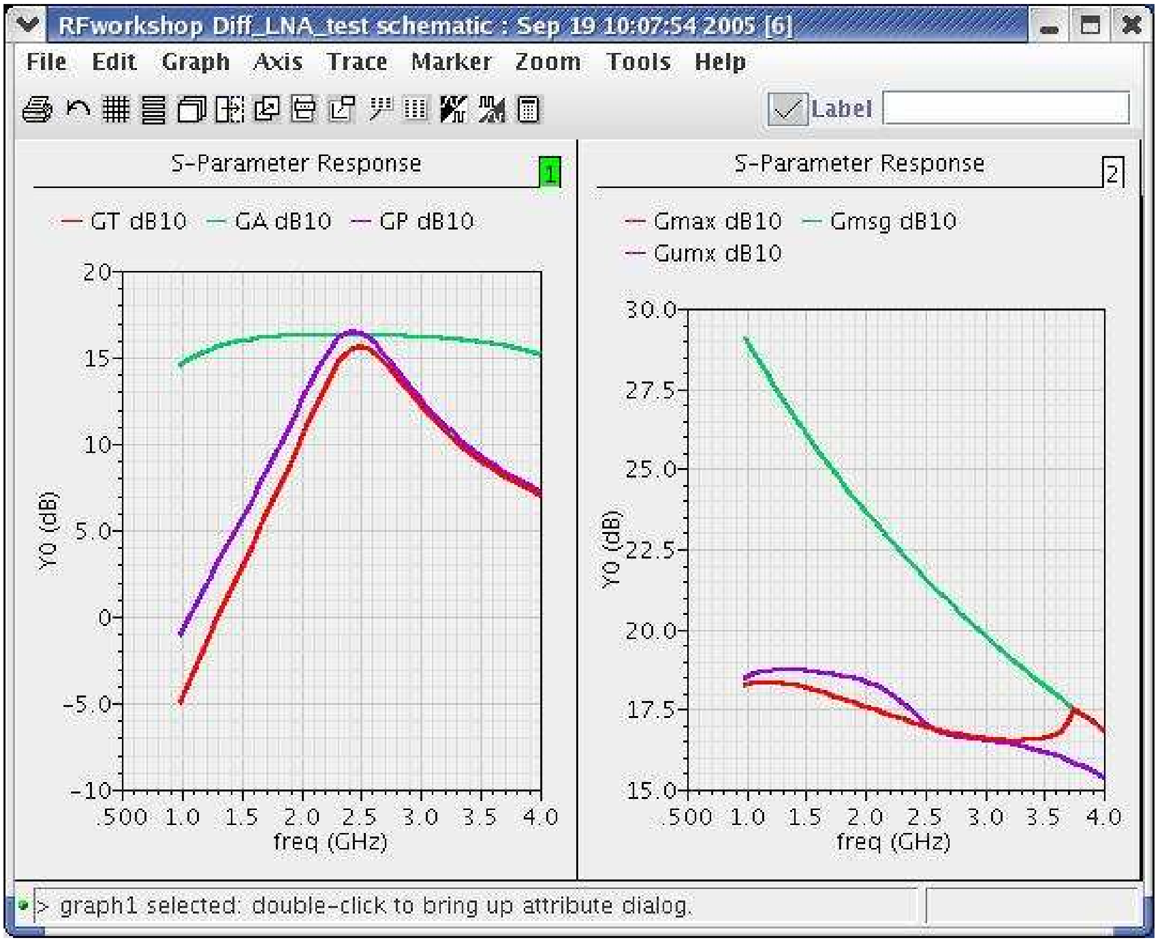
Action1-12: In Direct Plot Form window, set **Plotting Mode** to **Append**. In the **Analysis** field, select **sp**. In the **Function** field, select **GT** (for Transducer Gain). In the **Modifier** field, select **dB10.** Select **Add to Outputs**. The form should look like



Action1-13: Hit the **Plot** button. In the **Function** field, select **GA** (for Available Power Gain). Hit the **Plot** button again. In the **Function** field, select **GP** (for Operating Power Gain). Hit the **Plot** button once more. We plotted GT, GA and GP in one waveform window.

Action1-14: In waveform window, click on New Subwindow. Go back to Direct Plot Form. Select **Gmax** (for maximum Transducer Power Gain), Hit the **Plot** button. In the **Function** field, select **Gmsg** (for Maximum Stability Gain). Hit the **Plot** button. Select **Gumx** (for maximum Unilateral Transducer Power Gain), Hit the **Plot** button again.

Now you should get the following waveforms:



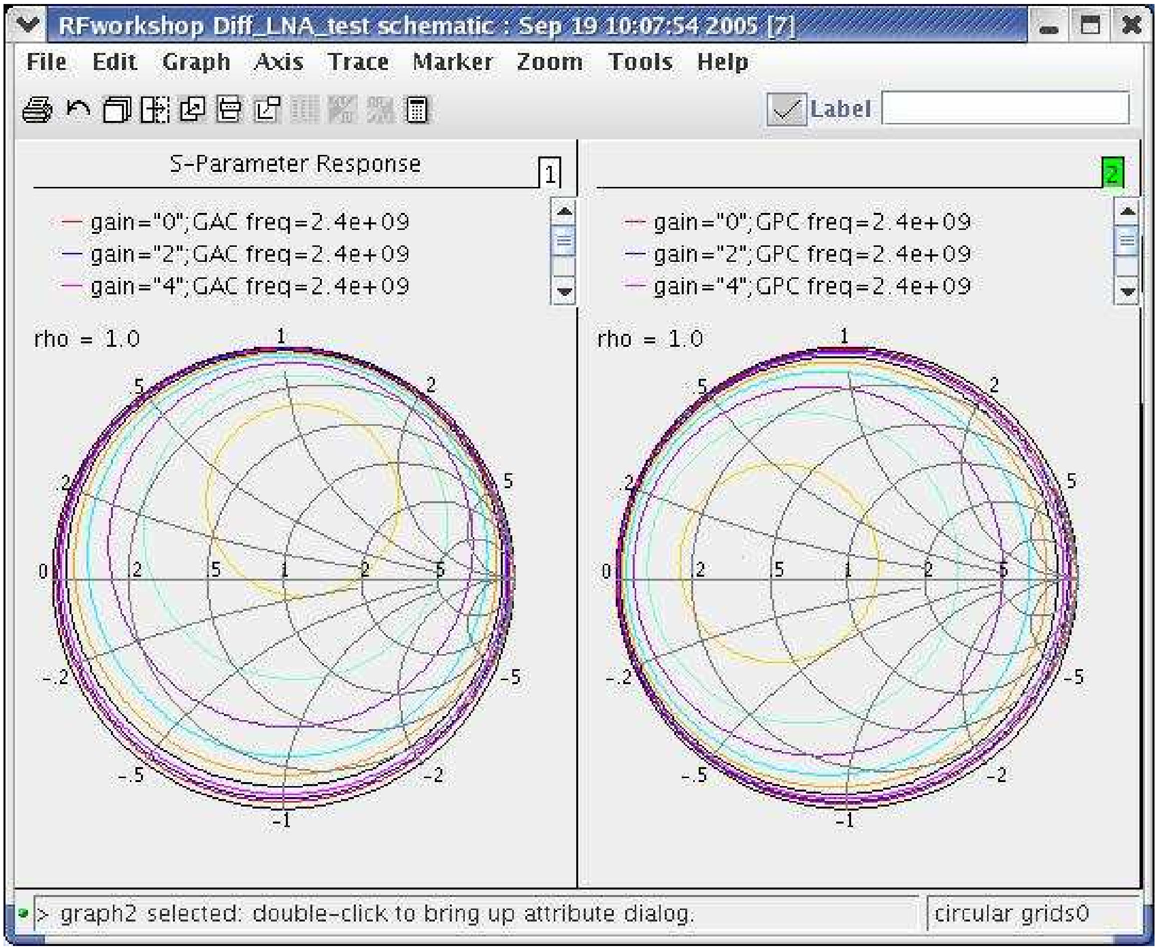
Action1-15: Close waveform window, go back to Direct Plot form. In the **Function** field, select **GAC** (Available Gain Circles). In **Plot Type**, choose **Z-Smith**, Sweep **Gain Level (dB)** at Frequency 2.4GHz from 0 to 18dB with step as 2 dB.

Action1-16: Press **plot** button.

Action1-17: In waveform window, click on **New Subwindow**.

Action1-18: Go back to Direct Plot Form, in Function field, select **GPC** (Power Gain Circles). And press Plot button.

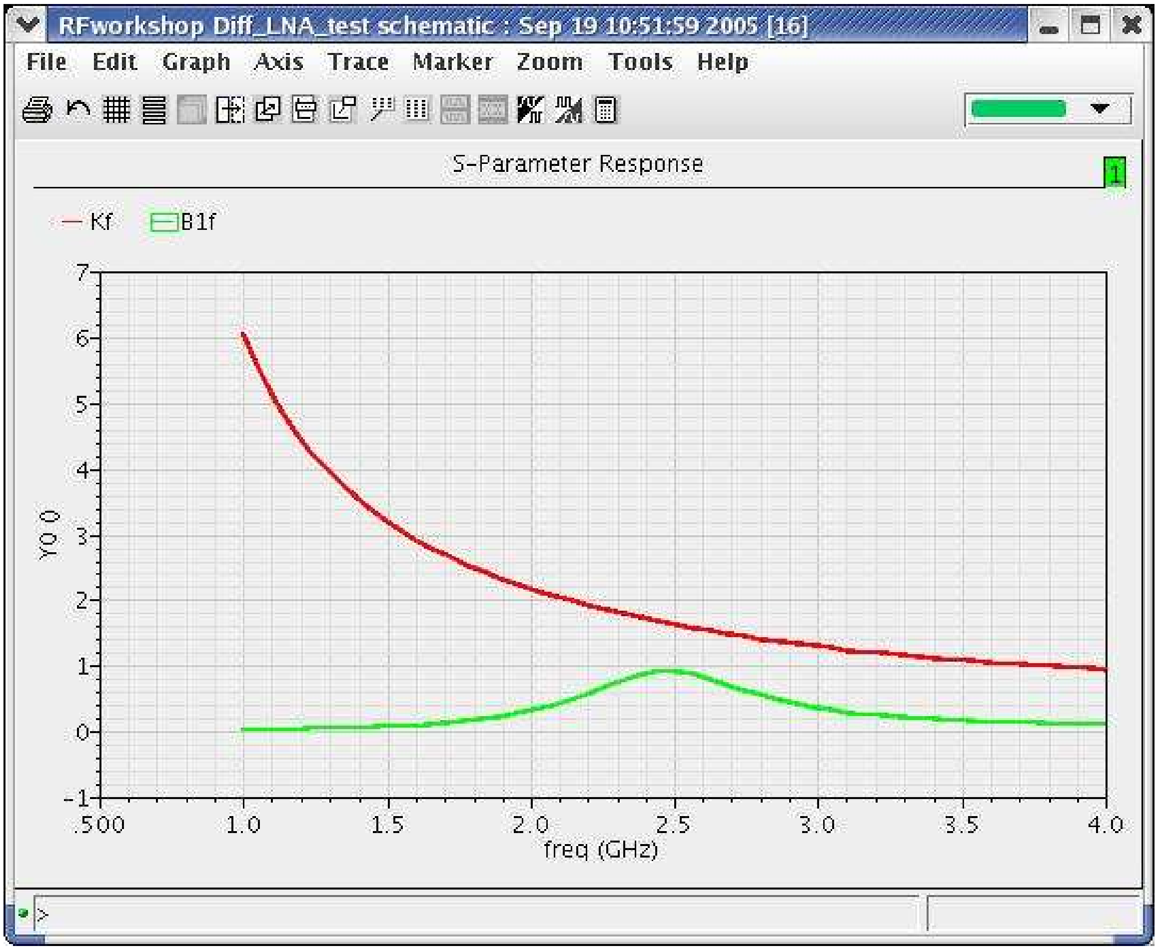
The waveforms should look like this:



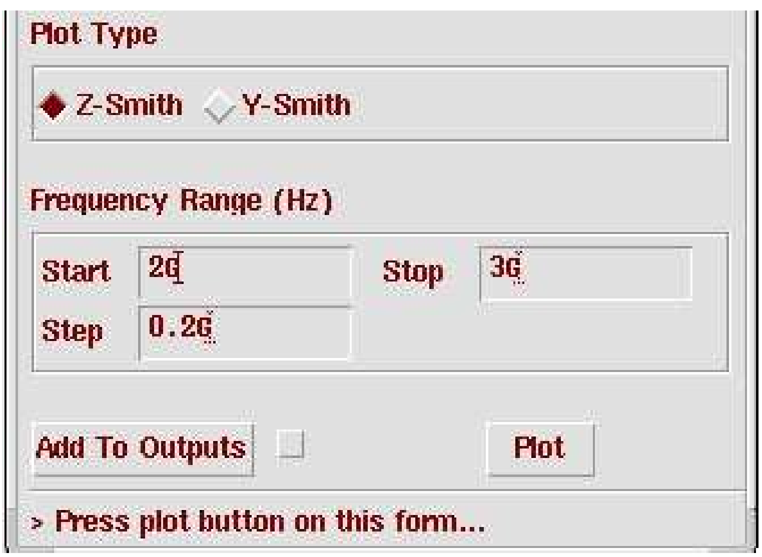
Action1-19: Close the waveform window, go back to **Direct Plot Window**. In **Function** field, choose **Kf**. Hit **Plot** button.

Action1-20: In **Function** field, choose **B1f**. Hit **Plot** button.

You will get the Stability Curves:



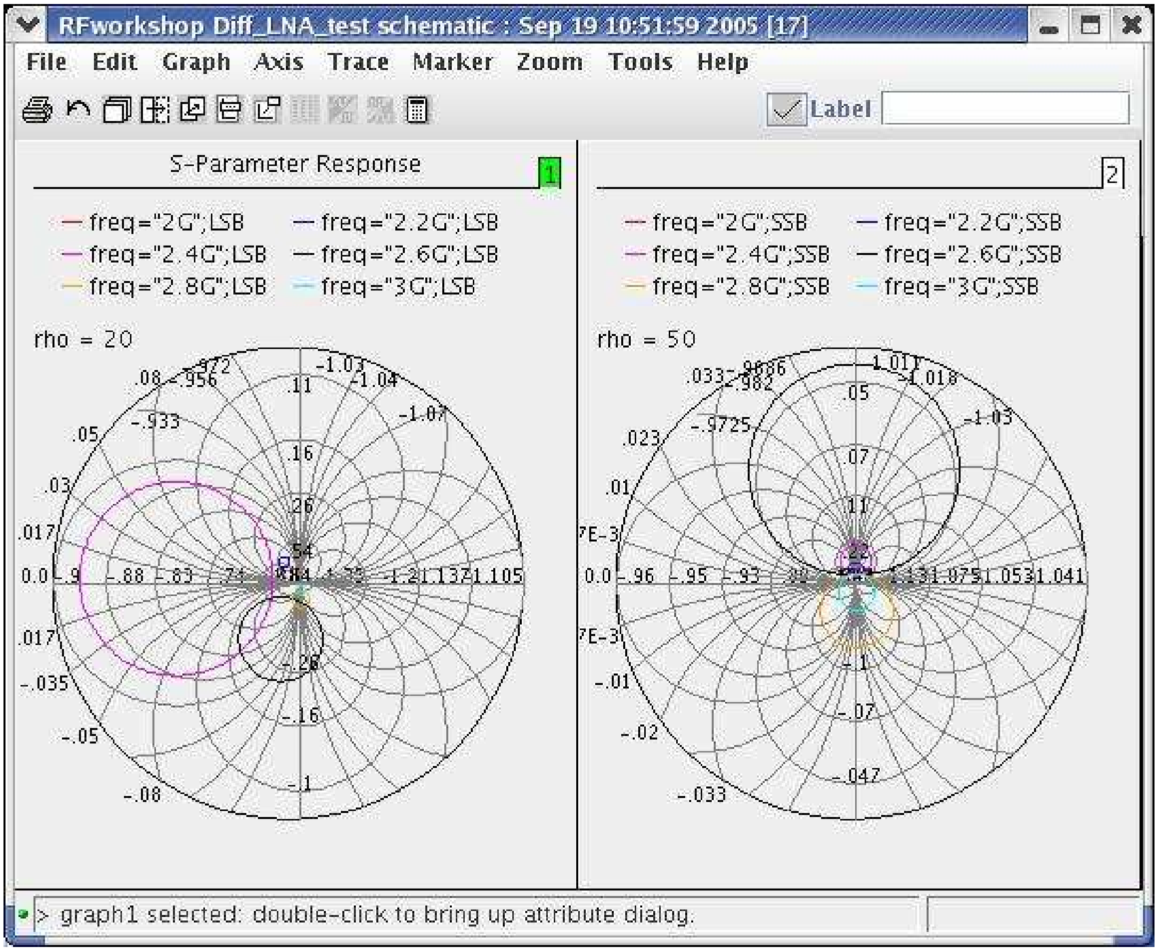
Action1-21: Close the waveform window; go back to **Direct Plot Window**. In **Function** field, choose **LSB** (Load Stability Circles)**.** In Plot Type, choose Z-Smith. Specify Frequency Rage from 2G to 3G with step as 0.2G. Hit **Plot** button.



Action1-22: In waveform window, click on **New Subwindow**.

Action1-23: Go back to **Direct Plot Form**, in **Function** field, select **SSB** (Source Stability Circles). And press **Plot** button.

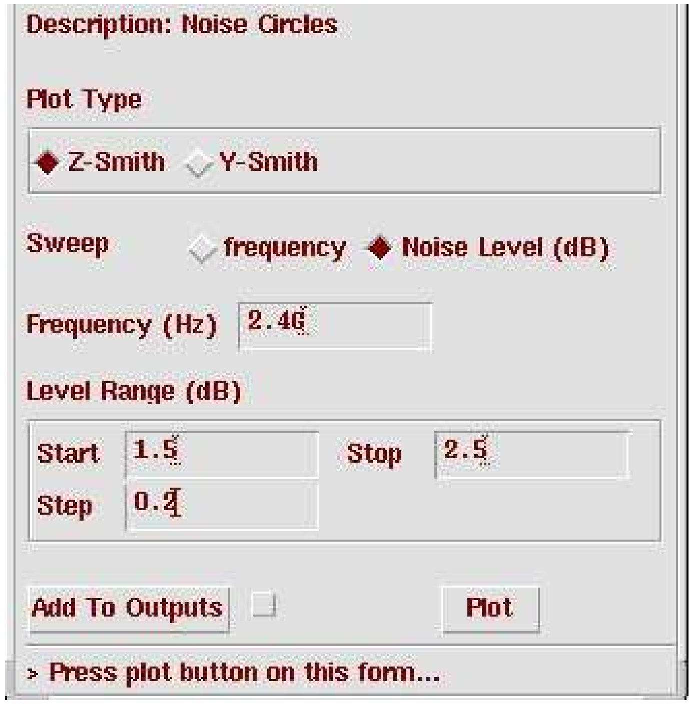
You will get the Load Stability Circles and Source Stability Circle:



Action1-27: In the **Direct Plot Form** window, select **NF (Noise Figure**) in the **Function** field. In the **Modifier** filed, select **dB10**. Hit the **Plot** button.

