RF Design Using Cadence Spectre

RFIC Tutorial

Contents

RF Design Using Spectre

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

Purpose

This tutorial describes how to use Spectre in the Analog Design Environment to measure parameters that are important in verifying mixers.
Audience

Users of Spectre in the Analog Design Environment

Overview

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

Introduction to Mixers

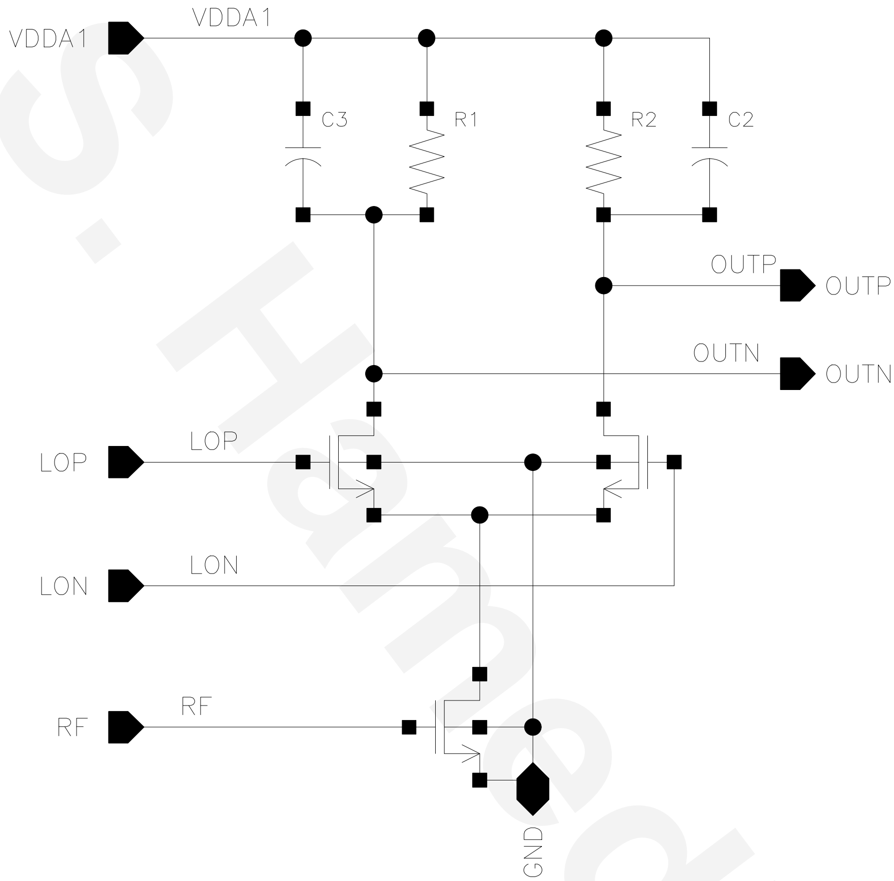
Mixers are key components in both receivers and transmitters. Mixers translate signals from one frequency band to another. The output of the mixer consists of multiple images of the mixer’s input signal where each image is shifted up or down by multiples of the local oscillator (LO) frequency. The most important mixer output signals are usually the signals translated up and down by one LO frequency.

In an ideal situation, the mixer is an exact replica of the input signal. In reality, mixer output is distorted by non-linearity in the mixer. In addition, the mixer components and a non-ideal LO signal add noise to the output. Leakage effects caused by bad mixer designs also complicate the design of the complete system.

Noise performance and the rejection of out-of-band interferers affect the sensitivity of receivers. Linearity affects transmitter performance, where error-free output signals are important.

The Design Example: A Differential Input Mixer

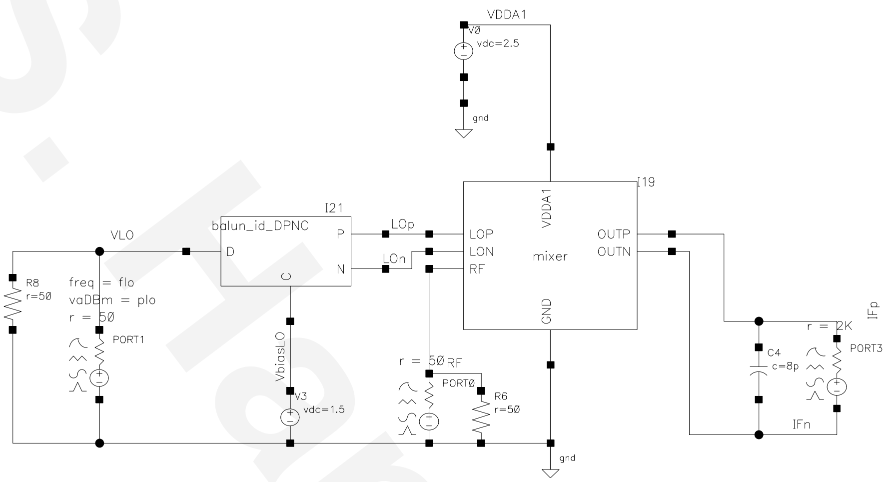
The mixer measurements described in this tutorial are calculated using Spectre in the Analog Design Environment. The design investigated is the mixer shown below.



The example circuit, a single balanced differential down-converting mixer, runs with a local oscillator at f(LO) = 5 GHz. The range of interest is the baseband output noise from 1 kHz to 10 MHz. The RF signal frequency used for the simulation is around 5001 MHz.

Testbench

In this tutorial, you will use the mixer measurements testbench shown below to measure typical mixer characteristics. You use a PORT component and match impedance for each of the inputs and the output.



- To supply a LO input to the mixer, the testbench uses a port (PORT1) with a matching resistor and transfers the single-ended signal into the differential with an ideal passband balun.

- To represent the RF input to the mixer, the testbench uses a port (PORT0) that is matched to the mixer input.

- To use the differential output for measurements, the testbench matches the output port (PORT3) to the output impedance of the mixer.

Simulate the resulting testbench as follows

- Set the LO bias voltage to 1.5 V and set the mixer supply to 2.5 V.

- Set the LO port to a sinusoidal source for all the measurements described in this tutorial.

- Set the RF port to either a dc or a sinusoidal source, depending on the requirements of each measurement. The RF port has a dc bias of 0.5 V.

- For both LO and RF ports, the amplitude and frequency of the signal are parameterized as plo, prf, and frf. You usually specify the amplitude in dBm. In addition, for the RF port, specify the small signal parameter PAC Magnitude. Use pacmag or pacdbm, depending on the units you prefer.

- Set the Output port to *dc* with no bias.

Example Measurements Using Spectre

The Mixer measurements described in the following labs are calculated using Spectre in the Analog Design Environment.

Begin your examination of the flow by bringing up the Cadence Design Framework II environment for a full view of the reference design:

To prepare to run the tutorial :

Action: 0-1 Move into the **./RFhomework** directory.

Action: 0-2 Start the tool **icfb&**.

Action: 0-3 In the CIW window, select **Tools — Library Manager**.

Lab 1: Voltage Conversion Gain Versus LO Signal Power (Swept PSS with PAC)

A mixer's frequency converting action is characterized by conversion gain or loss. The voltage conversion gain is the ratio of the RMS voltages of the IF and RF signals. The power conversion gain is the ratio of the power delivered to the load and the available RF input power.

When the mixer's input impedance and load impedance are both equal to the source impedance, the power and voltage conversion gains, in decibels, are the same. Note that when you load a mixer with a high impedance filter, this condition is not satisfied.

You can calculate the voltage convergence gain in two ways:

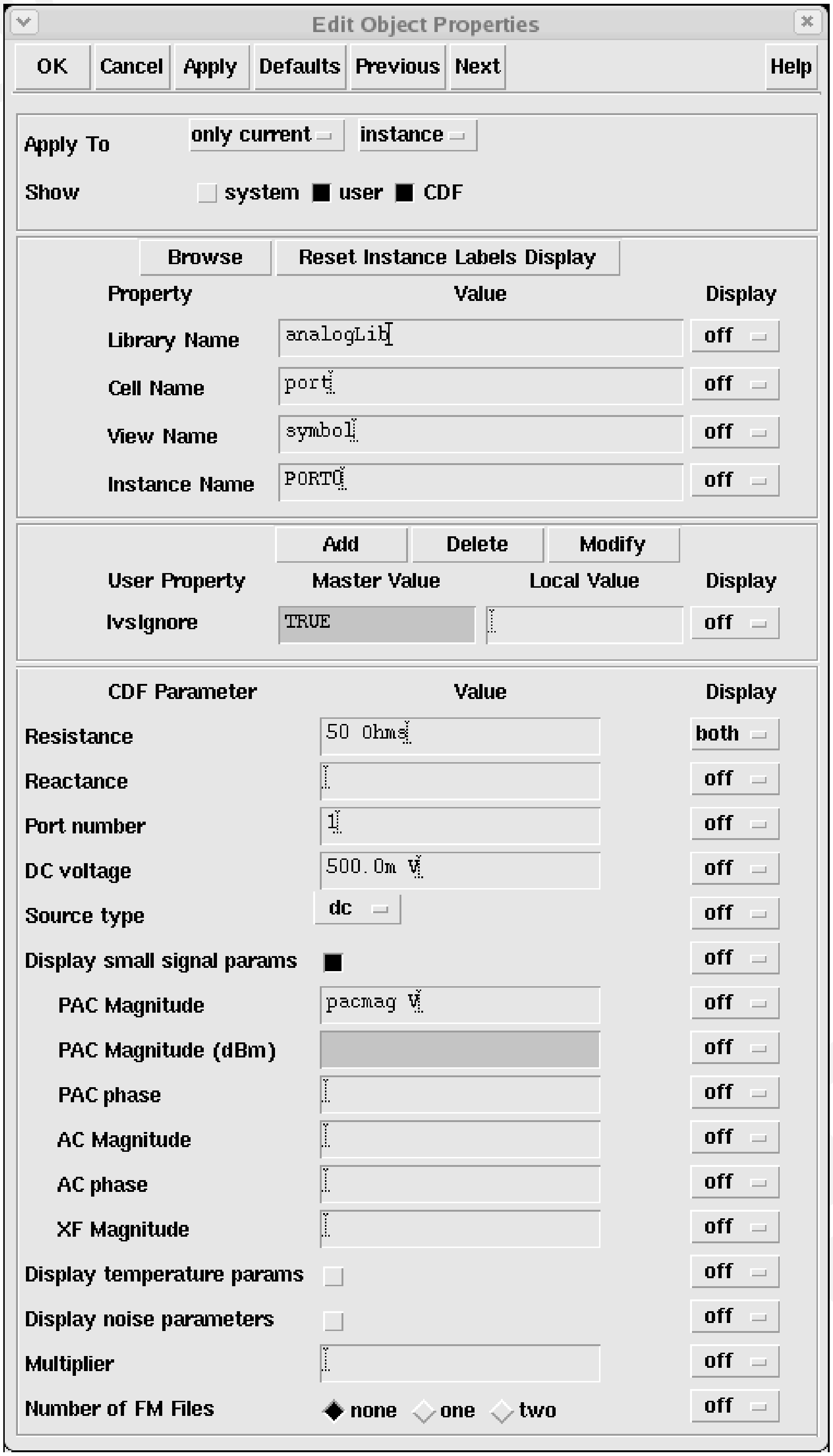
- Using a small-signal analysis, like PSS with PAC or PXF. The PSS with PAC or PXF analyses supply the small-signal gain information. You can use either PAC or PXF analysis to compute the voltage gain.
- Using a two-tone large-signal QPSS analysis, which is more time-consuming. The power convergence gain, in general, requires that you run the two-tone large-signal QPSS analysis.

