SpectreRF Workshop

LNA Design Using SpectreRF

MMSIM7.1

November 2008

Contents

LNA Design Using SpectreRF

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 (ADE) to measure parameters that are important in design verification of low noise amplifiers (LNAs).

Audience

Users of SpectreRF in the Analog Design Environment.

Overview

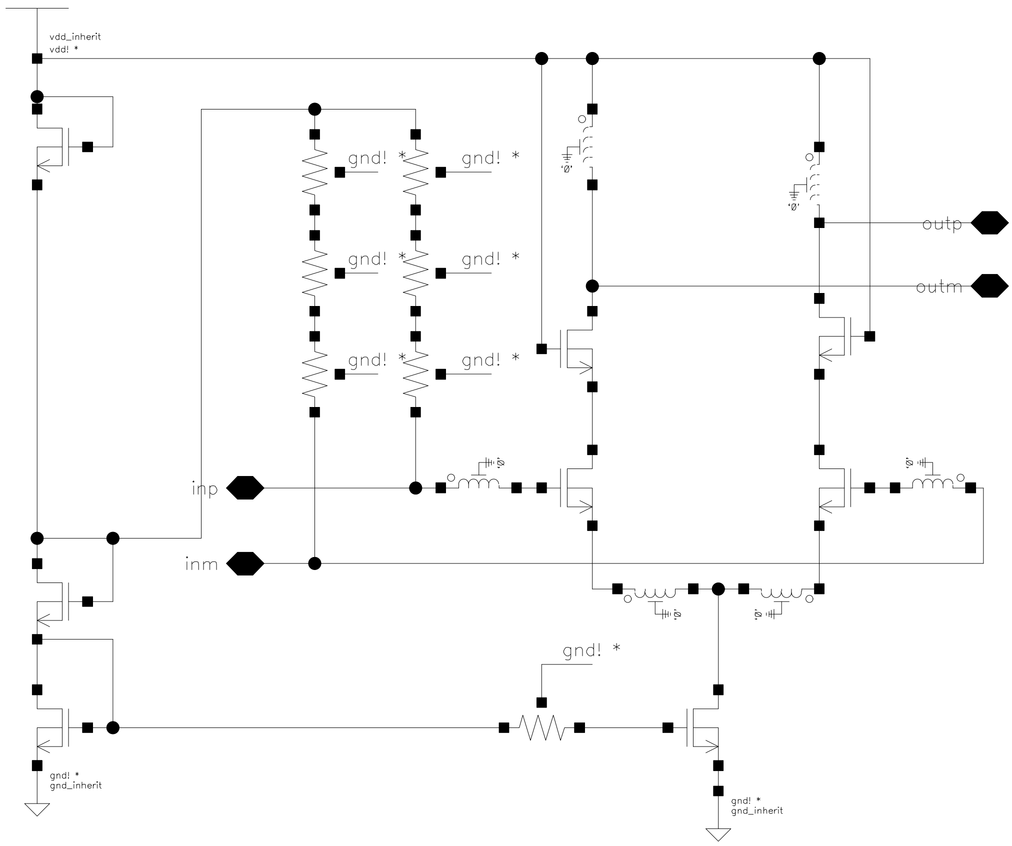
This workshop describes a basic set of the most useful measurements for LNAs.

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. Good matching is extremely important if a passive band-select filter and image-reject filter precedes and succeeds the LNA, because 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 ADE. The design investigated is the differential low noise amplifier shown below:



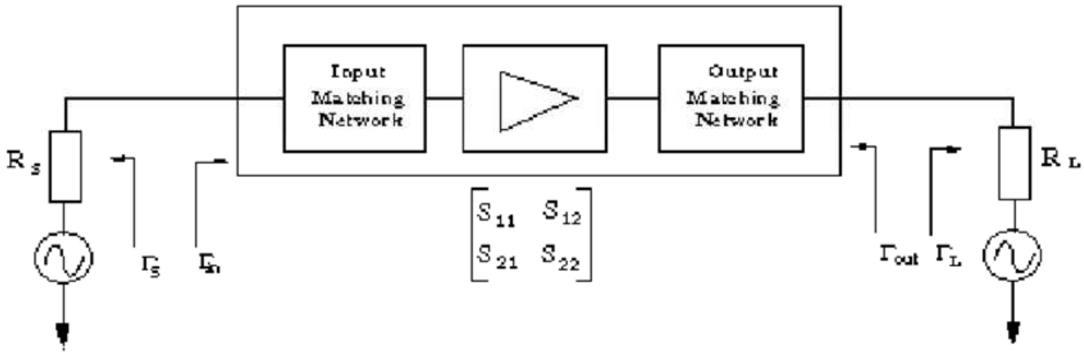
The following table lists typically acceptable values for the performance metrics of LNAs used in heterodyne architectures.

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

Figure 1-2 shows a generic two-port amplifier model. Its input and output are each terminated by a resistive port, like an amplifier measurement using a network analyzer.

Figure 1-2 A Generic Two-Port LNA



The LNA is characterized by the scattering matrix in Equation 1-1.

(1-1) $$\begin{bmatrix} b_S \\ b_L \end{bmatrix} = \begin{bmatrix} S_{11} S_{12} \\ S_{21} S_{22} \end{bmatrix} \begin{bmatrix} a_S \\ a_L \end{bmatrix}$$

where b_S and b_L are the reflected wave from the input and output of the LNA, a_S and a_L are the incident wave to the input and output of the LNA. They are defined in terms of the terminal voltage and current as follows

$$a_S = \frac{V_{in}}{2\sqrt{R_s}} + \frac{\sqrt{R_s}}{2} I_{in}$$

$$b_S = \frac{V_{in}}{2\sqrt{R_s}} - \frac{\sqrt{R_s}}{2} I_{in}$$

$$a_L = \frac{V_{out}}{2\sqrt{R_{Ls}}} + \frac{\sqrt{R_L}}{2} I_{out}$$

$$b_L = \frac{V_{out}}{2\sqrt{R_{Ls}}} - \frac{\sqrt{R_L}}{2} I_{out}$$

Spectre normalizes the LNA scattering matrix with respect to the source and load port resistance. Therefore, the source reflection coefficient Γ_S and load reflection coefficient Γ_L are both zero.

From network theory, the input and output reflection coefficients are expressed in Equations 1-2 and 1-3.

(1-2) $$\Gamma_{in} = S_{11} + \frac{S_{21}S_{12}\mathrm{T}_L}{1 - S_{22}\Gamma_L}$$

(1-3) $$\Gamma_{out} = S_{22} + \frac{S_{12}S_{21}\mathrm{T}_S}{1 - S_{11}\Gamma_S}$$

The LNA scattering matrix is normalized in terms of the source and load resistance in Equation 1-4.

(1-4) $$\Gamma_S = \Gamma_L = 0$$

Thus, the input and output reflection coefficients are simply expressed in Equations 1-5 and 1-6.

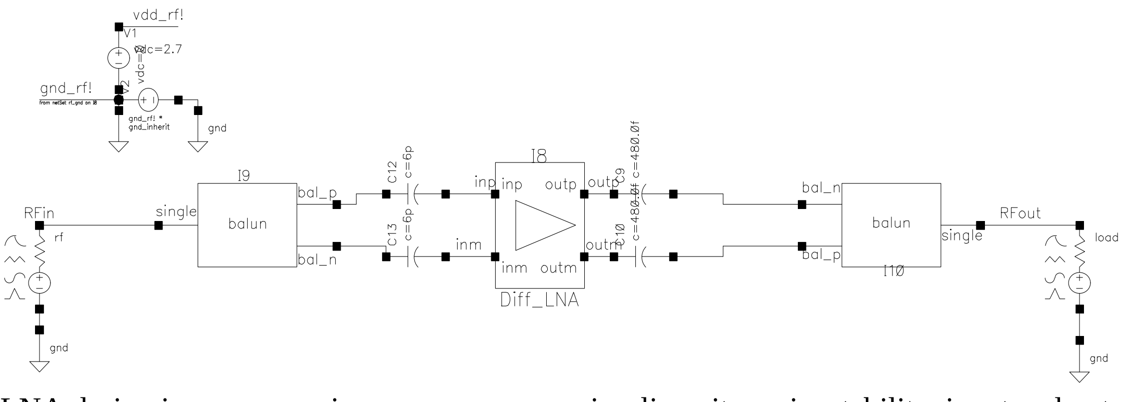
(1-5) $$\Gamma_{in} = S_{11}$$

(1-6) $$\Gamma_{out} = S_{22}$$

The main challenge of LNA design lies in the design of the input/output matching network to render Γ*in* and Γ*out* close to zero so that the LNA is matched to the source and load ports.

With the knowledge of a generic LNA model, Figure 1-3 shows the testbench for a differential LNA. The baluns used in the testbench are three-port devices. The baluns convert between single-ended and differential signals. Sometimes, they also perform the resistance transformation.

Figure 1-3 Testbench for a Double-Ended LNA



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

LNA Measurements and Design Specifications

Power Consumption and Supply Voltage

You must trade off gain, distortion, and noise performance against power dissipation. Total power dissipation for an operating LNA circuit should be within its design budget. Because most LNAs are operated in Class-A mode, power consumption is easily available by multiplying the DC supply voltage by the DC operating point current. Selecting the operating point is a critical stage of LNA design which affects the power consumption, noise performance, IP3, and dynamic range.

Gain

Three power gain definitions appear in the literature and are commonly used in LNA design.

- G_T, transducer power gain
- G_P, operating power gain
- G_A, available power gain

Besides these three gain definitions, there are three additional gain definitions you can use to evaluate the LNA design.

- G_{umx}, maximum unilateral transducer power gain
- $G_{\max}$, maximum transducer power gain
- G_{msg}, maximum stability gain

There are also two gain circles that are helpful to the design of input and output matching networks.

- *GPC*, power gain circle
- *GAC*, available gain circle

Transducer Power Gain

Transducer power gain, G_T, is defined as the ratio between the power delivered to the load and the power available from the source.

(1-9) $$G_T = \frac{1-|\Gamma_S|^2}{|1-S_{11}\Gamma_S|^2}|S_{21}|^2\frac{1-|\Gamma_L|^2}{|1-S_{22}\Gamma_L|^2}$$

In the test environment, from Equation 1-4 on page 6, you have

(1-10) $$G_T = |S_{21}|^2$$

Operating power gain

Operating power gain, G_T , is defined as the ratio between the power delivered to the load and the power input to the network.

(1-11) $$G_P = \frac{1}{1-|\Gamma_{in}|^2}|S_{21}|^2 \frac{1-|\Gamma_L|^2}{|1-S_{22}\Gamma_L|^2}$$

In the test environment, from Equations 1-4 and 1-5 on page 6, you have

(1-12) $$G_P = \frac{1}{1-|S_{11}|^2}|S_{21}|^2$$

Available power gain

Available power gain, G_T , is defined as the ratio between the power available from the network and the power available from the source.

(1-13) $$G_A = \frac{1-|\Gamma_S|^2}{|1-S_{11}\Gamma_S|^2}|S_{21}|^2 \frac{1}{1-|\Gamma_{out}\ |^2}$$

In the test environment, from Equations 1-4 and 1-6 on page 6, you have

(1-14) $$G_A = |S_{21}|^2 \frac{1}{1-|S_{22}|^2}$$

Because the power available from the source is greater than the power input to the LNA network, $G_P > G_T$. The closer the two gains are, the better the input matching is. Similarly, because the power available from the LNA network is greater than the power delivered to the load, $G_A > G_T$. The closer the two gains are, the better the output matching is.

Maximum Unilateral Transducer Power Gain

Maximum unilateral transducer power gain, G_{umx}, is the transducer power gain when you assume that the reverse coupling of the LNA, S_{12}, is zero, and the source and load impedances are conjugately matched to the LNA. That is $\Gamma_S = S_{11}$ and $\Gamma_L = S_{22}$. If $S_{12} = 0$, from Equations 1-2 and 1-3, the input and output reflection coefficients are $\Gamma_{in} = S_{11}$ and $\Gamma_{out} = S_{22}$. Thus from Equation 1-9 on page 7, you get Equation 1-15.

(1-15) $$G_{umx} = \frac{1}{1-|S_{11}|^2}|S_{21}|^2\frac{1}{1-|S_{22}|^2}$$

Maximum Transducer Power Gain

Maximum transducer power gain, $G_{\max}$, is the simultaneous conjugate matching power gain when both the input and output are conjugately matched. $\Gamma_S = \Gamma_{in}$ and $\Gamma_L = \Gamma_{out}$.When the reverse coupling, S_{12}, is small, G_{umx} is close to $G_{\max}$.

$$G_{\max} = \frac{|S_{21}|}{|S_{12}|}\left(K - \sqrt{K^2 - 1}\right)$$

The stability factor, *K*, is defined in “Stability” on page 12.

Maximum Stability Gain

Maximum stability gain, G_{msg}, is the maximum of $G_{\max}$ when the stability condition, $K > 1$, is still satisfied.

$$G_{msg} = \frac{|S_{21}|}{|S_{12}|}$$

Power Gain Circle

Power gain circle, *GPC*. From Equations 1-2 and 1-11, you can see that G_P is solely a function of the load reflection ΓL. Thus you can draw power gain contours on the Smith chart of Γ_L. The location for the peak of the contour corresponds to ΓL producing the maximum G_P. You can move the peak location by changing the design of the output matching network. The best location for the contour peak is at the center of the Smith chart, where $\Gamma_L = 0$.

Available Gain Circle

Available gain circle, *GAC*. From Equations 1-3 and 1-13, you can see that G_A is solely a function of the source reflection Γ_S. Thus you can draw available gain contours on the Smith chart of Γ_S. The location for the peak of the contour corresponds to Γ_S producing the maximum G_A. You can move the peak location by changing the design of the input matching network. The best location for the contour peak is at the center of the Smith chart, where $\Gamma_S = 0$.

Noise in LNAs

According to the Friis equation for cascaded stages, the overall noise figure is mainly determined by the first amplification stage, provided that it has sufficient gain. You achieve low noise performance by carefully selecting the low noise transistor, DC biasing point, and noise-matching at the input.

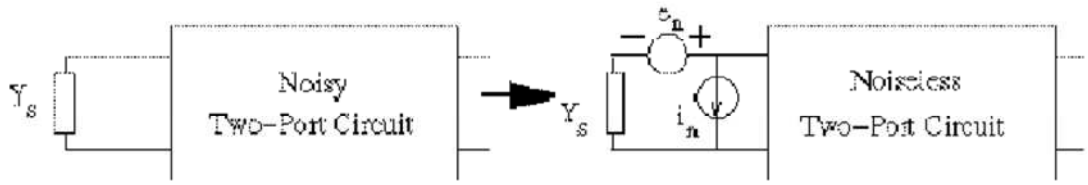
The noise performance is characterized by noise factor, *F*, which is defined as the ratio between the input signal-to-noise ratio and the output signal-to-noise ratio

(1-16) $$F = \frac{\left(\frac{S}{N}\right)_{out}}{\left(\frac{S}{N}\right)_{in}} = \frac{G_A N_{in}}{N_{out}}$$

where N_{in} is the available noise power from the source, $N_{in} = kT\Delta f$, and N_{out} is the available noise power to the load.

According to linear noise theory, you can model the noise of a noisy two-port system with two equivalent input noise generators: a series voltage source and a shunt current source. This is shown in Figure 1-4.

Figure 1-4 Two-Port Noise Theory



The two noise sources are related by the correlation admittance. The noise factor, *F*, is described by Equation 1-17.

(1-17) $$F = F_{\min} + \frac{R_n}{G_S}\left|Y_S - Y_{opt}\right|^2$$

where R_n is the equivalent noise resistance of the noisy two-port system

$$R_n = \frac{\overline{e_n^{\,2}}}{4kT\Delta f}$$

The source admittance is $Y_s = G_s + jB_s$, the optimum source admittance is $Y_{opt} = G_{opt} + jB_{opt}$, and the minimum noise factor is $F_{\min}$. The optimum source admittance Y_{opt}, the minimum noise factor $F_{\min}$, and R_n are solely determined by the two-port circuit itself.

From Equation 1-17, the noise factor, *F*, is a function of source admittance, *YS*. Thus you can plot the noise factor contour on the source admittance Smith chart. Where $Y_s = Y_{opt}$, the center of the noise factor contour corresponds to $F_{\min}$. You can move the center of the source admittance Smith chart, Y_{opt}, by changing the input matching network design. The best choice is to move the center of the noise circles to the center of the Smith chart so that $Y_{opt} = R_s$.

You perform noise-matching by designing the input-matching network so that the center of the LNA's noise circle (NC) moves to the center of the source admittance Smith chart. However, as previously mentioned, to maximize the available gain, you should also move the center of the available gain circle (GAC) to the center of the source admittance Smith chart. These two goals might turn out to be contradictory, in which case you must compromise so that the centers of the noise circles and the gain circle are both close to the Smith chart center.

Several design topologies are available to help you to balance noise and gain matching. The topologies include shunt-series feedback, common-gate and inductively-degenerated common-source [3] [4].

Input and Output Impedance Matching

The input and output are each connected to the LNA with filters whose performance relies heavily on the terminal impedance. Furthermore, input and output matching to the source and load can maximize the gain. Input and output impedance matching is characterized by the input and output return loss.

$$20\log|\Gamma_{in}| = 20\log|S_{11}|$$

$$20\log|\Gamma_{out}| = 20\log|S_{22}|$$

You can also characterize the LNA's input and output impedance matching by the voltage standing wave ratio (VSWR):

$$VSWR_{in} = \frac{1+|\Gamma_{in}|}{1-|\Gamma_{in}|} = \frac{1+|S_{11}|}{1-|S_{11}|}$$

$$VSWR_{out} = \frac{1+|\Gamma_{out}|}{1-|\Gamma_{out}|} = \frac{1+|S_{22}|}{1-|S_{22}|}$$

Your primary design goals are to minimize the return loss and make the VSWR close to 1

Reverse Isolation

The reverse isolation of an LNA determines the amount of the LO signal that leaks from the mixer to the antenna. LO signal leakage arises from capacitive paths, substrate coupling, and bond wire coupling. In a heterodyne receiver, because the LO signal is ωif away from the RF signal, the image-reject and band-select filters and the LNA can all work together to significantly attenuate the LO signal leaked from the VCO.

Insufficient isolation can cause feedback and even instability. Reverse isolation is characterized by the reverse transducer gain power, $|S_{12}|^2$. You should minimize the reverse transducer gain power as much as possible.

Stability

In the presence of feedback paths from the output to the input, the circuit might become unstable for certain combinations of source and load impedances. An LNA design that is normally stable might oscillate at the extremes of the manufacturing or voltage variations, and perhaps at unexpectedly high or low frequencies.

The Stern stability factor characterizes circuit stability as in Equation 1-18.

(1-18)
$$K = \frac{1 + |\Delta|^2 - |S_{11}|^2 - |S_{22}{}^2|}{2|S_{21}||S_{12}|}$$

where $\Delta = S_{11}S_{22} - S_{12}S_{21}$

When $K > 1$ and $\Delta < 1$, the circuit is unconditionally stable. That is, the circle does not oscillate with any combination of source and load impedances. You should perform the stability evaluation for the S parameters over a wide frequency range to ensure that K remains greater than one at all frequencies.

As the coupling (S_{12}) decreases, that is as reverse isolation increases, stability improves. You might use techniques such as resistive loading and neutralization to improve stability for an LNA. [2].

Equation 1-18 is valid for small-signal stability. If the circuit is unconditionally stable under small-signal conditions, the circuit is less likely to be unstable when the input signal is large.

Aside from the two metrics K and Δ, you can also use the source and load stability circles to check for LNA stability. The input stability circle draws the circle $\Gamma_{out} = 1$ on the Smith chart of Γ_S. The output stability circle draws the circle $\Gamma_{in} = 1$ on the Smith chart of Γ .

The non-stable regions of the two circles should be far away from the center of the Smith chart. In fact, it is better if the non-stable regions are located outside the Smith chart circles.

Linearity

Nonlinear LNAs can corrupt the RF input signal and cause the types of distortion [3] described in Table 1-2.

Table 1-2 RF Input Signal Distortion In Nonlinear LNAs

Harmonic Distortion	A nonlinear LNA might generate output with high order harmonic when the input is a pure sinusoid.
Cross Modulation	A nonlinear LNA might transfer the modulation on one channel's carrier to another channel's carrier.
Blocking	In a nonlinear LNA, one large signal on one channel might desensitize the amplification of a small signal on neighboring channels. Many RF receivers must be able to withstand blocking signals 60 to 70 dB greater than the wanted signal.
Gain Compression	In a nonlinear LNA, gain decreases as input power increases because of transistor saturation.
Intermodulation	In a nonlinear LNA, two large signals (interferers) on two adjacen channels might generate a 3rd-order intermodulation component that falls into the bandwidth of neighboring channels.

LNA linearity is characterized by the 1 dB compression point (P1 dB) and the 3rd order interception point (IP3).

Example Measurements Using SpectreRF

To test an LNA, place it into the testbenches described in page 6. You can then perform various analyses to determine the gain, noise, power, linearity, stability, and matching performance for the LNA.

This section demonstrates how to set up the required SpectreRF analyses and to make measurements on LNAs. It explains how to extract the design parameters from the data generated by the analyses.

The workshop begins by bringing up the Cadence Design Framework II environment for a full view of the reference design:

To prepare for the workshop,

Action P-1: cd into the **./rfworkshop** directory.

Action P-2: Run the tool **icfb&.**

Action P-3: In the CIW window, select **Tools — Library Manager….**

Lab 1: Small Signal Gain (SP)

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

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

Action 1-2: 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….** After these actions, the Edit Object Properties window for the port cell comes up. Set up the port properties as follows:

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

Action 1-3: Set the source type of **PORT load** to DC.

Action 1-4: Check and save the schematic.

Action 1-5: In the Virtuoso Schematic Editing window, select **Tools — Analog Environment**.

Action 1-6: (Optional) Choose **Session — Load State** in the Virtuoso Analog Design Environment window, select **Cellview** in **Load State Option** and load state "**Lab1_sp**", then skip to Action 1-10.

Action 1-7: In the Virtuoso Analog Design Environment window, select **Analyses — Choose….**

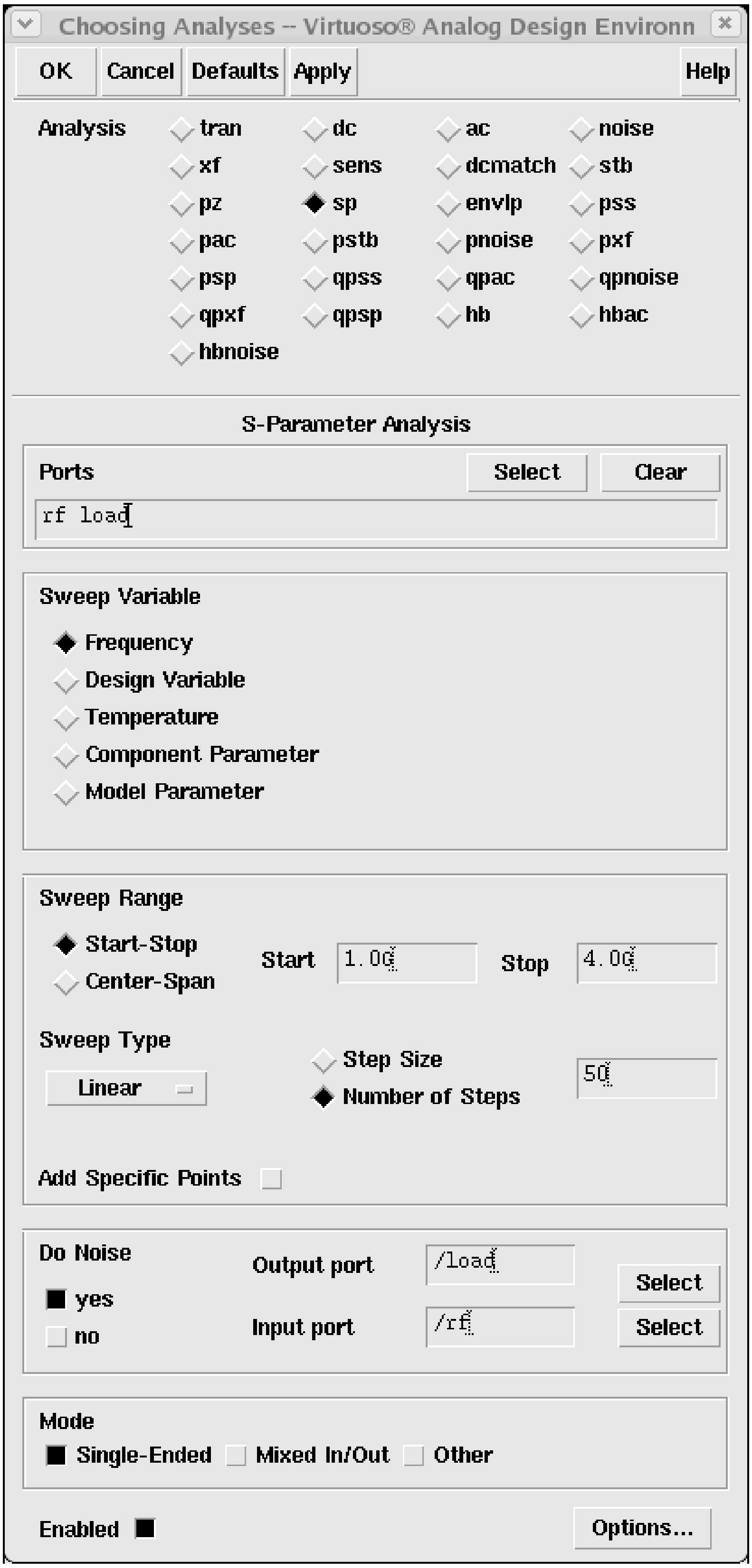
Action 1-8: In the Choosing Analyses window, select **sp** in the **Analysis** field of the window.