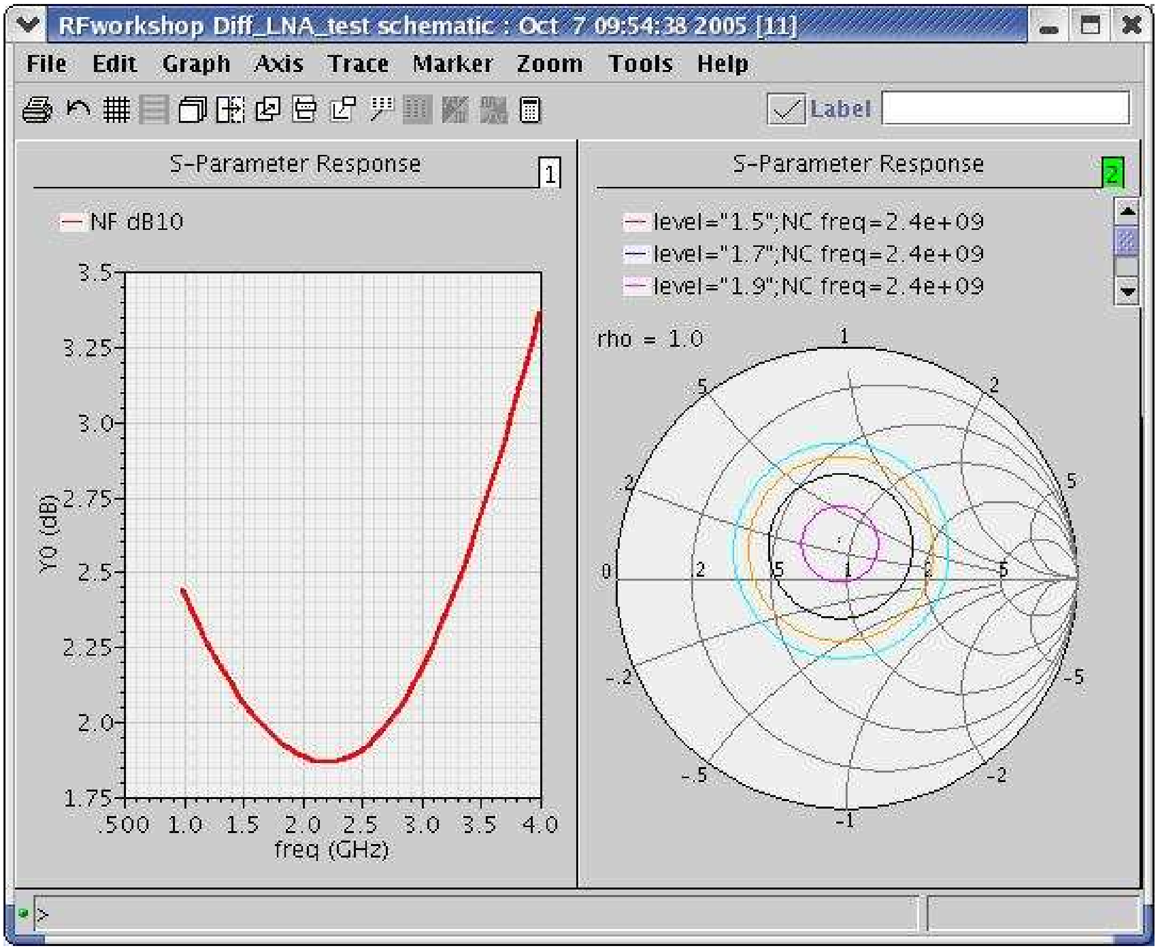
Action1-26: In the waveform window, click on **New Subwindow**.

Action1-24: Close the waveform window, go back to Direct Plot Form window, in the **function** field, and choose **NC** (Noise Circles). In the **Plot type** field, choose **Z-Smith**. Sweep Noise Level at Frequency 2.4G Hz staring from 0 to 10 dB with step of 1 1dB.



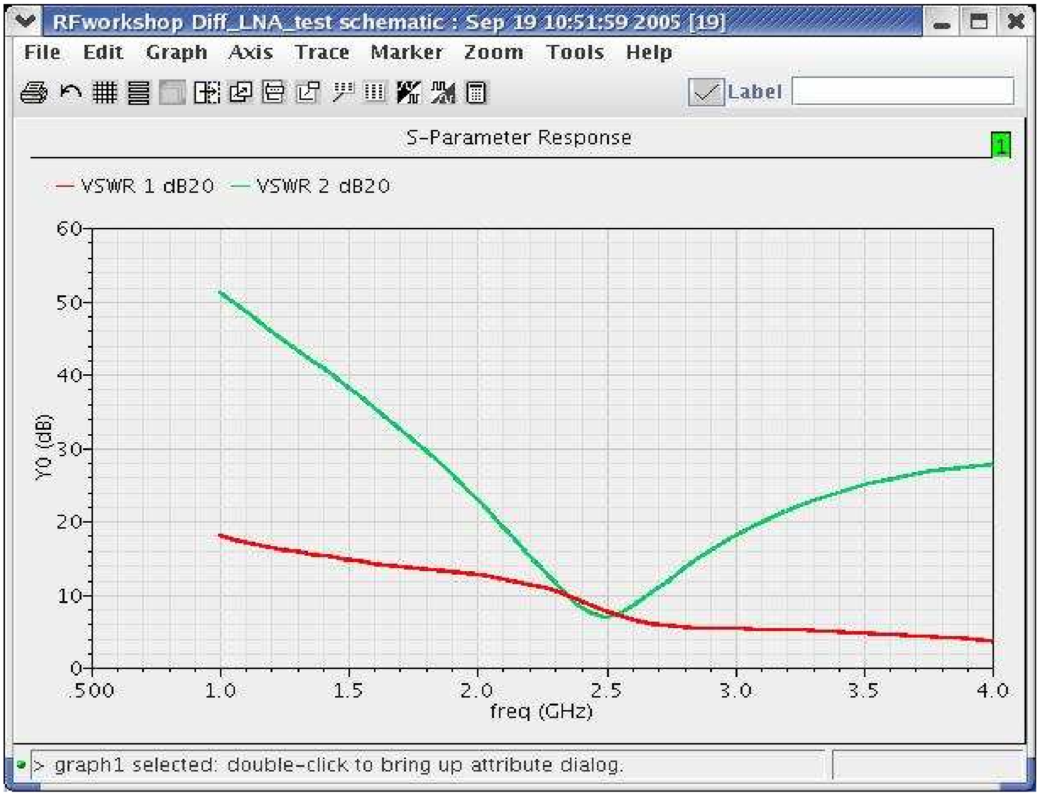
Action1-25: Hit the **Plot** button.

You should get the following plot:



Action1-28: Close the waveform window, go back to the Direct Plot Form window, in the **function** field, and choose **VSWR** (Voltage standing-wave ratio). In the **Modifier** field, select **dB20**. Press on **VSWR1**, then **VSWR2**.

You should get the following waveforms:



Action1-29: Close the waveform window and click **Cancel** on the Direct Plot form

Lab2: Large Signal Noise Simulation (PSS and Pnoise)

Use the PSS and Pnoise analyses for large-signal and nonlinear noise analyses, where the circuits are linearized around the periodic steady-state operating point. (Use the Noise and SP analyses for small-signal and linear noise analyses, where the circuits are linearized around the DC operating point.) As the input power level increases, the circuit becomes nonlinear, the harmonics are generated and the noise spectrum is folded. Therefore, you should use the PSS and Pnoise analyses. When the input power level remains low, the NF calculated from the Pnoise, PSP, Noise, and SP analyses should all match. For most cases, LNAs work with very low input power level, so SP or noise analysis is enough.

Action2-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action2-2: Select the **PORTrf** source. Use the **Edit—Properties—Objects** command to ensure that the port properties are set as described below:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	sine
Frequency name 1	RF
Frequency 1	frf
Amplitude 1 (dBm)	prf
Frequency name 2	(blank)
Frequency 2	(blank)
Amplitude 2 (dBm)	(blank)

Action2-3: Check and save the schematic.

Action2-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.

Action2-5: You can choose **Session—Load State**, load state "**Lab2_Pnoise**" and skip to Action2-15 or …

Action2-6: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

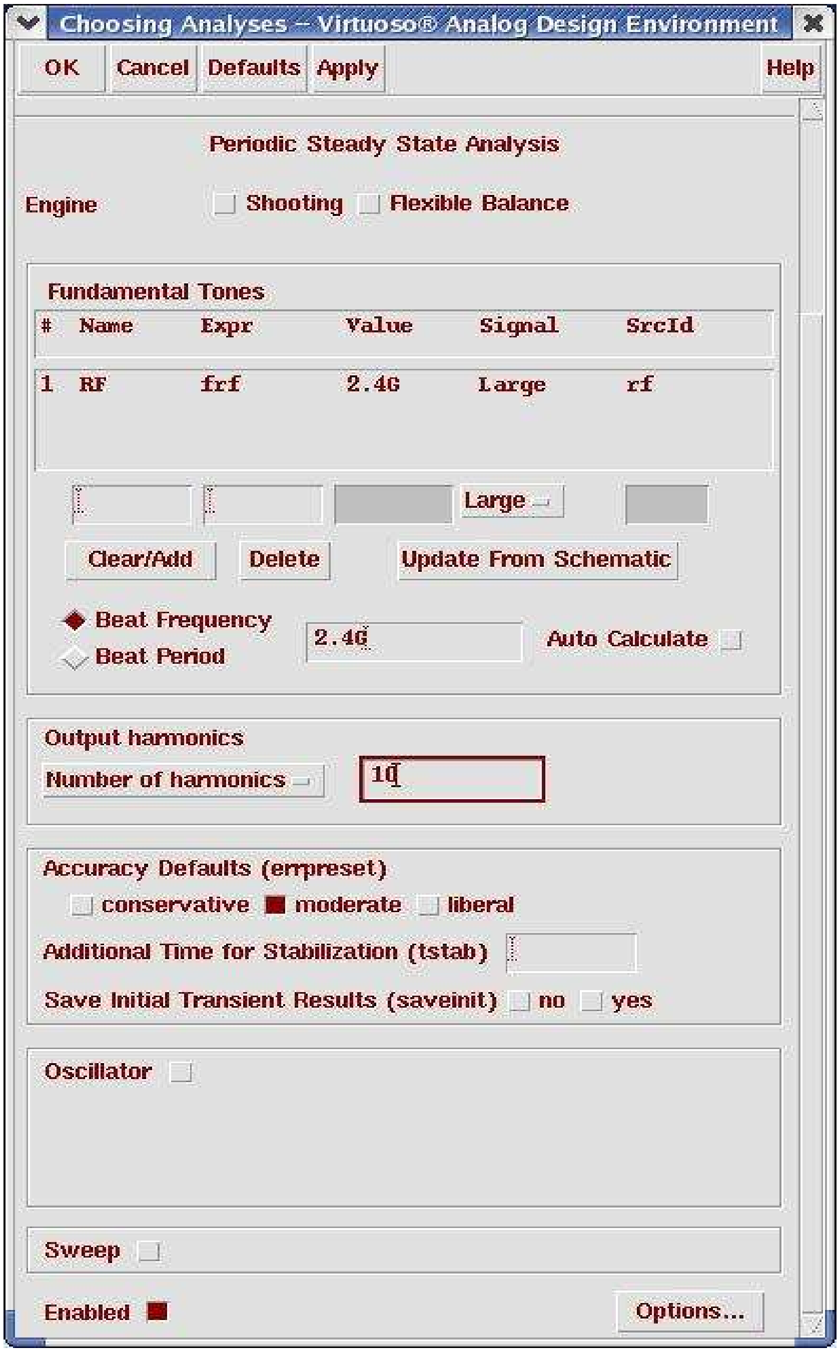
Action2-7: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window.

Action2-8: In pss analyses window, select **Auto Calculate** button. This automatically calculates either the Beat Frequency or Beat Period of the circuit. If the circuit contains frequency dividers or the input sources do not come from **analogLib**, it might be necessary to manually calculate the Beat Frequency (or Beat Period).

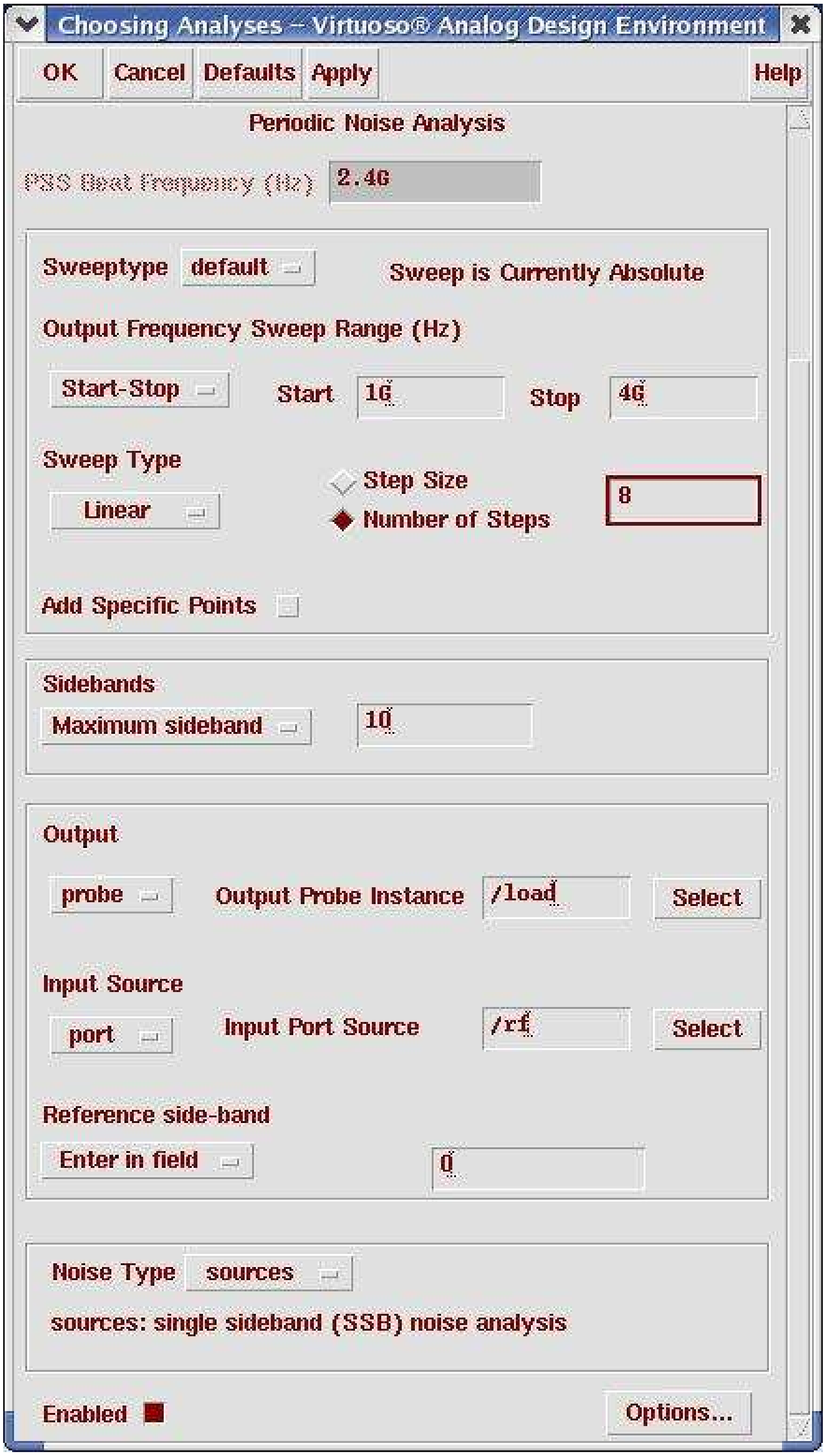
Action2-9: In the **Output Harmonics** field, set the cyclic to **Number of Harmonics** and set the number of harmonics to 3. This will allow us to look at, in the frequency domain results, 3 harmonics of the Beat Frequency.

Action2-10: In the **Accuracy Defaults (errpreset)** field, select the **moderate** button.

Your Choosing Analyses->PSS window should look like…

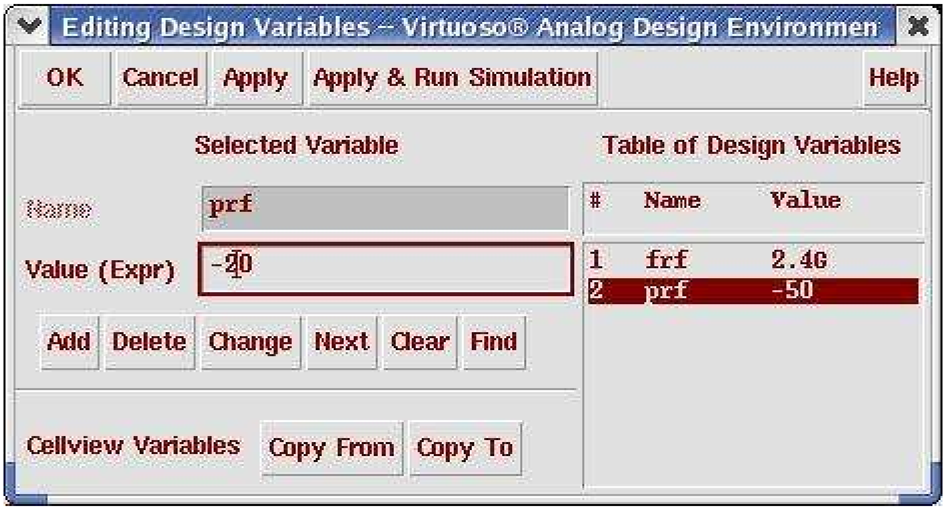


Action2-11: Now that you have set up the PSS analysis. Click **pnoise** in the Choosing Analyses form. You must specify the noise source and the number of sidebands for inclusion in the summation of the final results. The larger the number, the more accurate the results will be, until the point where the higher order harmonics are negligible. Spectre will give you warning message regarding accuracy for any maxsideband number lower than 7. You specify the reference sideband as 0 for an LNA because an LNA has no frequency conversion form input to output. The form should look like this:



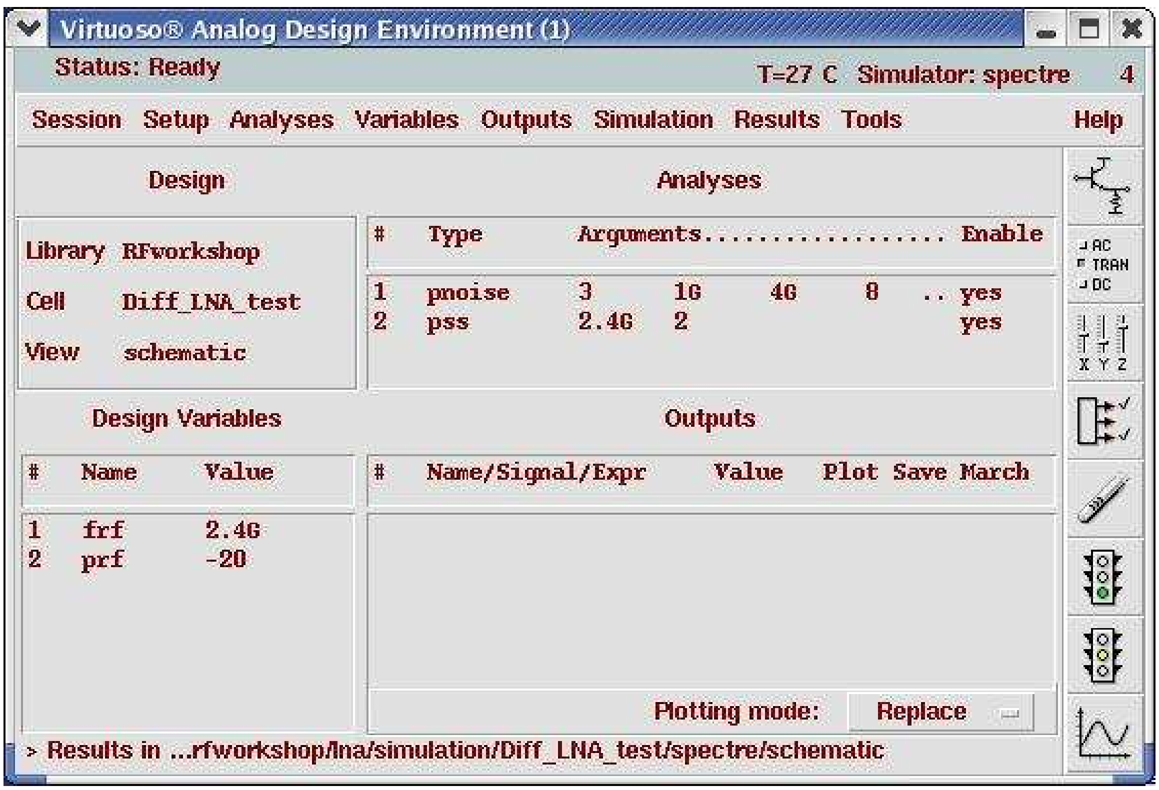
Action2-12: Make sure the **Enabled** button is active, and click **OK** in the Choosing Analyses form.