This example measures the variation of conversion gain with the power of the LO signal.

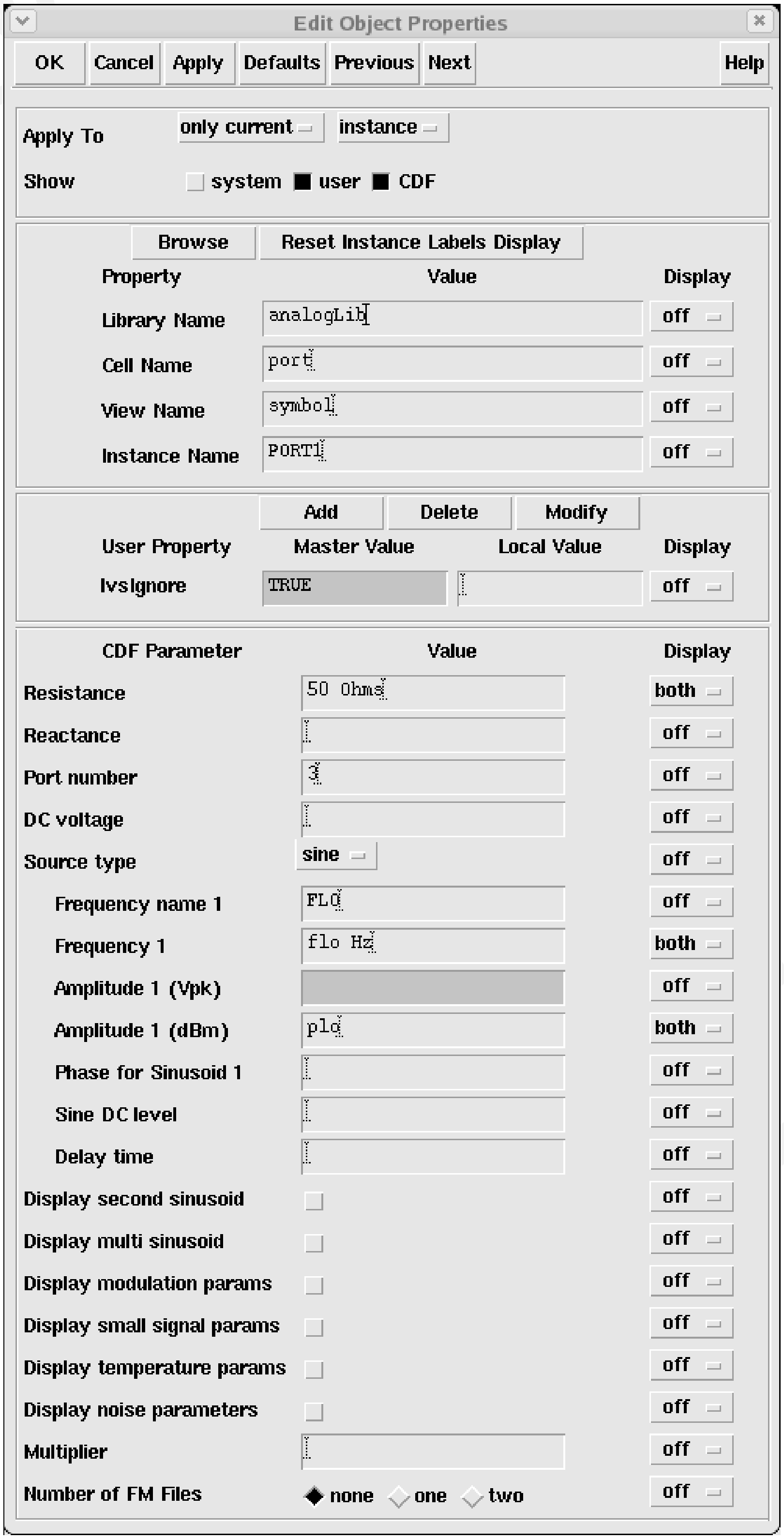
Action 1-1: In the Library Manager window, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework.*

Action 1-2: Use the mouse to select the `PORT0` source. Then, in the Virtuoso Schematic Editor, select **Edit — Properties — Objects.**

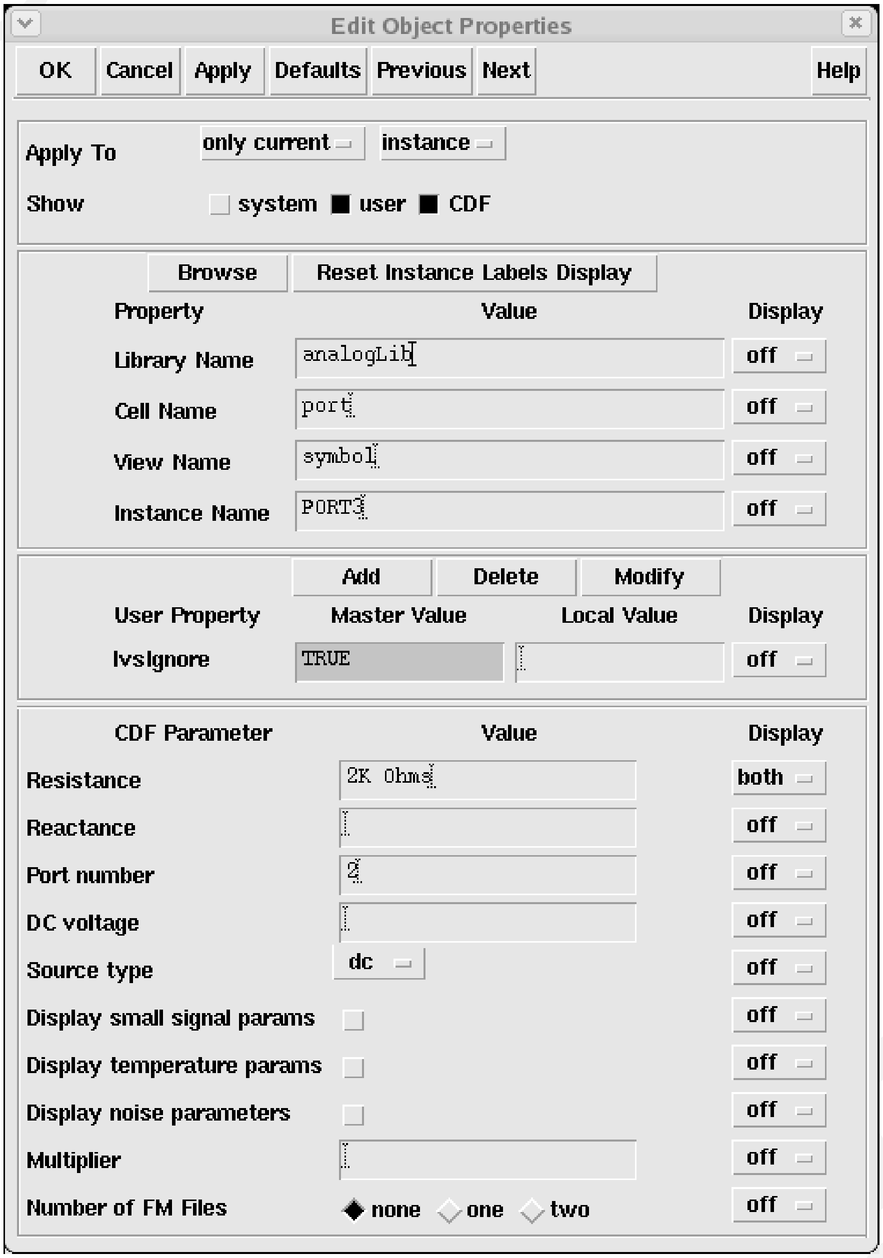
The Edit Object Properties window for the port cell appears.