Action 1-9: In the S-Parameter Analysis window, in the **Ports** field, click **Select**. 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, click the **ESC** key.

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

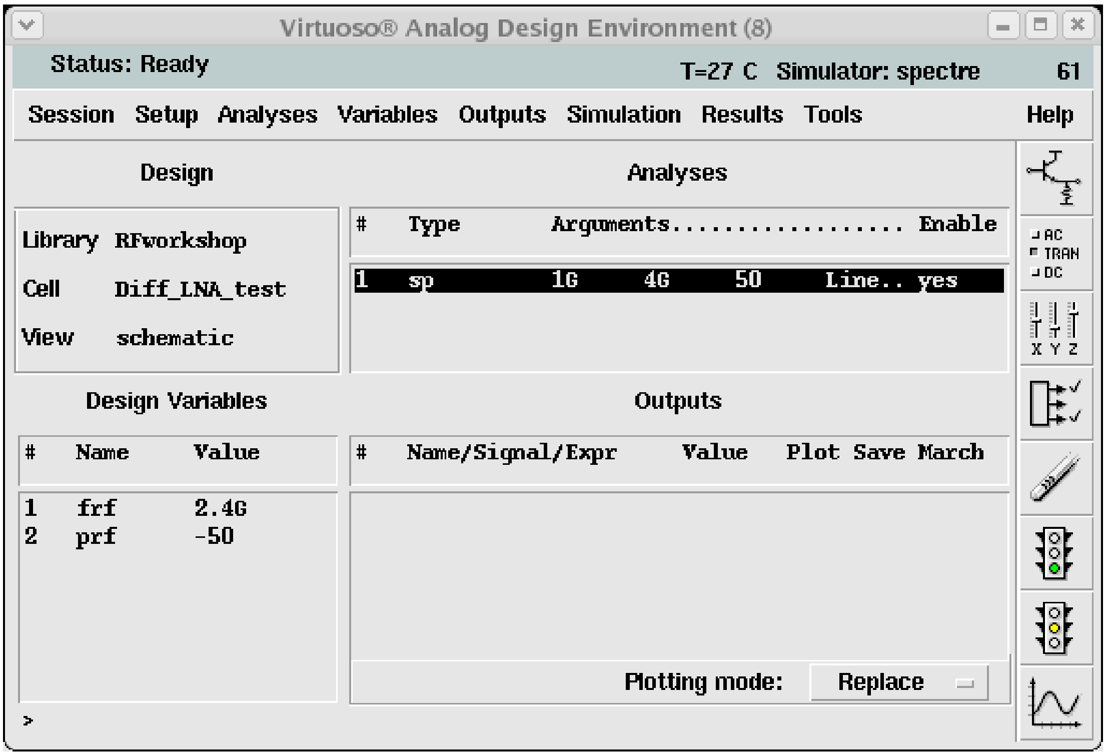
In the **Sweep Range** field, select **Start-Stop**, set **Start** to 1.0 G and **Stop** to 4.0G, set **Sweep Type** to Linear, select **Number of Steps** and set that to 50. In the **Do Noise** field, select **yes**, set **Output port** to /load and **Input port** to /rf.

After these actions, the form looks like this:


Choosing Analyses -- Virtuoso® Analog Design Environn
OK
Cancel
Defaults
Apply
Help
Analysis
tran
dc
ac
noise
xf
sens
dcmatch
stb
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
S-Parameter Analysis
Ports
Select
Clear
rf load
Sweep Variable
Frequency
Design Variable
Temperature
Component Parameter
Model Parameter
Sweep Range
Start-Stop
Center-Span
Start
1.0G
Stop
4.0G
Sweep Type
Linear
Step Size
Number of Steps
50
Add Specific Points
Do Noise
yes
no
Output port
/load
Input port
/rf
Select
Select
Mode
Single-Ended
Mixed In/Out
Other
Enabled
Options...

Note: Selecting **yes** under **Do Noise** sets up the Noise analysis. You can obtain small signal noise when the input power level is low and the circuits are considered linear.

The Virtuoso Analog Design Environment window looks like this:



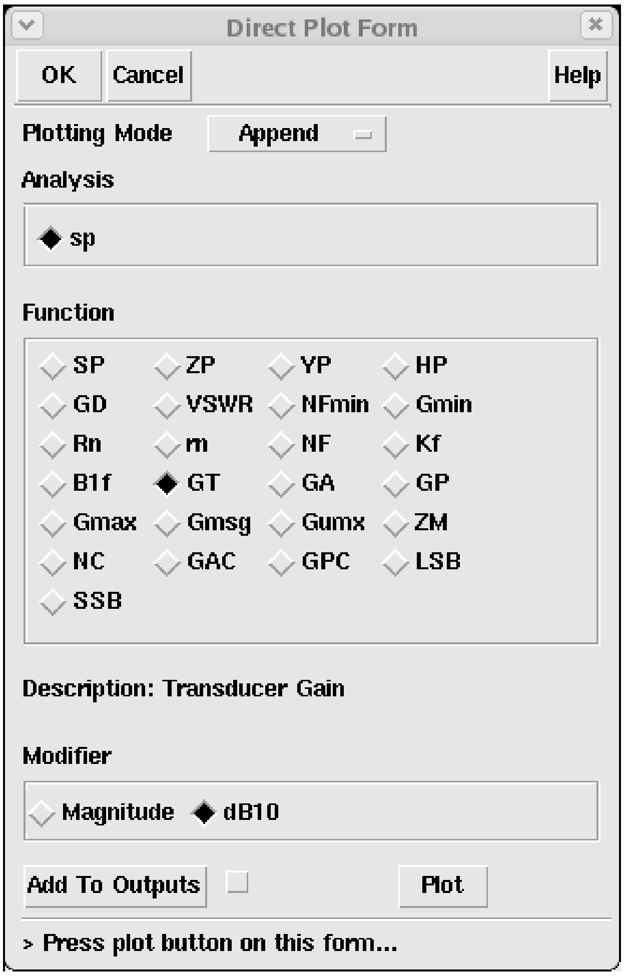
Action 1-10: Choose **Simulation — Netlist and Run** to start the simulation or click the **Netlist and Run** icon in the Virtuoso Analog Design Environment window.

Action 1-11: In the Virtuoso Analog Design Environment window, select **Results — Direct Plot — Main Form….**

A waveform window and a Direct Plot Form window open.

Action 1-12: In the 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.**

After these actions, the form looks like this:

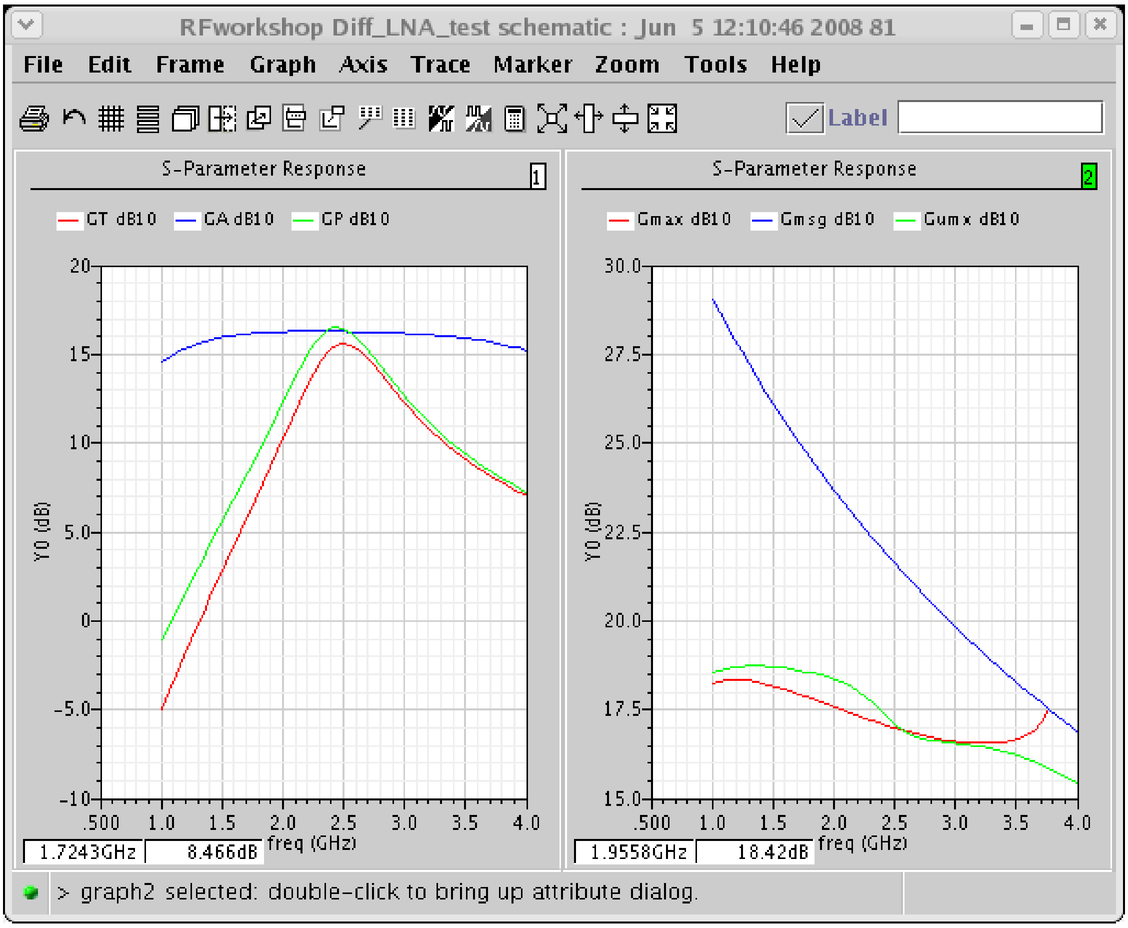


Action 1-13: Click **Plot**. In the **Function** field, select **GA** (for Available Power Gain). Click **Plot** again. In the **Function** field, select **GP** (for Operating Power Gain). Click **Plot** once more.

These actions plot GT, GA, and GP in one waveform window. G_T is the smallest gain. This is expected from the discussion about "Gain" on page 7. The power gain G_P is closer to the transducer gain G_T than the available gain G_A which means the input matching network is properly designed. That is, *S11* is close to zero.

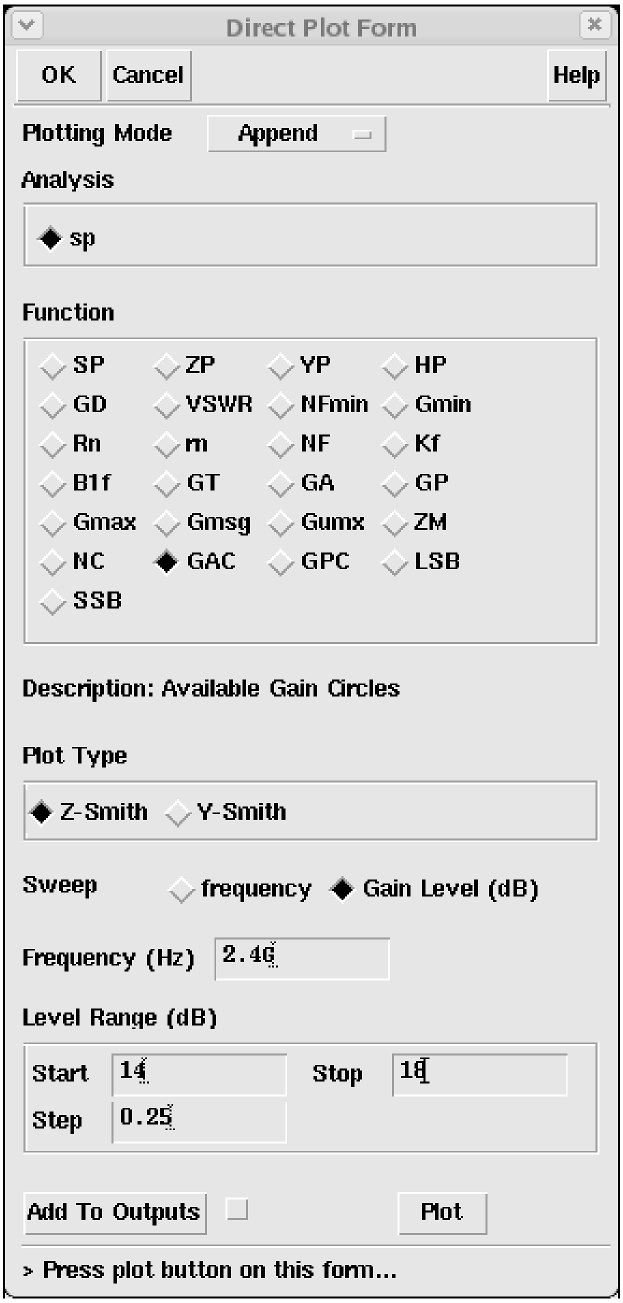
Action 1-14: In the waveform window, click **New Subwindow**. Go back to the Direct Plot Form. Select **Gmax** (for maximum Transducer Power Gain) and click **Plot**. In the Direct Plot Form window, set **Plotting Mode** to **Append**. In the **Function** field, select **Gmsg** (for Maximum Stability Gain). Click **Plot**. Select **Gumx** (for maximum Unilateral Transducer Power Gain), and click **Plot** again.

You get the following waveforms:



In the above plot, G_{umx} is very close to $G_{\max}$ which means the reverse coupling, S_{12}, is small. Obviously G_{msg} is the largest of the six gains plotted.

Action 1-15: Close the waveform window, and go back to the Direct Plot Form. In the **Function** field, select **GAC** (Available Gain Circles). In **Plot Type** field, choose **Z-Smith**, Sweep **Gain Level (dB)** at Frequency 2.4GHz from 14 to 18dB with steps set to 0.25 dB.

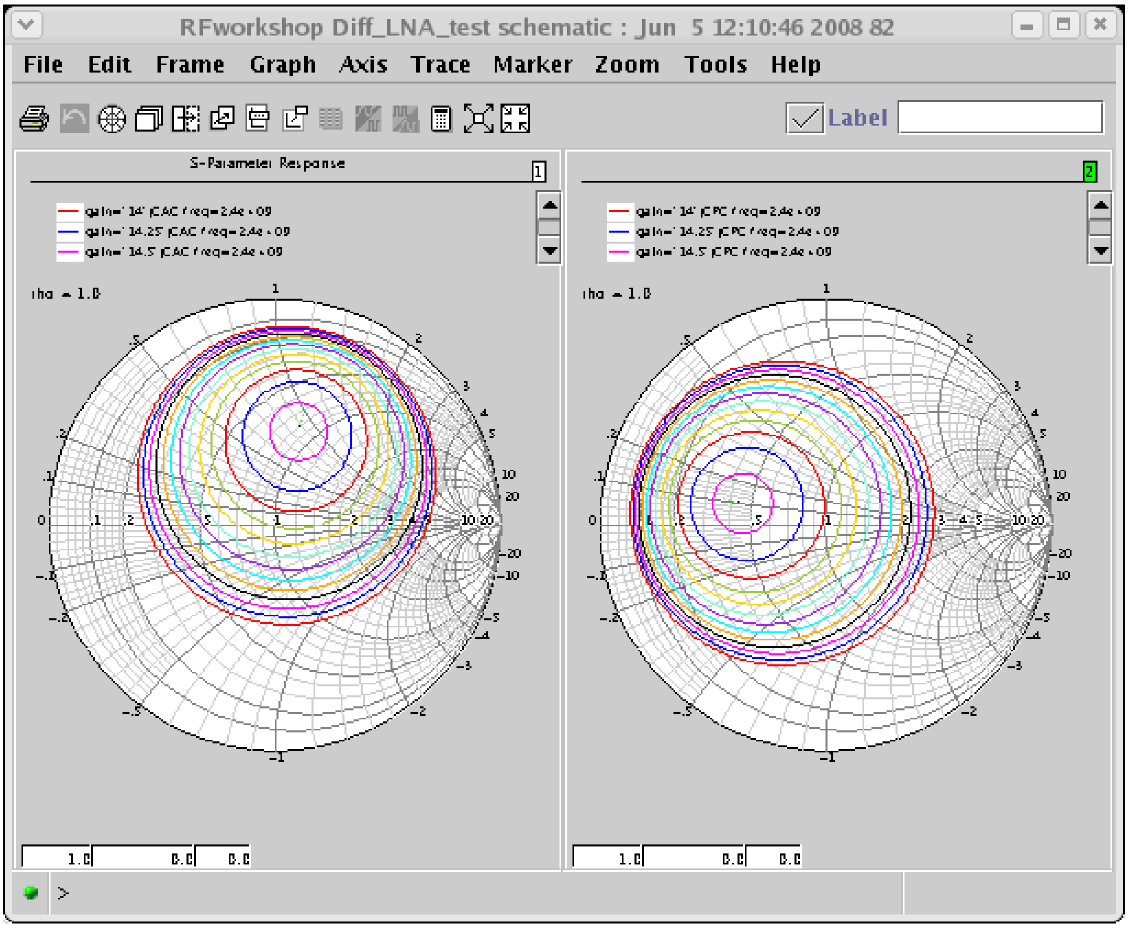


Action 1-16: Click **plot**.

Action 1-17: In the waveform window, click **New Subwindow**.

Action 1-18: Go back to the Direct Plot Form. In the **Function** field, select **GPC** (Power Gain Circles). Click **Plot**.

The waveforms look like this:

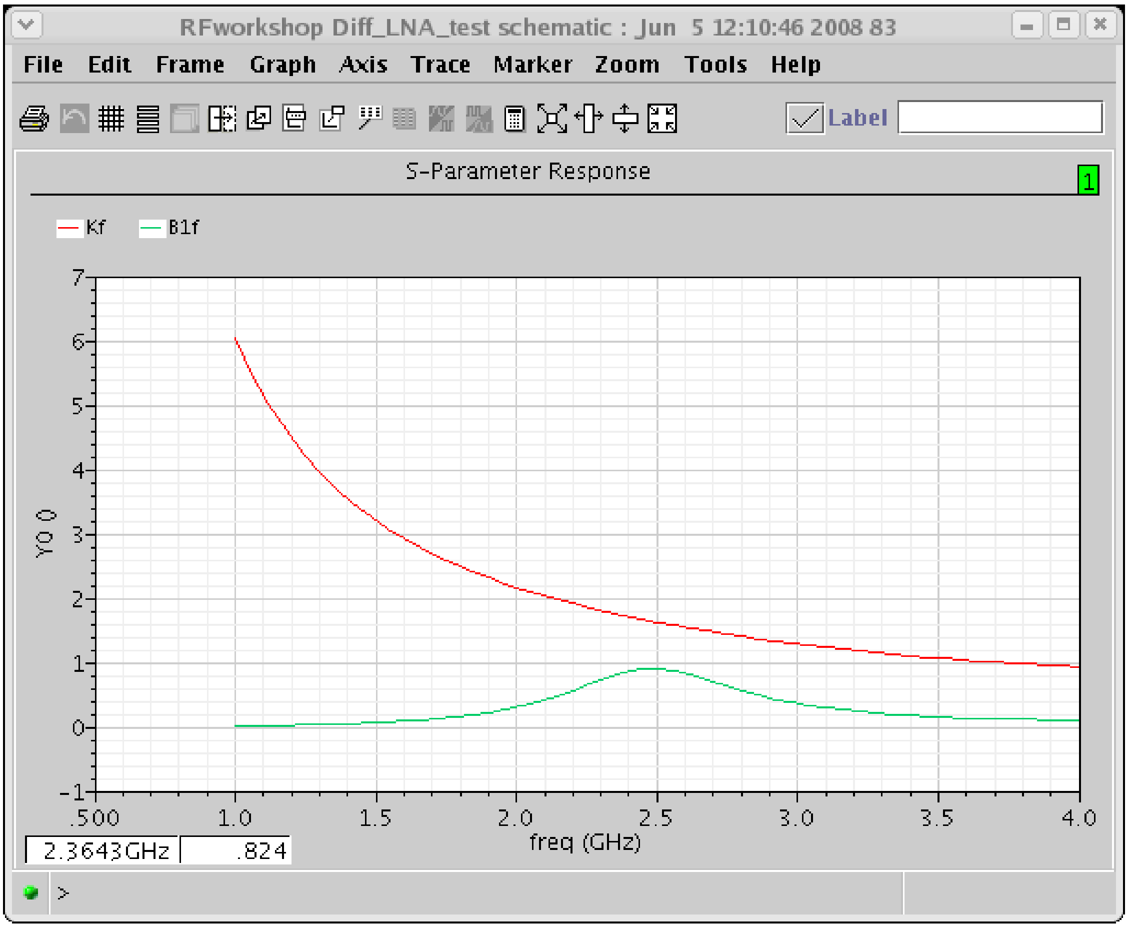


The contours in the above figure are plotted for freq=2.4GHZ. In the GPC plot, $G_P \approx 16.5$ at $\Gamma_L = 0$. In the GAC plot, $G_A \approx 16.25$ at $\Gamma_S = 0$. These results match the results in G_P / G_A on page 18. As has been discussed, the centers of the two contours are located close to the centers of the Smith charts.

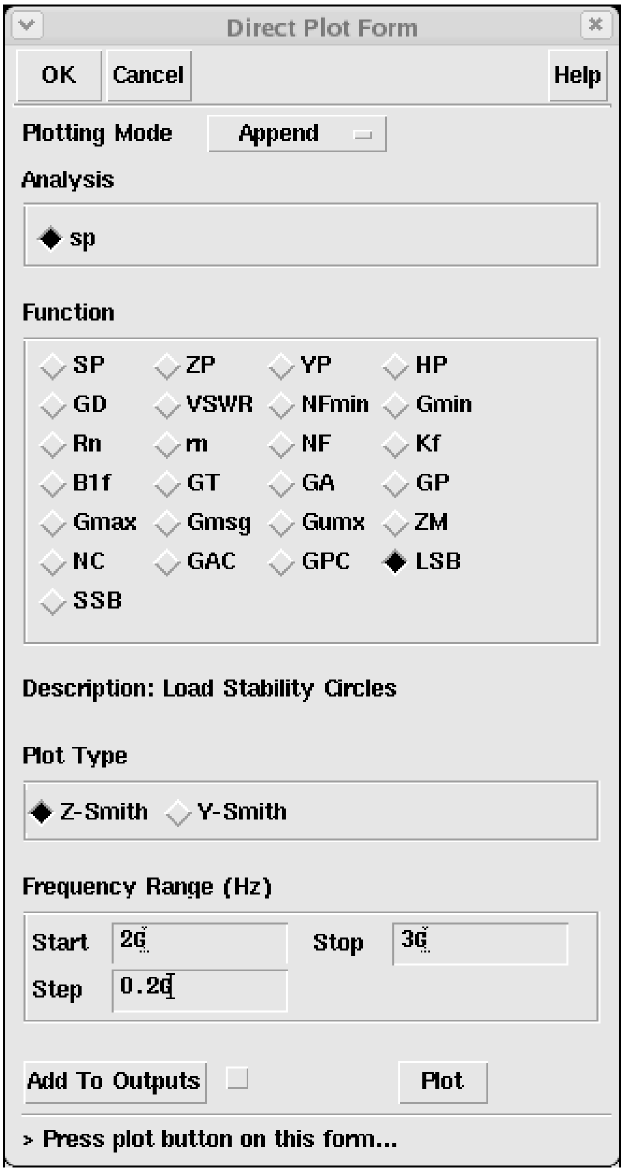
Action 1-19: Close the waveform window, and go back to the **Direct Plot Window**. In the Direct Plot Form window, set **Plotting Mode** to **Append**. In the **Function** field, choose **Kf**. Click **Plot**.

Action 1-20: In the **Function** field, choose **B1f**. Click **Plot**.