Action2-13: In the Virtuoso Analog Design Environment window, double click on prf in the field of Design Variables. Change the input power to -20.



Action2-14: Click **Change**. Click on OK to close the Editing Design Variables window.

Your Virtuoso Analog Environment will look like this:

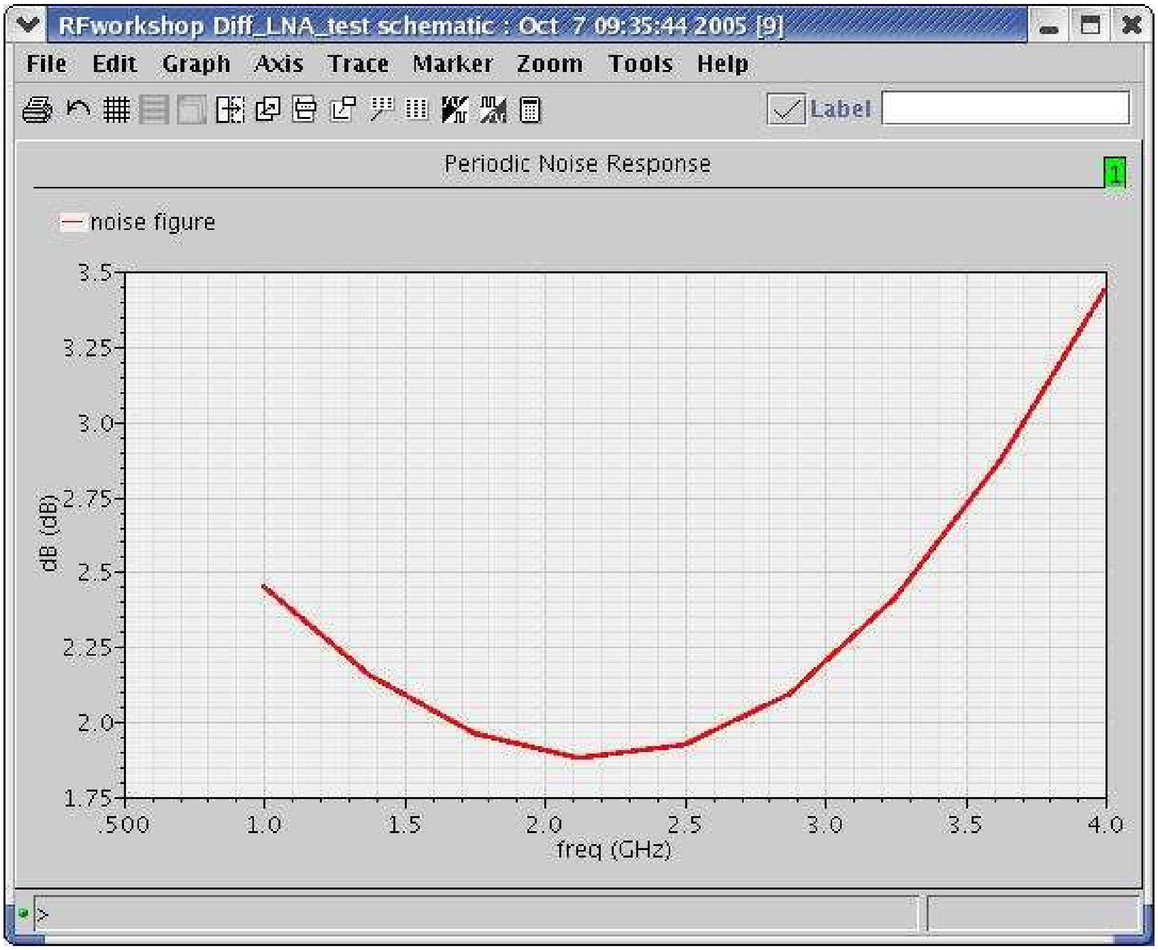


Action2-15: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action2-16: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action2-17: In the Direct Plot Form, select the **pnoise** button, and configure the form as follows:

Action2-18: Click the **Plot** button.



The Waveform window displays the Noise Figure. The NF from Pnoise is slightly larger than the NF from SP because at Pin = -20 dBm, the LNA demonstrates very weak nonlinearity and noise as other high harmonics are convoluted.

Action2-19: Close the waveform window and click **Cancel** on the Direct Plot form. Close the Virtuoso Analog Design Environment window.

Lab3: Gain Compression and Total harmonic Distortion (Swept PSS)

A PSS analysis calculates the operating power gain. That is, the ratio of power delivered to the load divided by the power available from the source. This gain definition is the same as that for GP. Therefore, the gain from PSS should match GP when the input power level is low and nonlinearity is weak. After the PSS analysis with swept input power level, plot the output power against the input power level. Determine the 1 dB compression point from the curve.

Action3-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action3-2: Select the **PORTrf** source. Use the **Edit—Properties—Objects** command to ensure that the port properties are set as described below:

Parameter	**Value**
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	sine
Frequency name 1	RF
Frequency 1	frf
Amplitude 1 (dBm)	prf

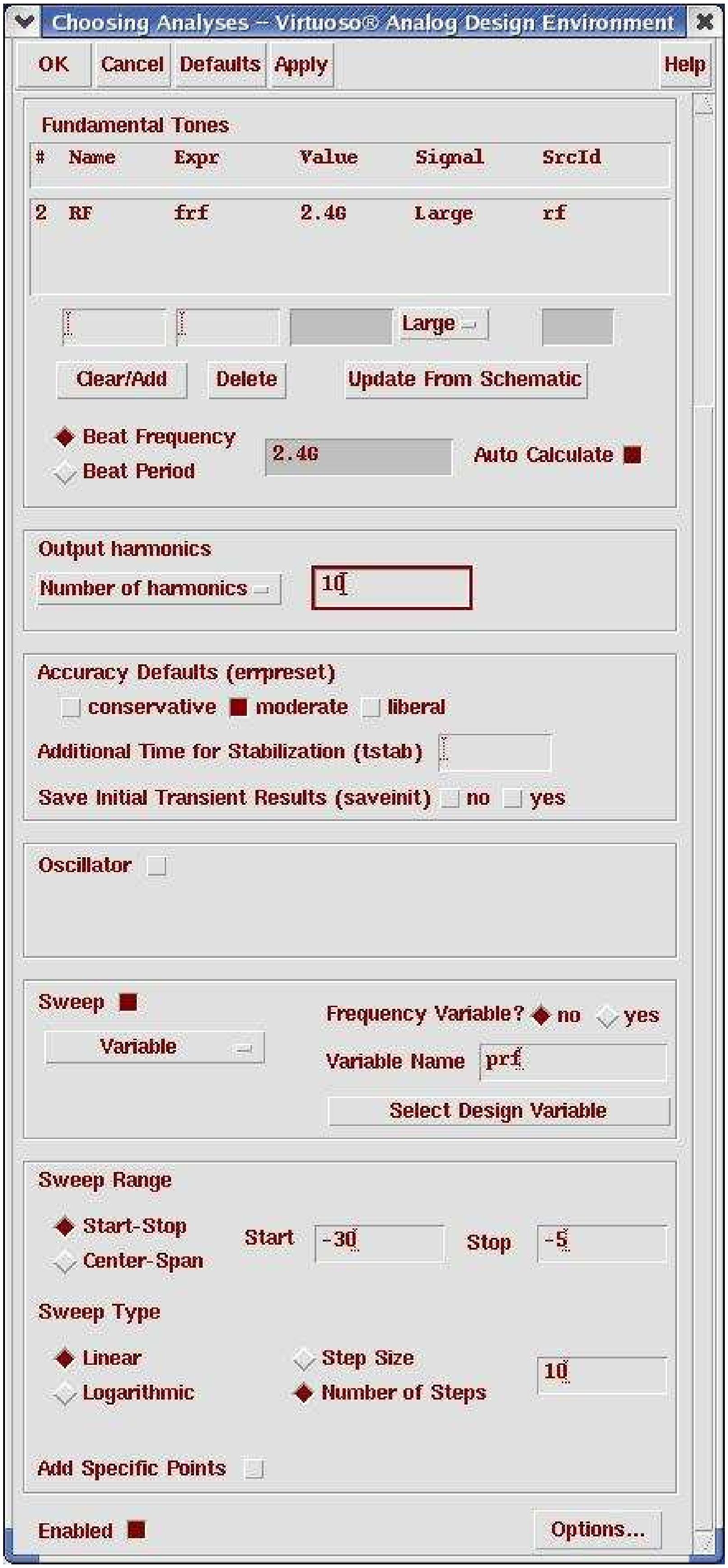
Action3-3: Check and save the schematic.

Action3-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.

Action3-5: You can choose **Session—Load State**, load state "**Lab3_P1dB**" and skip to Action3-9 or …

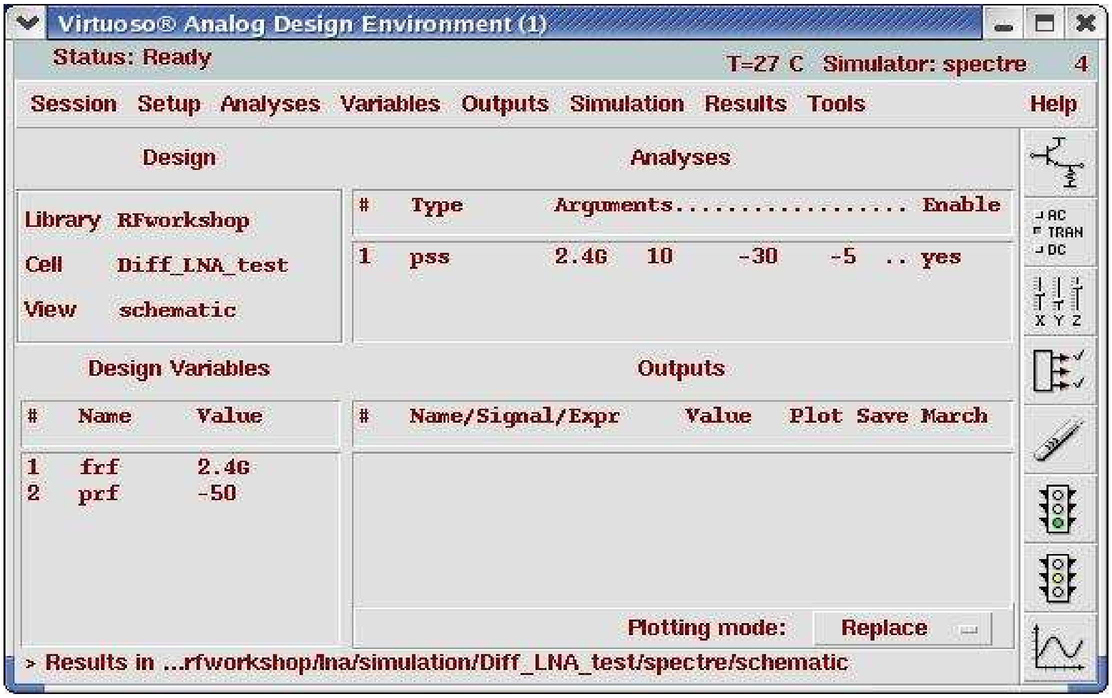
Action3-6: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action3-7: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window. Set up the form as follows:


Choosing Analyses – Virtuoso® Analog Design Environment
OK
Cancel
Defaults
Apply
Help
Fundamental Tones
Name Expr Value Signal SrcId
2 RF frf 2.4G Large rf
Large
Clear/Add
Delete
Update From Schematic
Beat Frequency
Beat Period
2.4G
Auto Calculate
Output harmonics
Number of harmonics
10
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
Save Initial Transient Results (saveinit)
no
yes
Oscillator
Sweep
Frequency Variable?
no
yes
Variable
Variable Name
prf
Select Design Variable
Sweep Range
Start-Stop
Center-Span
Start
-30
Stop
-5
Sweep Type
Linear
Logarithmic
Step Size
Number of Steps
10
Add Specific Points
Enabled
Options...

Action3-8: Make sure the **Enabled** button is on. Click on **OK** on the choosing analyses form.

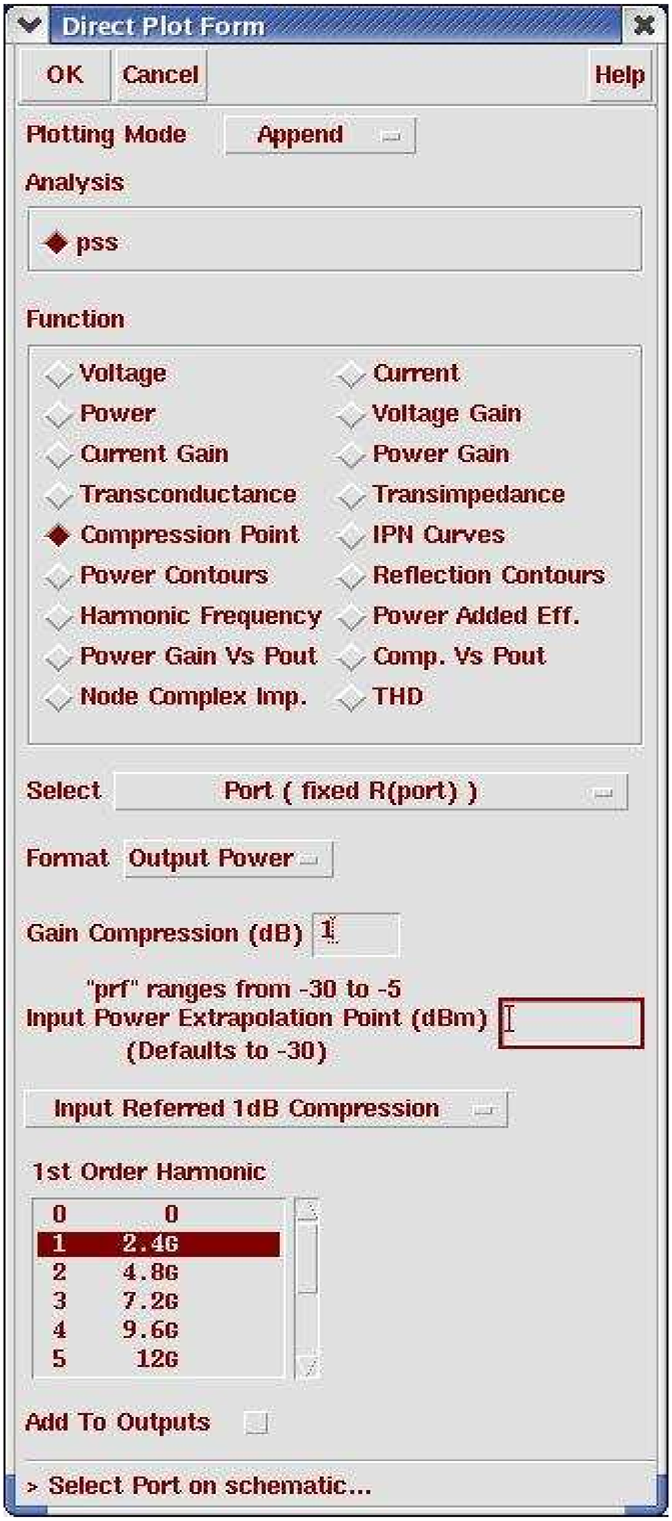
Now your Virtuoso Analog Design Environment will look like:



Action3-9: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

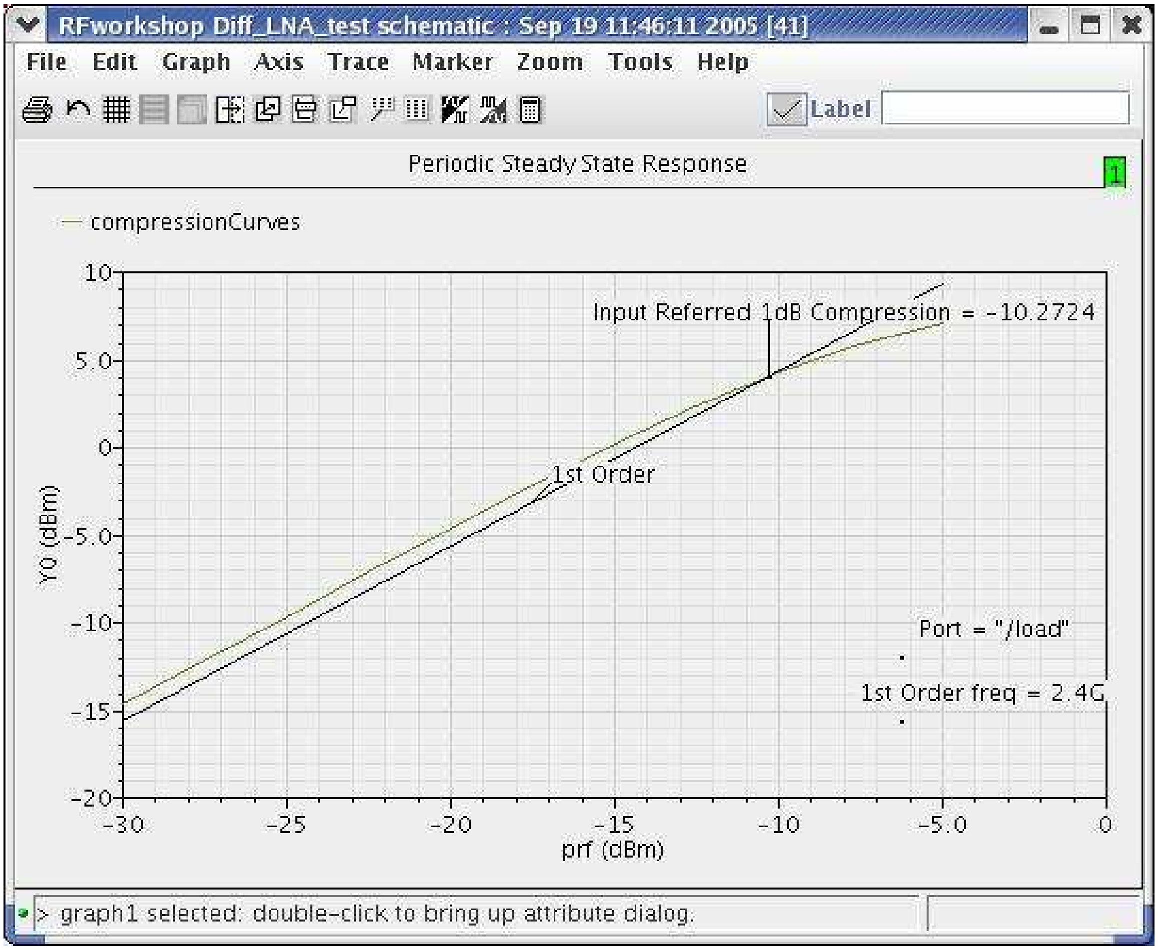
Action3-10: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action3-11: In the Direct Plot Form, select the **pss** button, and configure the form as follows:



Action3-12: Select output port **load** on schematic.