Edit Object Properties
OK
Cancel
Apply
Defaults
Previous
Next
Help
Apply To
only current
instance
Show
system
user
CDF
Browse
Reset Instance Labels Display
Property
Value
Display
Library Name
analogLib
off
Cell Name
port
off
View Name
symbol
off
Instance Name
PORT0
off
Add
Delete
Modify
User Property
Master Value
Local Value
Display
lvsIgnore
TRUE
off
CDF Parameter
Value
Display
Resistance
50 Ohms
both
Reactance
off
Port number
1
off
DC voltage
500.0m V
off
Source type
dc
off
Display small signal params
off
PAC Magnitude
pacmag V
off
PAC Magnitude (dBm)
off
PAC phase
off
AC Magnitude
off
AC phase
off
XF Magnitude
off
Display temperature params
off
Display noise parameters
off
Multiplier
off
Number of FM Files
none
one
two
off

Action 1-3: Click *OK* on the Edit Object Properties window to close it. Select `PORT1` and show the object properties:


Edit Object Properties
OK
Cancel
Apply
Defaults
Previous
Next
Help
Apply To
only current
instance
Show
system
user
CDF
Browse
Reset Instance Labels Display
Property
Value
Display
Library Name
analogLib
off
Cell Name
port
off
View Name
symbol
off
Instance Name
PORT1
off
Add
Delete
Modify
User Property
Master Value
Local Value
Display
lvsIgnore
TRUE
off
CDF Parameter
Value
Display
Resistance
50 Ohms
both
Reactance
off
Port number
3
off
DC voltage
off
Source type
sine
off
Frequency name 1
FLO
off
Frequency 1
flo Hz
both
Amplitude 1 (Vpk)
off
Amplitude 1 (dBm)
plo
both
Phase for Sinusoid 1
off
Sine DC level
off
Delay time
off
Display second sinusoid
off
Display multi sinusoid
off
Display modulation params
off
Display small signal params
off
Display temperature params
off
Display noise parameters
off
Multiplier
off
Number of FM Files
none
one
two
off

Action 1-4: Make sure the *Source type* for `PORT3` is set to *dc*.



Action 1-5: Check and save the schematic.

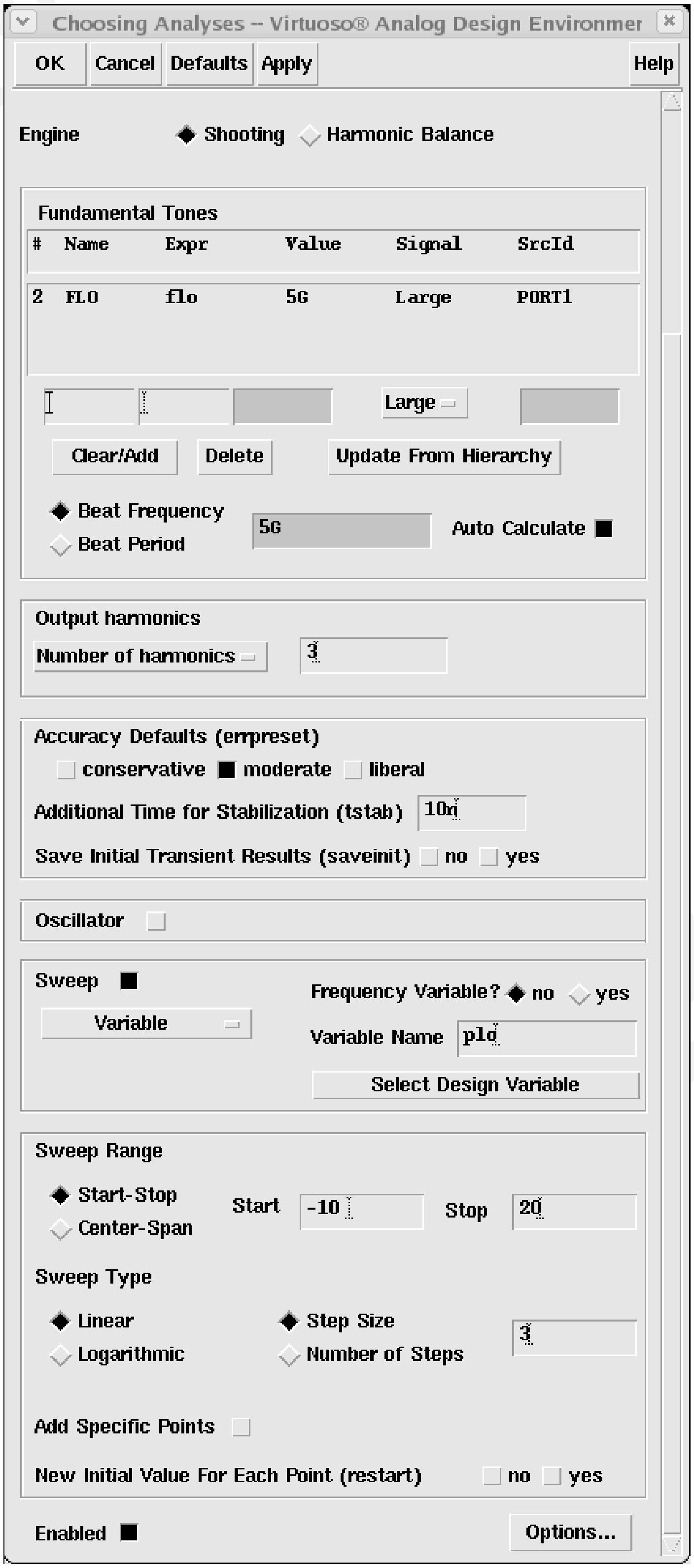
Action 1-6: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

The Virtuoso Analog Design Environment window appears.

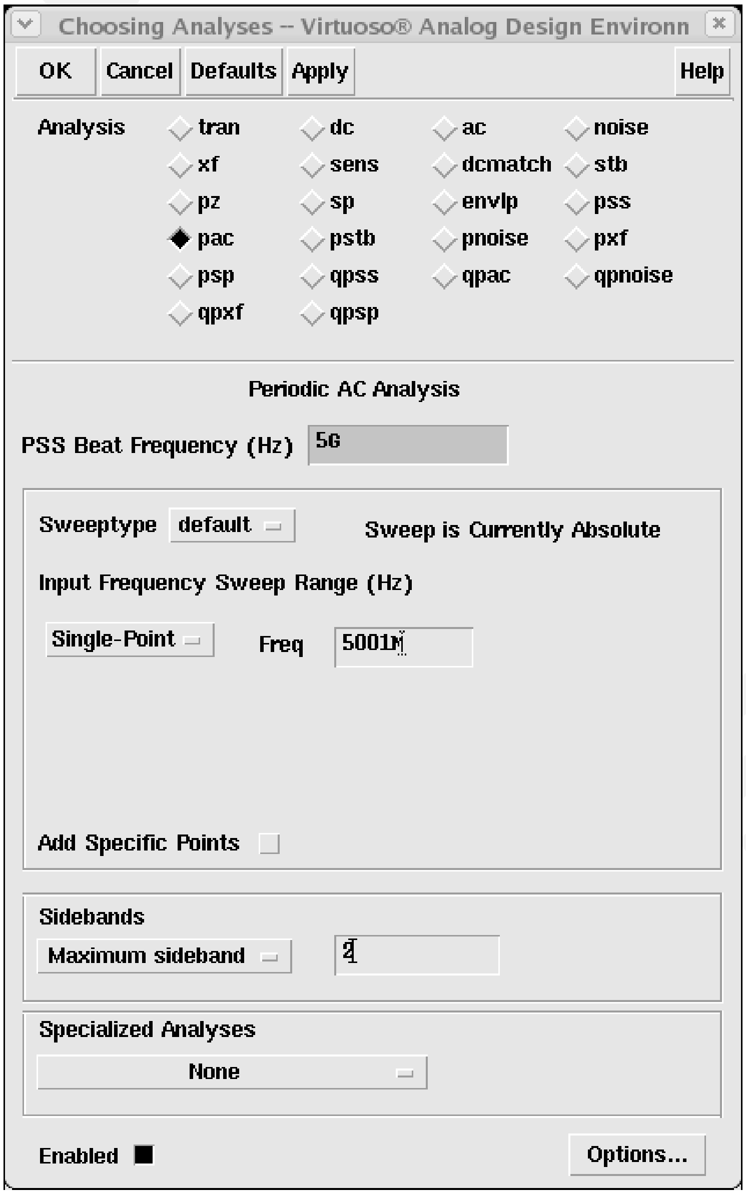
Action 1-7: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab1_VCGvsLO_PSSPAC**" and skip to Action 1-12 or …

Action 1-8: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

Action 1-9: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window. Set the fundamental frequency parameter, flo = 5 GHz. Set *errpreset* = *moderate*. Click *Sweep* and enter plo as the *Variable Name* parameter to sweep LO power. Click *Sweep Range* and set *Start* = -10 dBm and *Stop* = 20 dBm. This sweeps LO power from a small value to a value above the expected gain saturation. The form looks like this:

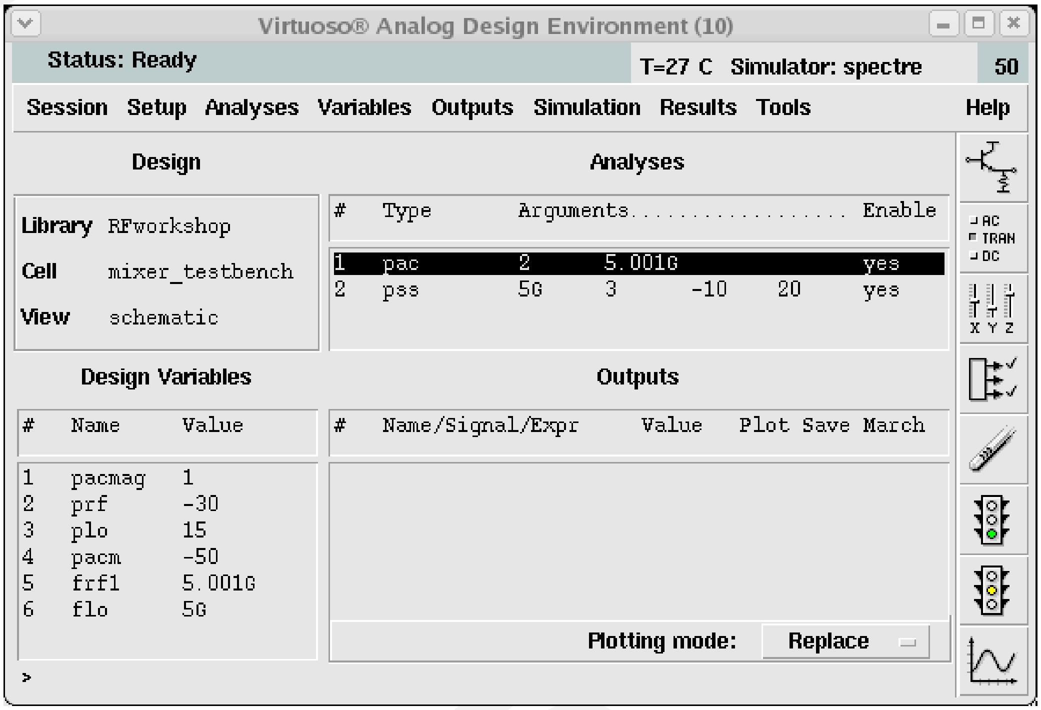

Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId
2 FLO flo 5G Large PORT1
Large
Clear/Add
Delete
Update From Hierarchy
Beat Frequency
Beat Period
5G
Auto Calculate
Output harmonics
Number of harmonics
3
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
10n
Save Initial Transient Results (saveinit)
no
yes
Oscillator
Sweep
Frequency Variable?
no
yes
Variable
Variable Name
plo
Select Design Variable
Sweep Range
Start-Stop
Center-Span
Start
-10
Stop
20
Sweep Type
Linear
Logarithmic
Step Size
Number of Steps
3
Add Specific Points
New Initial Value For Each Point (restart)
no
yes
Enabled
Options...

Action 1-10: In the Choosing Analyses window, select the *pac* button in the *Analysis* field of the window. Set the fixed input frequency point to the RF signal frequency, 5001 MHz. Select sidebands either by specifying *Maximum sideband* = 2 or using *Select from range*. Set the *Maximum sideband* to 2 because you are only interested in the first harmonics of LO. The form looks like this.



Action 1-11: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

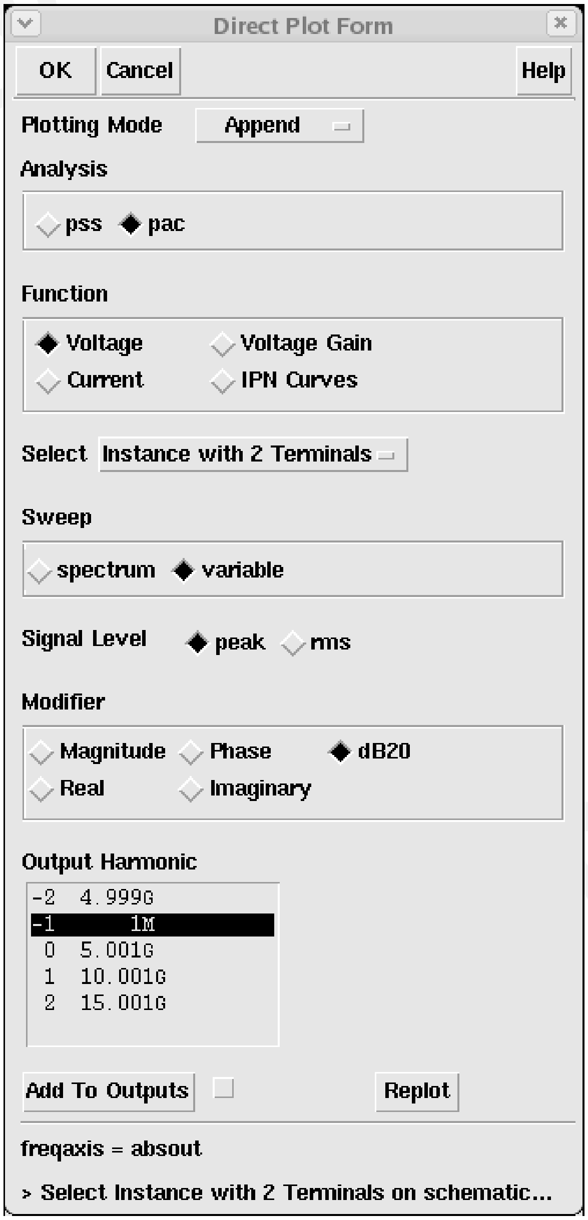
The Virtuoso Analog Design Environment window looks like this.



Action 1-12: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

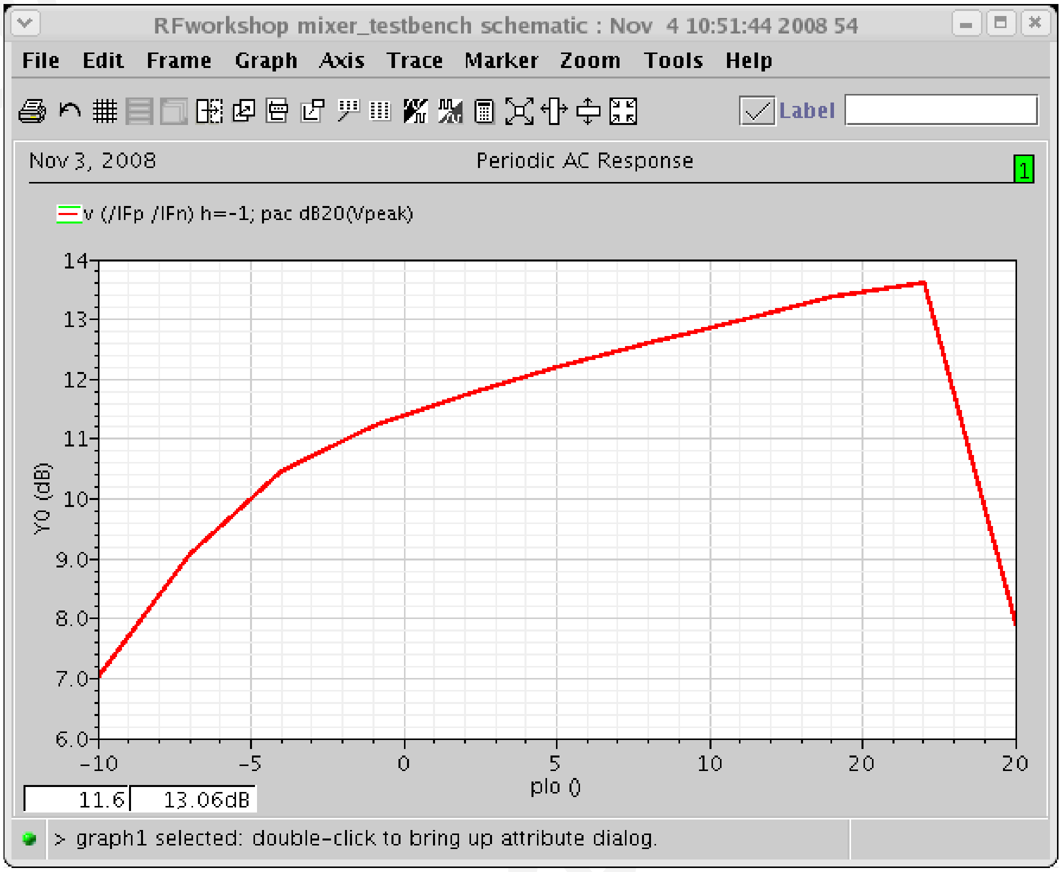
Action 1-13: After the simulation completes, in the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 1-14: In the Direct Plot Form, select the *pac* button, and configure the form as follows:



Action 1-15: Select `port3` on the schematic.

You get the following waveform:



The PAC analysis computes gain directly only when you set the pacmag parameter to 1 V. Otherwise, take a ratio of the output and input. The maximum conversion gain value is reached somewhere above 15 dBm. Use this value for the plo parameter in the following measurements. However, for most CMOS transceiver design, the RF LO power is about 0~3 dBm.

Action 1-16: After viewing the waveforms, click *Cancel* in the Direct Plot Form. Close the waveform window.

Lab 2: Voltage Conversion Gain Versus RF Frequency (PSS and Swept PAC)

This lab measures how conversion gain varies with the frequency of the stimuli.

Action 2-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework.*

Action 2-2: Set port as you did in Lab 1.