The Stability Curves are plotted:



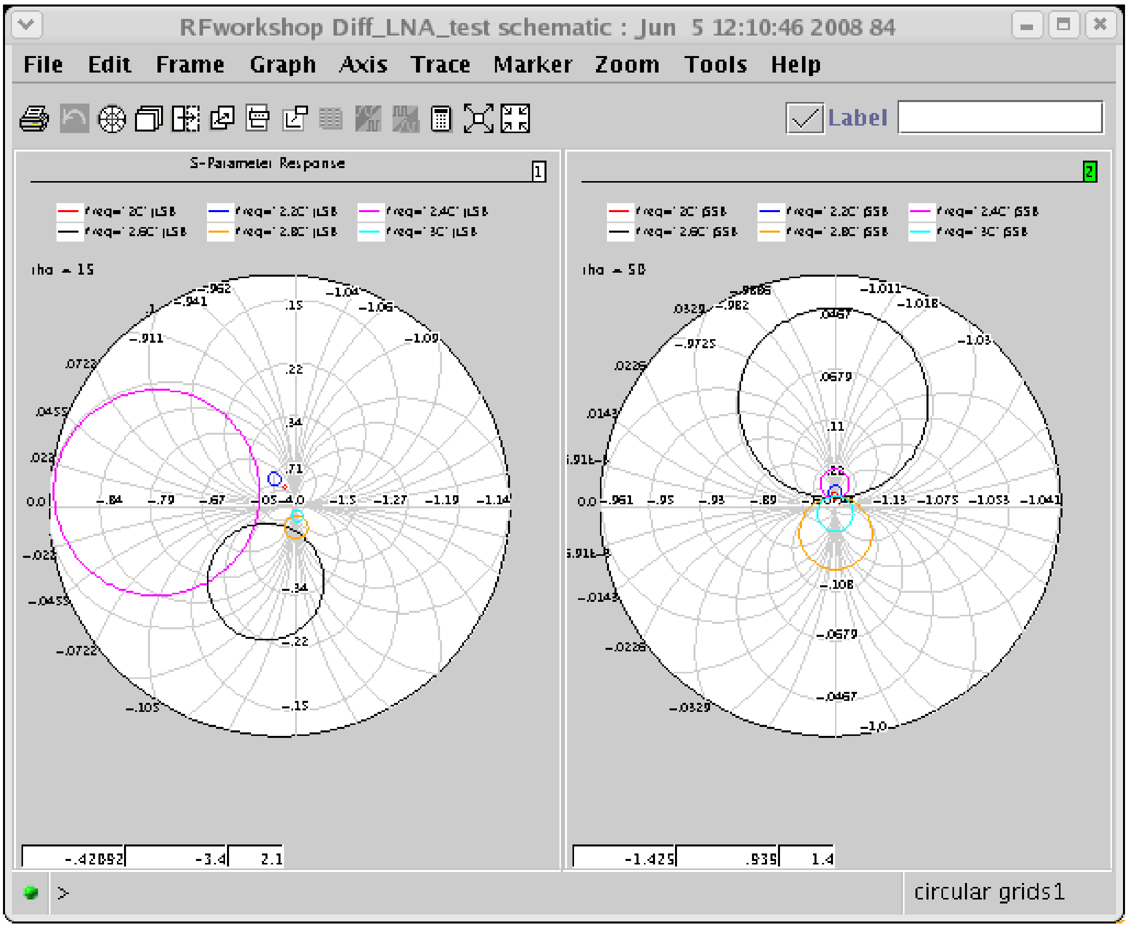
Action 1-21: Close the waveform window, go back to the Direct Plot Form window. In the **Function** field, choose **LSB** (Load Stability Circles)**.** In **Plot Type**, choose Z-Smith. Specify Frequency Range from 2G to 3G with the step set to 0.2G. Click **Plot**.



Action 1-22: In the waveform window, click **New Subwindow**.

Action 1-23: Go back to the Direct Plot Form window. In the **Function** field, select **SSB** (Source Stability Circles). Click **Plot**.

The Load Stability Circles and Source Stability Circle are plotted:



Action 1-24: Close the waveform window, and go back to the Direct Plot Form window. In the Direct Plot Form window, set **Plotting Mode** to **Append**. In the Direct Plot Form window, select **NF (Noise Figure**) in the **Function** field. In the **Modifier** field, select **dB10**. Click **Plot**.

Action 1-25: In the waveform window, click **New Subwindow**.

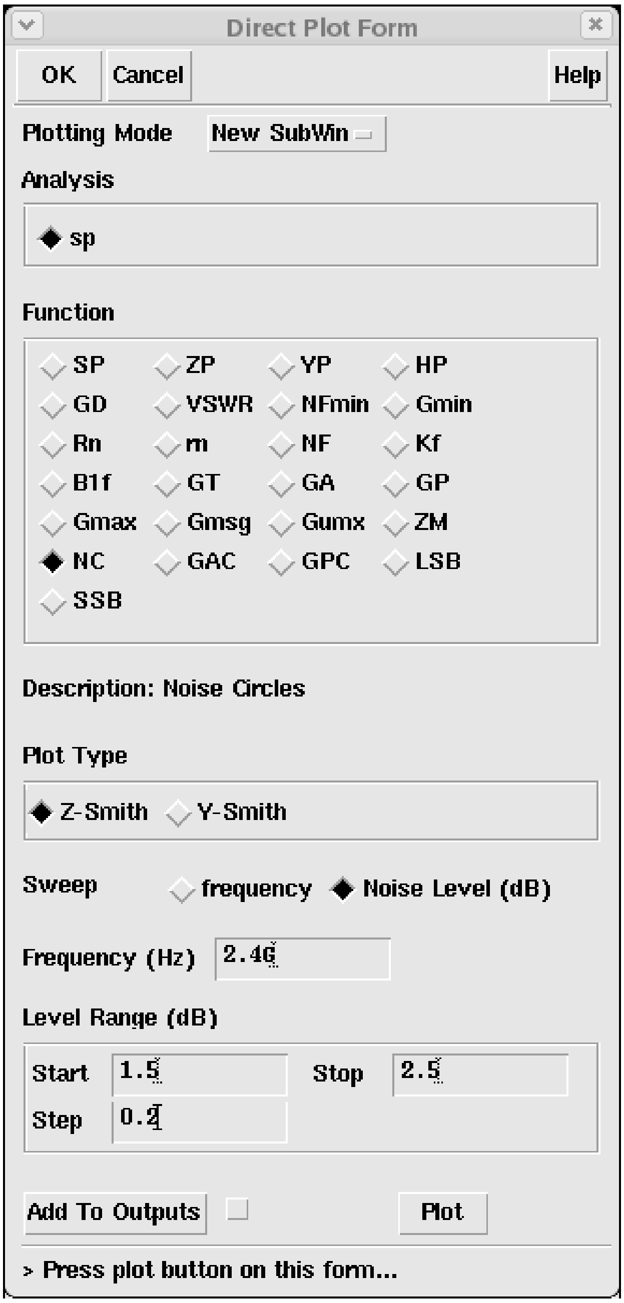
Action 1-26: In the **function** field, choose **NC** (Noise Circles). In the **Plot type** field, choose **Z-Smith**. Sweep Noise Level at Frequency 2.4G Hz starting from 1.5 to 2.5 dB with steps set to 0.2 dB.

Note: You can perform small signal noise simulation using either the SP or the Noise analyses. The Noise analysis provides only the noise figure, *NF*. The SP analysis provides:

- $NF_{\min}$, minimum noise figure
- R_S , noise resistance
- $G_{\min}$, optimum noise reflection coefficient
- Y_{opt} , optimum source admittance which is related to $G_{\min}$ as shown

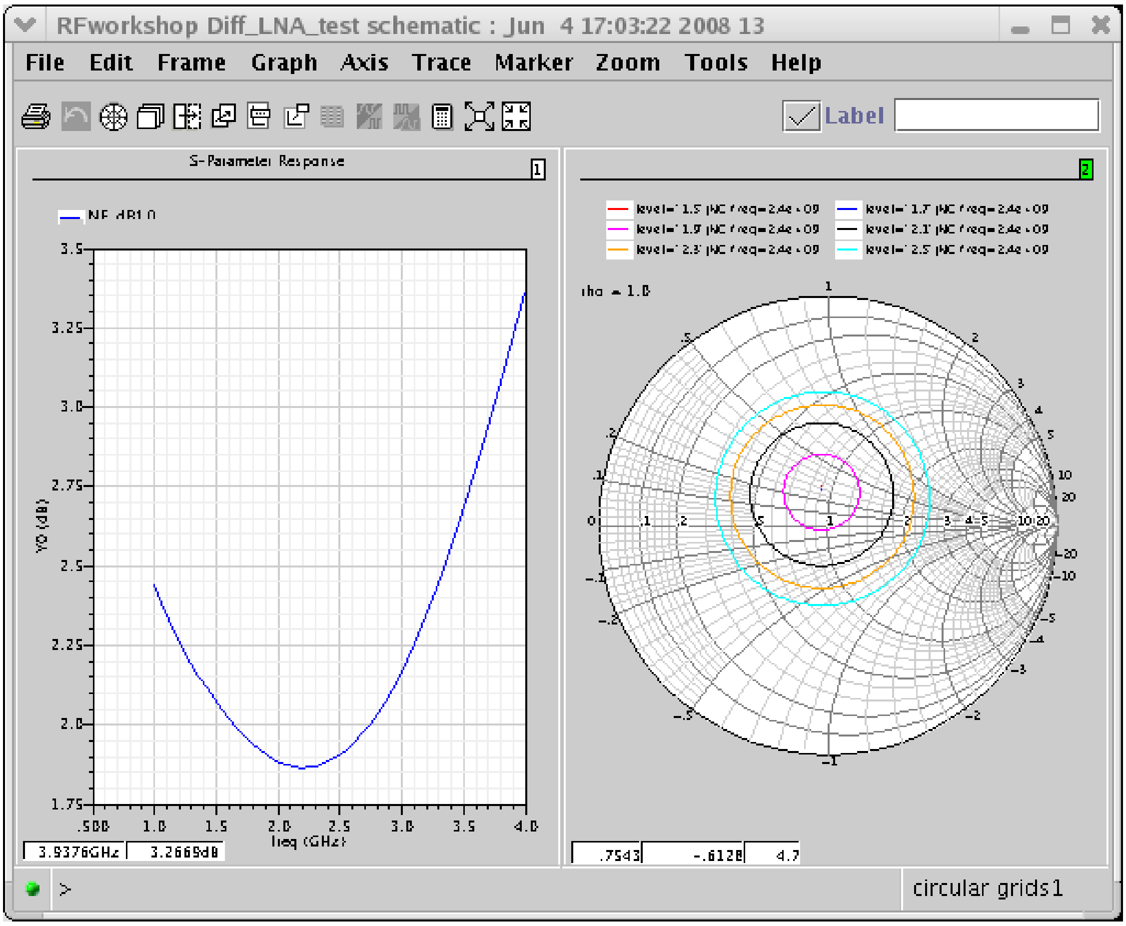
in the equation.

$$G_{\min} = \frac{Y_S - Y_{opt}}{Y_S + Y_{opt}}$$



Action 1-27: Click **Plot**.

You get the following plot:

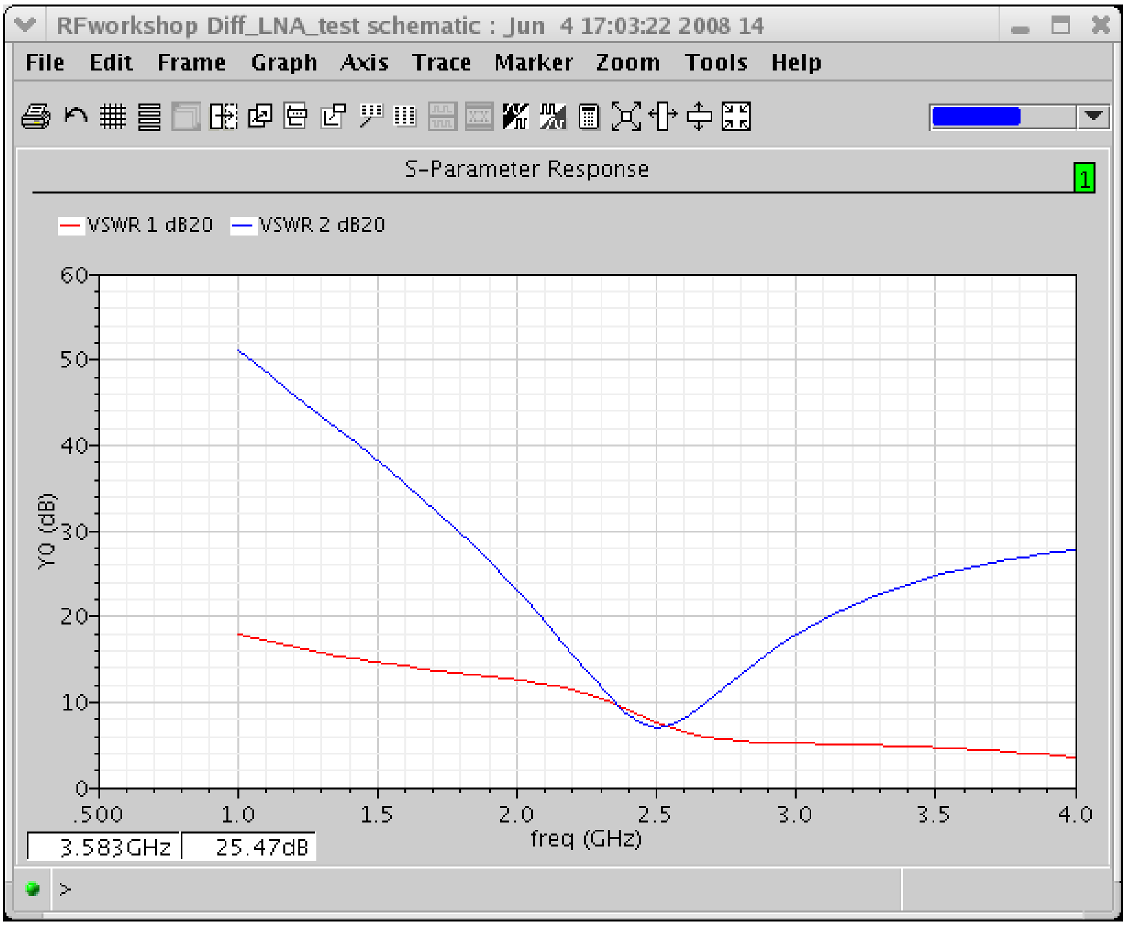


In the above figure, the noise circle, *NC*, draws the *NF* on the Smith chart of the source reflection coefficient, Γ_S. The result in the NC plot where $\Gamma_S = 0$ and *NF* = 1.9 dB matches the result in the *NF* plot. The center of the *NC* corresponds to Γ_S (that is, $G_{\min}$) which generates $NF_{\min}$. The optimum location for the center of the noise circle is at the center of the Smith chart. However it is hard to center both the available gain circle, *GAC*, and the noise circle, *NC*, in the Smith chart.

When you design an LNA, plot *NC*, *GAC*, and the source stability circle, *SSB*, together in the same plot. Use this plot to trade off the gain, noise, and stability for the input matching network design.

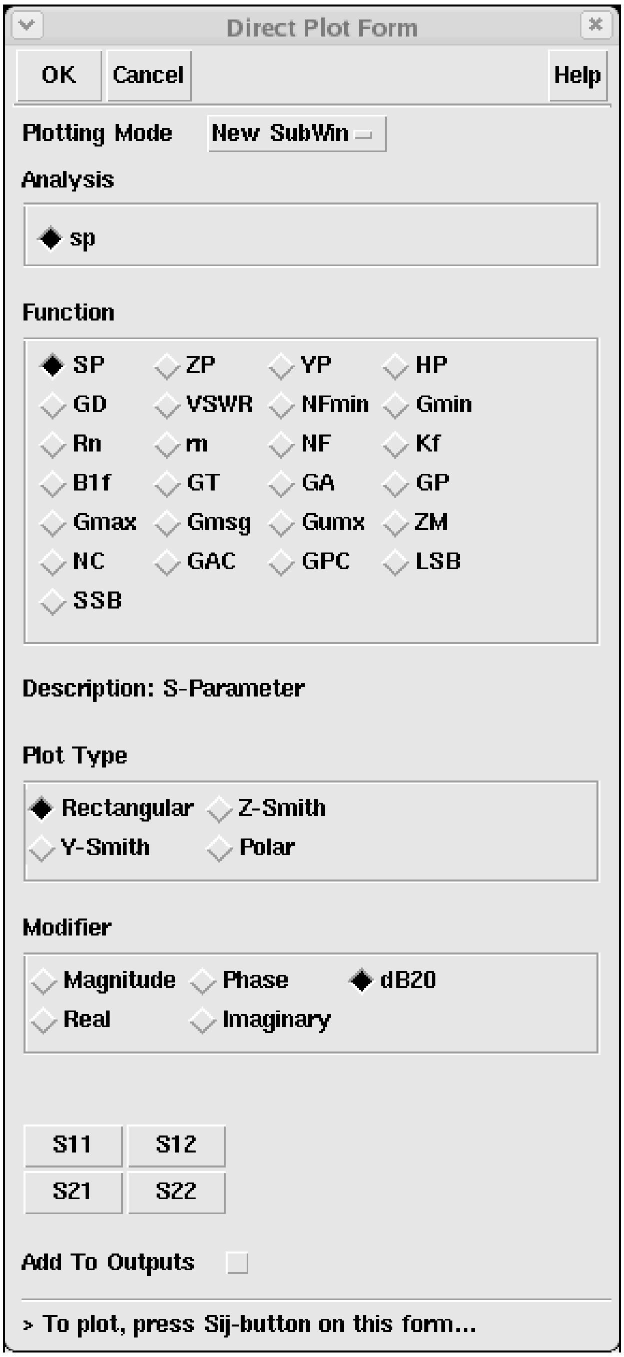
Action 1-28: Close the waveform window and go back to the Direct Plot Form window. In the Direct Plot Form window, set **Plotting Mode** to **Append**. In the **function** field, choose **VSWR** (Voltage standing-wave ratio). In the **Modifier** field, select **dB20**. Click **VSWR1**, then **VSWR2**.

You get the following waveforms:

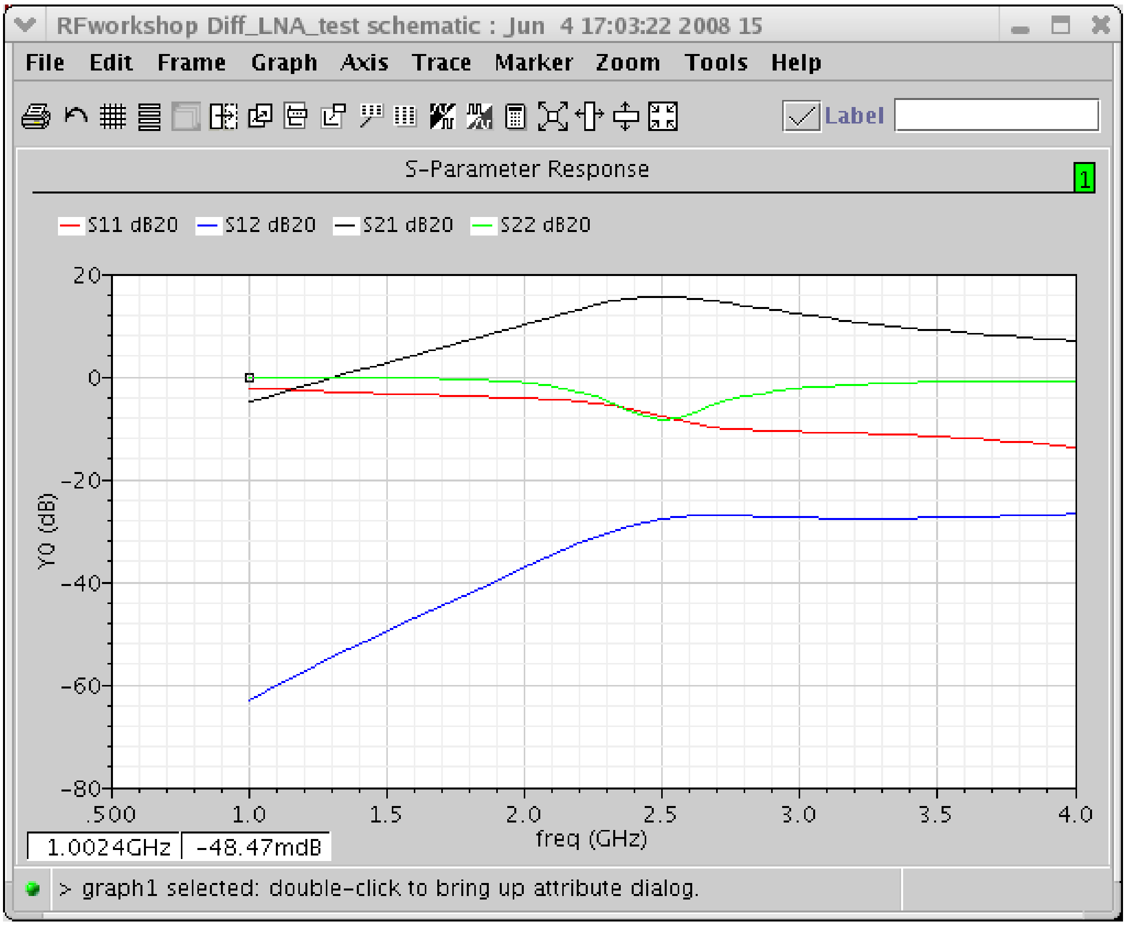


Action 1-29: Close the waveform window and go back to the Direct Plot Form. In the **function** field, choose **SP**. In the **Plot Type** field, choose **Rectangular**. In the **Modifier** field, select **dB20**. Click **S11, S12, S21**, then **S22**.

After these actions, the form looks like this:



You get the following waveforms:



Action 1-30: Close the waveform window and click **Cancel** in the Direct Plot Form.

Lab 2: 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, 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.

Action 2-1: If it is not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action 2-2: 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)

Action 2-3: Check and save the schematic.

Action 2-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools — Analog** Environment command.

Action 2-5: (Optional) Choose **Session — Load State**, select **Cellview** in **Load State Option** and load state “**Lab2_Pnoise**” and skip to Action 2-15.

Action 2-6: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose…**

Action 2-7: In the Choosing Analyses window, select **pss** in the **Analysis** field of the window.

Action 2-8: In the pss analyses window, click **Auto Calculate**.

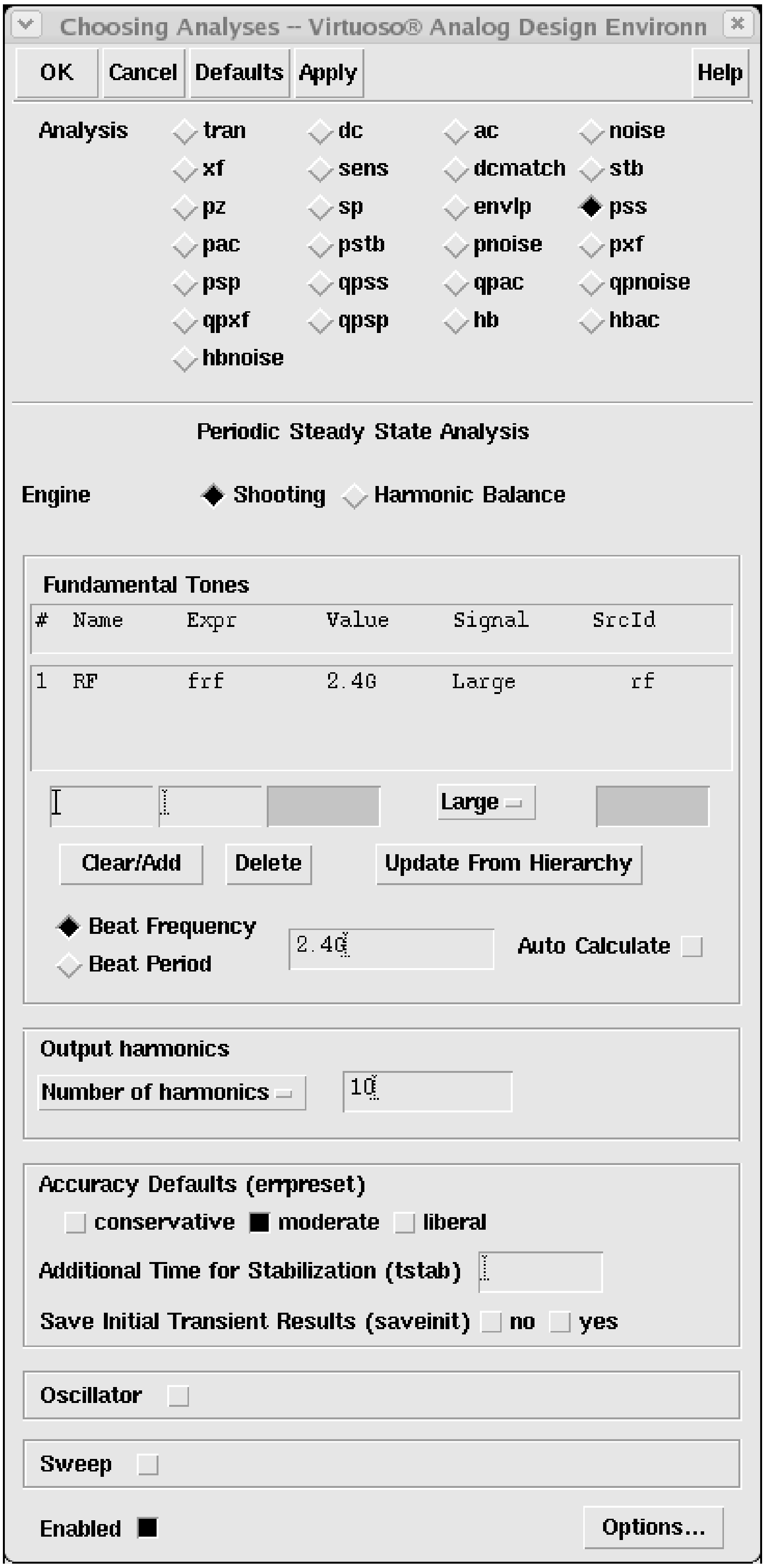
This automatically calculates either the beat frequency or beat period of the circuit. If the circuit contains frequency dividers or the input sources do not come from **analogLib**, it might be necessary to manually calculate the beat frequency (or beat period).

Action 2-9: In the **Output Harmonics** field, set the cyclic to **Number of Harmonics** and set the number of harmonics to 10.