The P1dB plot appears in the Waveform window.

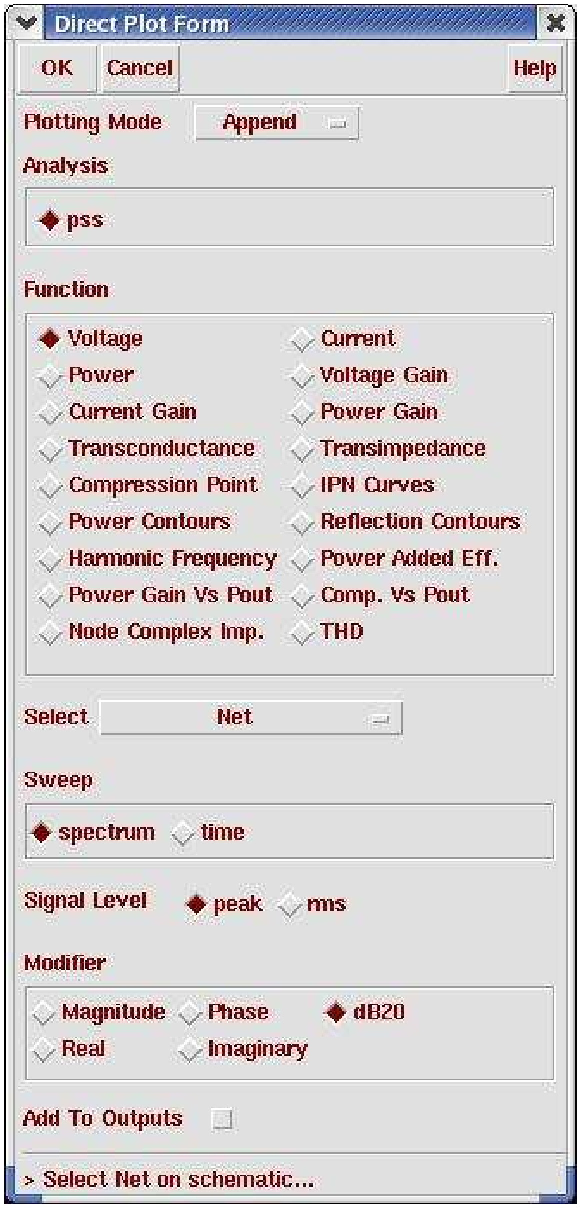


The gain at -30 dBm input power level is -14.7 - (-30) = 15.3 dBm which is a good match for the small signal gain.

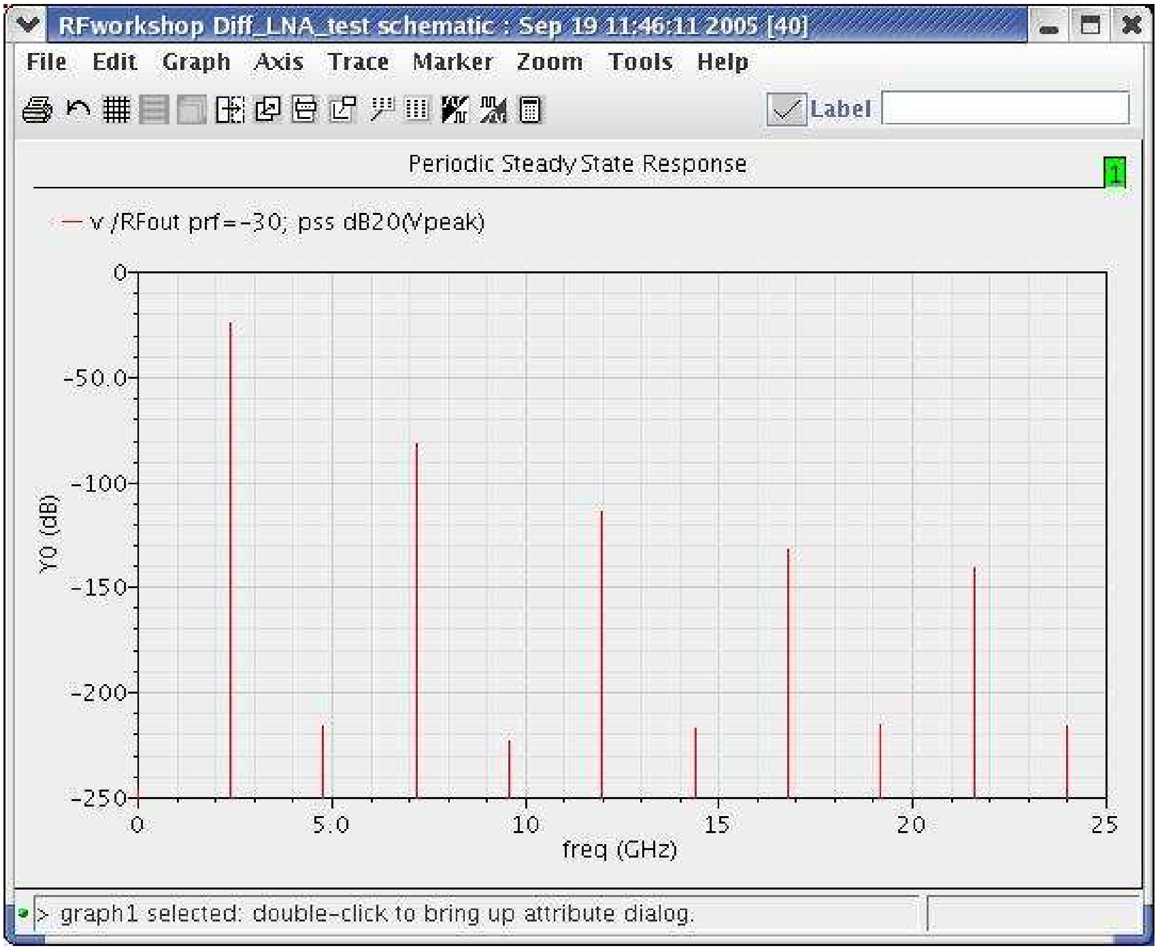
After the PSS analysis, you can observe the harmonic distortion of the LNA by plotting the spectrum of any node voltage. Harmonic distortion is characterized as the ratio of the power of the fundamental signal divided by the sum of the power at the harmonics.

Action3-13: Close the waveform window.

Action3-14: In the Direct Plot Form, select the **pss** button, and configure the form as follows:



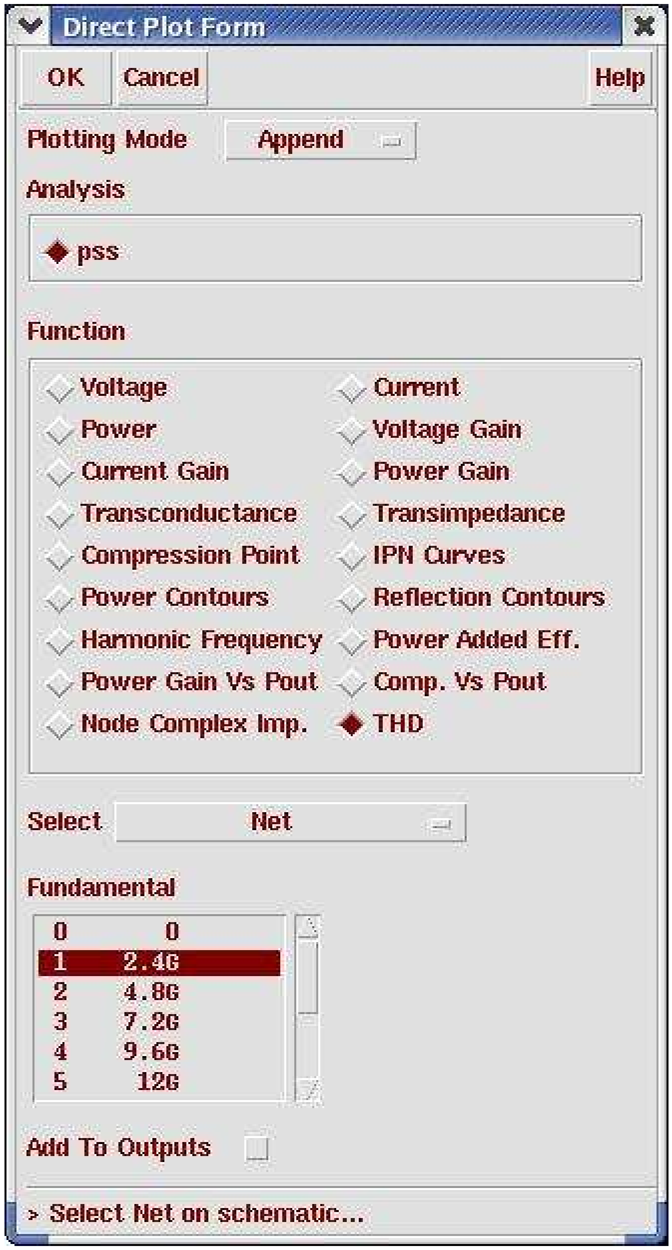
Action3-15: Select output **net RFout** on schematic.



it is obvious that the DC and all the even modes at the output are suppressed because the LNA we investigated is a differential LNA.

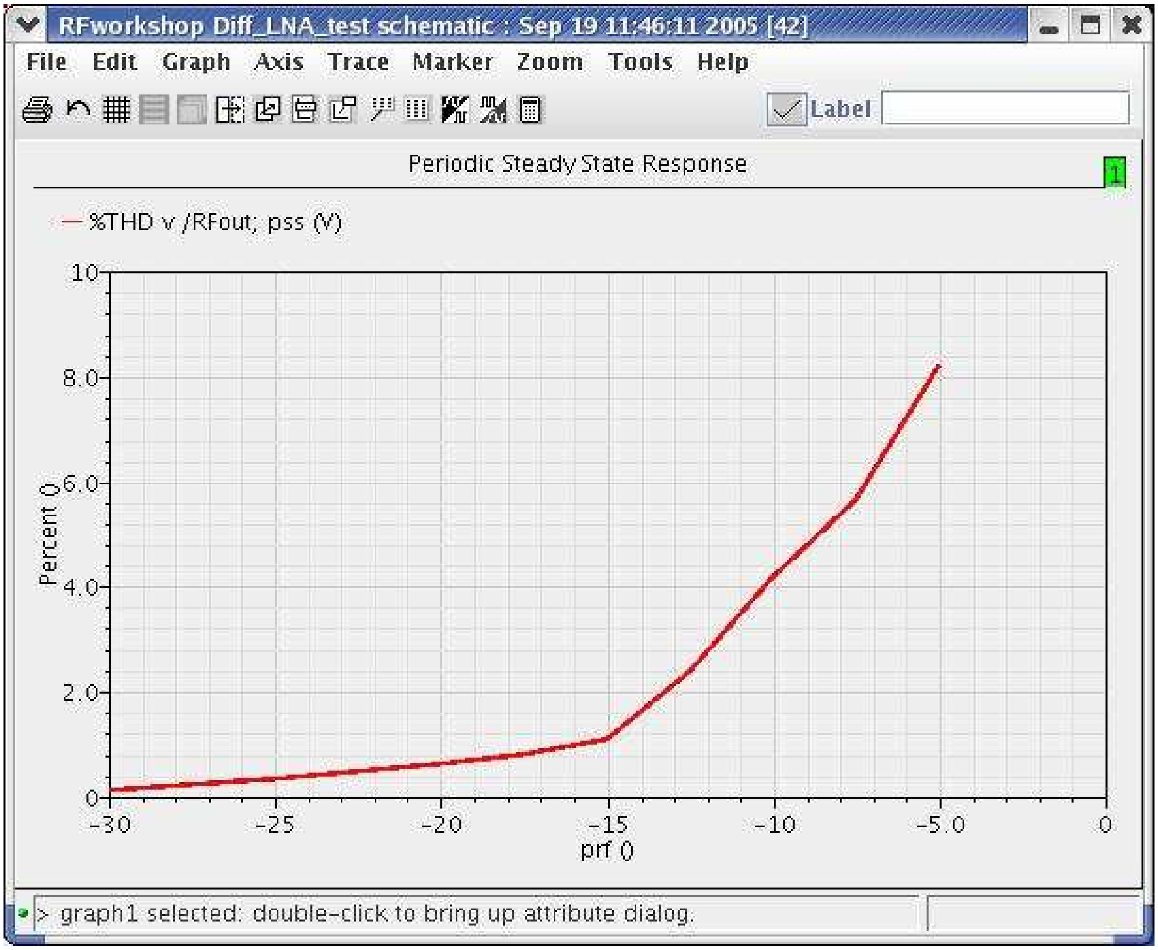
Action3-16: After viewing the waveforms, close waveform window.

Action3-17: In the Direct Plot Form, select the **pss** button, and choose **THD** function.



Action3-18: Select output **net RFout** on schematic.

The THD plot appears in the Waveform window.



Action3-19: Close the waveform window and click **Cancel** on the Direct Plot form**.**

IP3 measurements

SpectreRF offers Several ways to simulate IP3.

The first method treats one tone as a large signal, for example ω_1, and performs a PSS analysis on this signal. The other tone, for example ω_2, is treated as a small signal and a PAC analysis is performed based on the linear time-varying systems obtained after the PSS analysis. The IP3 point is the intercept point between the power for the signal ω_2 and the power for the signal $2\omega_1 - \omega_2$. Since the magnitude of this component is $0.75\alpha_3 A_1^2 A_2$, it has a linear relationship with the power level of the tone ω_2. Thus the ω_2 component can be treated as a small signal. It is necessary to set the power level of both tones the same.

The second method treats both tones as large signals and uses a QPSS analysis.

Both the first and second methods are equivalent because of the linear dependence of the output component's magnitude, $2\omega_1 - \omega_2$, on the input component's magnitude, ω_2. The recommended method is to use the PSS and PAC analyses for IP3 simulation because the PSS with PAC analysis method is more efficient than the QPSS analysis, and because the calculated IP3 is theoretically expected to be the same and is actually very close numerically.

The third method uses the PSS analysis with the beat frequency set to be the commensurate frequency of the two tones. Because the commensurate frequency can be very small, the simulation time for this method can be very long. This method is not recommended.

MMSIM6.0 USR2 provides new solution to calculate Rapid IP3 based on PAC or AC simulation. It is the fastest way for IP3 calculation. Rapid IP3 is a perturbative approach to solve weakly nonlinear circuit based on Born approximation. The method does not require explicit high order derivatives from device model. All equations are formulated in the form of RF harmonics. They can be implemented in both time and frequency domains.

For nonlinear system, the circuit equation can be expressed as:

$$L \cdot v + F_{NL}(v) = \varepsilon \cdot s$$

Here the first term is the linear part, the second one is the nonlinear part, and *s* is RF input source. Parameter ε is introduced to keep track of order of perturbation expansion. Under weakly nonlinear condition, nonlinear part is small compared to the linear part, so the above equation can be solved by using Born approximation iteratively:

$$u^{(n)} = v^{(1)} - L^{-1} \cdot F_{NL}\left(u^{(n-1)}\right)$$

where $u^{(n)}$ is the approximation of *v* and it accurate to the order or $O(\varepsilon^n)$.

Since the evaluation of F_{NL} takes full nonlinear device evaluation of *F* and its first derivative, no higher order derivative is needed. This allows us to carry out higher order perturbations without modifications in current device models. Also, the dynamic range of perturbation calculations covers only RF signals. It gives the perturbative method advantages in terms of accuracy.

Lab4: IP3 Measurement---PSS Plus PAC analysis

Action4-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action4-2: Select the **PORTrf** source. Use the **Edit—Properties—Objects** command to ensure that the port properties are set as described below:

Parameter	**Value**
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	sine
Frequency name 1	RF
Frequency 1	frf
Amplitude 1 (dBm)	prf
PAC magnitude (dBm)	prf

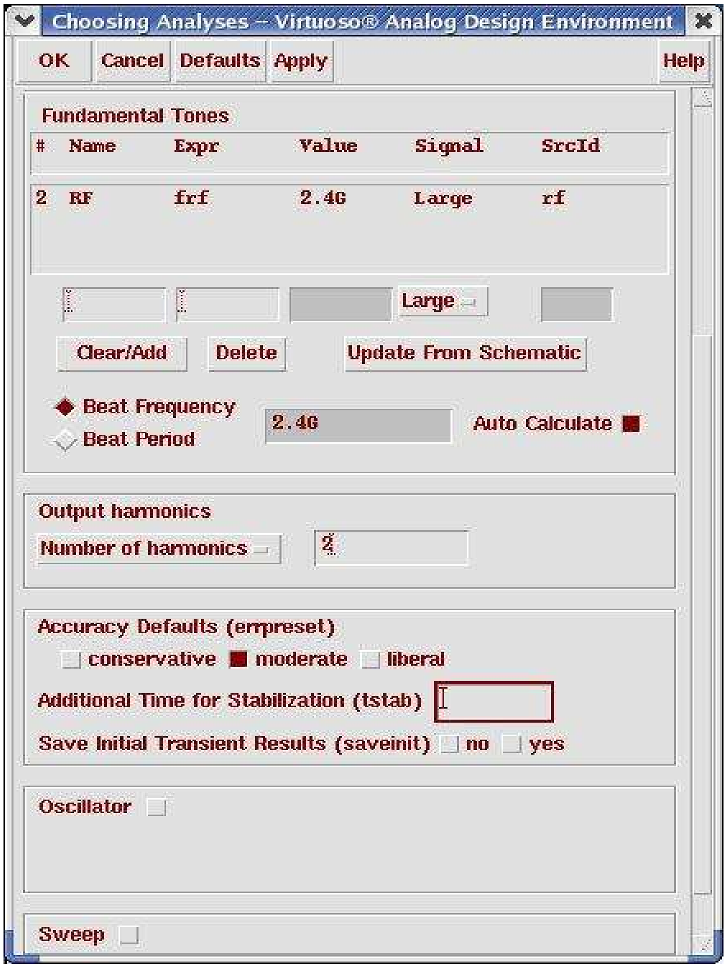
Action4-3: Check and save the schematic.