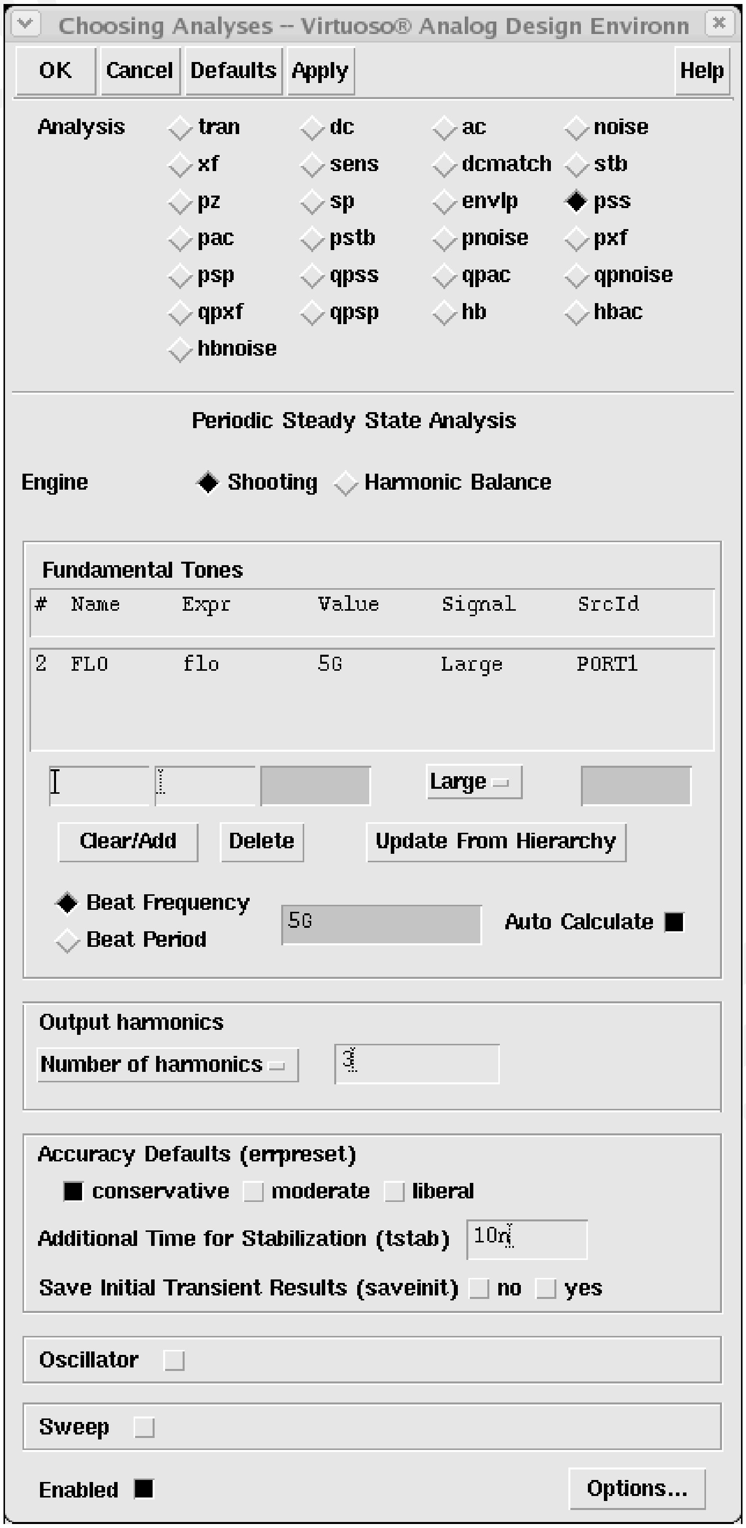
Action 2-3: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment.**

The Virtuoso Analog Design Environment window appears.

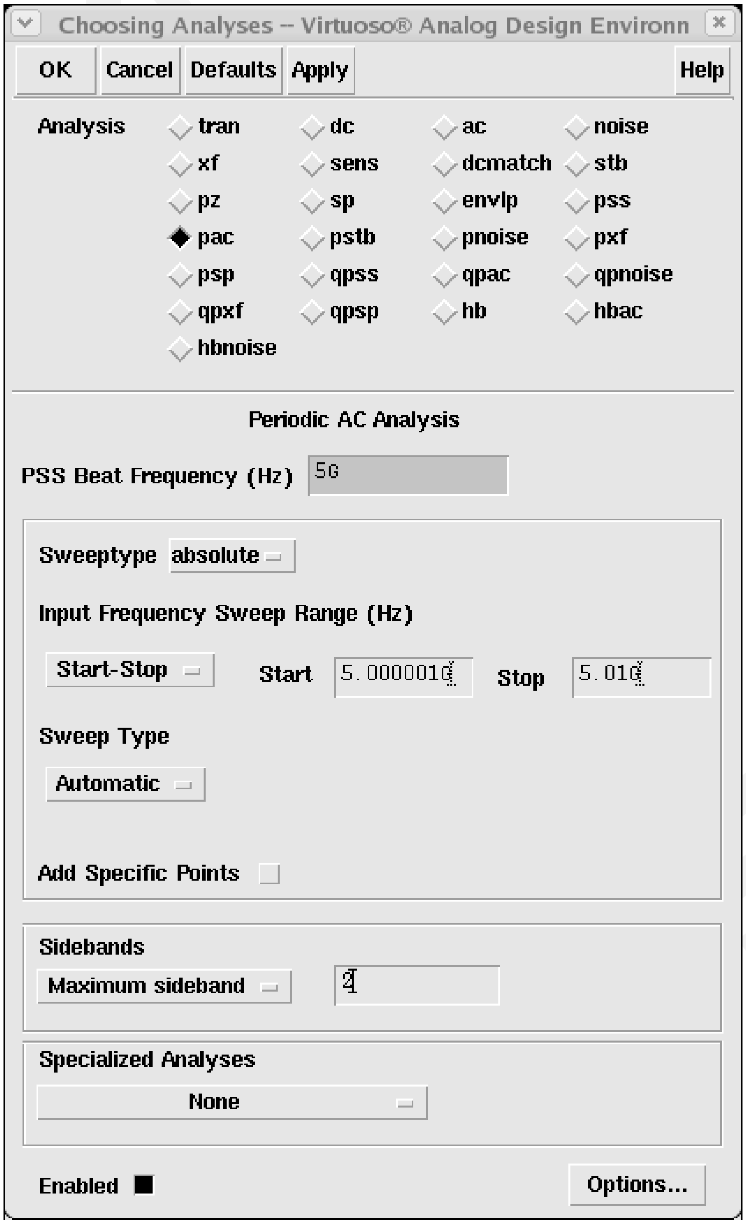
Action 2-4: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab2_VCGvsRF_PSSPAC**" and skip to Action 2-9 or …

Action 2-5: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

Action 2-6: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window and set the form as follows:

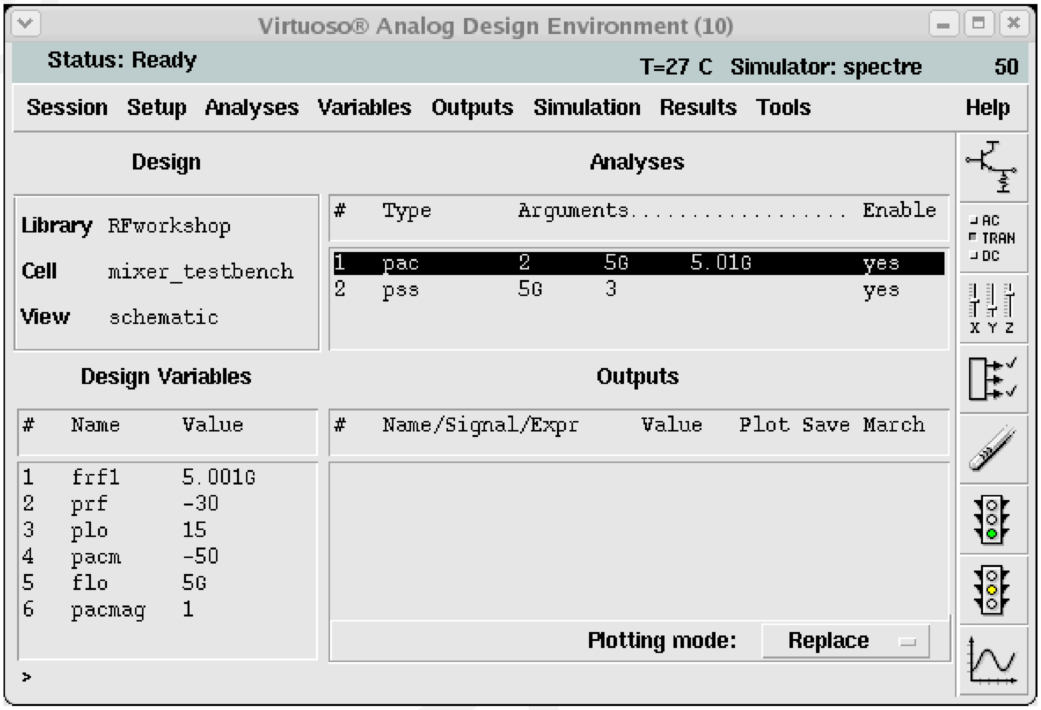

Choosing Analyses -- Virtuoso® Analog Design Environn
OK
Cancel
Defaults
Apply
Help
Analysis
tran
dc
ac
noise
xf
sens
dcmatch
stb
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Periodic Steady State Analysis
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId
2 FLO flo 5G Large PORT1
Large
Clear/Add
Delete
Update From Hierarchy
Beat Frequency
Beat Period
5G
Auto Calculate
Output harmonics
Number of harmonics
3
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
10n
Save Initial Transient Results (saveinit)
no
yes
Oscillator
Sweep
Enabled
Options...

Action 2-7: In the Choosing Analyses window, select the *pac* button in the *Analysis* field of the window. Set the RF input frequency to sweep from 5 G + 1 kHz to 5 G + 10 MHz. Select sidebands either by specifying *Maximum sideband* = 2 or using *Select from range*. The form looks like this.