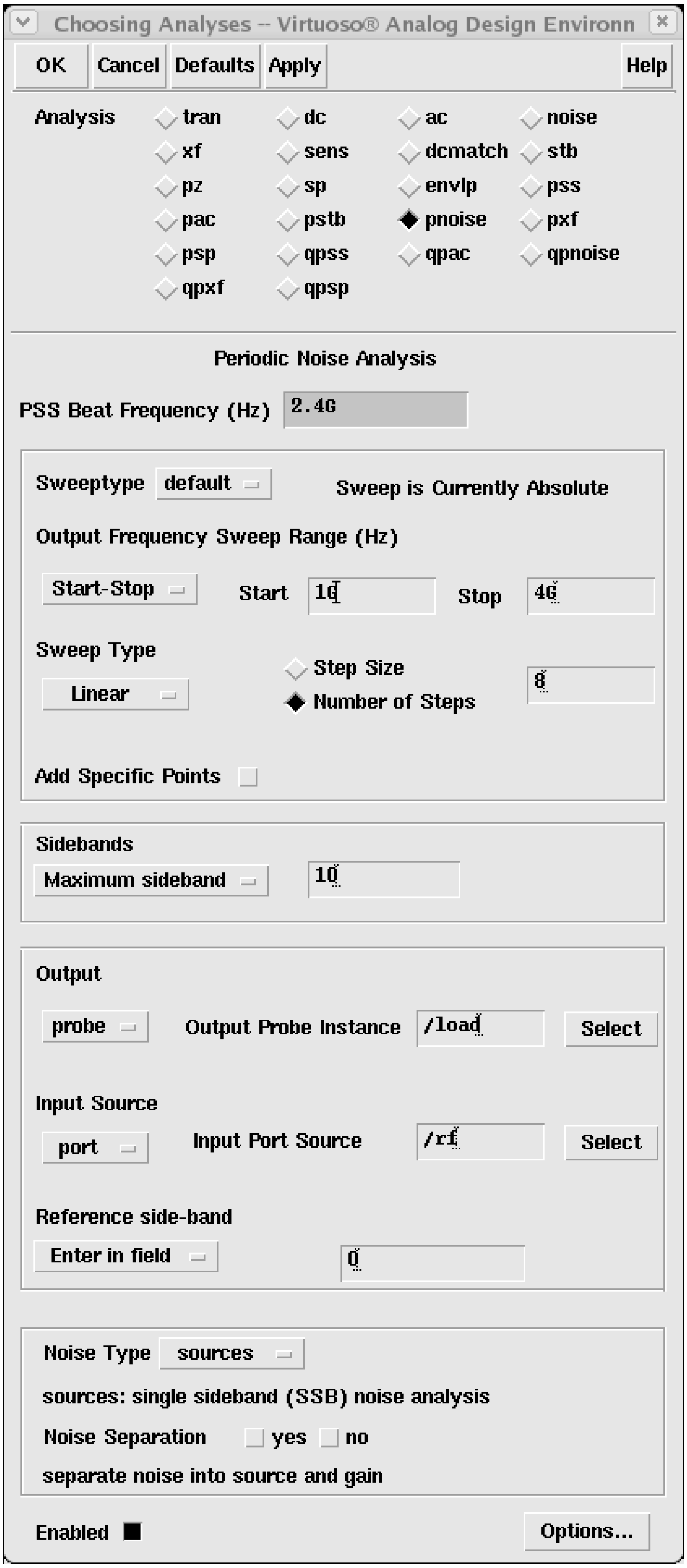
This allows us to look at, in the frequency domain results, 10 harmonics of the beat frequency.

Action 2-10: In the **Accuracy Defaults (errpreset)** field, select **moderate**.

The Choosing Analyses **—** PSS window looks like…

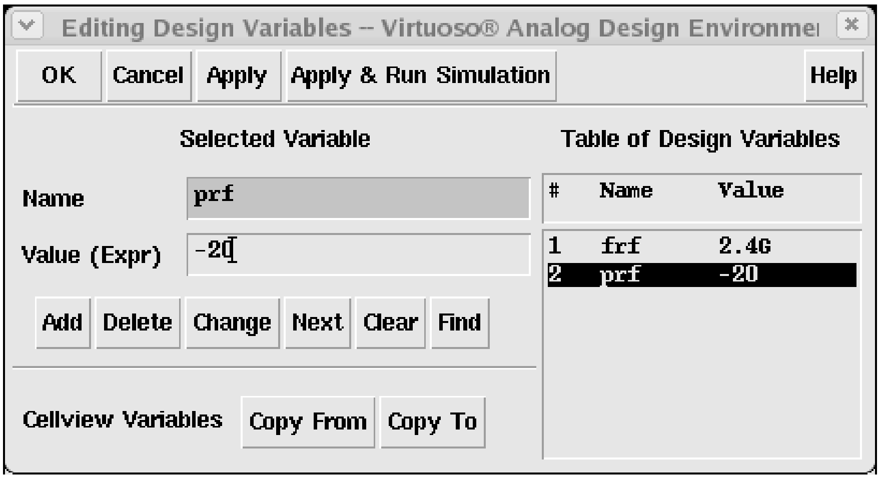

Choosing Analyses -- Virtuoso® Analog Design Environn
OK
Cancel
Defaults
Apply
Help
Analysis
tran
dc
ac
noise
xf
sens
dcmatch
stb
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Periodic Steady State Analysis
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId
1 RF frf 2.4G Large rf
Large
Clear/Add
Delete
Update From Hierarchy
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
Enabled
Options...

Action 2-11: Now you 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 are, until the point where the higher order harmonics are negligible. Spectre gives you a 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 looks like this:


Choosing Analyses -- Virtuoso® Analog Design Environn
OK
Cancel
Defaults
Apply
Help
Analysis
tran
dc
ac
noise
xf
sens
dcmatch
stb
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
Periodic Noise Analysis
PSS Beat Frequency (Hz)
2.4G
Sweeptype
default
Sweep is Currently Absolute
Output Frequency Sweep Range (Hz)
Start-Stop
Start
1G
Stop
4G
Sweep Type
Linear
Step Size
Number of Steps
8
Add Specific Points
Sidebands
Maximum sideband
10
Output
probe
Output Probe Instance
/load
Select
Input Source
port
Input Port Source
/rf
Select
Reference side-band
Enter in field
0
Noise Type
sources
sources: single sideband (SSB) noise analysis
Noise Separation
yes
no
separate noise into source and gain
Enabled
Options...

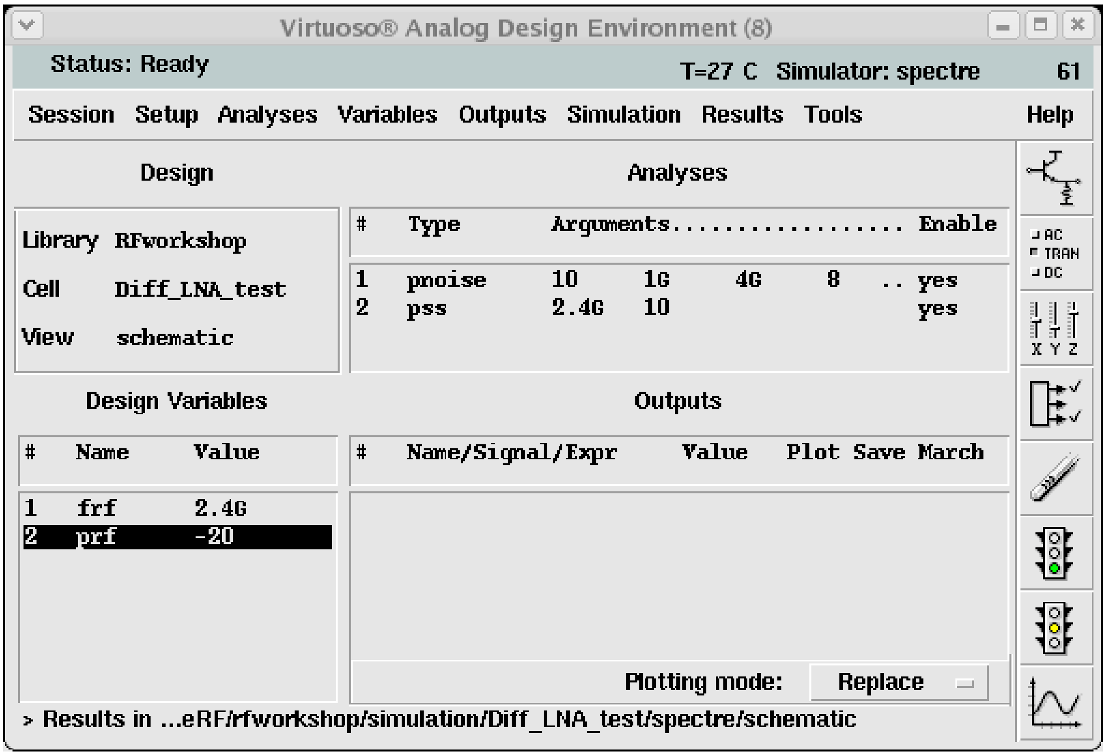
Action 2-12: Make sure **Enabled** is selected, and click **OK** in the Choosing Analyses form.

Action 2-13: In the Virtuoso Analog Design Environment window, double click prf in the field of Design Variables. Change the input power to -20.



Action 2-14: Click **Change**. Click OK to close the Editing Design Variables window.

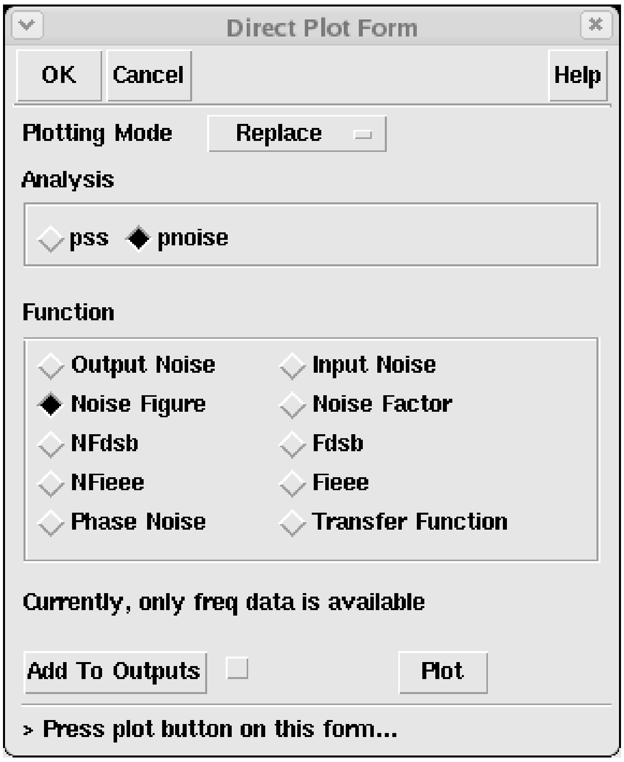
The Virtuoso Analog Environment looks like this:



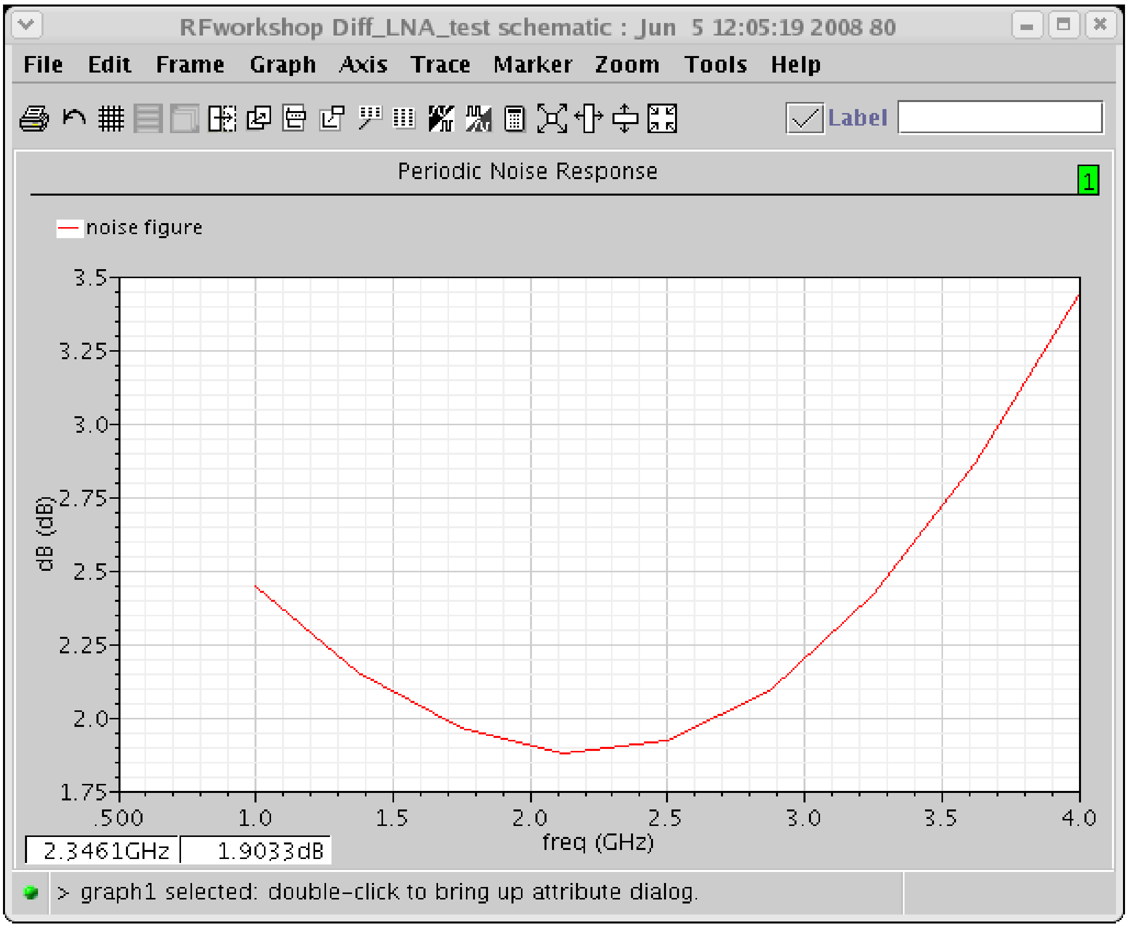
Action 2-15: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action 2-16: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 2-17: In the Direct Plot Form, select **pnoise**, and configure the form as follows:



Action 2-18: Click **Plot**.



The waveform window displays the noise figure. Comparing the above figure with the figure on page 27, the noise figure plot matches very well. The noise figure from Pnoise is slightly larger than the noise figure from SP because at Pin = -20 dBm, the LNA demonstrates very weak nonlinearity and noise as other high harmonics are convoluted.

Action 2-19: Close the waveform window. Click **Cancel** on the Direct Plot Form. Close the Virtuoso Analog Design Environment window.

Lab 3: Gain Compression and Total harmonic Distortion (Swept PSS)

A PSS analysis calculates the operating power gain, which 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 G_p, so the gain from PSS should match G_p when the input power level is low and nonlinearity is weak.

In the following actions, you perform a PSS analysis with a swept input power level, plot the output power against the input power level, and determine the 1 dB compression point from the curve.

Action 3-1: If it is not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*.

Action 3-2: Select the **PORTrf** source. Choose **Edit — Properties — Objects** and 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

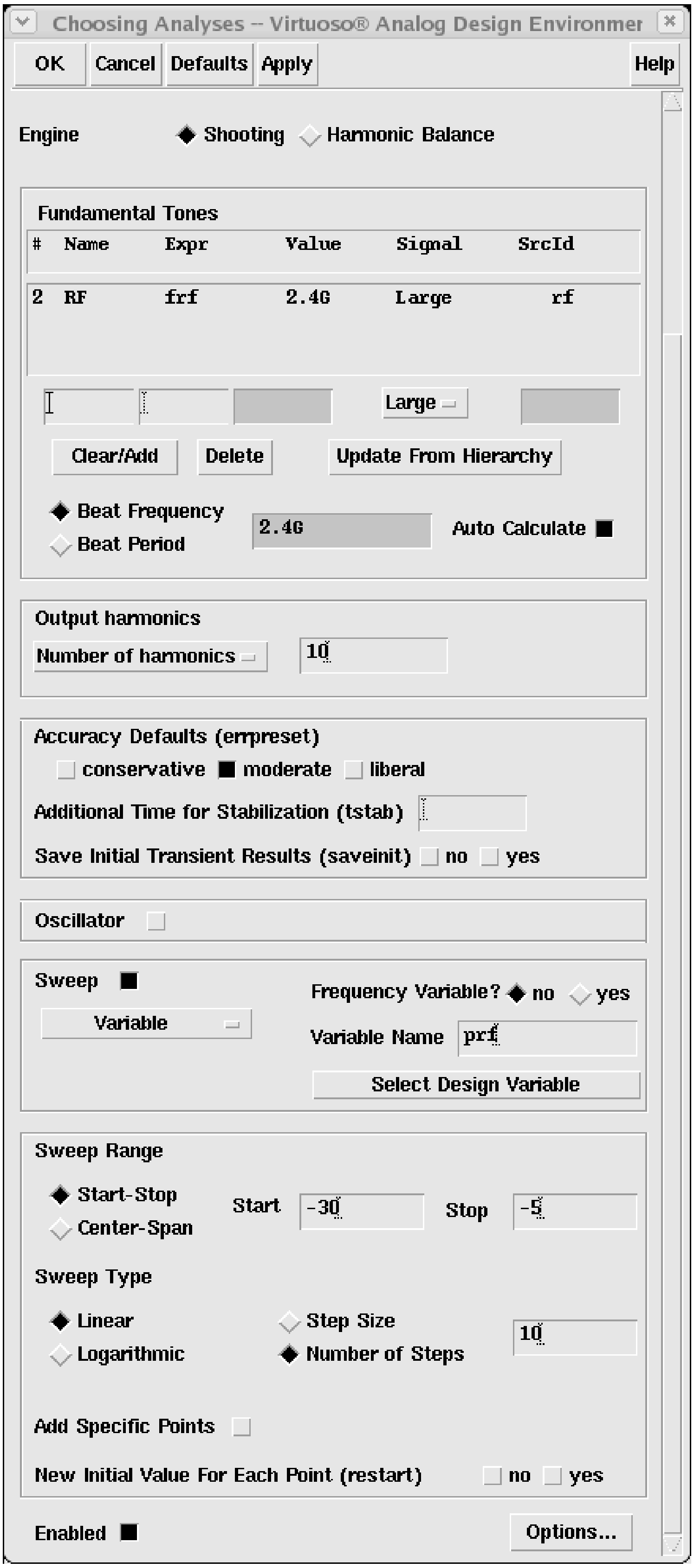
Action 3-3: Check and save the schematic.

Action 3-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment by choosing **Tools — Analog Environment**.

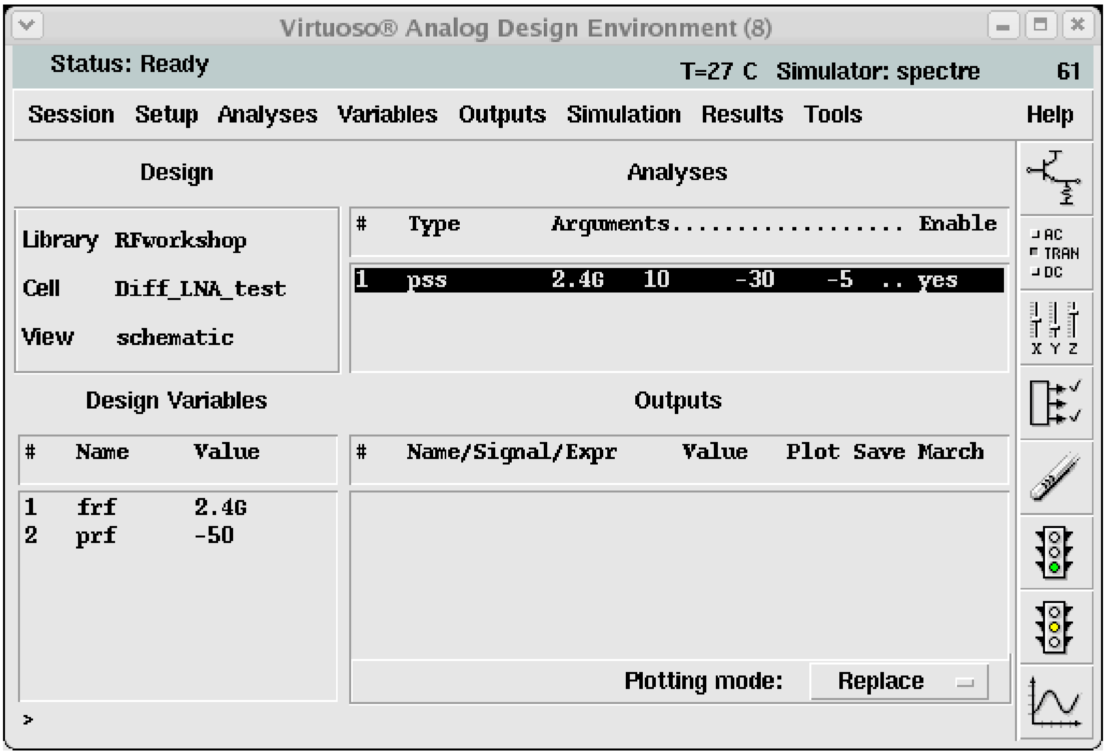
Action 3-5: (Optional) Choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab3_P1dB**", and skip to Action 3-9.

Action 3-6: In the Virtuoso Analog Design Environment window, select **Analyses — Choose…**

Action 3-7: In the Choosing Analyses window, select **pss** in the **Analysis** field of the window. Set up the form as follows:


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId
2 RF frf 2.4G Large rf
Large
Clear/Add
Delete
Update From Hierarchy
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
Variable
Frequency Variable?
no
yes
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
New Initial Value For Each Point (restart)
no
yes
Enabled
Options...

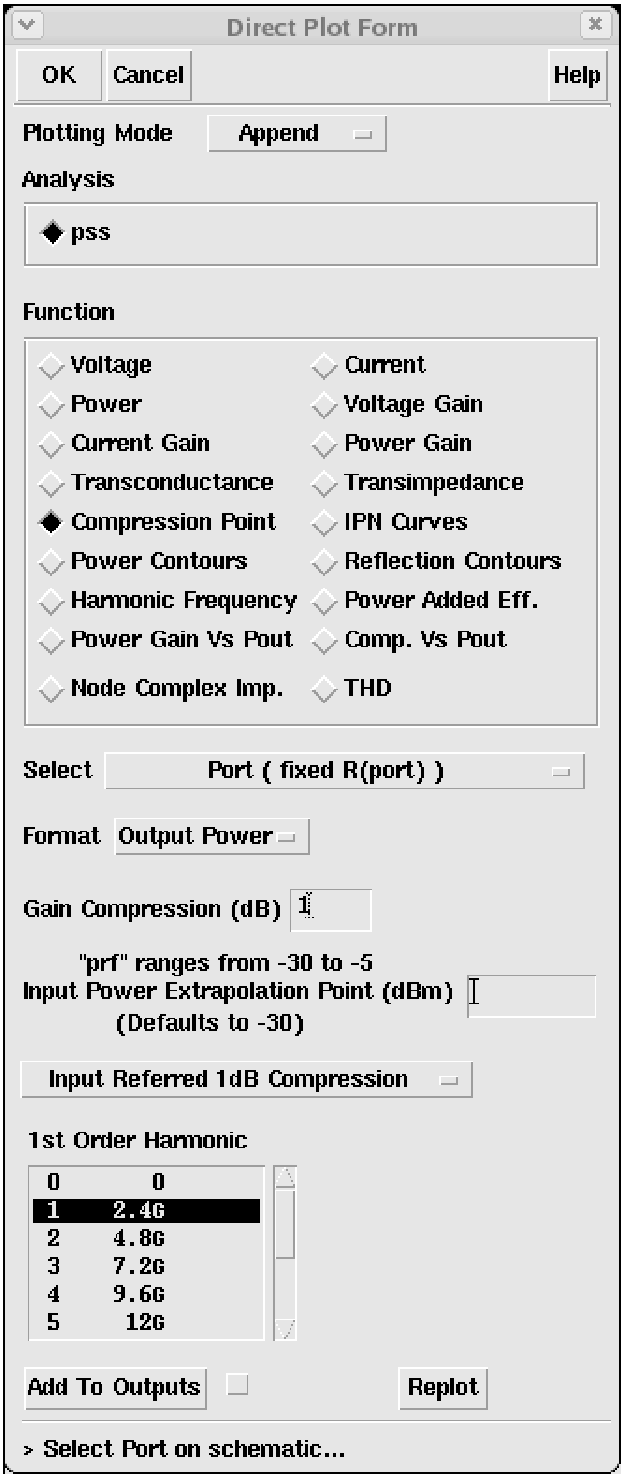
Action 3-8: Make sure **Enabled** is selected. Click **OK** on the Choosing Analyses form. The Virtuoso Analog Design Environment window looks like this:



Action 3-9: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

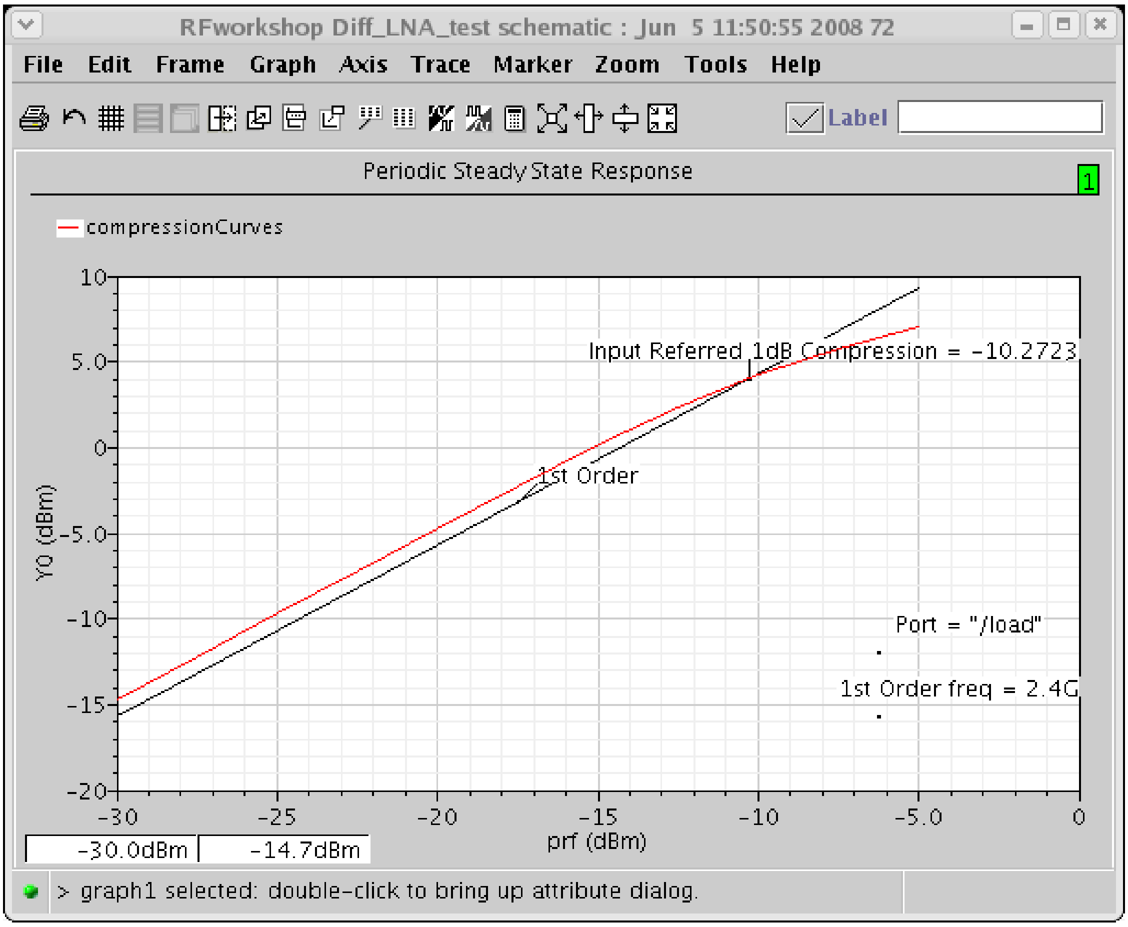
Action 3-10: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 3-11: In the Direct Plot Form, select **pss**, and configure the form as follows:



Action 3-12: Select output port **load** on the schematic.