Action4-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.

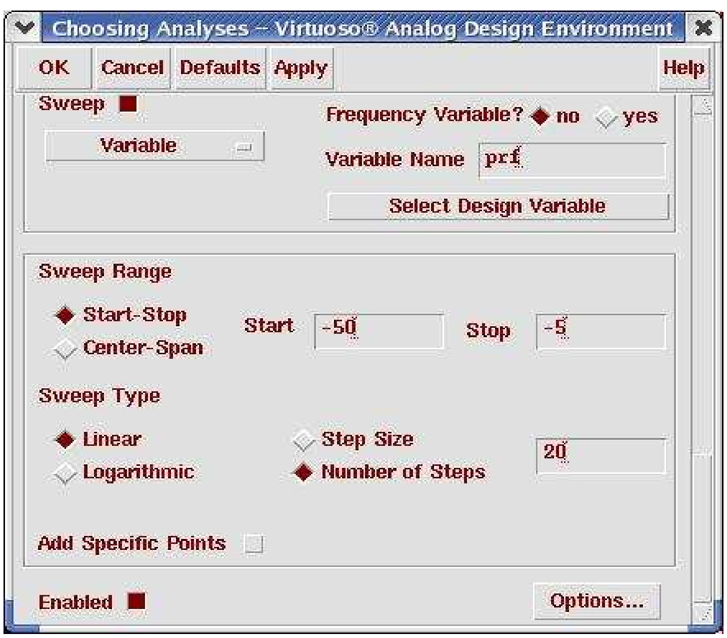
Action4-5: You can choose **Session—Load State**, load state “**Lab4_IP3_PSSPAC_shooting**” and skip to Action4-12 or …

Action4-6: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action4-7: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set up the form as follows:

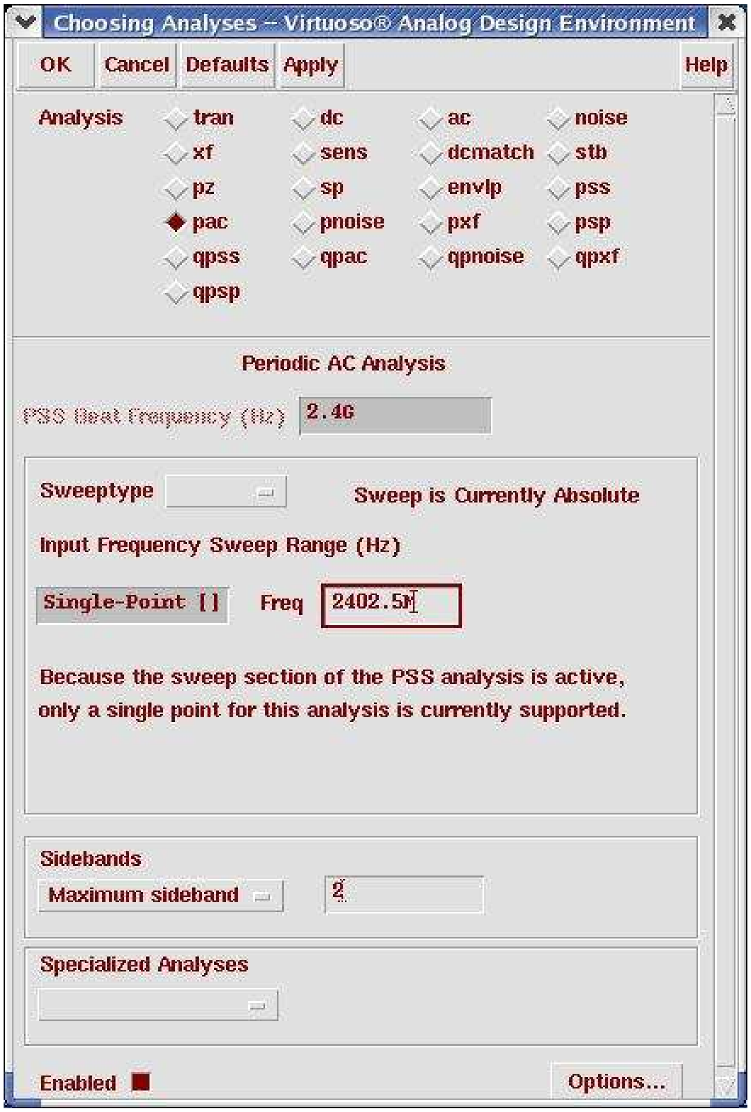


Action4-8: Enable the Sweep button and set the sweep values as follows:



Action4-9: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window.

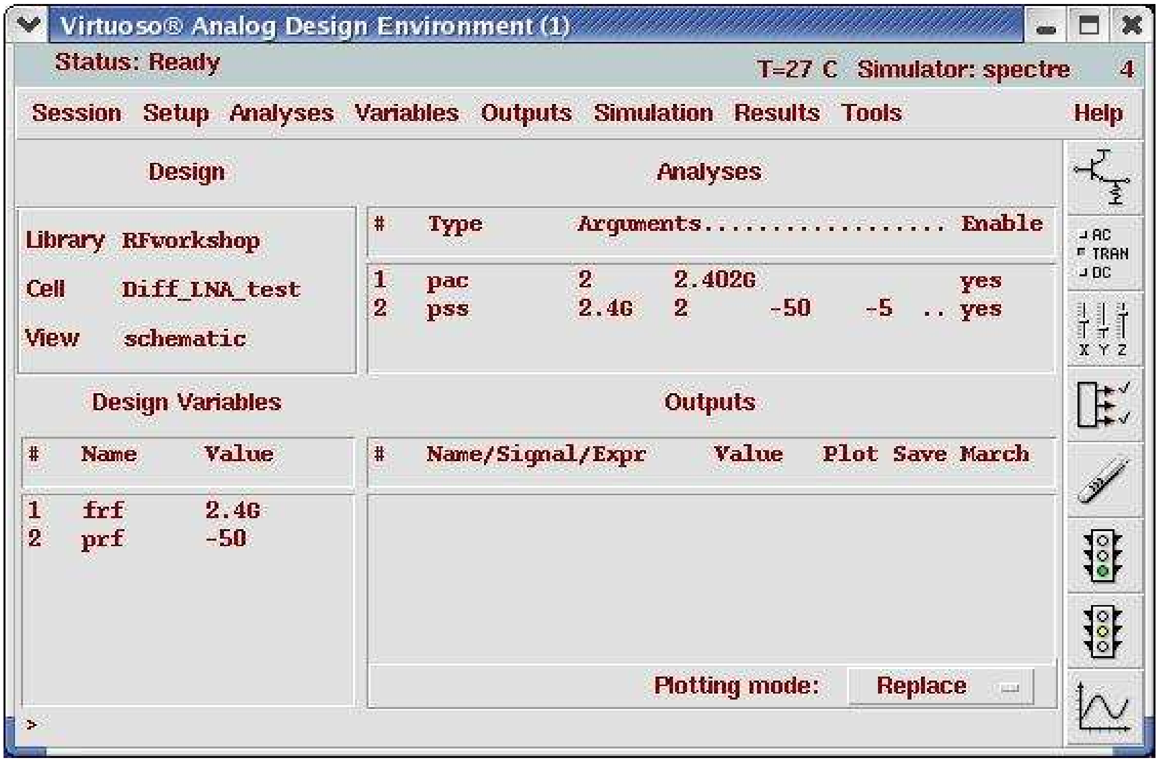
Action4-10: Set up the form as shown here:



Action4-11: Click **Ok** in the choosing Analyses form.

Your Simulation Environment should look like this:

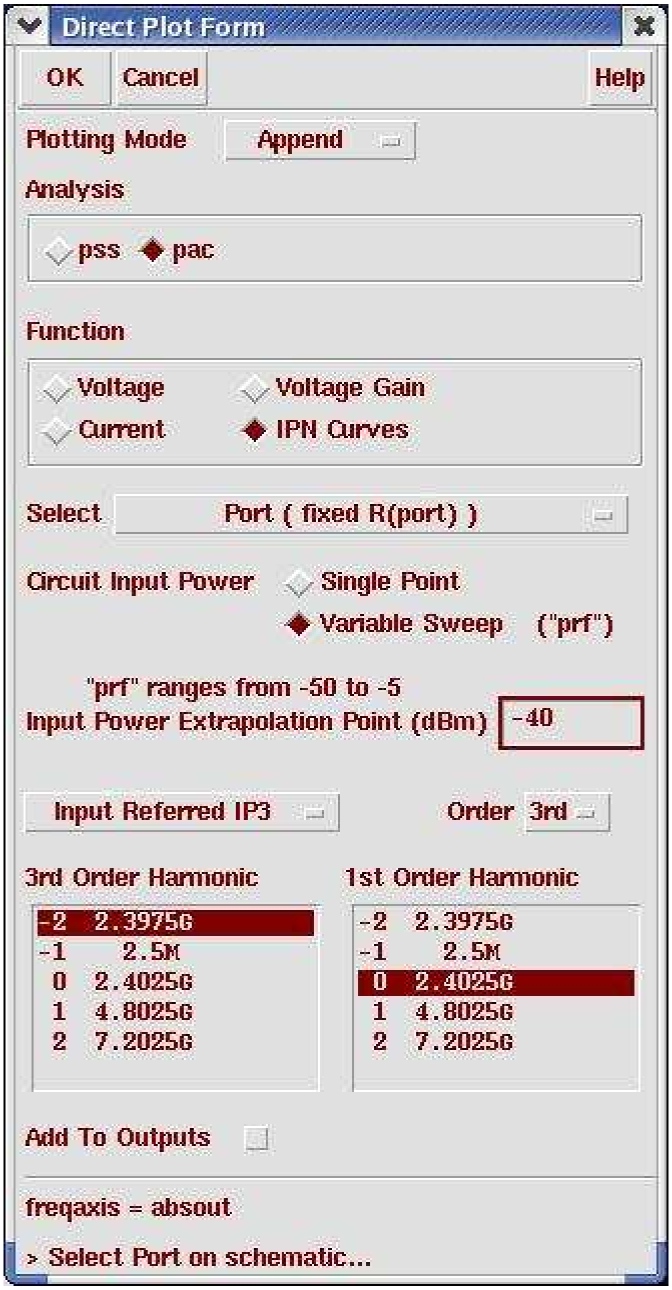
=



Action4-12: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

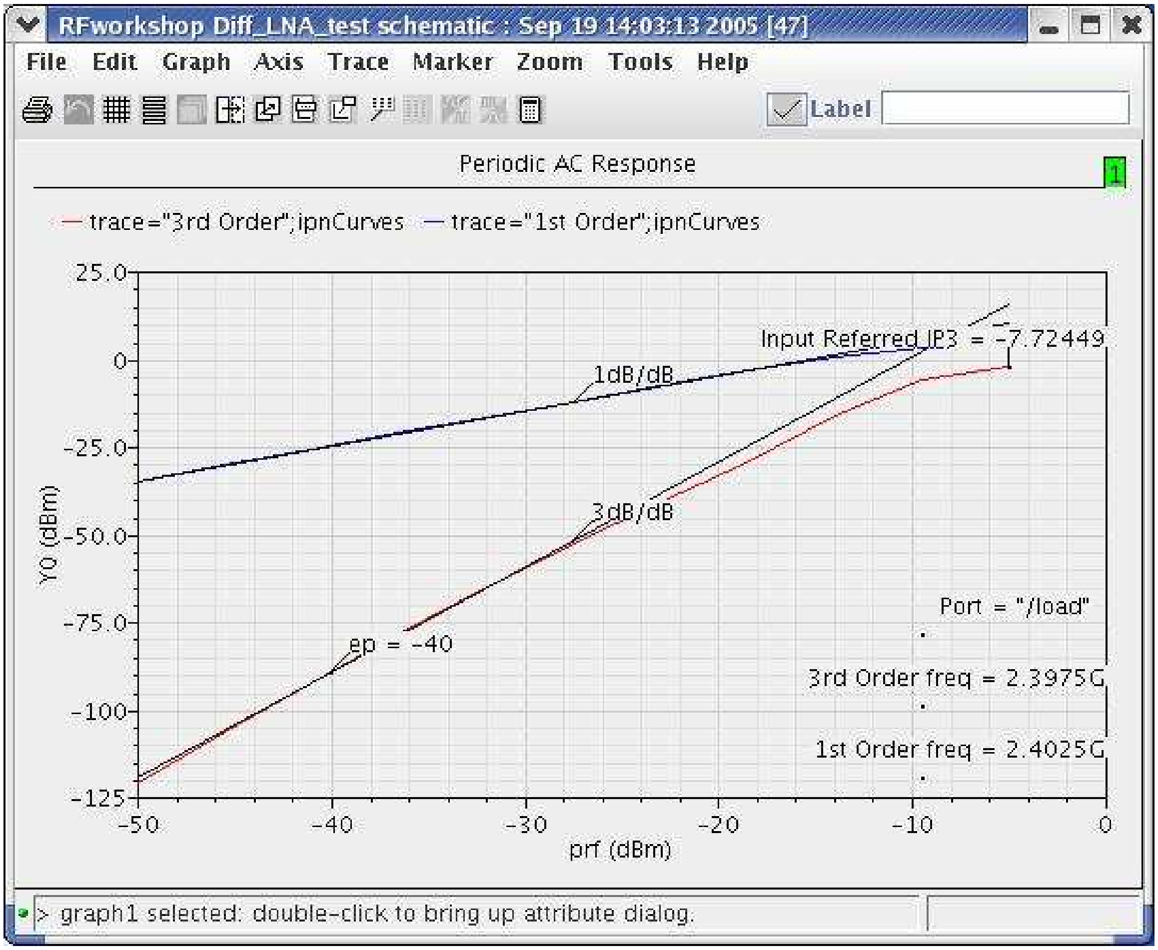
Action4-13: After the simulation ends, in the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action4-14: Choose **pac** and Set up the forms as follows:



Action4-15: Select **port load** in the Diff_LNA_test schematic.

The IP3 plot appears in the Waveform window.



Action4-16: Click Cancel in the Direct Plot form and close the waveform window.

Lab5: IP3 Measurement---QPSS Analysis with Shooting or Flexible Balance Engine

Action5-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action5-2: Select the **PORT rf** source. Use the Edit—Properties—Objects command to ensure that the port properties are set as described below:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	500 mV
Source type	sine
Frequency name 1	RF
Frequency 1	frf
Amplitude 1 (dBm)	prf
Frequency name 2	RF2
Frequency 2	frf+2.5M
Amplitude 2 (dBm)	prf

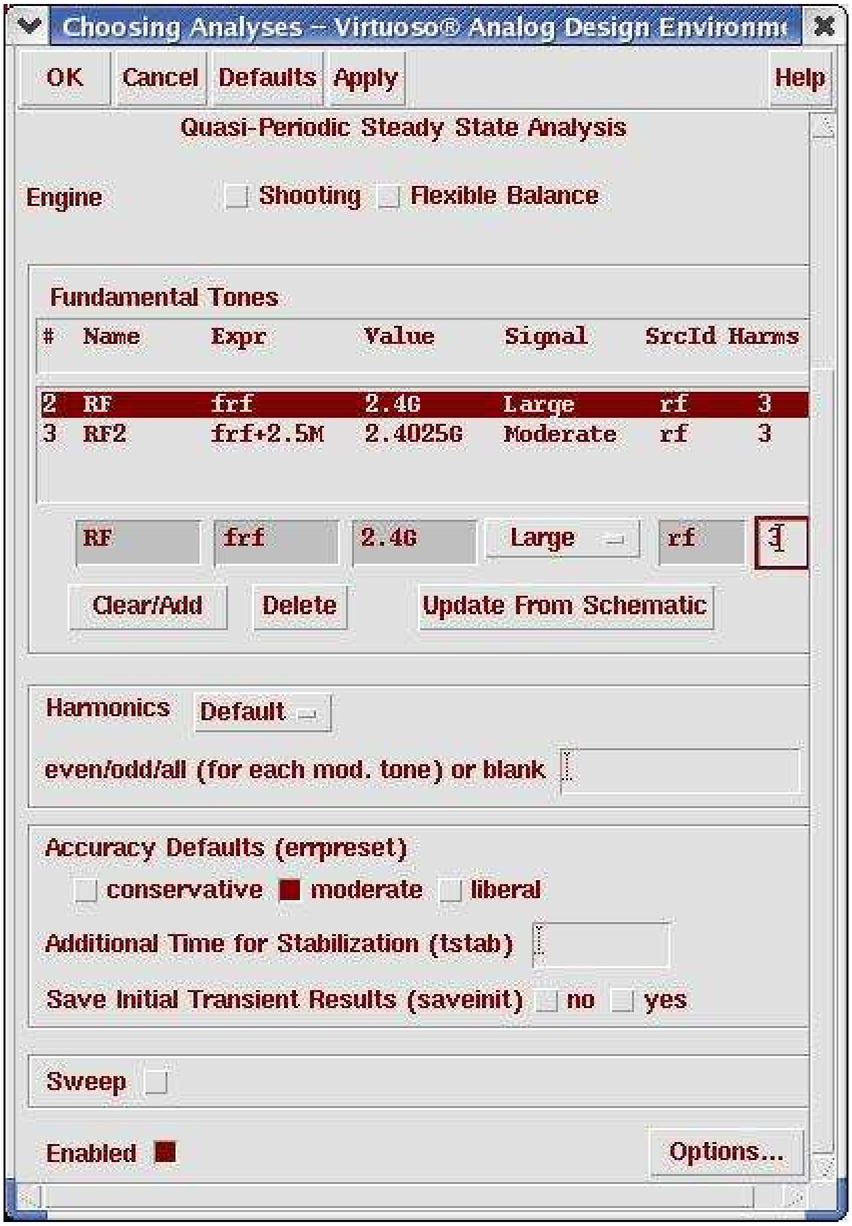
Action5-3: Check and save the schematic.

Action5-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.