Action 2-8: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

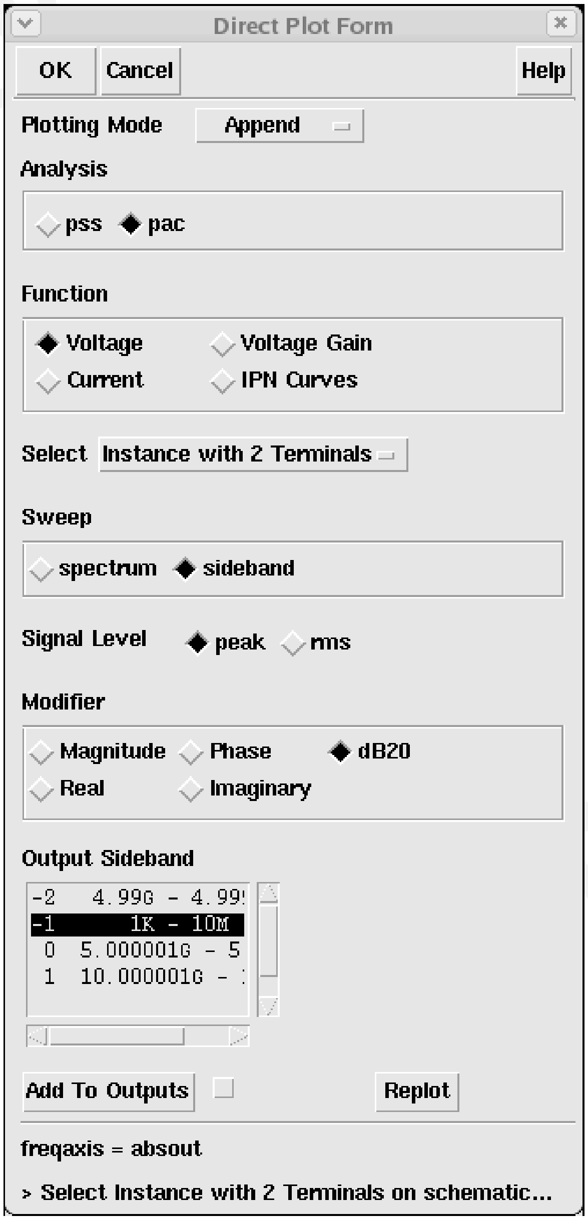
The Virtuoso Analog Design Environment window looks like this.



Action 2-9: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action 2-10: After the simulation completes, in the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 2-11: In the Direct Plot Form, select the *pac* button, and configure the form as follows:



Action 2-12: Select `port3` on the schematic.

The waveform window appears:

Because the sweep type in the analysis is linear by default, uniform frequency points display along the X-axis in the above plot. For a large frequency range, set the sweep type to logarithmic.

The same PAC analysis generates results you can use to measure RF to LO isolation. The results are also used in the measurements that follow.

Action 2-13: Close the waveform window and click *Cancel* in the Direct Plot Form.

Lab 3: Voltage Conversion Gain Versus RF Frequency (PSS and Swept PXF)

This example uses PXF analysis to measure the small-signal voltage conversion gain.

Action 3-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework*

Action 3-2: Set the ports as you did in Lab1.

Action 3-3: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

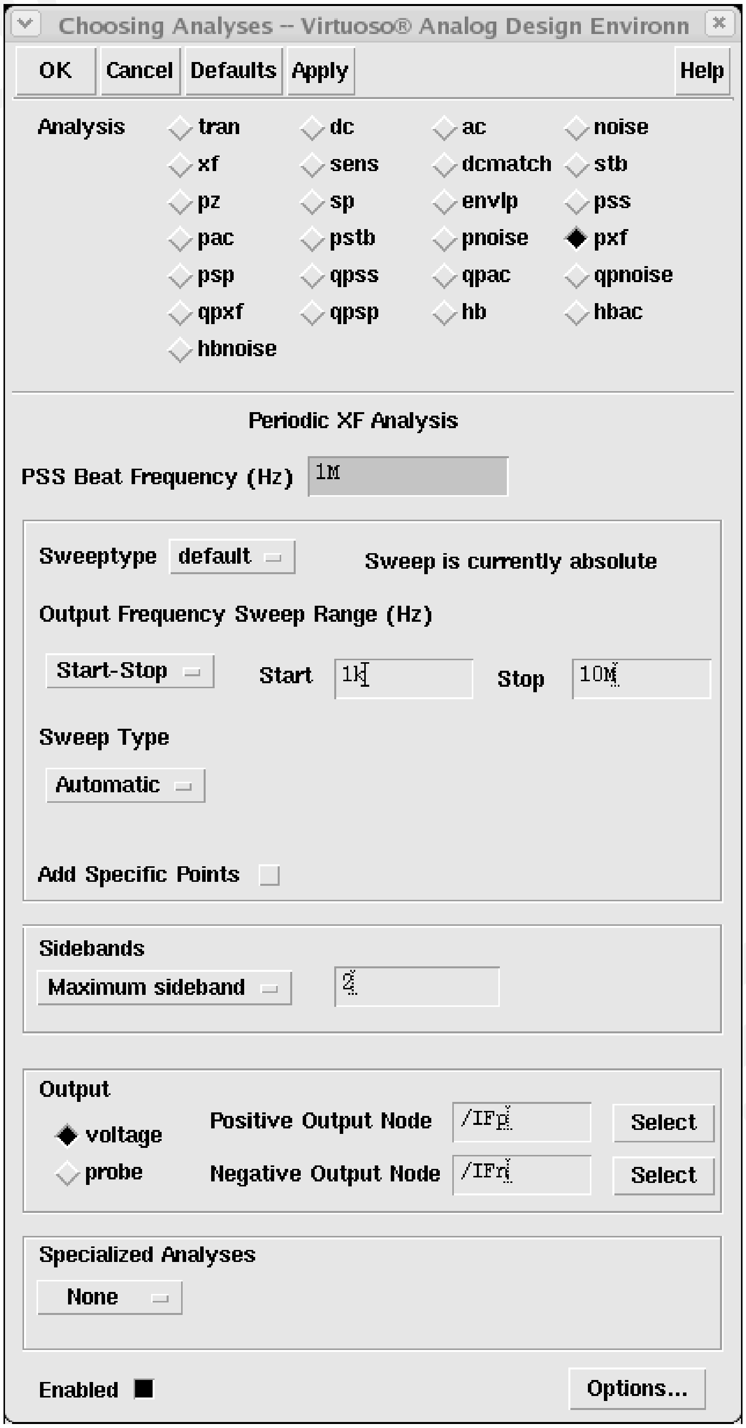
The Virtuoso Analog Design Environment window appears.

Action 3-4: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab3_VCGvsRF_PSSPXF**" and skip to Action 3-9 or …

Action 3-5: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

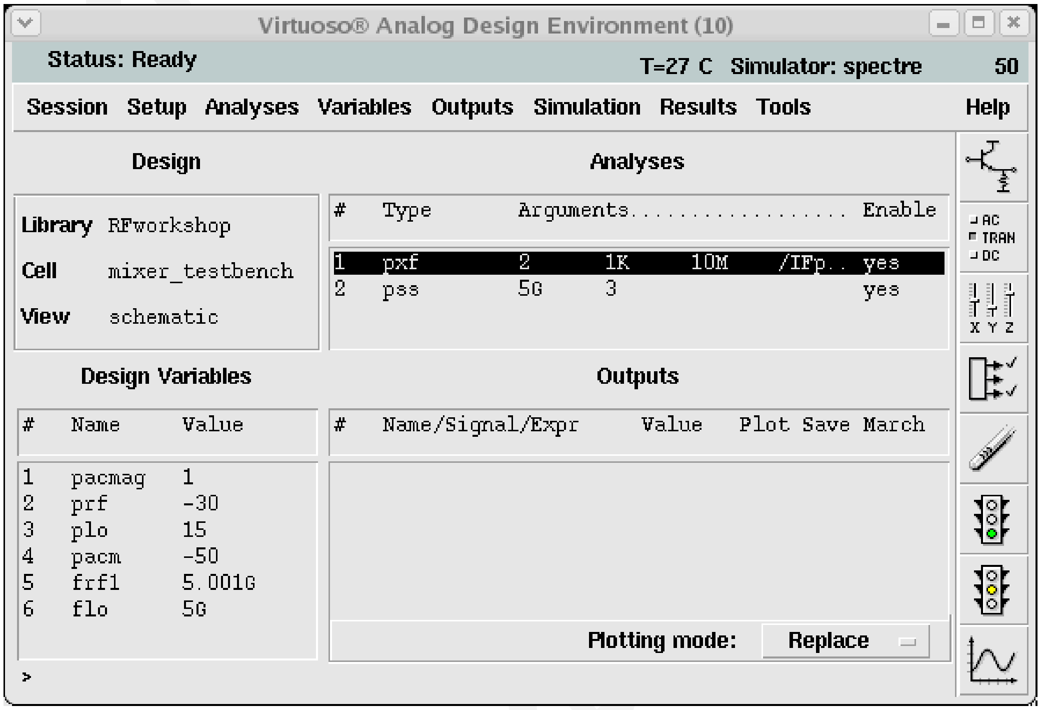
Action 3-6: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window. Set the fundamental frequency parameter, flo = 5 GHz or use the *Auto Calculate* button. Set *errpreset* = *moderate*. The form is the same as in Action 2-6.

Action 3-7: In the Choosing Analyses window, select the *pxf* button in the *Analysis* field of the window. Sweep the output frequency from 1 kHz to 10 MHz. Select sidebands either by specifying *Maximum sideband* = 2 or by using *Select from range*. The form looks like this:



Maximum sideband = 2 is good for this example, but other circuits might require a different value. Set the *Output* by specifying output as *voltage* with *Positive Output Node* being the Ifp net and *Negative Output Node* being the Ifn net.

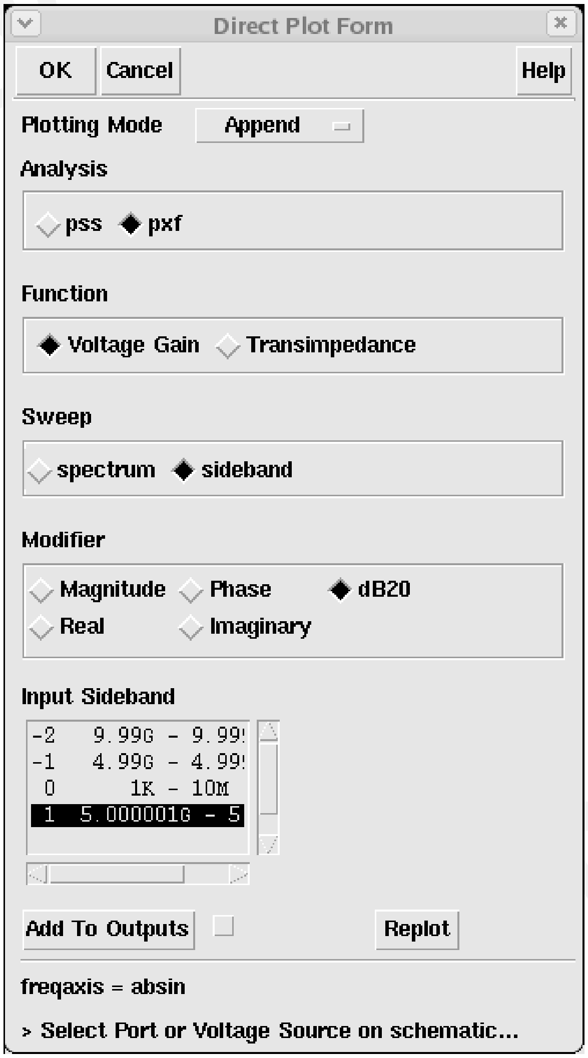
Action 3-8: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.
The Virtuoso Analog Design Environment window looks like this.