The P1dB plot appears in the waveform window.

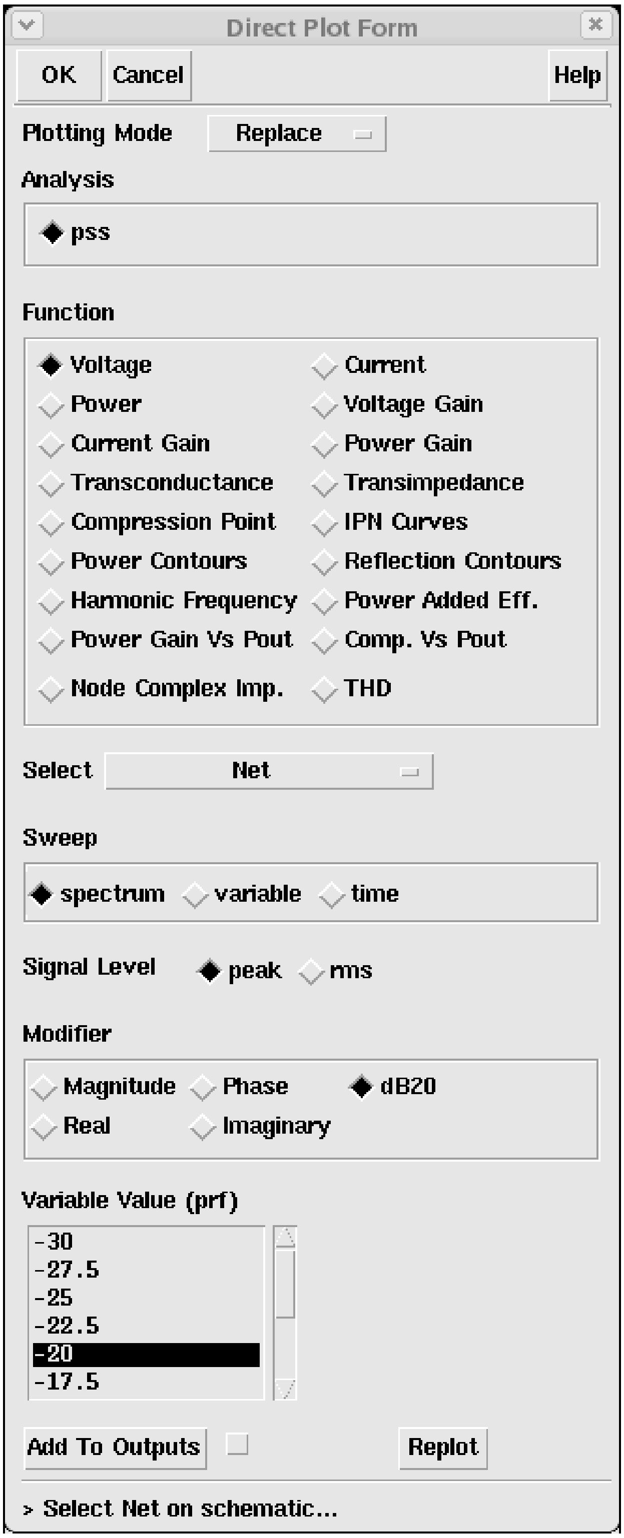


The gain at -30 dBm input power level is -14.7 - (-30) = 15.3 dBm which is a good match for the small signal gain.

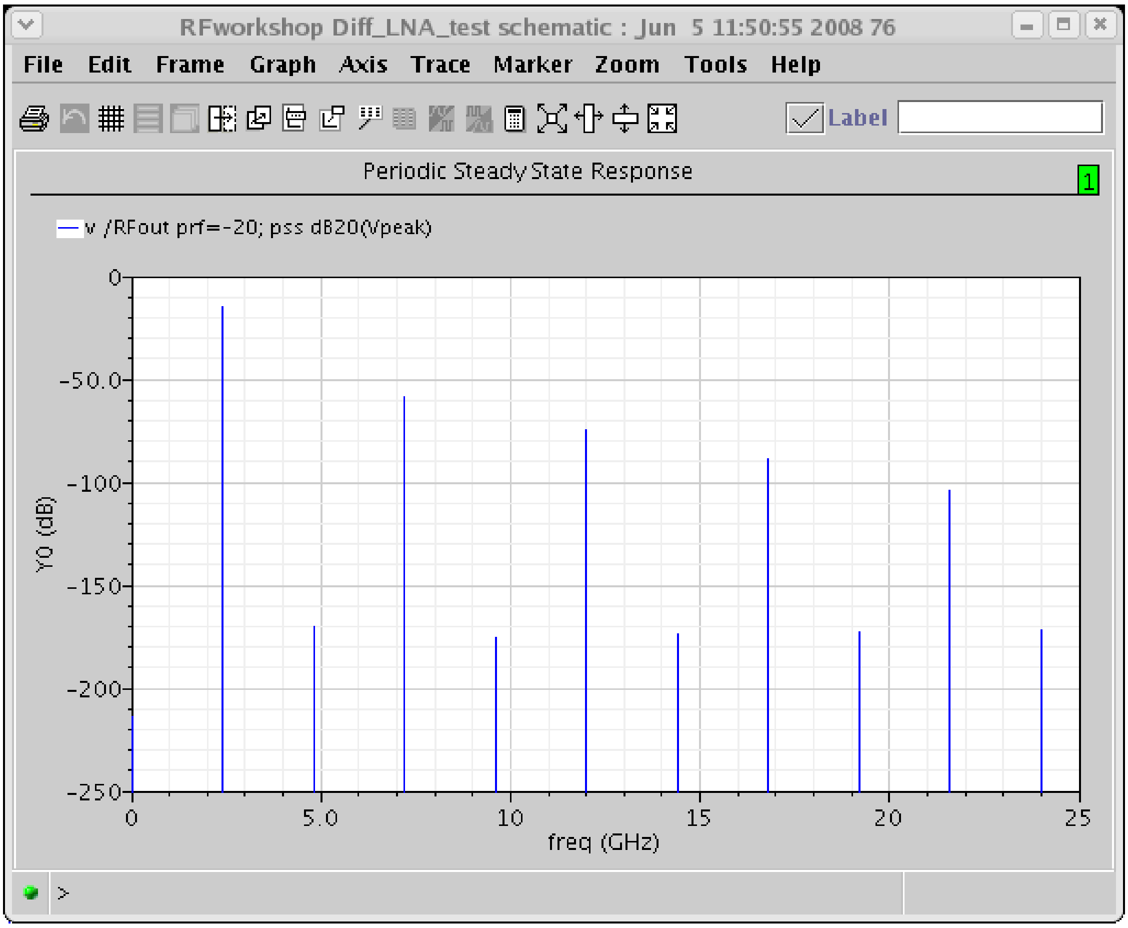
Action 3-13: Close the waveform window.

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. In the following steps, you plot the spectrum of a load when the input power level is -20 dBm.

Action 3-14: In the Direct Plot Form, select **pss**, and configure the form as follows:



Action 3-15: Select output **net RFout** on the schematic.



The plot shows that the DC and all the even modes at the output are suppressed because the LNA is a differential LNA.

If you write the nonlinear response of one side amplification as

$$y(x) = \alpha_0 + \alpha_1 x + \alpha_2 x^2 + \alpha_3 x^3 ......$$

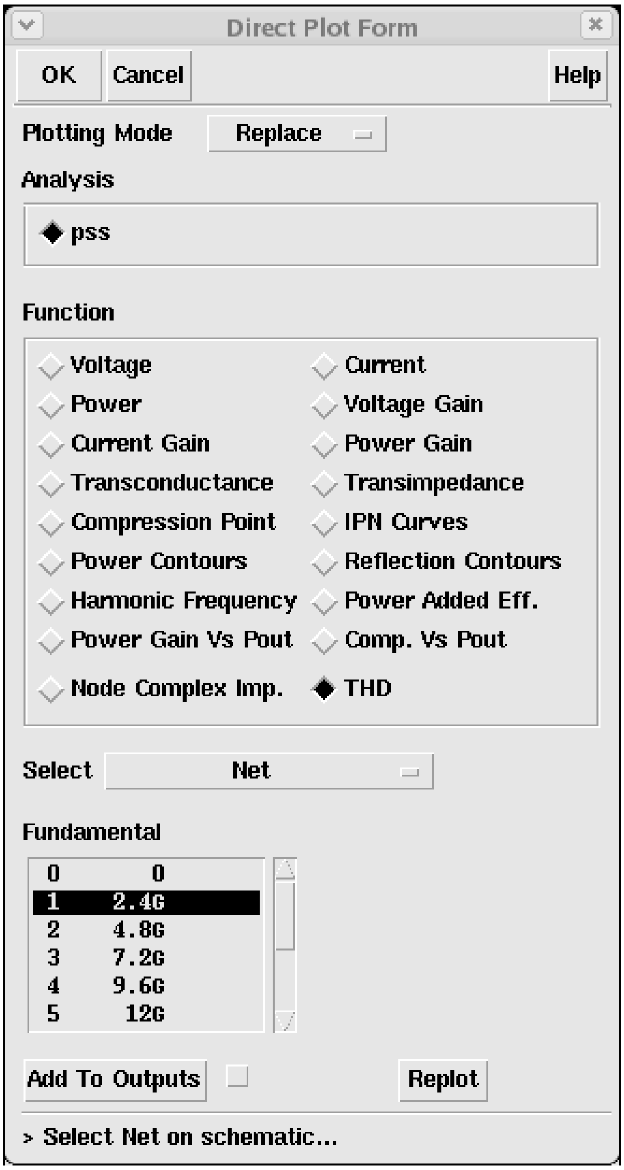
the output is

$$y = y(x/2) - y(-x/2) = \alpha_1 x + \alpha_3 x^3 / 4$$

For the differential LNA, the common mode disturbance is rejected.

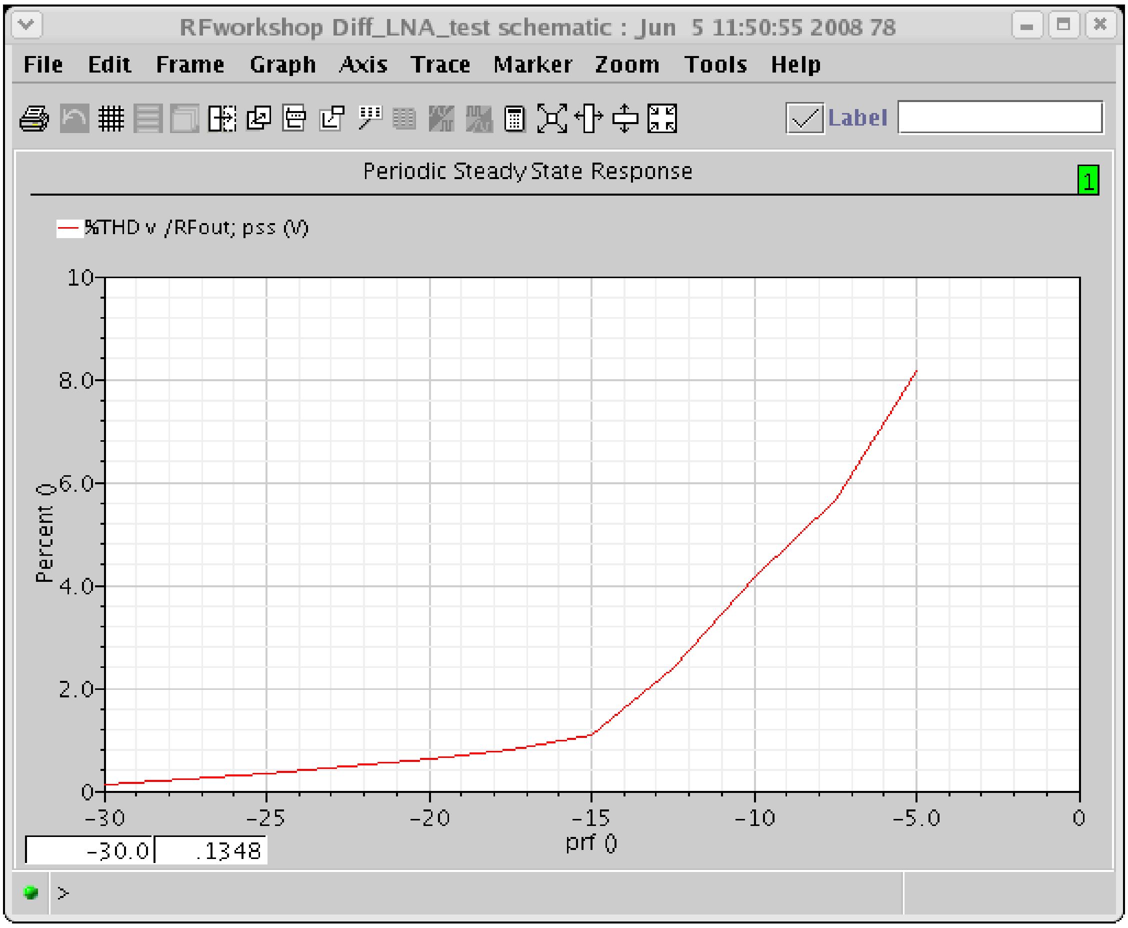
Action 3-16: After viewing the waveforms, close the waveform window.

Action 3-17: In the Direct Plot Form, select the **pss** analysis and the **THD** function.



Action 3-18: Select output net **RFout** on the schematic.

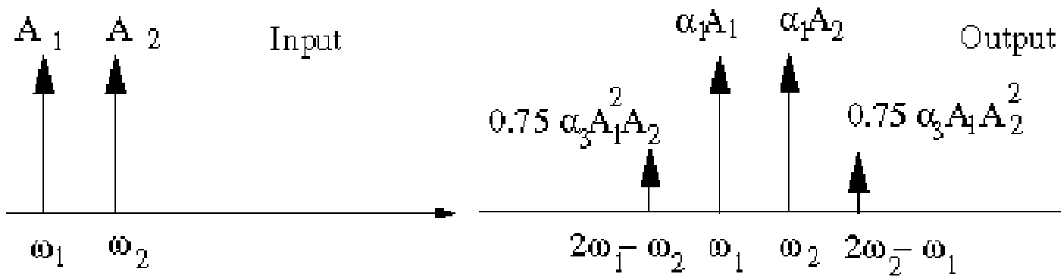
The THD plot appears in the waveform window.



Action 3-19: Close the waveform window and click **Cancel** on the Direct Plot Form.

IP3 measurements

IP3 is an important RF specification. The IP3 measurement is defined as the cross point of the power for the 1st order tones, ω_1 and ω_2, and the power for the 3rd order tones, $2\omega_1 - \omega_2$ and $2\omega_2 - \omega_1$, on the load.



As shown in the above figure, when you assume the input signal, *x*, is

$$x = A_1 \cos \omega_1 t + A_2 \cos \omega_2 t$$

and the nonlinear response, *y*, is

$$y = \alpha_1 x + \alpha_2 x^2 + \alpha_3 x^3$$

then, the linear and third order components at the output are

$$\alpha_1 A_1 \cos \omega_1 t,\ \alpha_1 A_2 \cos \omega_2 t,\ \frac{3\alpha_3 A_1^{\,2} A_2}{4} \cos(2\omega_1 - \omega_2)t,\ \frac{3\alpha_3 A_1\ A_2^{\,2}}{4} \cos(2\omega_2 - \omega_1)t,$$

When $A_1 = A_2$, the two first-order components have the same amplitude and the two third-order components also have the same amplitude.

Because the first-order components grow linearly and the third-order components grow cubically, they eventually intersect as the input power level *A* increases. The third-order intercept point is the point where the two output power curves intersect.

SpectreRF offers several ways to simulate IP3. The following 4 labs, for example, illustrate different methods that can be used to calculate IP3 for LNAs.

Lab 4: IP3 Measurement---PSS/ PAC analysis

The first method treats one tone, for example ω_1, as a large signal and performs a PSS analysis on the signal. The method treats the other tone, for example ω_2, as a small signal and performs a PAC analysis on the signal based on the linear time-varying systems obtained after the PSS analysis. The IP3 point is the intersect point between the power for the signal ω_2 and the power for the signal $2\omega_1 - \omega_2$. Because 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. The power level of both tones must be set to the same value.

Action 4-1: If it is not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*.

Action 4-2: Select the **PORTrf** source. Choose **Edit — Properties — Objects** and 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

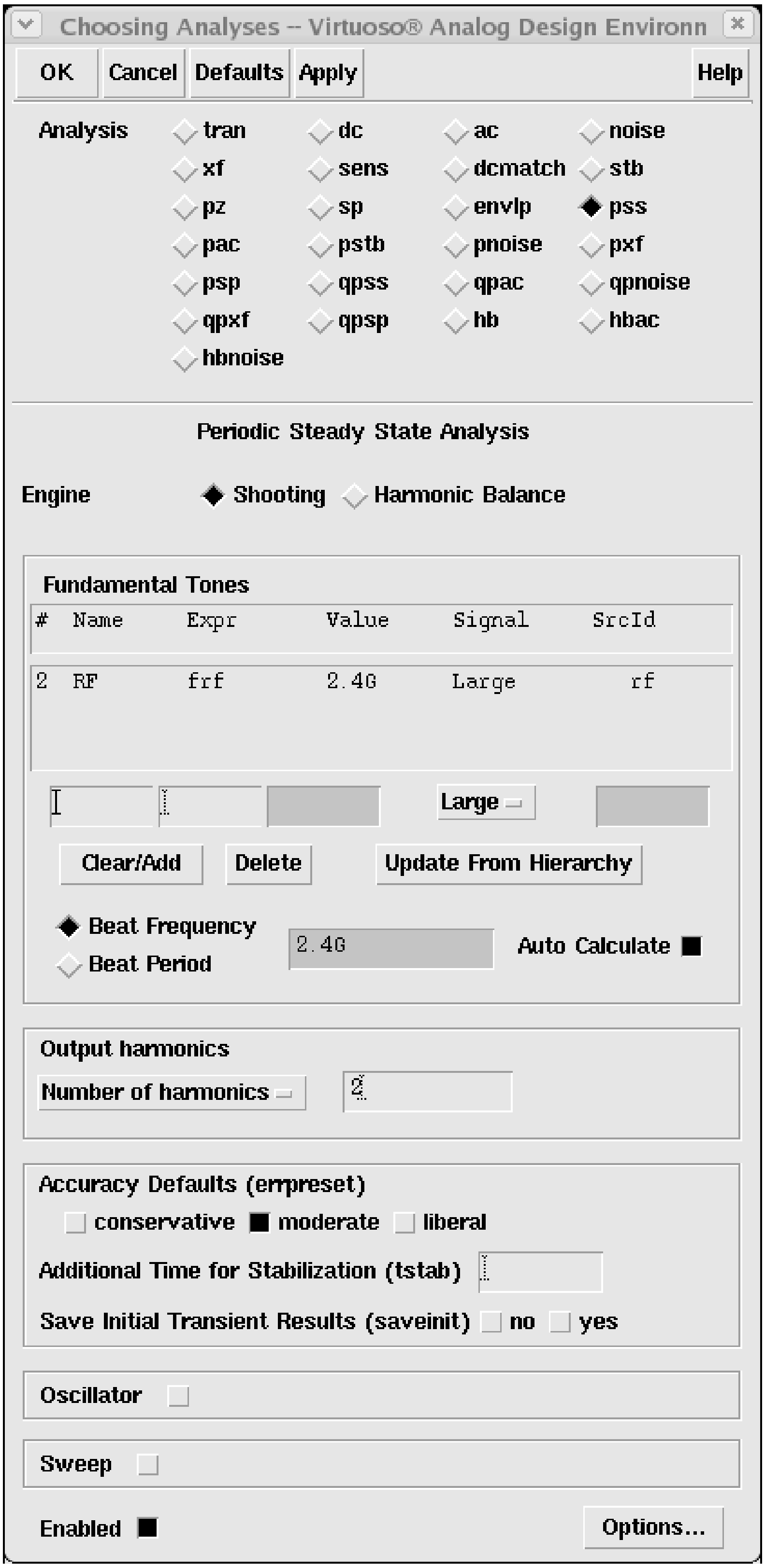
Action 4-3: Check and save the schematic.

Action 4-4: From the Diff_LNA_test schematic, choose **Tools — Analog** to start the Virtuoso Analog Design Environment.

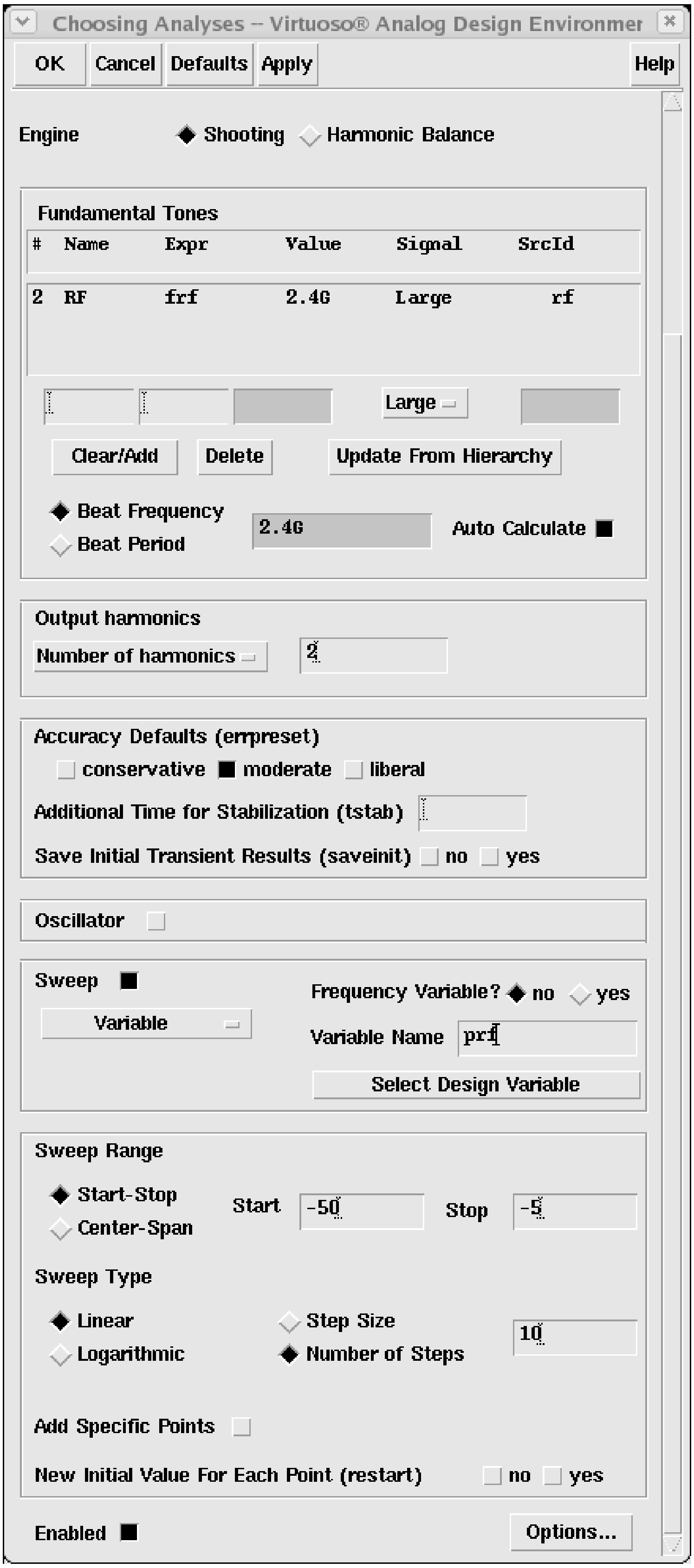
Action 4-5: (Optional) Choose **Session — Load State**, select **Cellview** in **Load State Option** and load state “**Lab4_IP3_PSSPAC_shooting**”, skip to Action 4-12.