Action5-5: You can choose **Session—Load State**, load state "**Lab5_IP3_QPSS_shooting**" and skip to Action5-9 or …

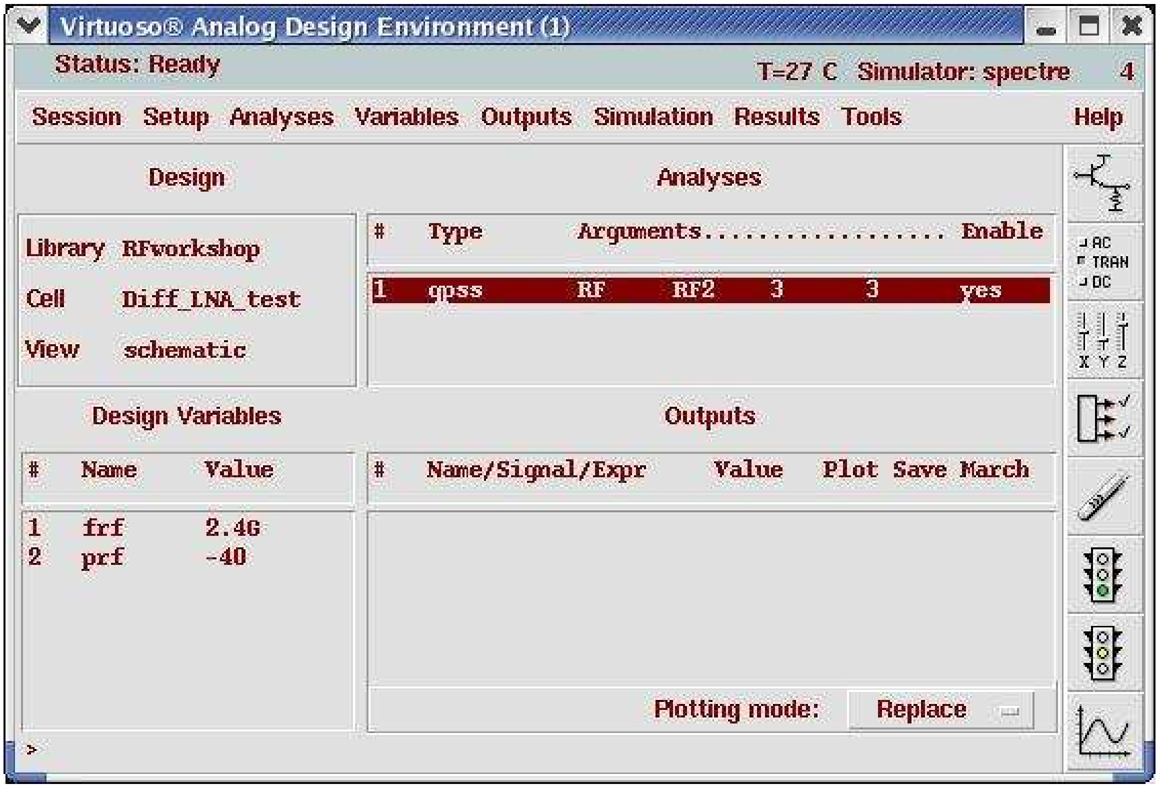
Action5-6: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action5-7: In the Choosing Analyses window, select the **qpss** button in the **Analysis** field of the window and set the form as follows:



Action5-8: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

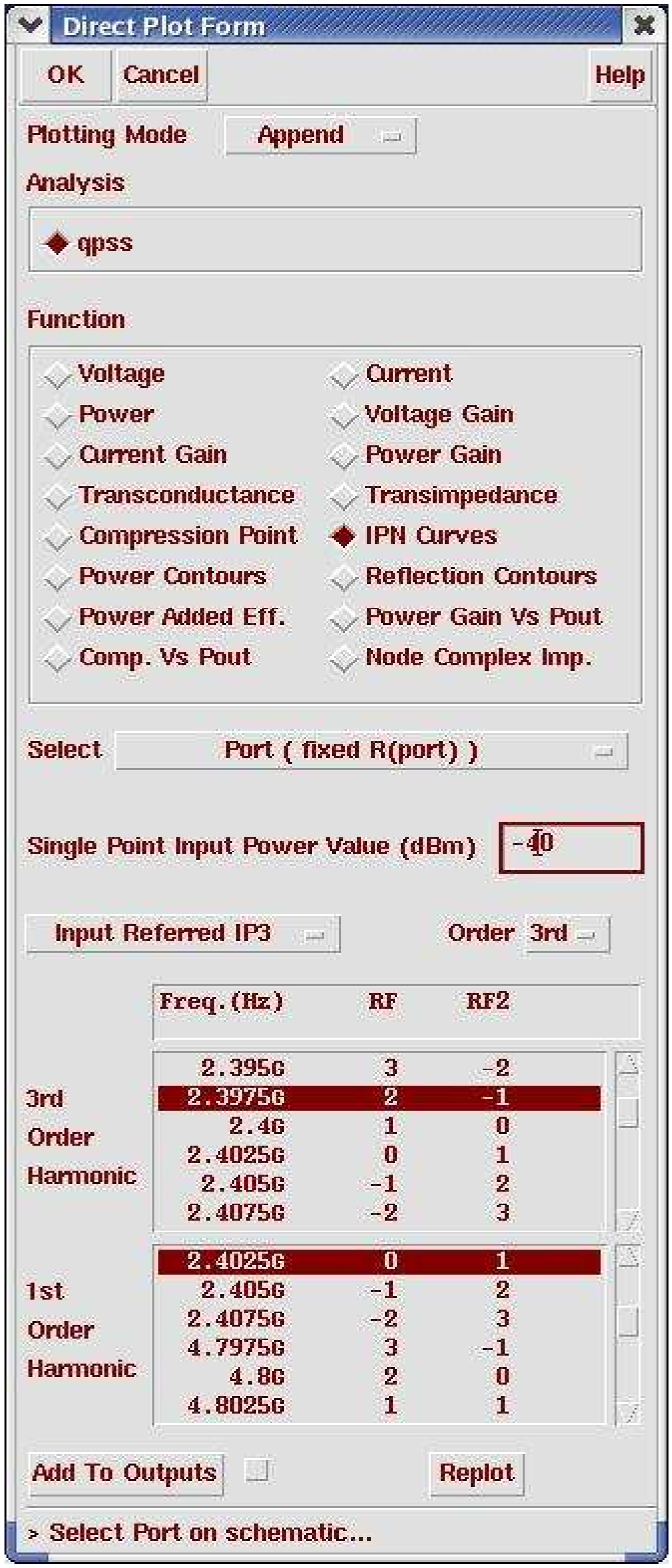
Now your Virtuoso Analog Design Environment will look like:



Action5-9: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

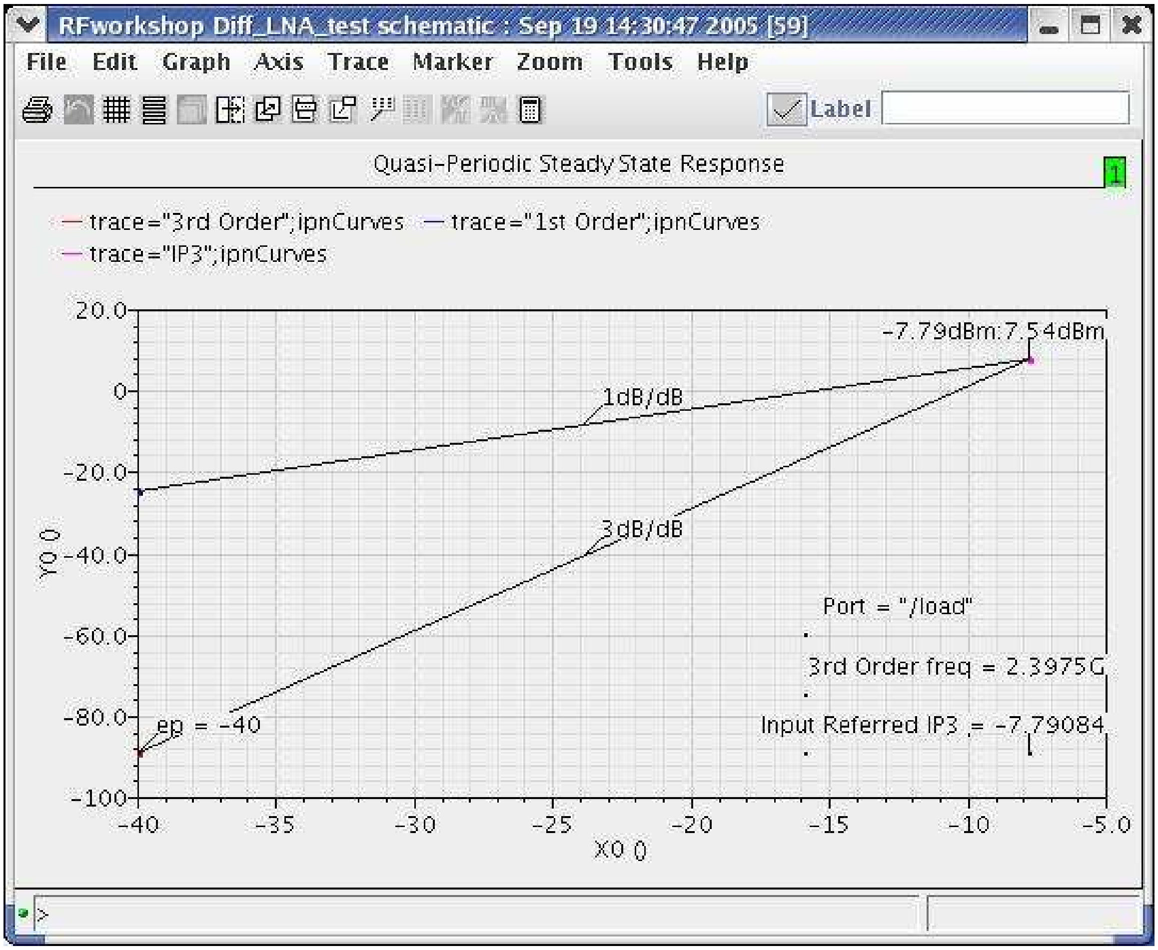
Action5-10: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action5-11: In the Direct Plot Form, select the **qpss** button, and configure the form as follows:



Action5-12: Select output port **load** on schematic.

The IP3 plot shows in the waveform window.



Action5-13: Close the waveform window. Click on Cancel on the Direct Plot form.

We are going to simulate the IP3 with Flexible Balance engine and compare its results with Shooting engine.

Action5-14: You can choose **Session—Load State**, load state "**Lab5_IP3_QPSS_FB**" and skip to Action5-21 or …

Action5-15: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

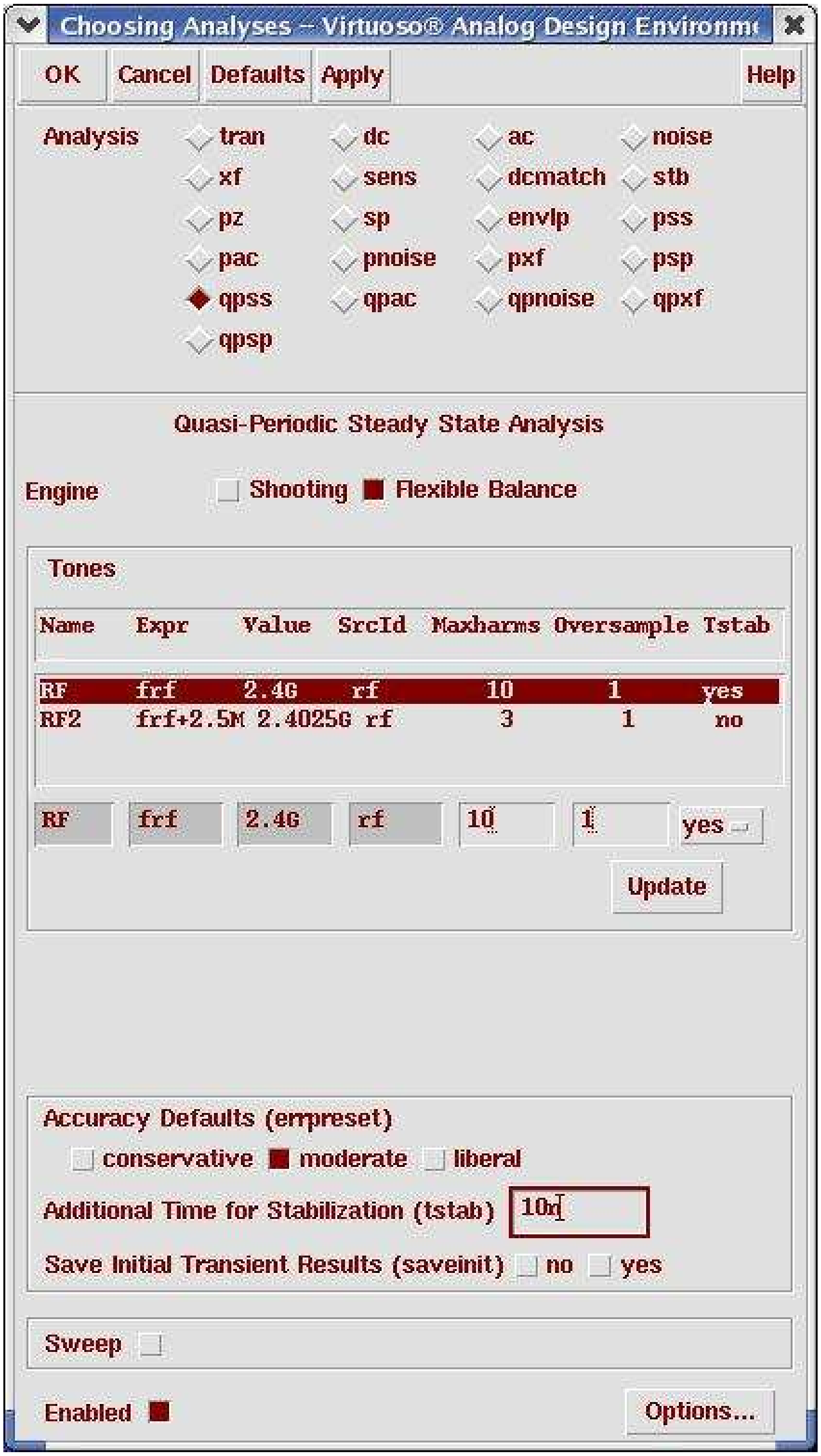
Action5-16: In the Choosing Analyses window, select the **qpss** button in the **Analysis** field of the window.

Action5-17: In **Engine** filed, choose **Flexible Balance**.

Action5-18: In field of Function Tones, choose RF. Change the Maxharms to 10, because Flexible balance need more harmonics to calculate. Click Update.

Action5-19: Put **10n** in the field of **Additional Time for Stabilization (stab)**.

The form should look like this:



Action5-20: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Note: Harmonic balance engine shares the same PSS/QPSS statement with time-domain engine. A toggle button is available for user to switch between time domain shooting and HB in ADE PSS and QPSS set up form. When setting up HB QPSS/PSS analyses, the following parameter should pay attention to:

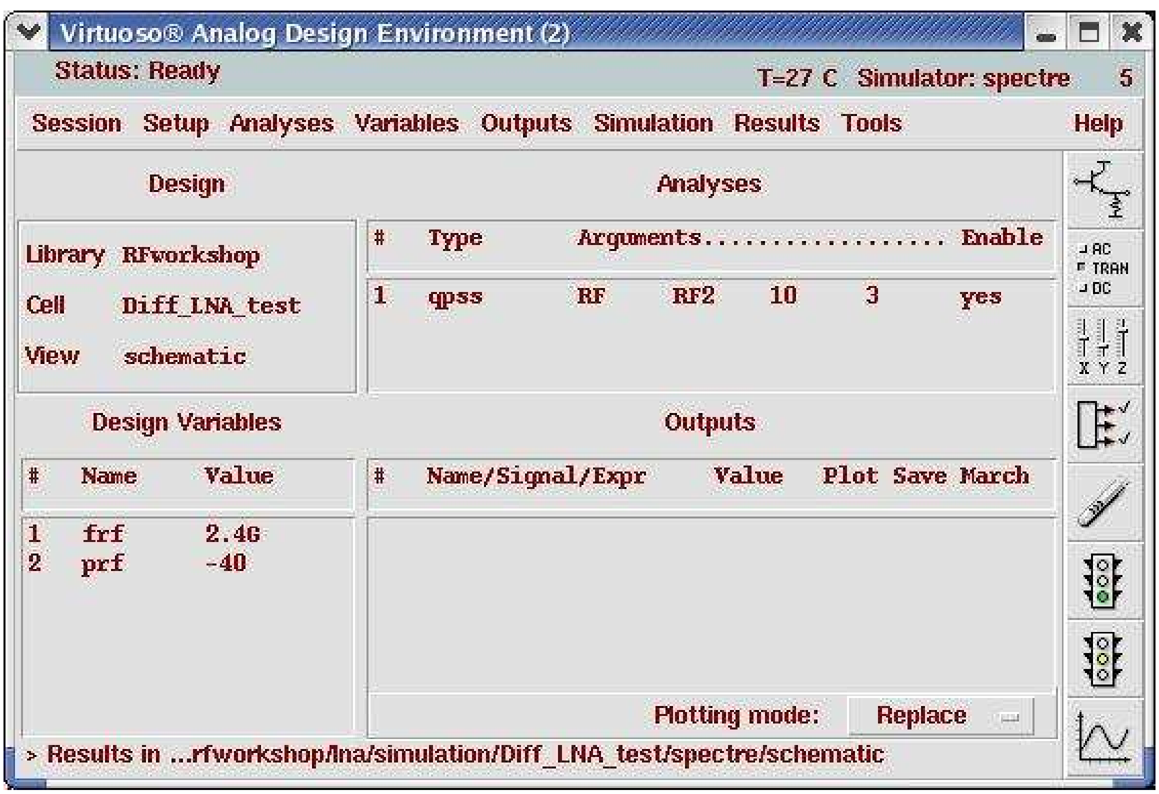
1**. Maximum harmonic**: Maximum harmonic ("**harms**" in PSS and "**maxharms**" in QPSS) has the most impact on HB accuracy. When inadequate harmonics are used, spectrum outside of **maxharm** will be folded back into harmonics inside by aliasing effect and cause error. To obtain accurate results, **maxharm** should be big enough to cover the signal bandwidth.

Parameter "**reltol**" or "**errpreset**" also determines the simulation accuracy. HB uses the same convergence criteria as the shooting method.

2. **tstab**: Similar to time domain shooting method, **tstab** is a valid parameter for initial transient analysis in HB. The default **tstab** for both PSS and QPSS is one cycle of signal period. For QPSS analysis, user can choose the specific tone during tstab period and only one tone is allowed. One additional cycle is run for FFT. If **tstab** is set to 0, dc results will be used as initial condition for HB.

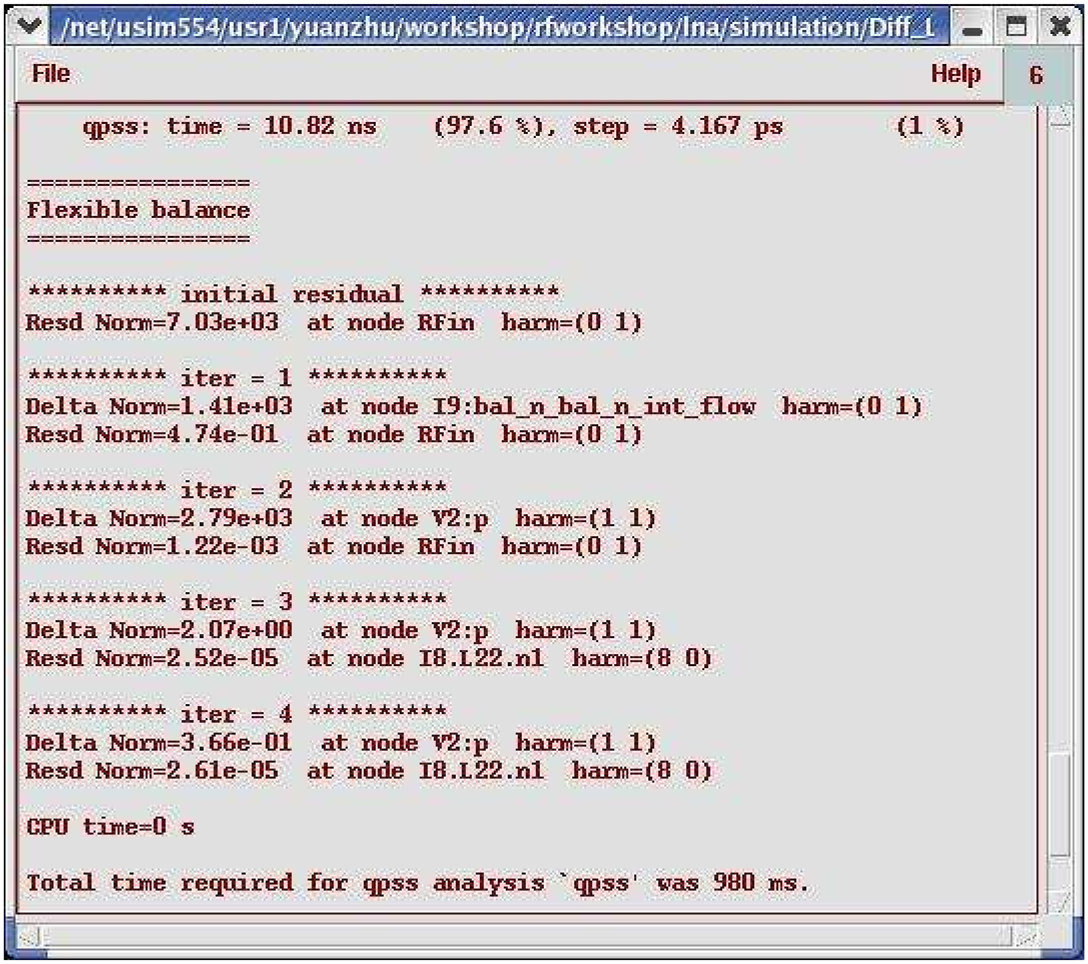
3: **Oversample Factor:** In general oversampling is not needed. For extremely nonlinear circuits driven by sources at very high power level, it might help with convergence.