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.

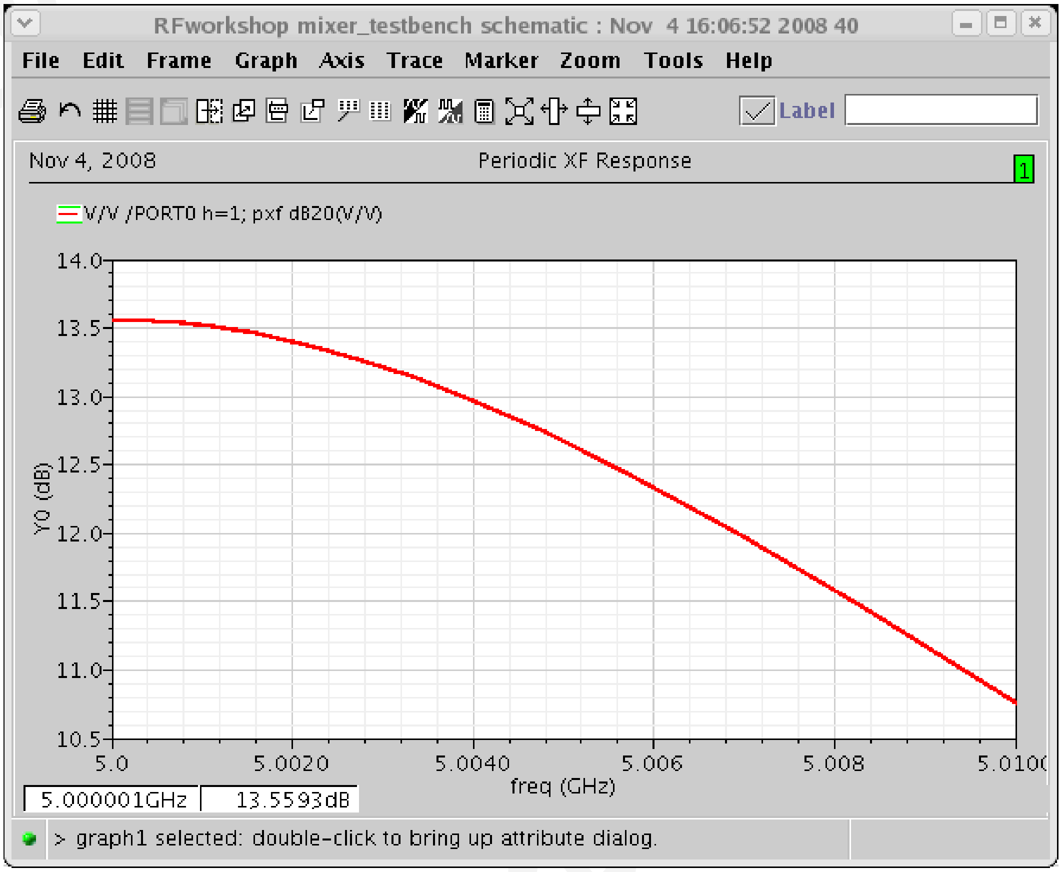
Action 3-10: After the simulation completes, in the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

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



Action 3-12: Select input `port0` on the schematic.

The waveform window appears:



Action 3-13: After viewing the waveforms, click *Cancel* in the Direct Plot Form.

Another way to measure small-signal gain is to use the PSS and PSP analyses to get the gain and noise parameters with one simulation. For more information, refer to the Spectre user guide Appendix L (using psp and pnoise analysis).

You can also set up an appropriate QPSS analysis to measure large-signal gain. Set LO as a large tone on the Plo port. Use a sinusoidal voltage source for the Prf port. This analysis models the signal at a particular frequency going through the mixer. In the Direct Plot Form for QPSS, the Voltage and Power Gain provide all the needed information.

Lab 4: Power Conversion Gain Versus RF Frequency (QPSS)

You can measure the Power Conversion Gain and Power Dissipation for an unmatched source and load using a QPSS analysis. If the effect of the RF tone is small, you might use a PSS analysis instead, as mentioned in previous sections.

The QPSS and PSS analyses provide only spectrum data, not a scalar value, of the total power. To get a scalar value for total power, work through the summation over the harmonics and sidebands. In general, most of the power is in the main output harmonics.

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

Action 4-2: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

The Virtuoso Analog Design Environment window appears.

Action 4-3: Use the mouse to select the `PORT0` source. Then, in the Virtuoso Schematic Editor, select **Edit — Properties — Objects**. The Edit Object Properties window for the port cell appears. Make sure the properties are set as follows:

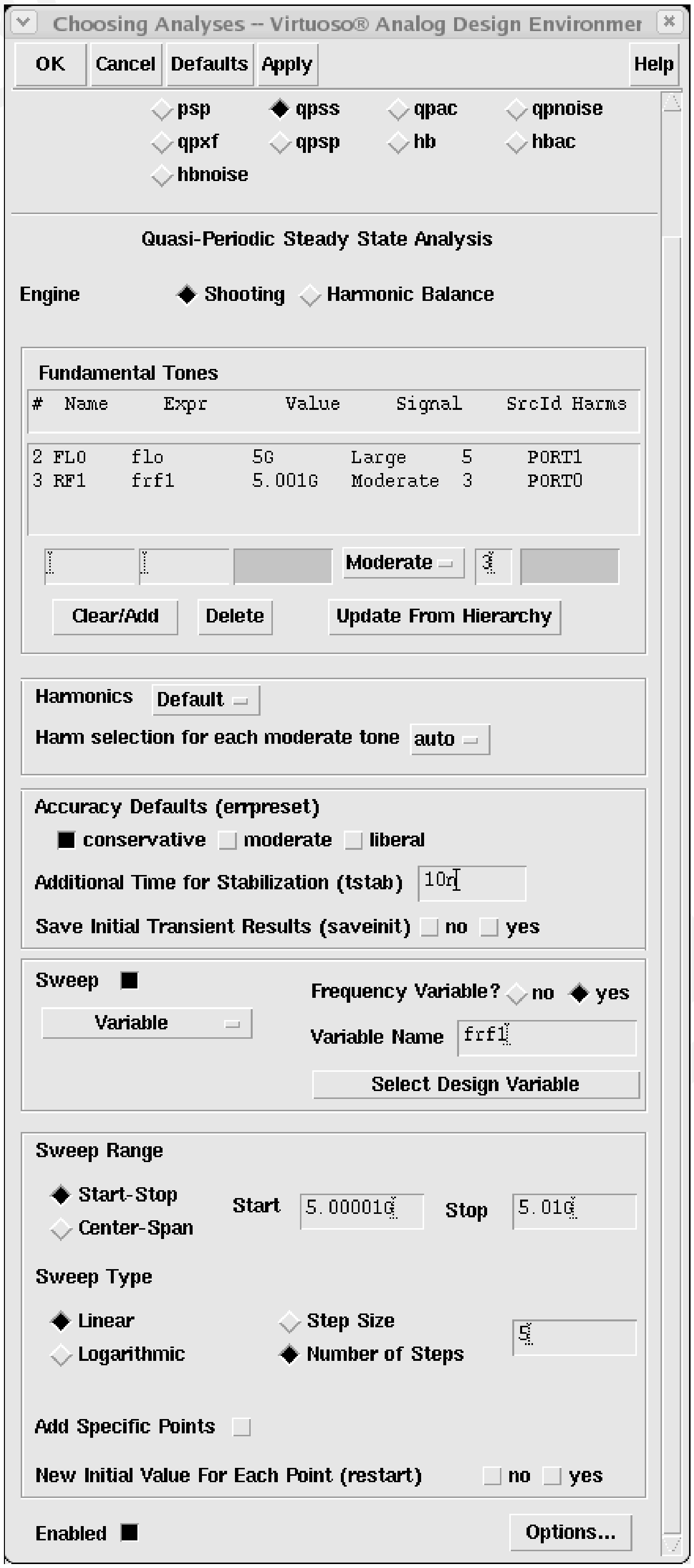
Parameter	**Value**
Resistance	50 ohm
Port Number	1
DC voltage	500 mV
Source type	*sine*
Frequency name 1	RF1
Frequency 1	frf1
Amplitude 1 (dBm)	prf

Action 4-4: Check and save the schematic.

Action 4-5: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab4_PCG_QPSS**" and skip to Action 4-12 or …

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

Action 4-7: In the Choosing Analyses window, select the *qpss* button in the *Analysis* field of the window. Set LO as a *Large* tone with flo = 5 GHz. Set RF as a *Moderate* tone with frf = 5.01 GHz. Set *errpreset* to *moderate*. Limit the maximum harmonic value to the maximum index of interest plus one more. For this example, set LO = 5 and RF = 3. You can select any reasonable number of LO harmonics because the large tone is modeled in the time domain and you can analyze up to 40 harmonics without a runtime penalty.