Action 4-6: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose…**

Action 4-7: In the Choosing Analyses window, select **pss** in the **Analysis** field of the window and set up the form as follows:

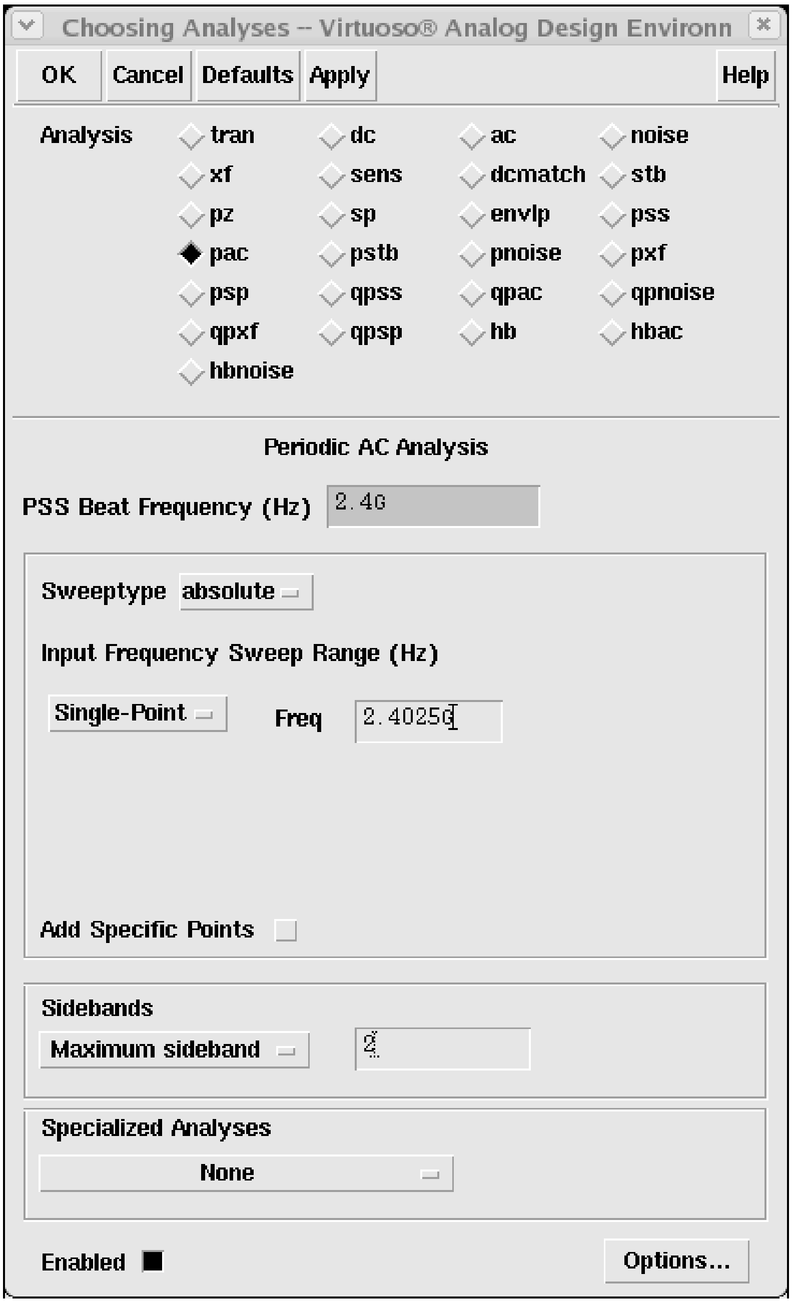


Action 4-8: Select **Sweep** and set the sweep values as follows:


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId
2 RF frf 2.4G Large rf
Large
Clear/Add
Delete
Update From Hierarchy
Beat Frequency
Beat Period
2.4G
Auto Calculate
Output harmonics
Number of harmonics
2
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
-50
Stop
-5
Sweep Type
Linear
Logarithmic
Step Size
Number of Steps
10
Add Specific Points
New Initial Value For Each Point (restart)
no
yes
Enabled
Options...

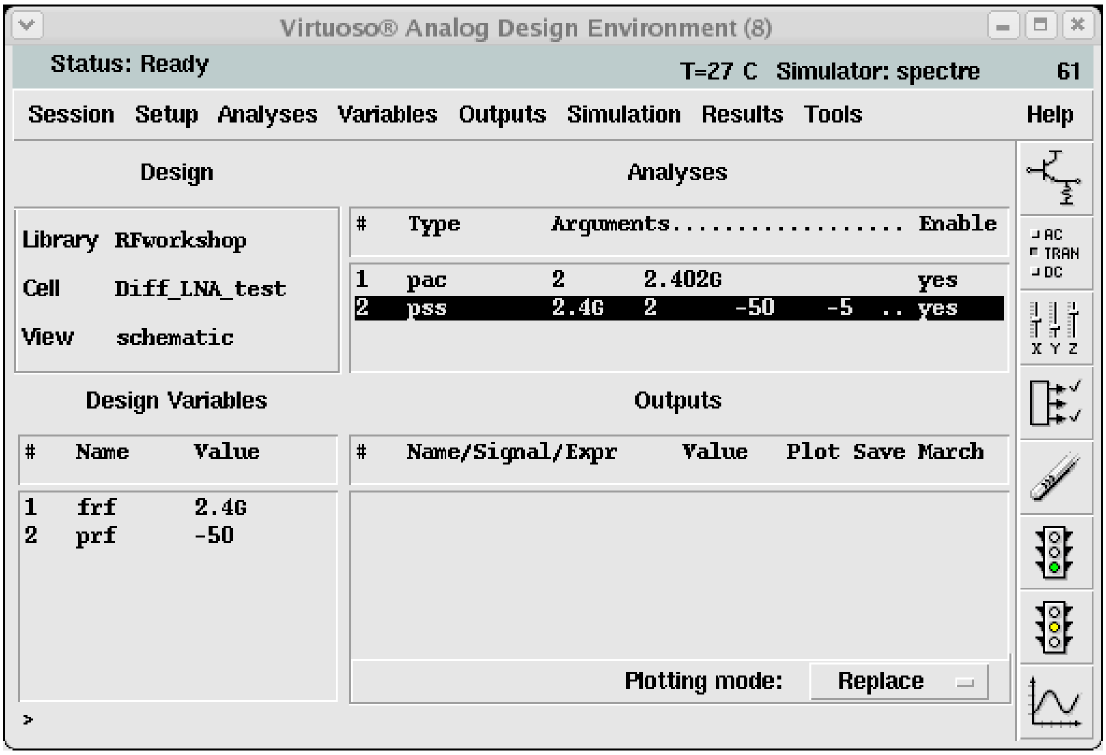
Action 4-9: In the Choosing Analyses window, select **pac** in the **Analysis** field of the window.

Action 4-10: Set up the form as shown here:



Action 4-11: Click **OK** in the Choosing Analyses form.

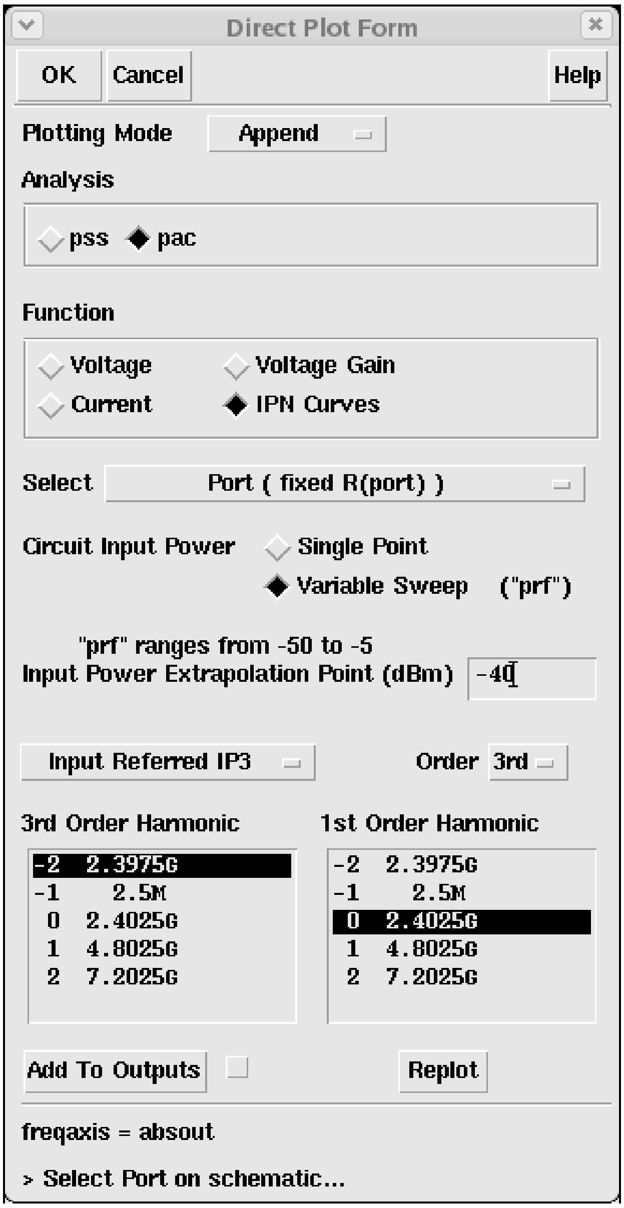
The Virtuoso Analog Design Environment window looks like this:



Action 4-12: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action 4-13: After the simulation ends, in the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

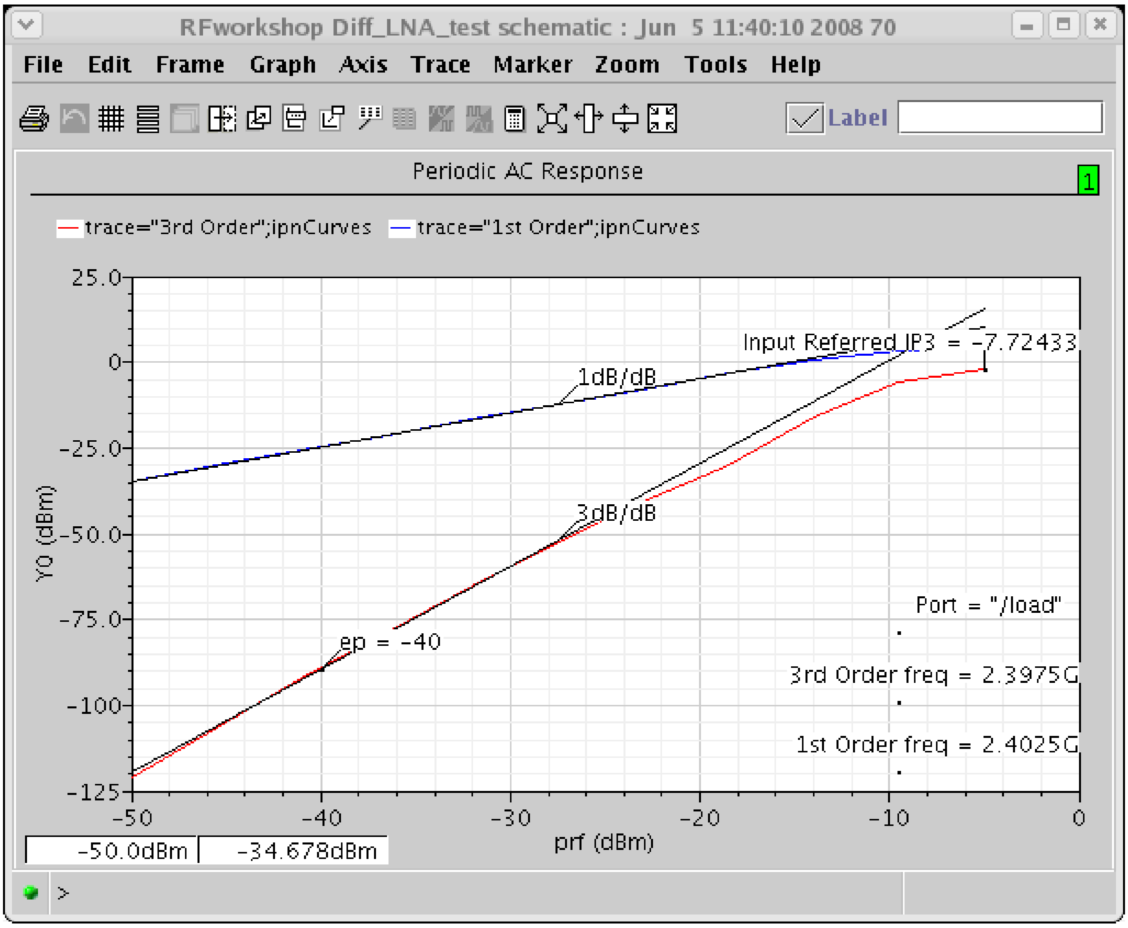
Action 4-14: Choose **pac** and set up the form as follows:



Note: As defined, the IP3 point is the intersection point between the power for the signal ω_2 and the power for the signal $2\omega_1 - \omega_2$. So here you choose ω_2 as the 1st order harmonic, and $2\omega_1 - \omega_2$ the 3rd order harmonic.

Action 4-15: Select **port load** in the Diff_LNA_test schematic.

The IP3 plot appears in the waveform window.



Action 4-16: Click **Cancel** in the Direct Plot Form and close the waveform window.

Although it is possible to use the PSS analysis with the beat frequency set to be the commensurate frequency of the two tones, this method is not recommended. Because the commensurate frequency can be very small, the simulation time for this method can be very long.

Lab 5: IP3 Measurement---QPSS Analysis with Shooting or Harmonic Balance Engine

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

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. Cadence recommends using the PSS and PAC analyses for IP3 simulation because that 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.

Action 5-1: If it is not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action 5-2: Select the **PORT rf** source. Choose **Edit — Properties — Objects** and 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

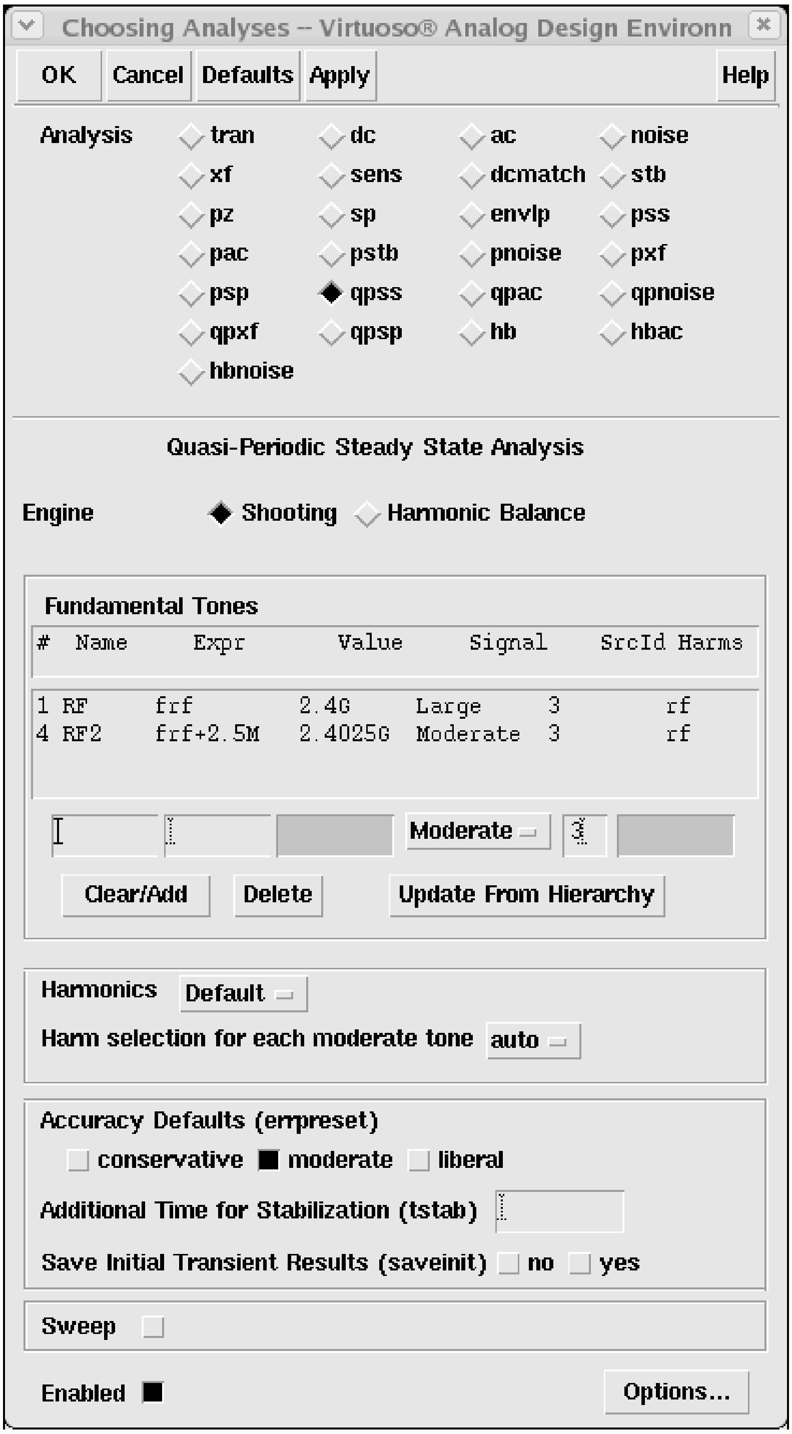
Action 5-3: Check and save the schematic.

Action 5-4: From the Diff_LNA_test schematic, choose **Tools — Analog** to start the Virtuoso Analog Design Environment.

Action 5-5: (Optional) Choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab5_IP3_QPSS_shooting**" and skip to Action 5-9.

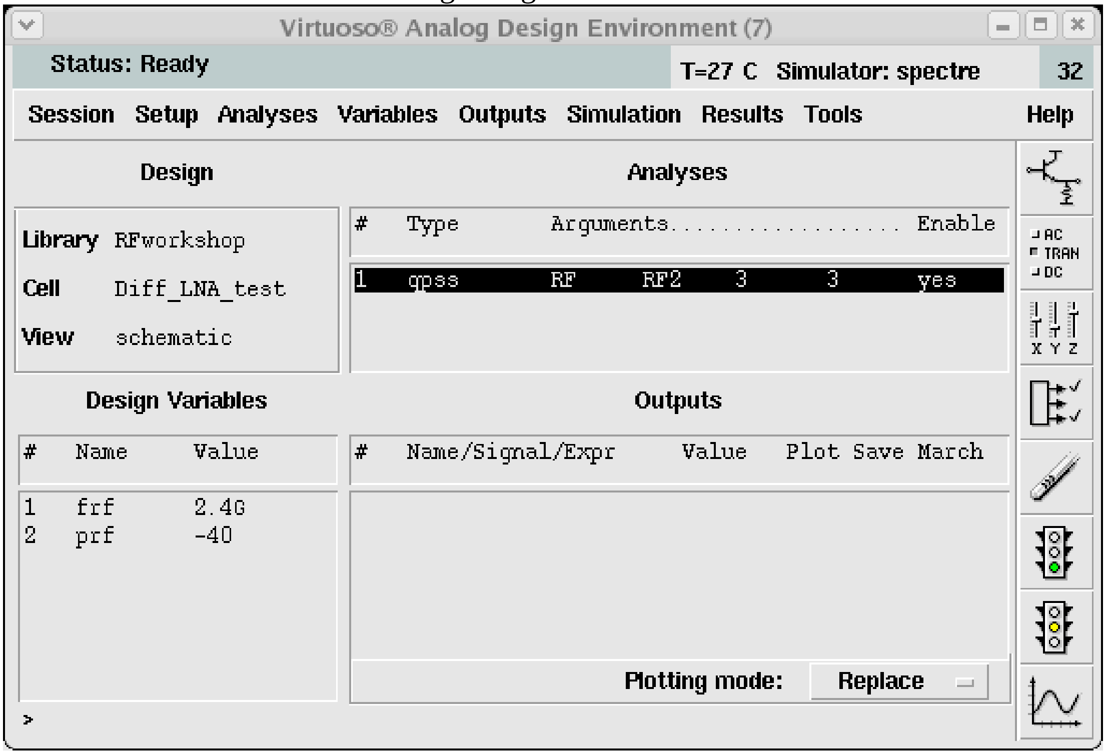
Action 5-6: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose…**

Action 5-7: In the Choosing Analyses window, select **qpss** in the **Analysis** field of the window and set the form as follows:



Action 5-8: Make sure **Enabled** is selected. In the Choosing Analyses window, click **OK**.

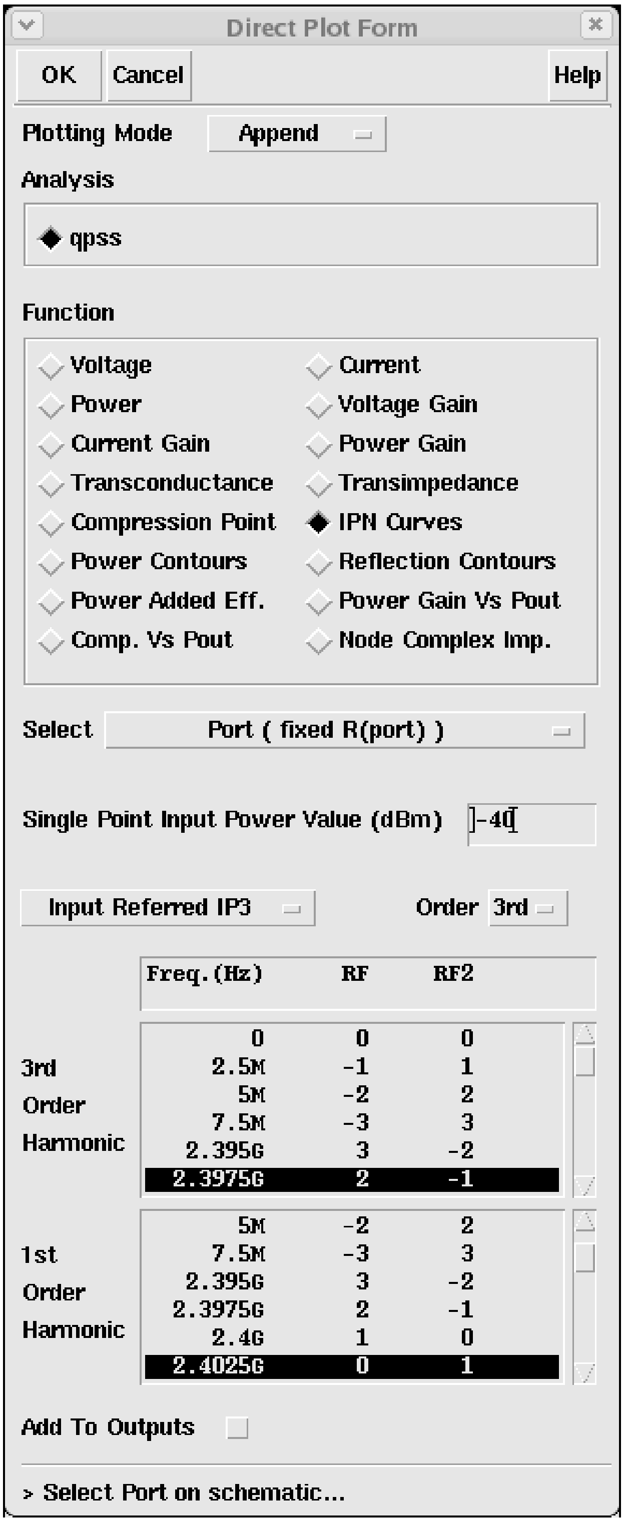
The Virtuoso Analog Design Environment window looks like this:



Action 5-9: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action 5-10: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

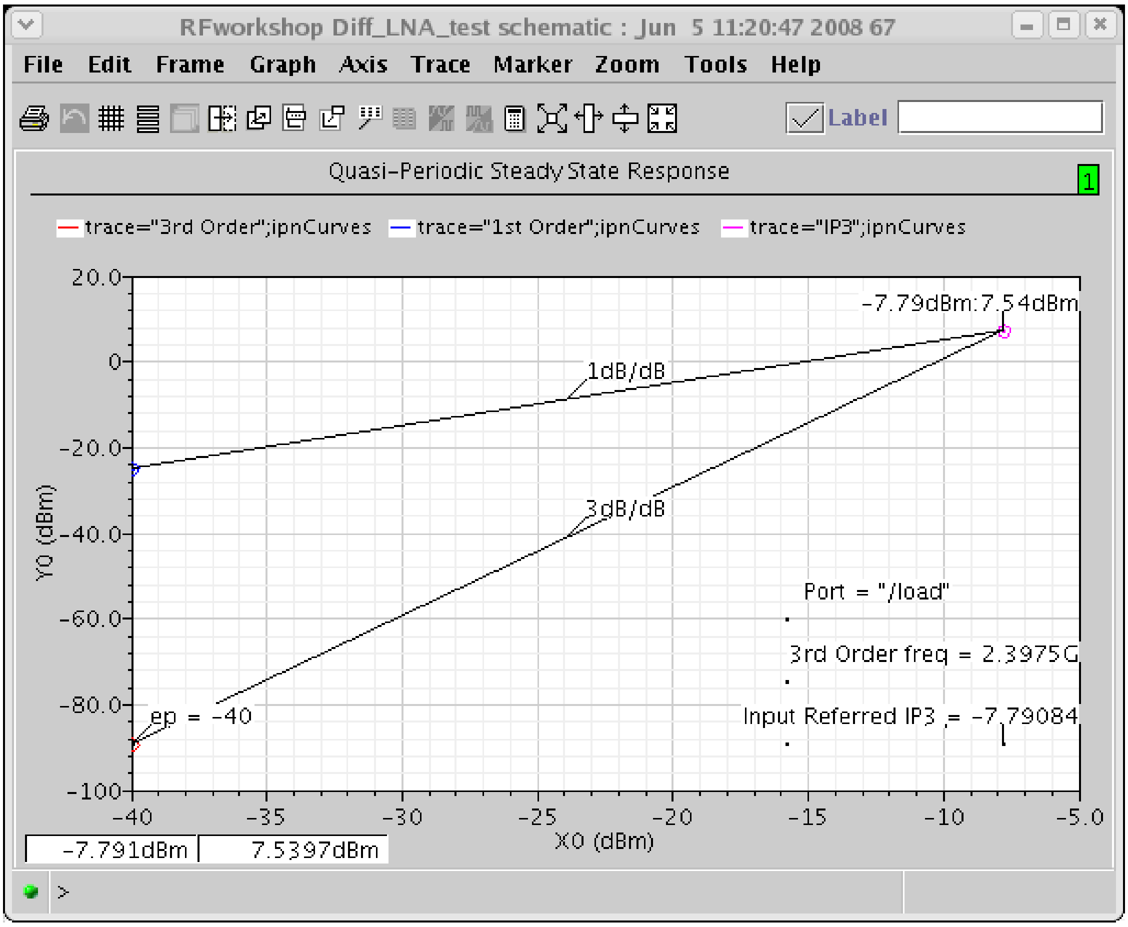
Action 5-11: In the Direct Plot Form, select **qpss**, and configure the form as follows:



Note: As defined, the IP3 point is the intersection point between the power for the signal ω_2 and the power for the signal $2\omega_1 - \omega_2$. So here you choose $\omega_2 + 0\times\omega_1 = 2.4025GHz$ as the 1st order harmonic, and $2\omega_1 - \omega_2 = 2.3975GHz$ as the 3rd order harmonic.