Now your Virtuoso Analog Design Environment will look like:



Action5-21: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

As the simulation progresses, note messages in the simulation output log window that are different from time domain qpss:

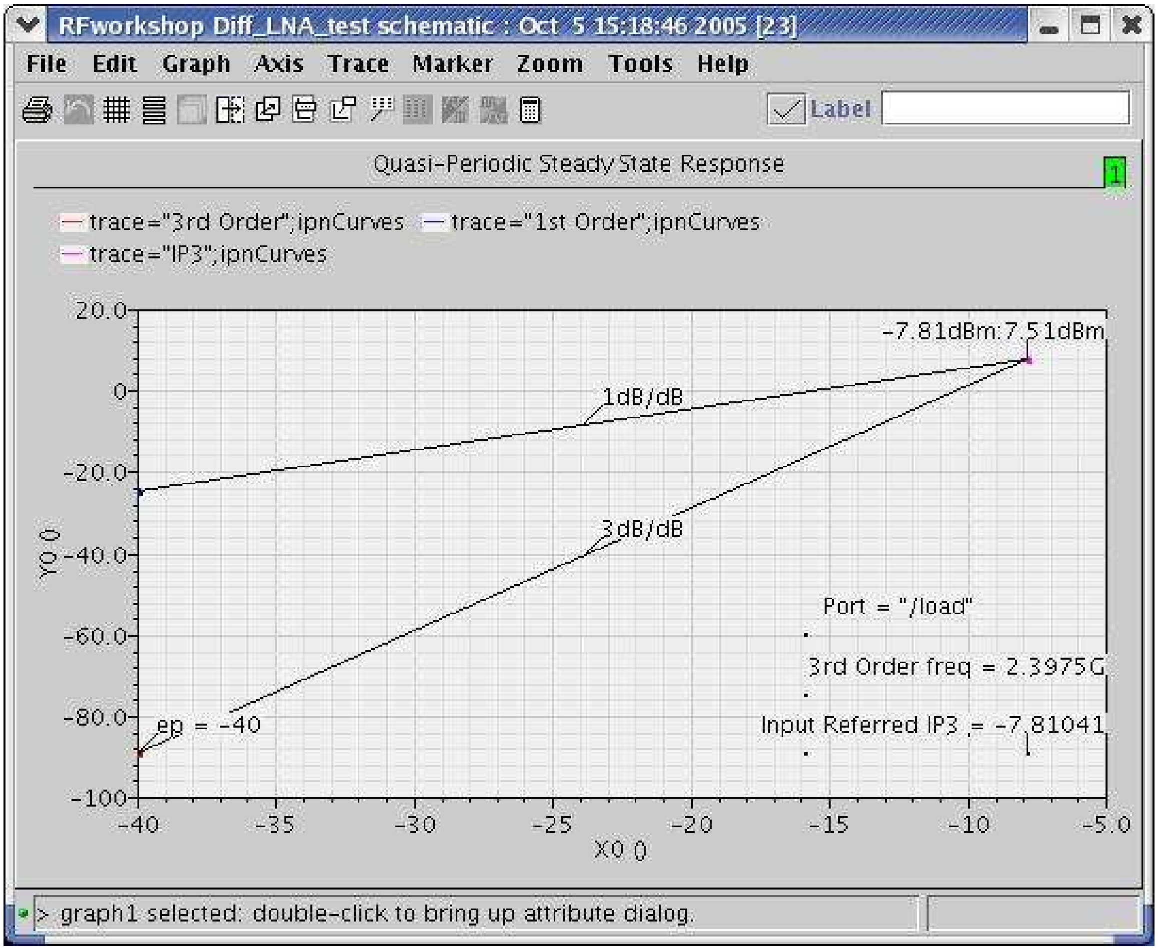


Action5-22: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action5-23: In the Direct Plot Form, select the **qpss** button. The form will the same as the one using shooting engine.

Action5-24: Select output port **load** on schematic.

The results are plotted in the Wavescan window.



Note: Flexible Balance is new feature of MMSIM6.0USR2. Flexible balance complements shooting method. It provides efficient and robust simulation for linear and weakly nonlinear circuits.

HB method is very efficient in simulating weakly nonlinear circuits such as LNA. Only a few harmonics are needed to represent accurately the solution. For highly nonlinear circuits with sharply raising/falling signals, time domain shooting is considered more suitable. However, HB may still has an advantage when exploring design trade-offs using a few harmonics where accuracy is not of primary concern.

Action5-25: After viewing the waveforms, close the waveform window and click **Cancel** in the Direct Plot form.

Lab6: IP3 Measurement---Rapid IP3 using AC analysis

Action6-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action6-2: Select the **PORTrf** source. Use the Edit—Properties—Objects command to ensure that the port properties are set as described below:

Parameter	**Value**
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	dc

Action6-3: Click **ok** on the Edit Object Properties window to close it.

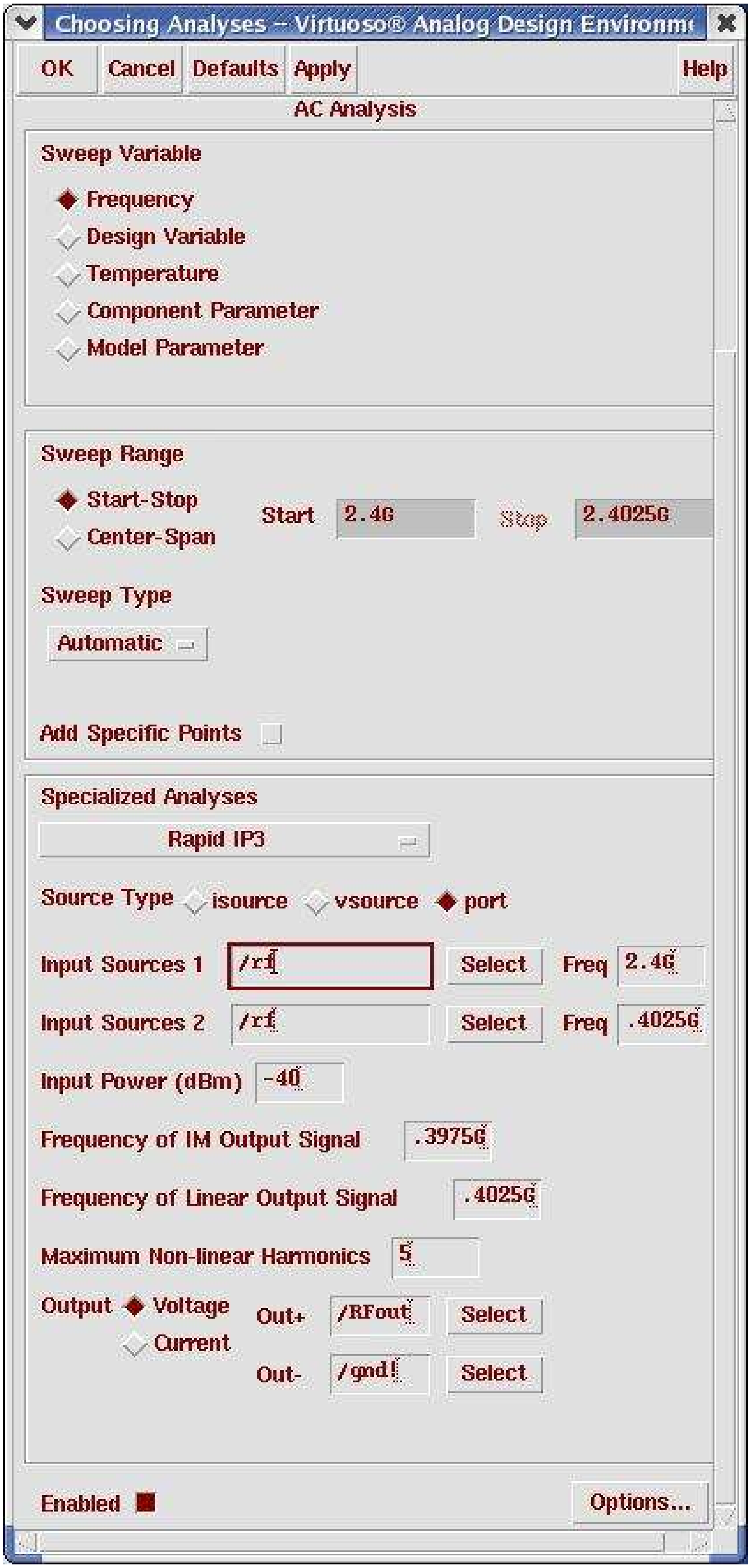
Action6-4: Check and save the schematic.

Action6-5: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.

Action6-6: You can choose **Session—Load State**, load state "**Lab6_RapidIP3_AC**" and skip to Action6-10 or …

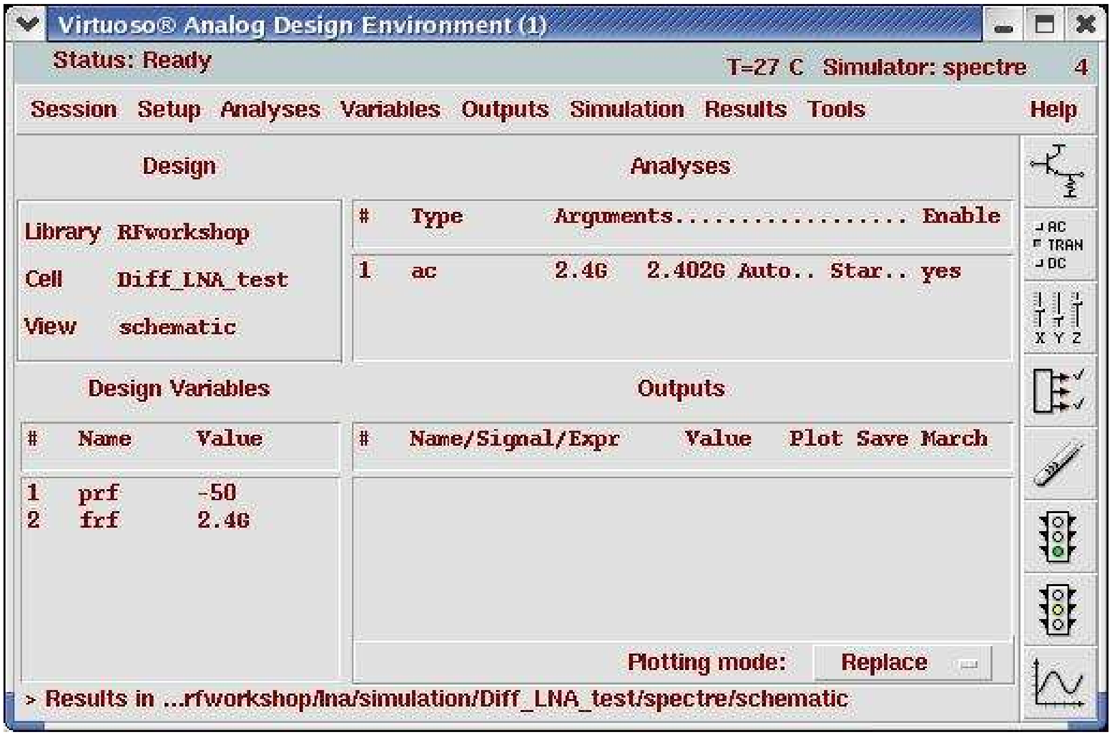
Action6-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action6-8: In the Choosing Analyses window, select the **ac** button in the **Analysis** field of the window. Choose Rapid IP3 as Specialized Analyses. Set the Input Source 1 to /rf by select PORT rf on the schematic. Push the ESC key on your keyboard to terminate the selection process. Set the Freq of source 1 to 2.4G and Freq of Source 2 to 2.4025G. Set the Frequency of IM Output Signal as 2.3975G and the frequency of Linear Output Signal as 2.0425G. If the Maximum Non-linear Harmonics is not specified, the default value is 4. The form should look like:


Choosing Analyses -- Virtuoso® Analog Design Environme
OK
Cancel
Defaults
Apply
Help
AC Analysis
Sweep Variable
Frequency
Design Variable
Temperature
Component Parameter
Model Parameter
Sweep Range
Start-Stop
Center-Span
Start
2.4G
Stop
2.4025G
Sweep Type
Automatic
Add Specific Points
Specialized Analyses
Rapid IP3
Source Type
isource
vsource
port
Input Sources 1
/rf
Select
Freq
2.4G
Input Sources 2
/rf
Select
Freq
.4025G
Input Power (dBm)
-40
Frequency of IM Output Signal
.3975G
Frequency of Linear Output Signal
.4025G
Maximum Non-linear Harmonics
5
Output
Voltage
Current
Out+
/RFout
Select
Out-
/gnd!
Select
Enabled
Options...

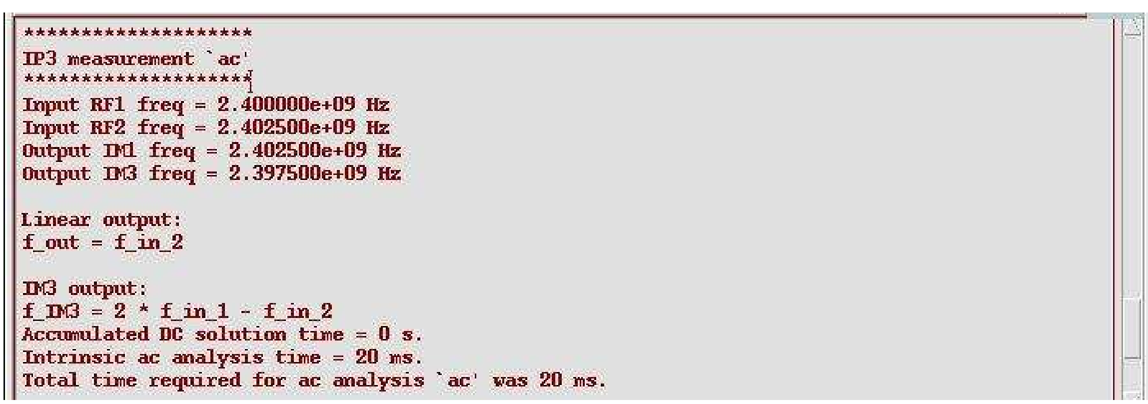
Action6-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:



Action6-10: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

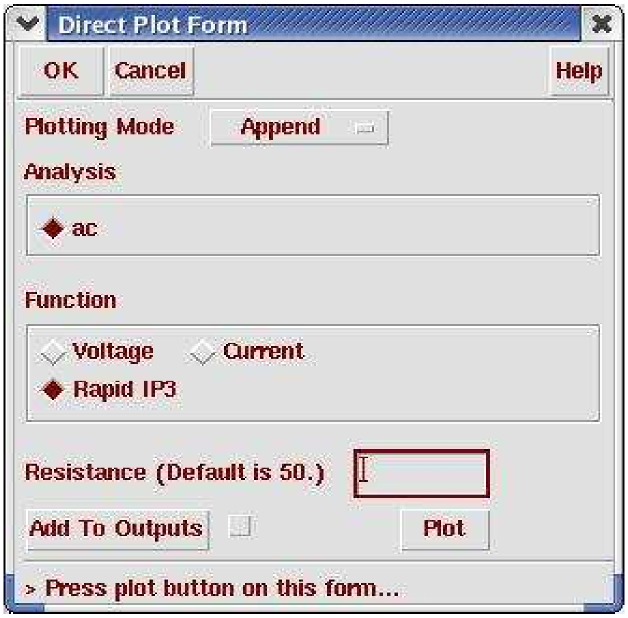
As the simulation progresses, note messages in the simulation output log window that are similar to the following:



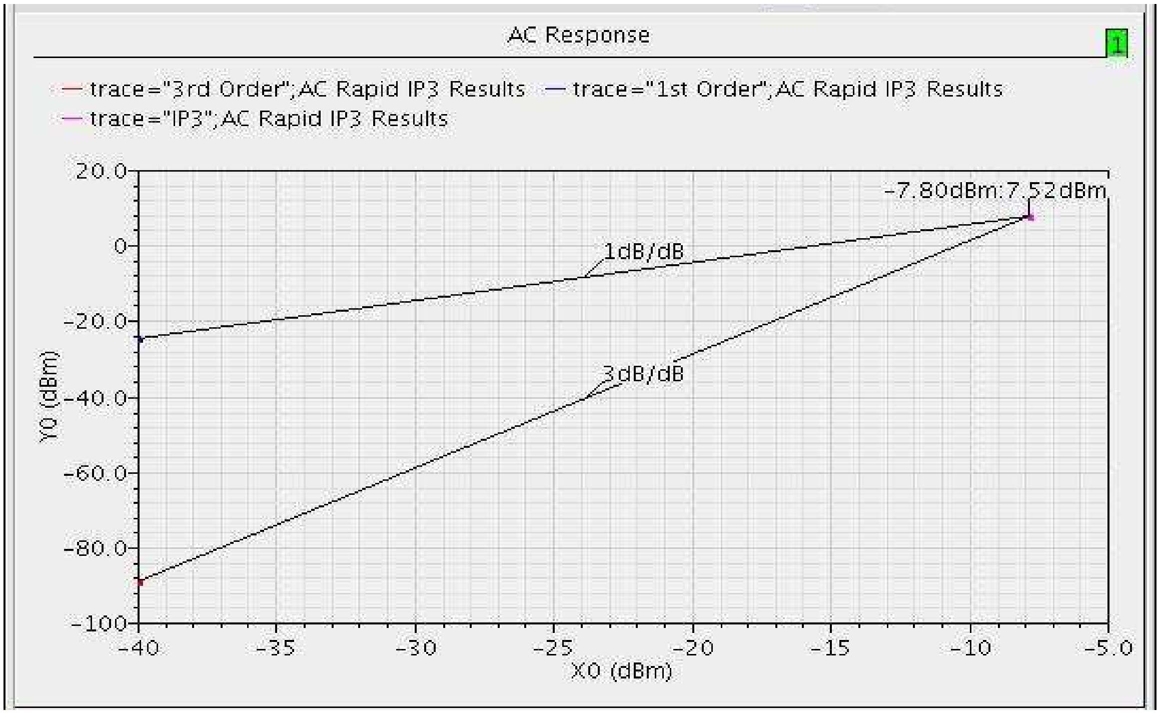
```
********************
IP3 measurement `ac'
********************
Input RF1 freq = 2.400000e+09 Hz
Input RF2 freq = 2.402500e+09 Hz
Output IM1 freq = 2.402500e+09 Hz
Output IM3 freq = 2.397500e+09 Hz

Linear output:
f_out = f_in_2

IM3 output:
f_IM3 = 2 * f_in_1 - f_in_2
Accumulated DC solution time = 0 s.
Intrinsic ac analysis time = 20 ms.
Total time required for ac analysis `ac' was 20 ms.
```

Action6-11: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action6-12: In the Direct Plot Form, select the **ac** button. Choose Rapid IP3 in the function field.