Action 5-12: Select output port **load** on the schematic.

The IP3 plot appears in the waveform window.



Action 5-13: Close the waveform window. Click **Cancel** on the Direct Plot Form.

You are going to simulate the IP3 with the harmonic balance engine and compare its results with the shooting Newton engine.

Action 5-14: (Optional) Choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab5_IP3_QPSS_FB**" and skip to Action 5-21.

Action 5-15: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose…**

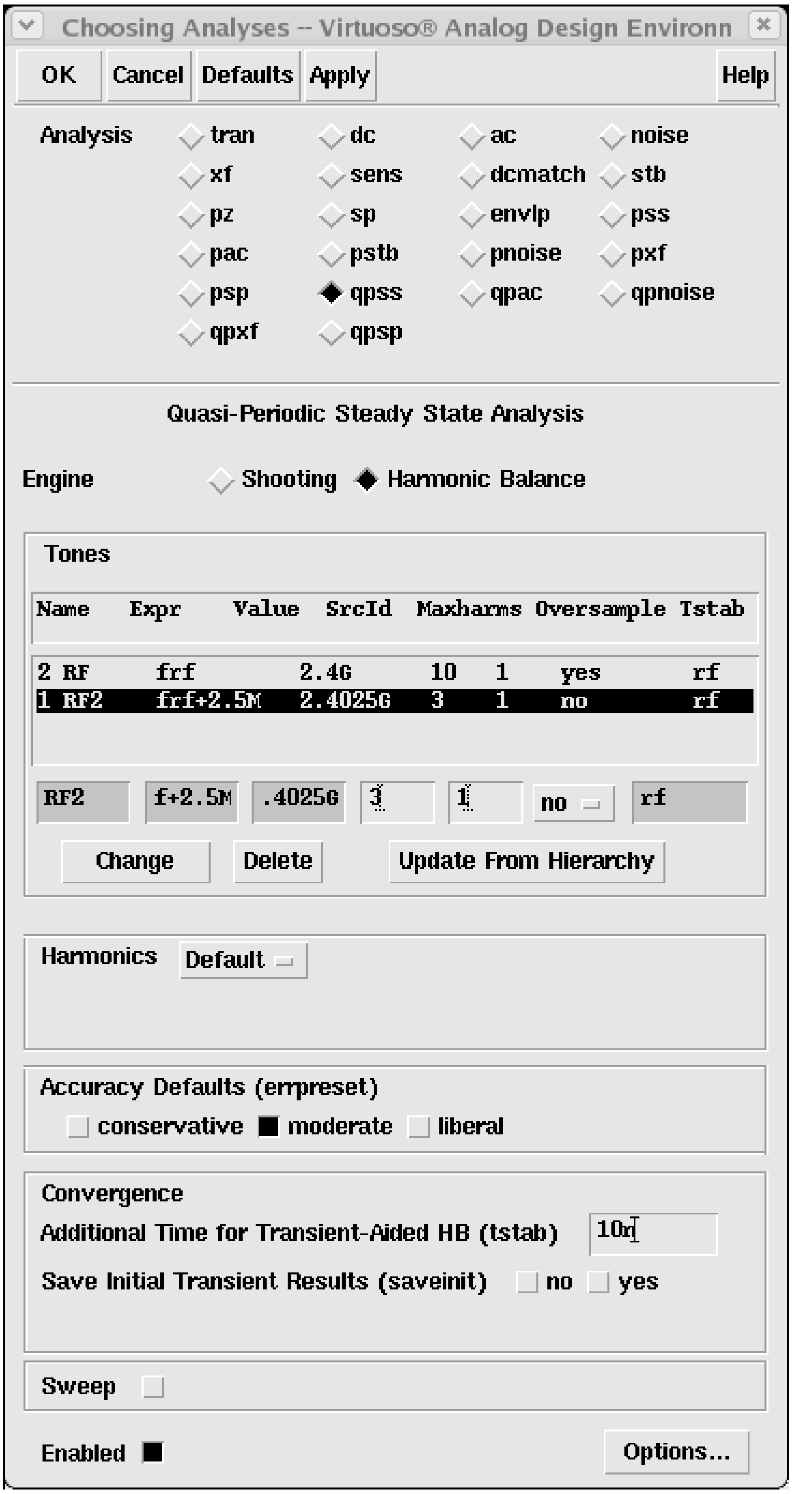
Action 5-16: In the Choosing Analyses window, select **qpss** in the **Analysis** field of the window.

Action 5-17: In the **Engine** field, choose **Harmonic Balance**. The default engine is **Shooting**.

Action 5-18: In the **Tones** field, choose **RF**. Change the **Maxharms** to 10, because the harmonic balance engine needs more harmonics to calculate. Click **Update**.

Action 5-19: Put **10n** in the **Additional Time for Stabilization (stab)** field.

The form looks like this:



Action 5-20: Make sure **Enabled** is selected. In the Choosing Analyses window, click **OK**.

Note: The harmonic balance (HB) engine uses the same PSS/QPSS statement as the time-domain engine. A toggle button is available for switching between time domain shooting and

HB in the ADE PSS and QPSS set up form. When setting up an HB QPSS/PSS analyses, pay attention to the following parameters:

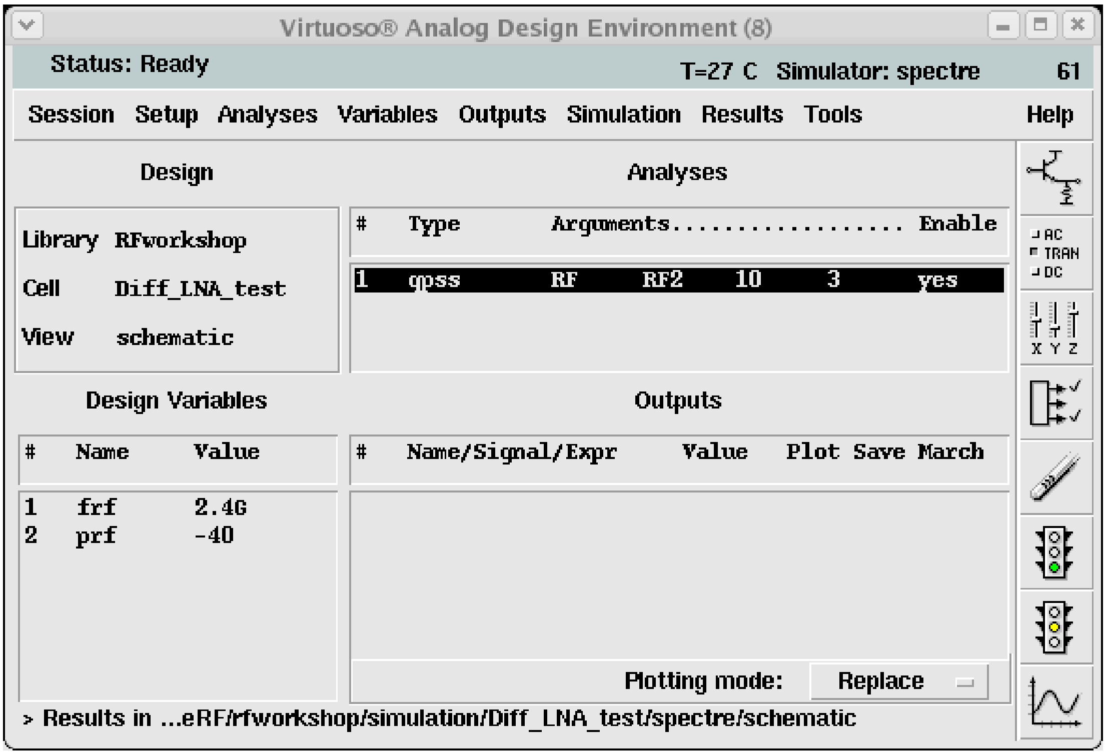
1**. Maximum harmonic**: Maximum harmonic (“**harms**” in PSS and “**maxharms**” in QPSS) has the most impact on HB accuracy. Using inadequate harmonics causes errors because the aliasing effect causes spectrum outside of **maxharms** to be folded back into harmonics inside. To obtain accurate results, **maxharms** should be big enough to cover the signal bandwidth.

The **reltol** and **errpreset** parameters also affect the simulation accuracy. HB uses the same convergence criteria as the shooting method.

2. **tstab**: Similar to the 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, you can choose the specific tone during the tstab period. Only one tone is allowed. One additional cycle is run for FFT. If **tstab** is set to 0, dc results are used as the initial condition for HB.

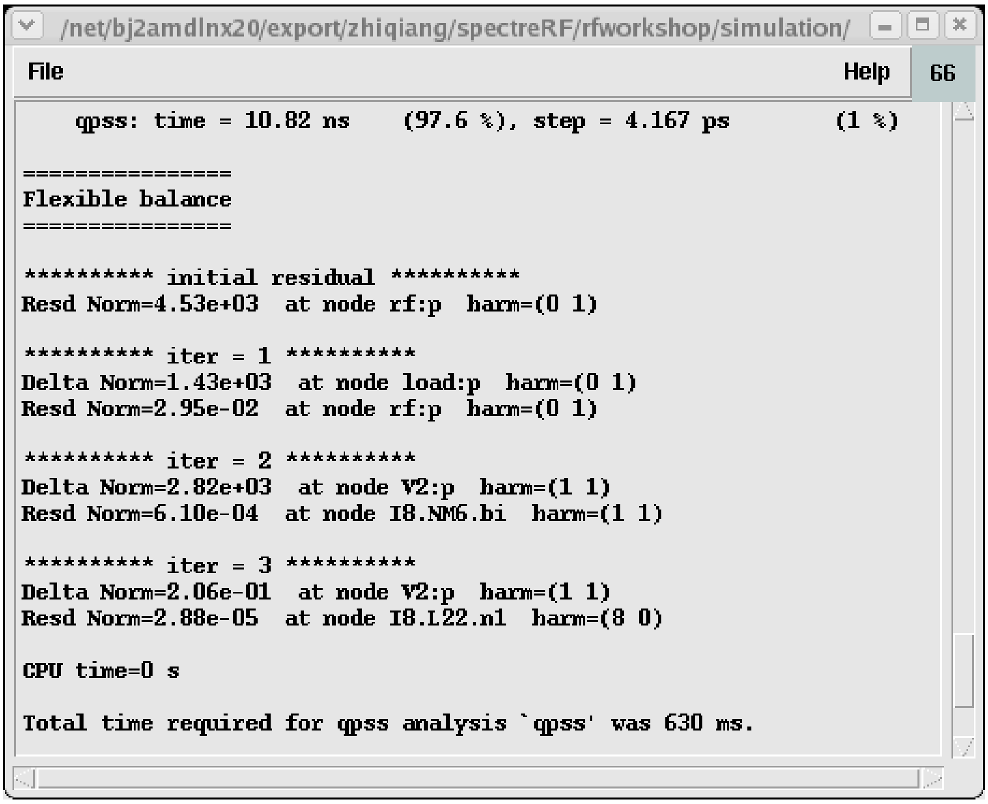
3: **Oversample Factor:** Oversampling is usually not needed, but for extremely nonlinear circuits driven by sources at very high power levels, oversampling might help with convergence.

The Virtuoso Analog Design Environment window looks like this:



Action 5-21: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

As the simulation progresses, messages appear in the simulation output log window. Note that the messages differ from those generated for the time domain qpss:

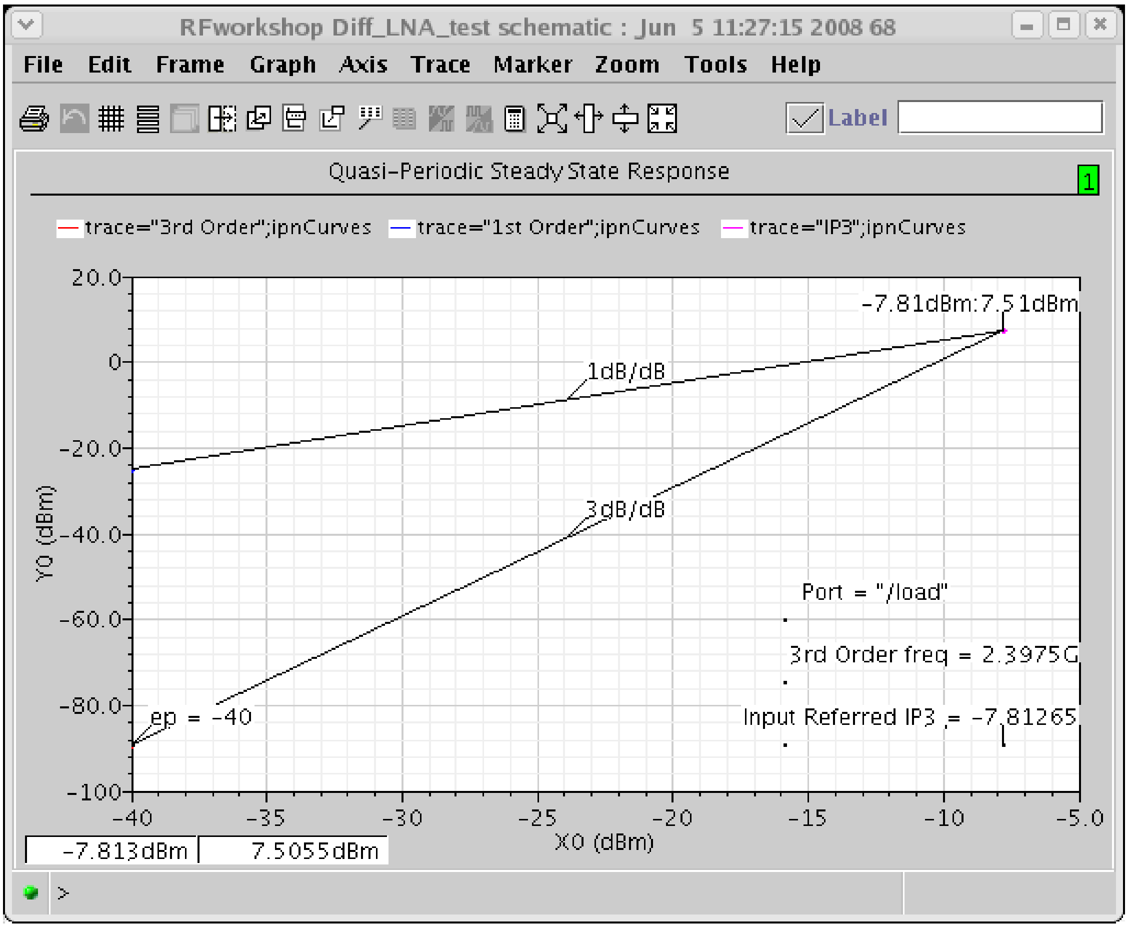


Action 5-22: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 5-23: In the Direct Plot Form, select **qpss**. The form is the same as the form used for the shooting engine.

Action 5-24: Select the output port **load** on the schematic.

The results are plotted in the waveform window.



Note: The harmonic balance (HB) engine is a feature introduced in MMSIM6.0 USR2. Harmonic balance complements the shooting method by providing efficient and robust simulation for linear and weakly nonlinear circuits.

The HB method is very efficient in simulating weakly nonlinear circuits such as LNAs. Only a few harmonics are needed to represent the solution accurately. For highly nonlinear circuits with sharply raising or falling signals, time domain shooting is more suitable. However, HB might still be the better choice when you are exploring design trade-offs using a few harmonics where accuracy is not the primary concern.

Action 5-25: After viewing the waveforms, close the waveform window and click **Cancel** in the Direct Plot Form.

Lab 6: IP3 Measurement---Rapid IP3 using PSS/PAC Analysis

Beginning with MMSIM6.0 USR2, SpectreRF supports Rapid IP3 calculation based on PAC or AC simulation. Rapid IP3, which is the fastest way to accomplish IP3 calculation, is a perturbative approach, based on the Born approximation, for solving weakly nonlinear circuits. The Rapid IP3 method does not require explicit high order derivatives from device models. All equations are formulated in the form of RF harmonics. They can be implemented in both time and frequency domains.

For a 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 the RF input source. Parameter ε tracks the order of the perturbation expansion. Under weakly nonlinear conditions, the nonlinear part is small compared to the linear part, so the above equation can be solved by using the 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 is accurate to the order of $O(\varepsilon^n)$.

Because the evaluation of F_{NL} takes full nonlinear device evaluation of F and its first derivative, no higher order derivative is needed. This makes it possible to carry out higher order perturbations without modifying current device models. The dynamic range of the perturbation calculations covers only RF signals, which gives the perturbative method advantages in terms of accuracy.

This lab shows you how to calculate the IP3 of LNAs using perturbation technology. With a similar setup and procedure, you can calculate IP2, compression Distortion Summary, and IM2 Distortion Summary.

Action 6-1: If it is not already open, open the *schematic* view of the *Diff_LNA_test* in the library *Rfworkshop*.

Action 6-2: Select the **PORT rf** source. Choose **Edit — Properties — Objects** and ensure that the port properties are set as described below:

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

Action 6-3: Click **OK** on the Edit Object Properties window to close it.

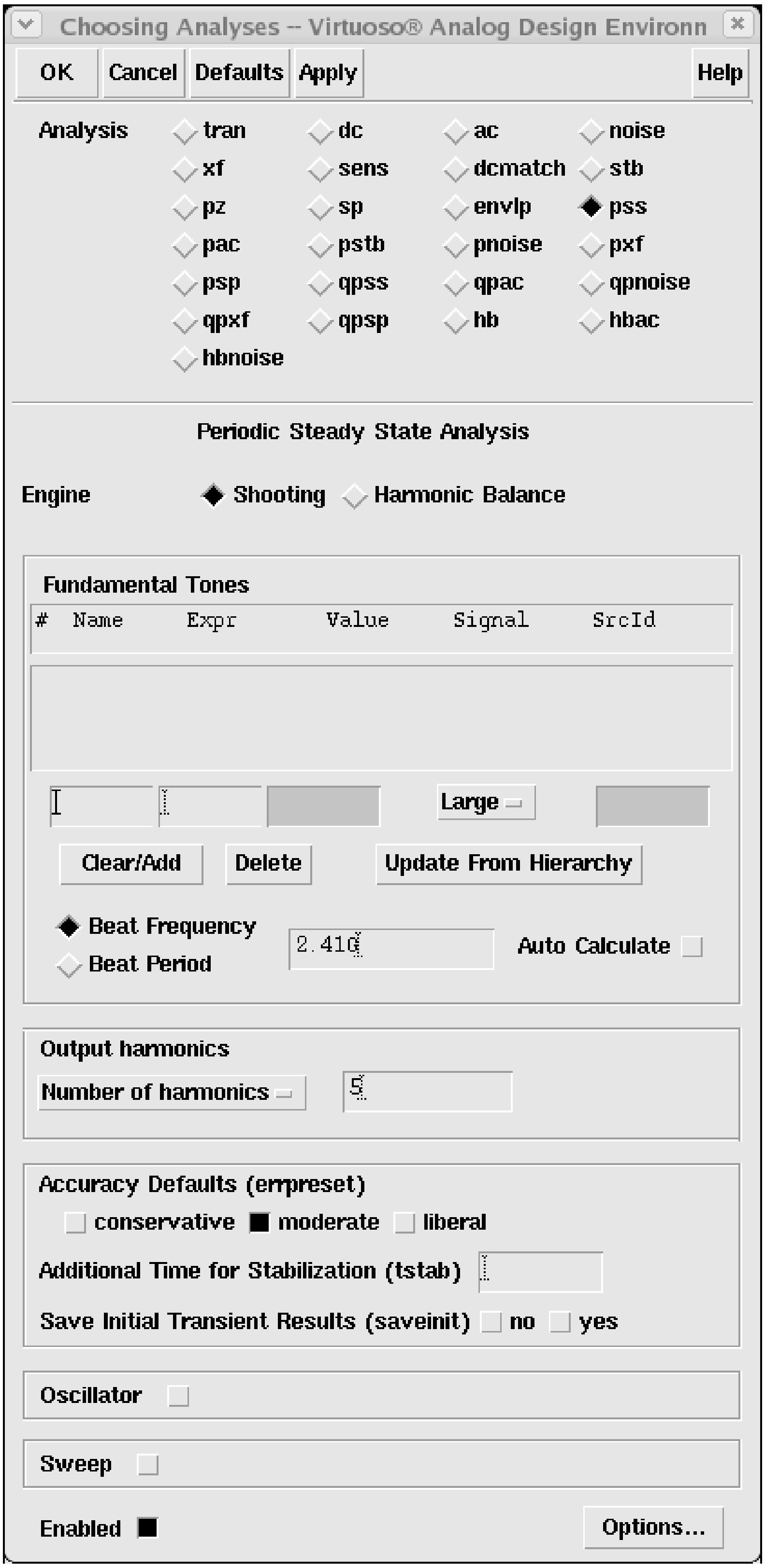
Action 6-4: Check and save the schematic.

Action 6-5: From the Diff_LNA_test schematic, choose **Tools — Analog Environment** to start the Virtuoso Analog Design Environment.

Action 6-6: (Optional) Choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab6_RapidIP3_PAC**" and skip to Action 6-12.

Action 6-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose…**

Action 6-8: In the Choosing Analyses window, select **pss** in the **Analysis** field of the window and set the form as follows:


Choosing Analyses -- Virtuoso® Analog Design Environn
OK
Cancel
Defaults
Apply
Help
Analysis
tran
dc
ac
noise
xf
sens
dcmatch
stb
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Periodic Steady State Analysis
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId
Large
Clear/Add
Delete
Update From Hierarchy
Beat Frequency
Beat Period
2.41G
Auto Calculate
Output harmonics
Number of harmonics
5
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
Save Initial Transient Results (saveinit)
no
yes
Oscillator
Sweep
Enabled
Options...