Action6-13: Click Plot button to get IP3 calculation results:



Action6-14: After viewing the waveforms, click **Cancel** in the Direct Plot form.

Lab7: IP3 Measurement ---Rapid IP3 using PSS Plus PAC Analysis

Action7-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action7-2: Select the **PORT rf** source. Use the Edit—Properties—Objects command to ensure that the port properties are set as described below:

Parameter	**Value**
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	dc

Action7-3: Click **ok** on the Edit Object Properties window to close it.

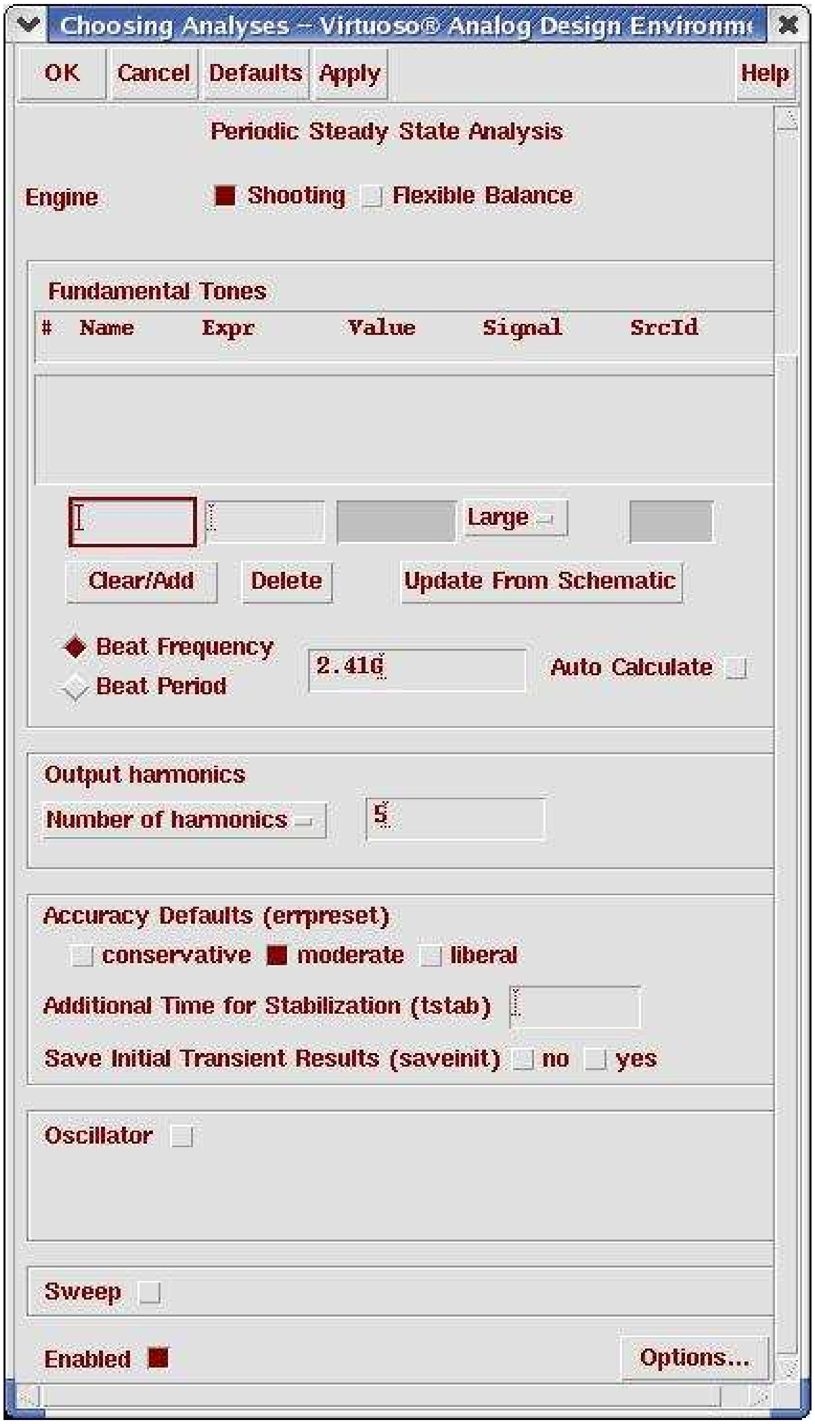
Action7-4: Check and save the schematic.

Action7-5: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action7-6: You can choose **Session—Load State**, load state "**Lab7_RapidIP3_PAC**" and skip to Action7-12 or …

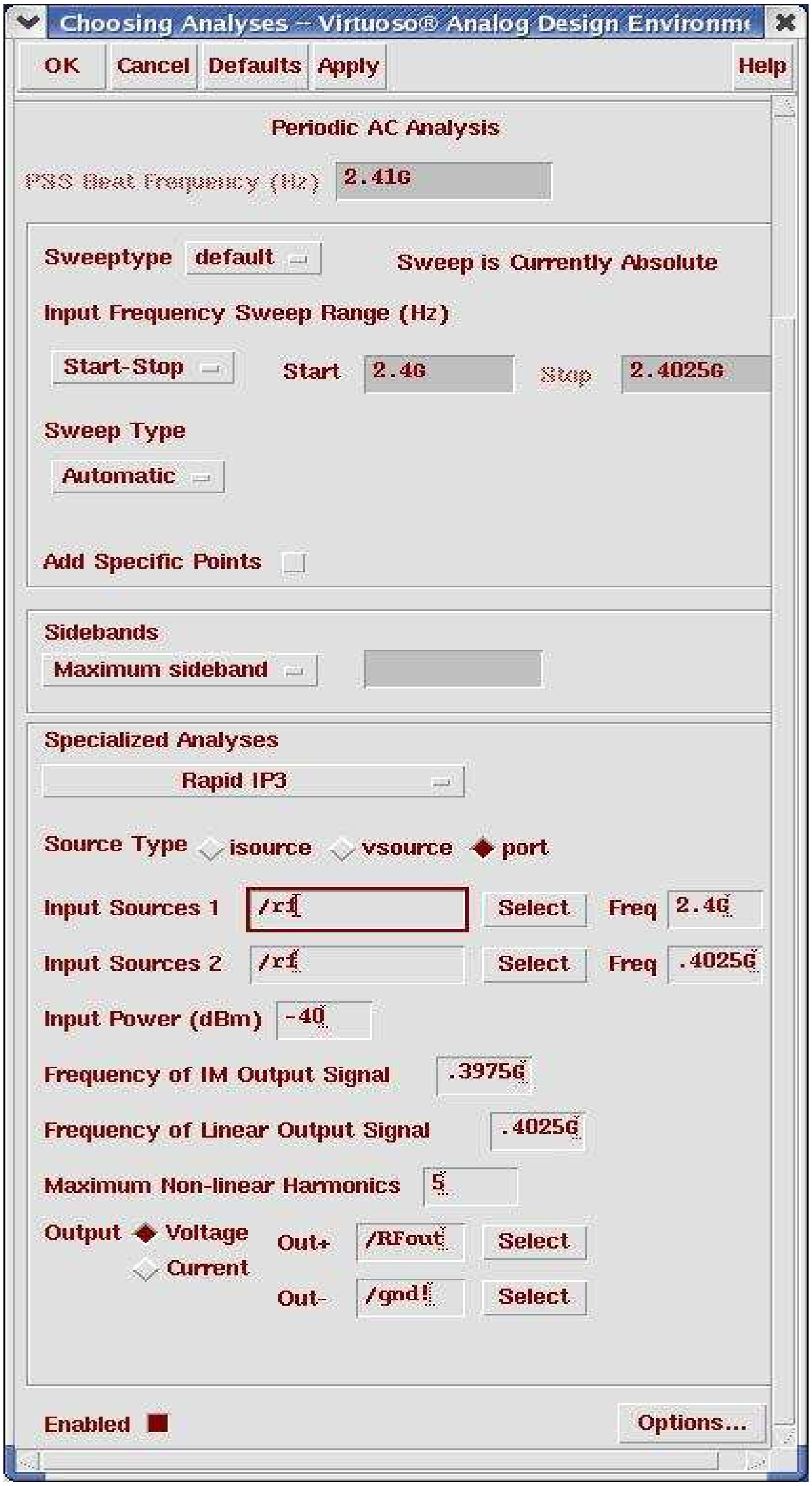
Action7-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action7-8: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set the form as follows:



Action7-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **apply** button.

Action7-10: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window. Choose Rapid IP3 as Specialized Analyses. Set the Input Source 1 to /rf by select PORT rf on the schematic. Push the ESC key on your keyboard to terminate the selection process. Set the Freq of source 1 to 2.4G and Freq of Source 2 to 2.4025G. Set the Frequency of IM Output Signal as 2.3975G and the frequency of Linear Output Signal as 2.0425G. . If the Maximum Non-linear Harmonics is not specified, the default value is 4. The form should look like:

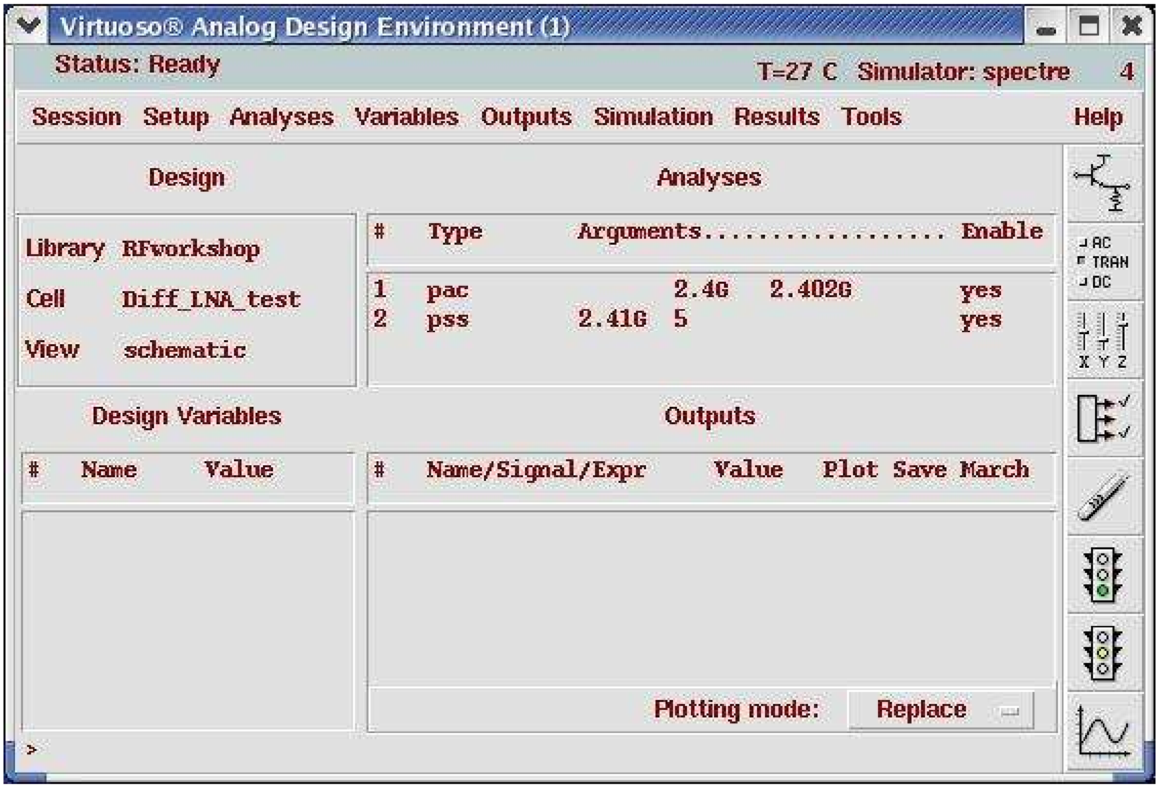


Note: Incommensurate frequencies should be used for all tones. If multiple combinations of tone frequencies match *Frequency of Linear Output Signal* or *Frequency of IM output Signal*, Spectre will complain because it can't figure out which one user wants to use as IM1 or IM3.

If output is current in a port, user must use the ‘save’ statement to indicate that the port current needs to be computed. Otherwise, Spectre won’t calculate it.

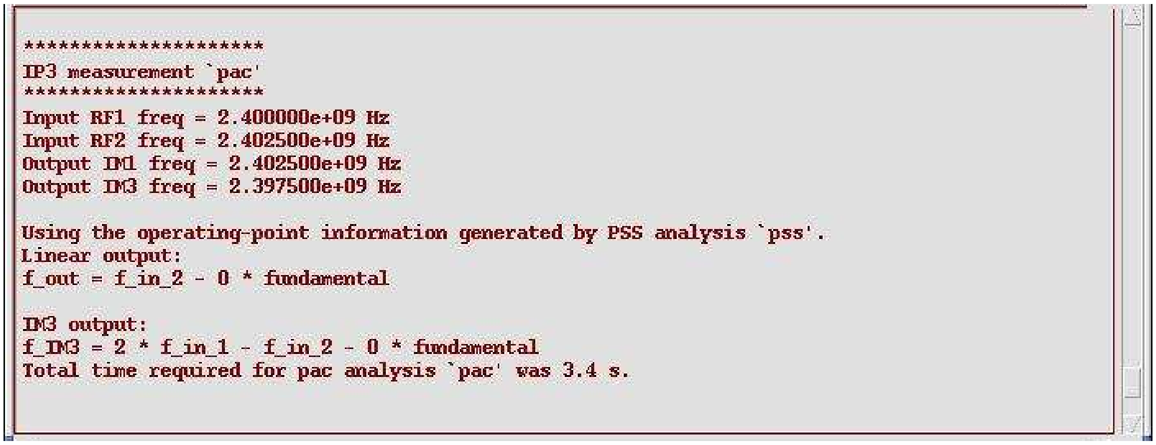
Action7-11: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:



Action7-12: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

As the simulation progresses, note messages in the simulation output log window that are similar to the following:

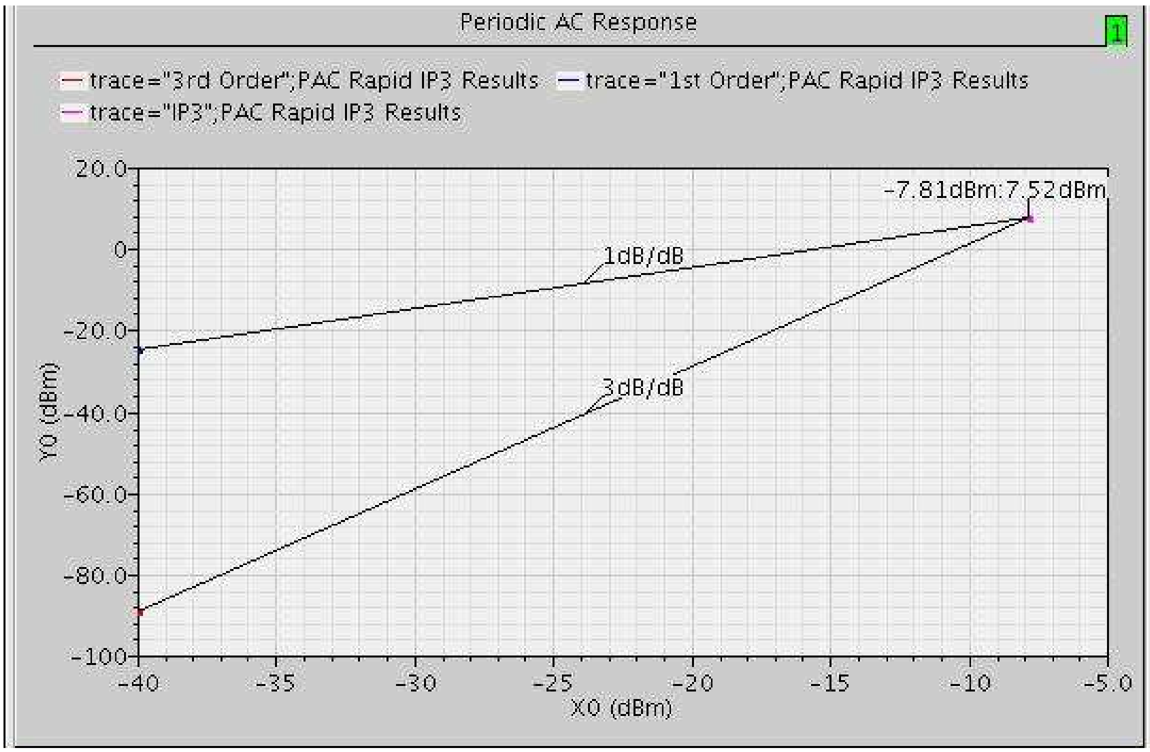


```
*********************
IP3 measurement `pac'
*********************
Input RF1 freq = 2.400000e+09 Hz
Input RF2 freq = 2.402500e+09 Hz
Output IM1 freq = 2.402500e+09 Hz
Output IM3 freq = 2.397500e+09 Hz

Using the operating-point information generated by PSS analysis `pss'.
Linear output:
f_out = f_in_2 - 0 * fundamental

IM3 output:
f_IM3 = 2 * f_in_1 - f_in_2 - 0 * fundamental
Total time required for pac analysis `pac' was 3.4 s.
```

Action7-13: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action7-14: In the Direct Plot Form, select the **pss** button, and configure the form as follows:

Action7-15: Select output port **load** on schematic.



Action7-16: After viewing the waveforms, close the waveform window. Click **Cancel** in the Direct Plot form

Conclusion

This application note discusses LNA testbench setup, LNA design parameters, and how to use SpectreRF to simulate an LNA and extract design parameters. Some useful SpectreRF analysis tools for LNA design, such as SP, PSS, Pnoise, PAC and QPSS analyses are addressed. The results from the analyses are interpreted.

Reference

[1] "The Designer's Guide to Spice & Spectre", Kenneth S. Kundert, Kluwer Academic Publishers, 1995.

[2] "Microwave Transistor Amplifiers", Guillermo Gonzalez, Prentice Hall, 1984.

[3] "RF Microelectronics", Behzad Razavi. Prentice Hall, NJ, 1998.

[4] "The Design of CMOS Radio Frequency Integrated Circuits", Thomas H. Lee. Cambridge University Press, 1998.