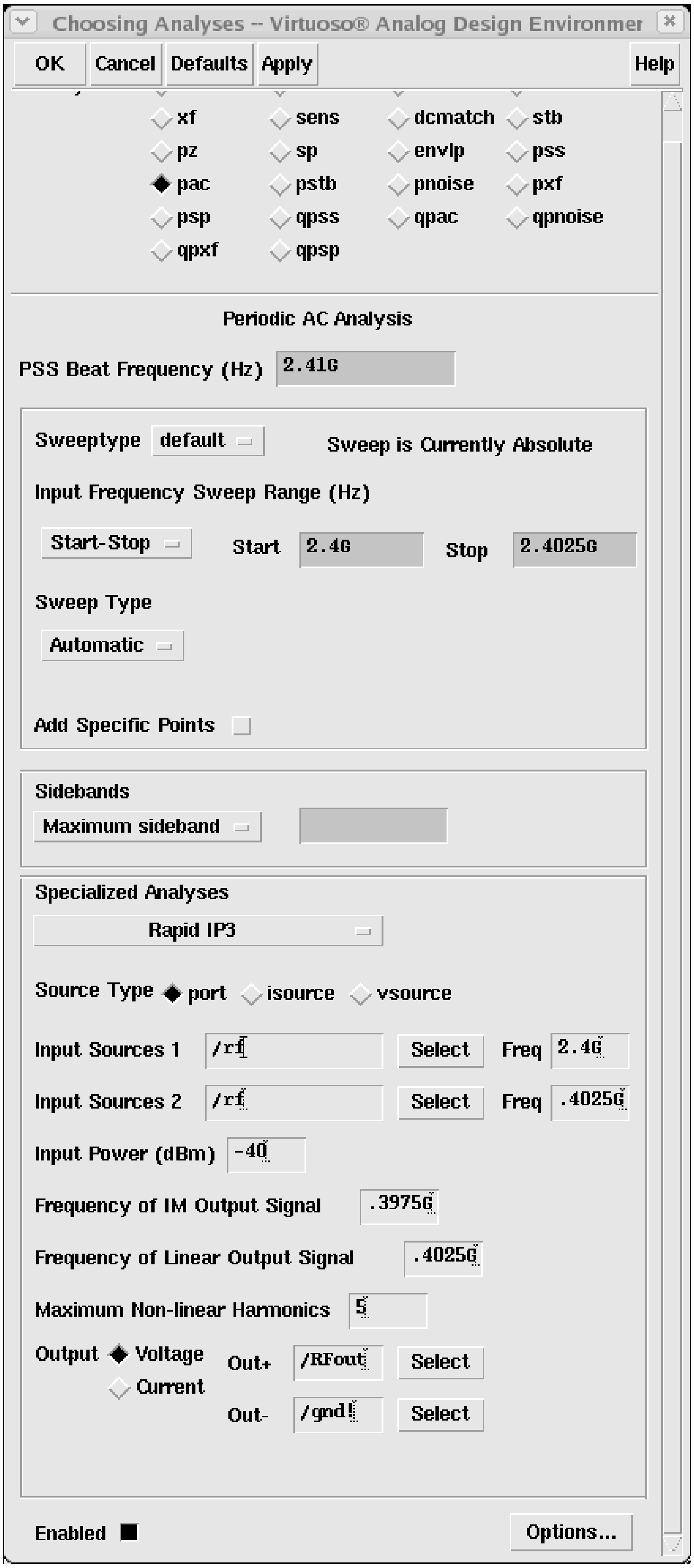
Note: In this example, there is no large signal in the circuit and so there is no particular need to run PSS. This psudo PSS is set up to define the operating point of the circuit. You need to choose a reasonable frequency which will not coincide with either of the PAC input frequencies, so 2.41GHz is used here. The purpose of this simultion is to show you can do RapidIP2 on an LNA either with PSS/PAC or better still in AC which will be illustrated later.

Action 6-9: Make sure **Enabled** is selected. In the Choosing Analyses window, click **Apply**.

Action 6-10: In the Choosing Analyses window, select **pac** in the **Analysis** field of the window. Choose **Rapid IP3** in the **Specialized Analyses** field. Set the **Input Sources 1** to **/rf** by selecting **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** to 2.3975G and the **Frequency of Linear Output Signal** to 2.0425G.

If **Maximum Non-linear Harmonics** is not specified, the default value is 4.

After these actions, the form looks like this:


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
xf
sens
dcmatch
stb
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
Periodic AC Analysis
PSS Beat Frequency (Hz)
2.41G
Sweeptype
default
Sweep is Currently Absolute
Input Frequency Sweep Range (Hz)
Start-Stop
Start
2.4G
Stop
2.4025G
Sweep Type
Automatic
Add Specific Points
Sidebands
Maximum sideband
Specialized Analyses
Rapid IP3
Source Type
port
isource
vsource
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...

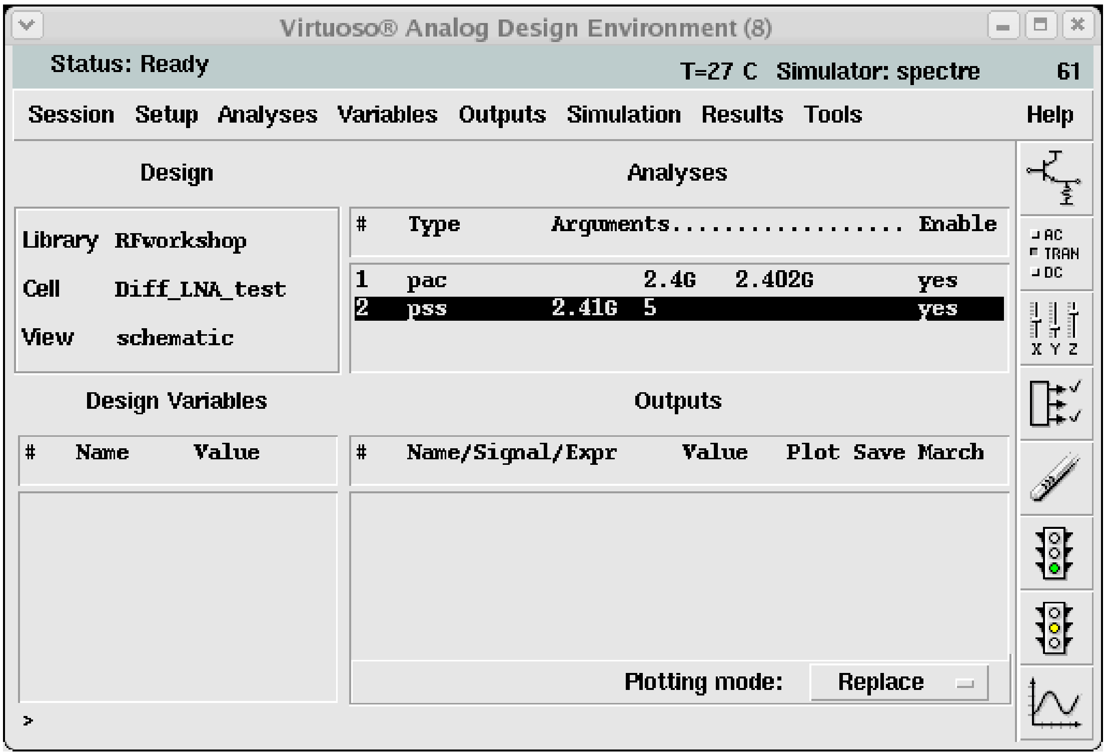
Note: Incommensurate frequencies should be used for all tones. If multiple combinations of tone frequencies match **Frequency of Linear Output Signal** or **Frequency of IM output Signal**, the Spectre simulator cannot determine which frequency to use as IM1 or IM3.

If output is current in a port, you must use the ‘save’ statement to indicate that the
port current needs to be computed. Otherwise, the simulator does not calculate it.

The perturbation method works under weakly nonlinear conditions. High input power that induces significant nonlinearity gives unreasonable results. Input power that is too low blends with the numerical noise floor. To avoid these difficulties, Cadence recommends using an input power range of -50dBm~-20dBm for general circuits.

Action 6-11: Make sure **Enabled** is selected. In the Choosing Analyses window, click **OK**.

The Virtuoso Analog Design Environment window looks like this:



Action 6-12: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

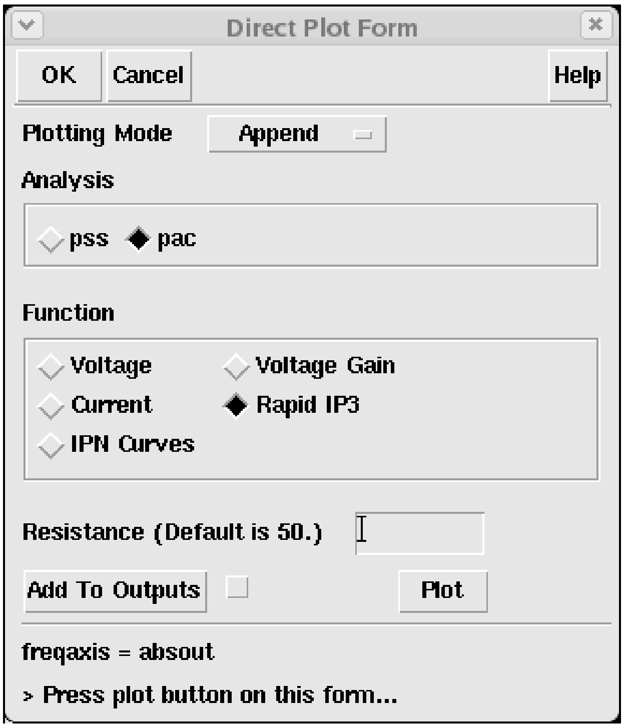
As the simulation progresses, messages similar to the following appear in the simulation output log window:

```
**********************
IP3 measurement `pac'
**********************
Input RF1 freq = 2.4 GHz
Input RF2 freq = 2.4025 GHz
Output IM1 freq = 2.4025 GHz
Output IM3 freq = 2.3975 GHz

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 2.49 s.
```

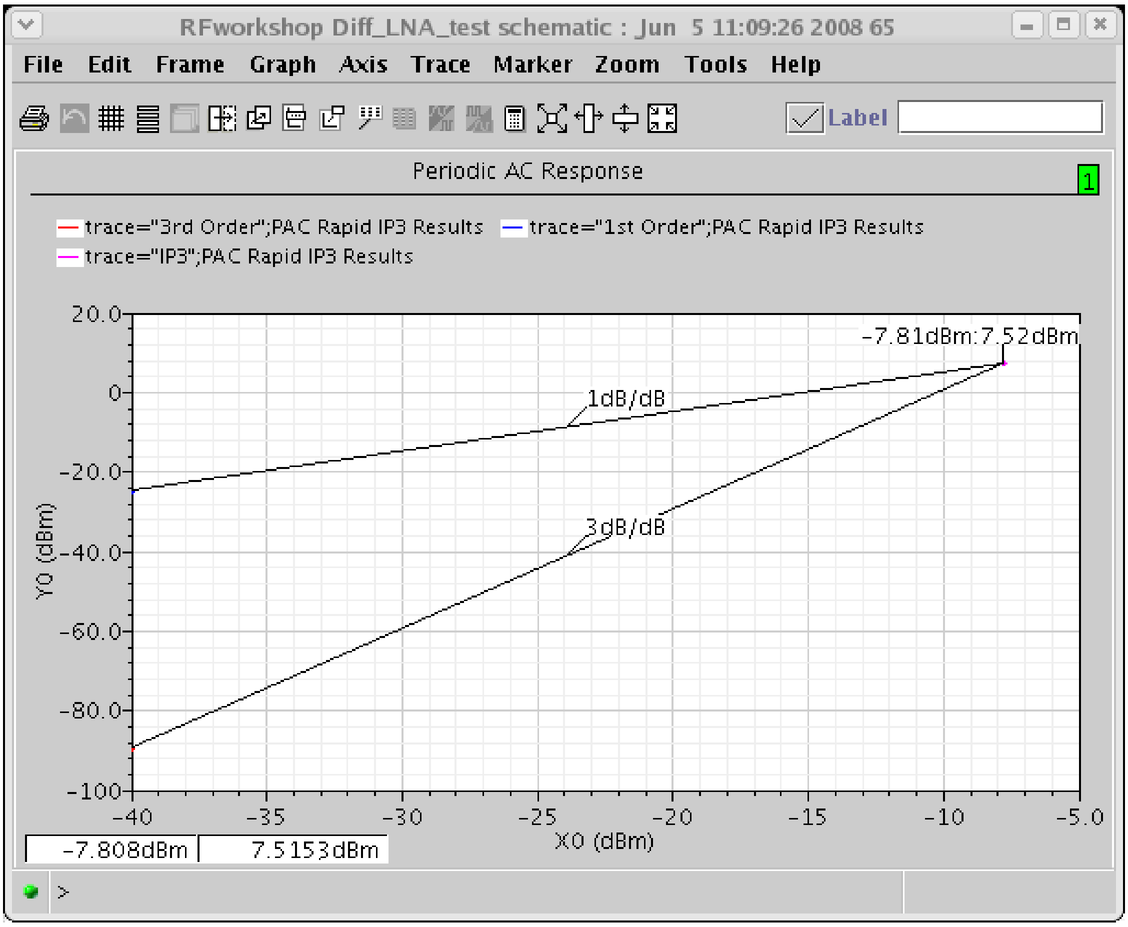
Compare the total time required for the pac analysis with the time required to do the ac analysis in Lab 7. The latter analysis is more efficient for LNA IP3 calculation.

Action 6-13: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 6-14: In the Direct Plot Form, select **pac**, and configure the form as follows:



Action 6-15: Press **plot** button on the Direct Plot Form.



Action 6-16: After viewing the waveforms, close the waveform window. Click **Cancel** in the Direct Plot Form

Lab 7: IP3 Measurement---Rapid IP3 using AC analysis

Action 7-1: If it is not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*

Action 7-2: Select the **PORTrf** source. Choose **Edit — Properties — Objects** and ensure that the port properties are set as described below:

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

Note: The RF input source should be set to DC. The perturbation method is a type of nonlinear small signal analysis that treats the RF signal as a small signal. If the RF input source is set to sinusoidal (or some other type of large signal), the PSS and later small signal results are affected.

Action 7-3: Click **OK** in the Edit Object Properties window to close it.

Action 7-4: Check and save the schematic.

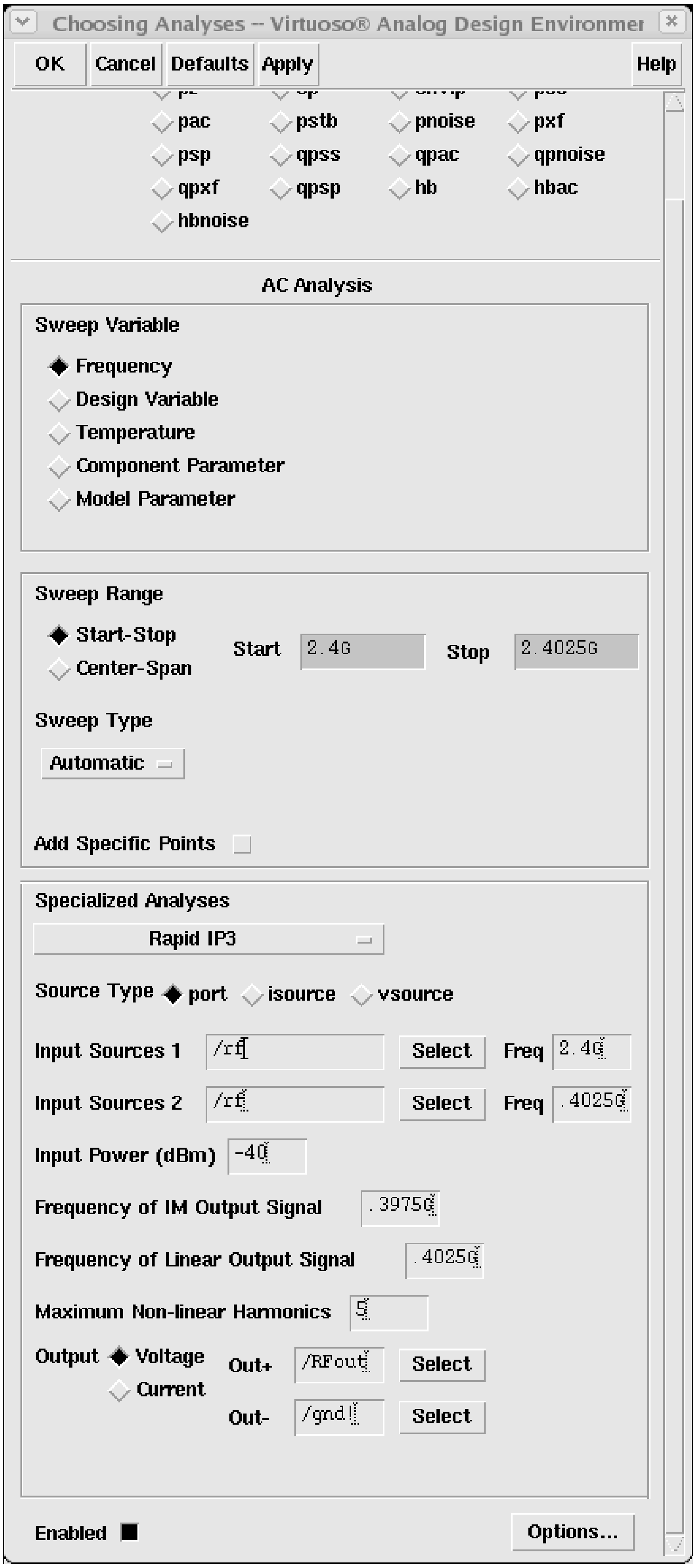
Action 7-5: From the Diff_LNA_test schematic, choose **Tools — Analog** to start the Virtuoso Analog Design Environment.

Action 7-6: (Optional) Choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab7_RapidIP3_AC**" and skip to Action 7-10.

Action 7-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose…**

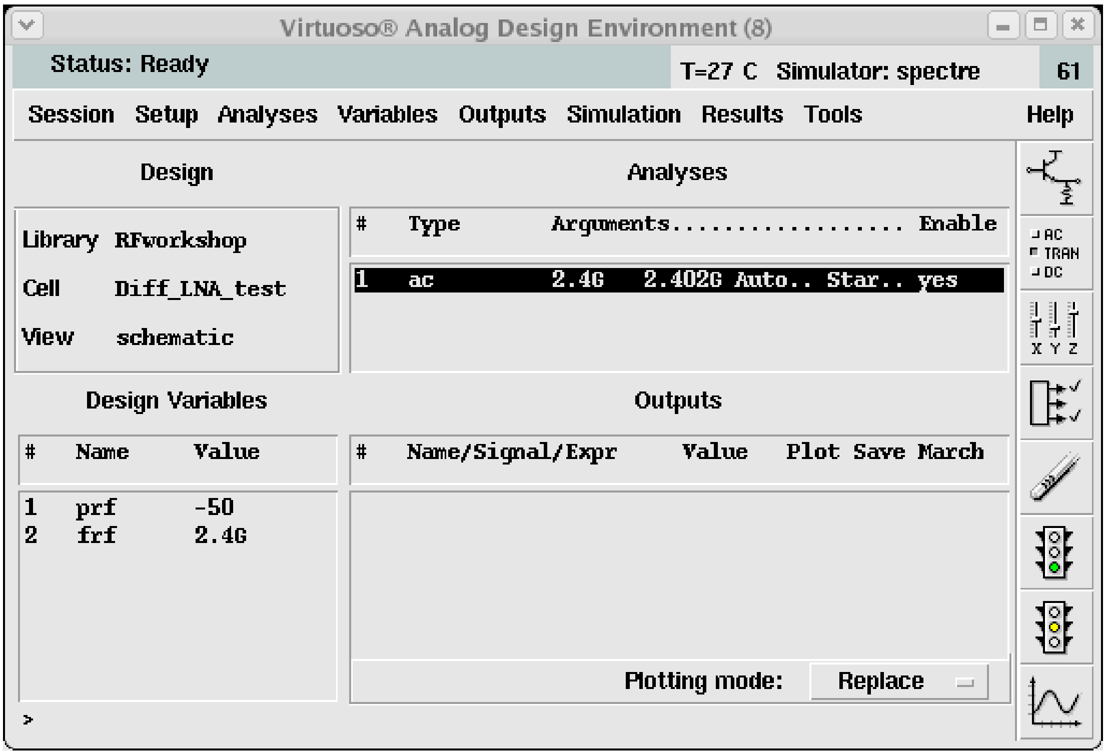
Action 7-8: In the Choosing Analyses window, select **ac** in the **Analysis** field of the window. Choose **Rapid IP3** in the **Specialized Analyses** field. Set the **Input Sources 1** to /rf by selecting PORT rf on the schematic. Push the **ESC** key on your keyboard to terminate the selection process. 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.

After these actions, the form looks like this:


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
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
port
isource
vsource
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...

Action 7-9: Make sure **Enabled** is selected. In the Choosing Analyses window, click **OK**.

The Virtuoso Analog Design Environment window looks like this:



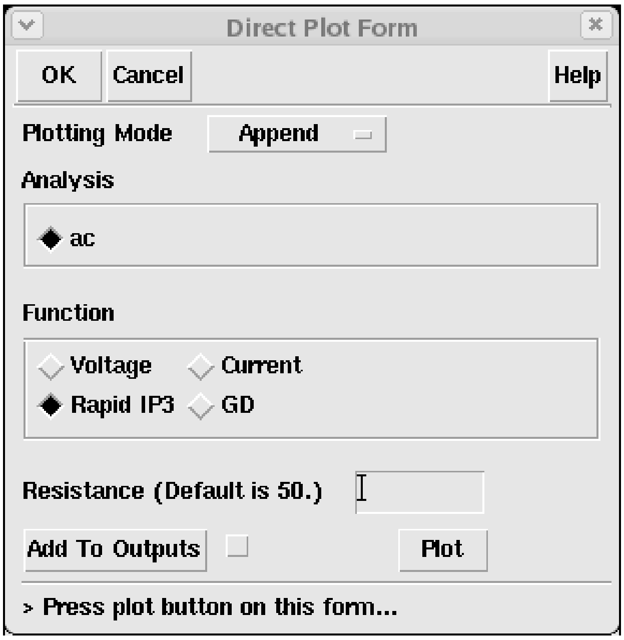
Action 7-10: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

As the simulation progresses, messages similar to the following appear in the simulation output log window:

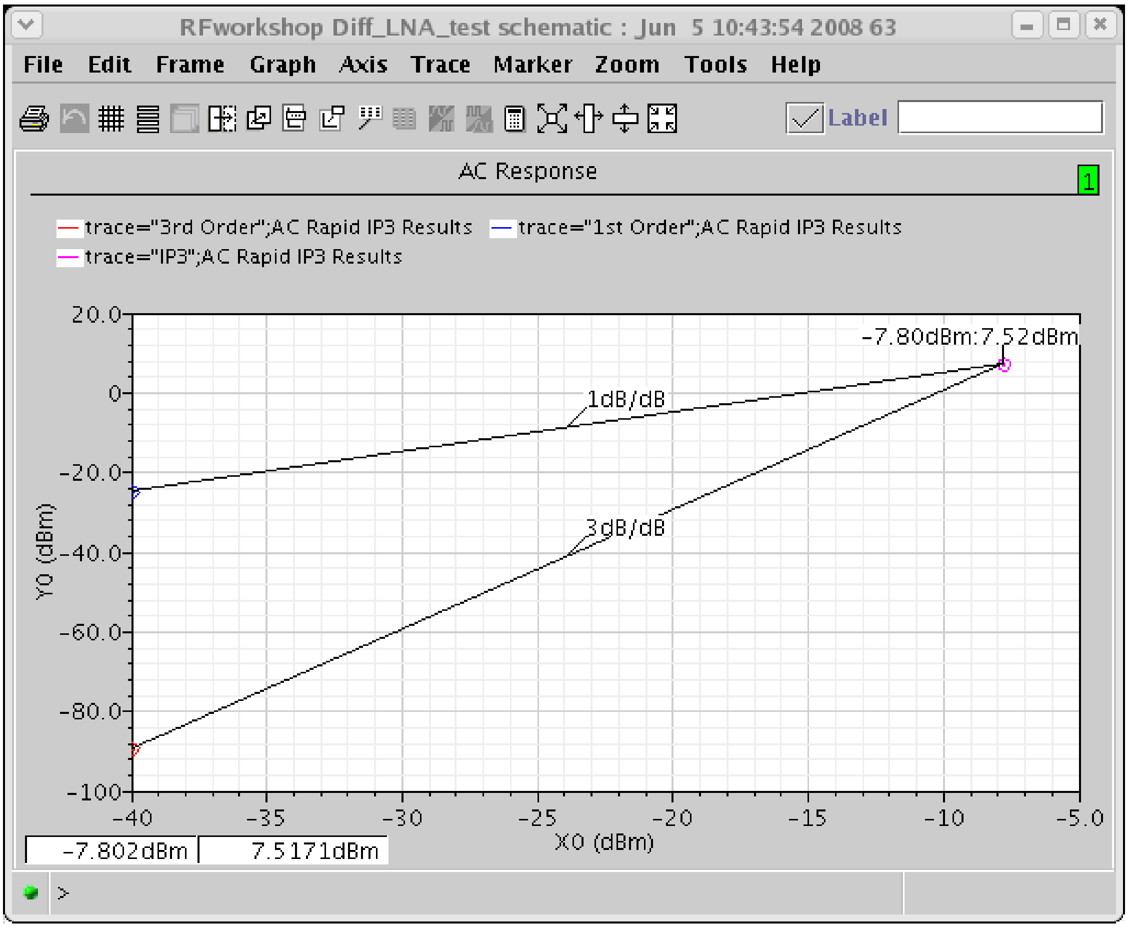
```
********************
IP3 measurement `ac'
********************
Input RF1 freq = 2.4 GHz
Input RF2 freq = 2.4025 GHz
Output IM1 freq = 2.4025 GHz
Output IM3 freq = 2.3975 GHz

Linear output:
f_out = f_in_2

IM3 output:
f_IM3 = 2 * f_in_1 - f_in_2
Accumulated DC solution time = 10 ms.
Intrinsic ac analysis time =          40 ms.
Total time required for ac analysis `ac' was 50 ms.
```

Action 7-11: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 7-12: In the Direct Plot Form, select the **ac** analysis. Choose **Rapid IP3** in the **Function** field.



Action 7-13: Click **Plot** to get the IP3 calculation results:



Action 7-14: After viewing the waveforms, click **Cancel** in the Direct Plot Form.

Conclusion

This application note discusses:

- LNA testbench setup
- LNA design parameters
- How to use SpectreRF to simulate an LNA and extract design parameters
- Useful SpectreRF analysis tools for LNA design, such as SP, PSS, Pnoise, PAC and QPSS analyses

The results from the analyses are interpreted.

References

[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.