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Quasi-Periodic Steady State Analysis
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId Harms
2 FLO flo 5G Large 5 PORT1
3 RF1 frf1 5.001G Moderate 3 PORT0
Moderate
3
Clear/Add
Delete
Update From Hierarchy
Harmonics
Default
Harm selection for each moderate tone
auto
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
10n
Save Initial Transient Results (saveinit)
no
yes
Sweep
Frequency Variable?
no
yes
Variable
Variable Name
frf1
Select Design Variable
Sweep Range
Start-Stop
Center-Span
Start
5.00001G
Stop
5.01G
Sweep Type
Linear
Logarithmic
Step Size
Number of Steps
5
Add Specific Points
New Initial Value For Each Point (restart)
no
yes
Enabled
Options...

Action 4-8: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

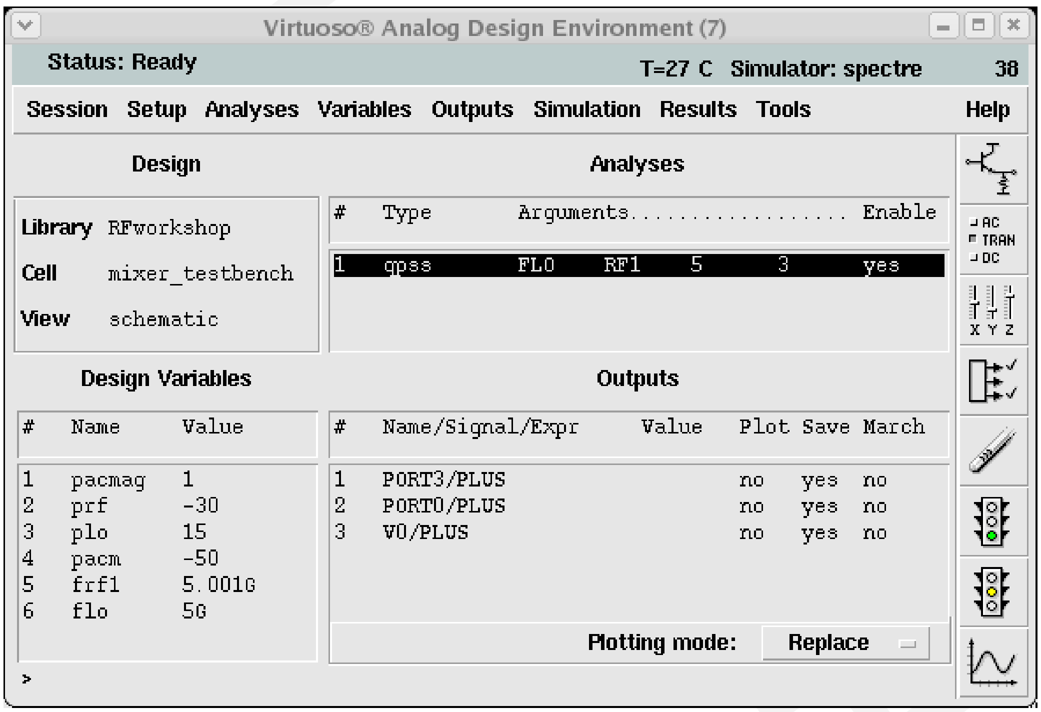
Action 4-9: In the Virtuoso Analog Design Environment window, choose **Outputs — To Be Saved — Select on Schematic**.

Action 4-10: In the schematic, select the positive terminals of `port0`, `port3`, and `V0`.

After you select them, the terminals are circled in the schematic window, indicating that you are saving the currents at these nodes.

Action 4-11: With your cursor in the schematic window, press **Esc** to end the selections.

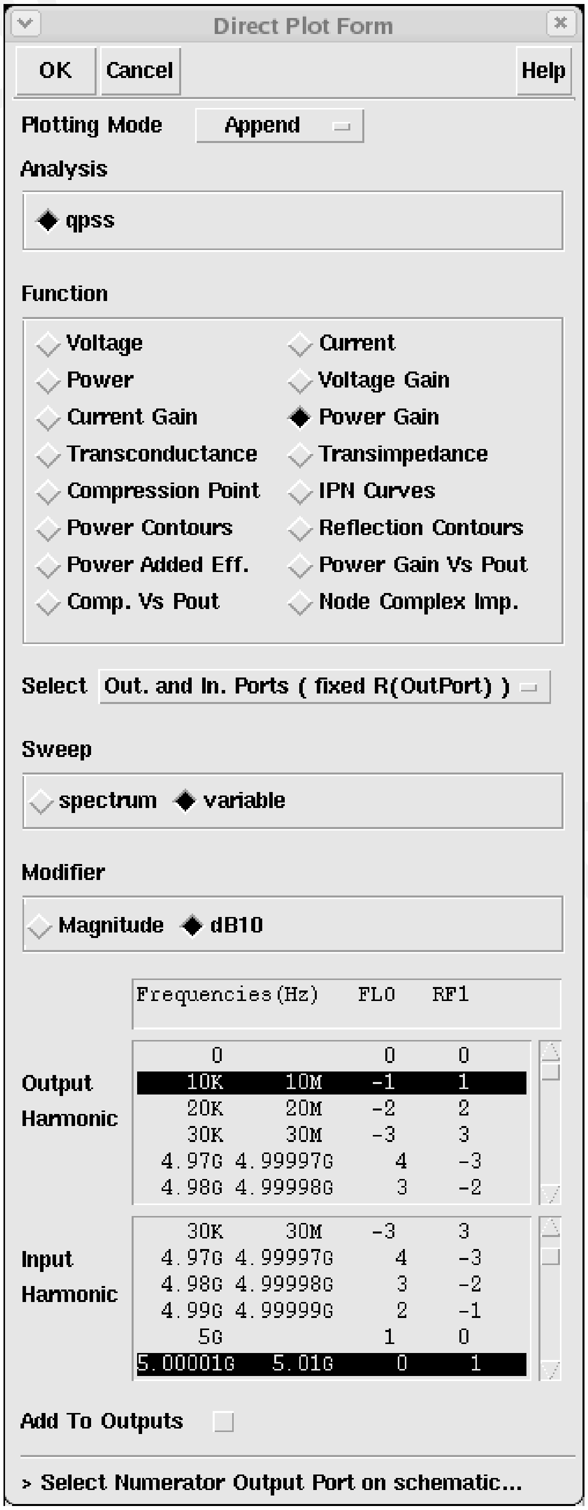
The Virtuoso Analog Design Environment window looks like this.



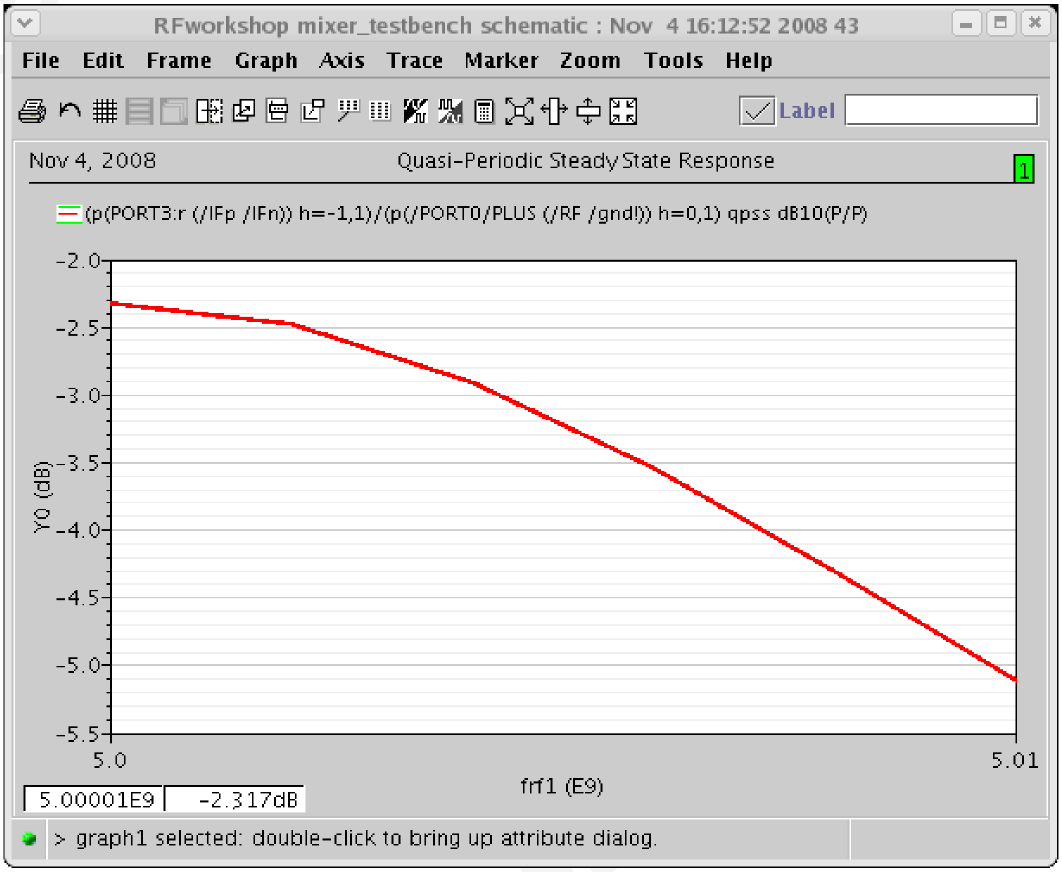
Action 4-12: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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

Action 4-14: In the Direct Plot Form, select the *qpss* button, and configure the form as follows:



Action 4-15: Select output `port3` and input `port0` on the schematic. The waveform window appears.



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

Lab 5: Periodic S-Parameters (PSS and PSP)

The receiver amplifies the small input signals to the point where they can be processed by the baseband section. You develop a gain budget where every stage in the receiver is assigned the gain it is expected to provide. Therefore, the signal gain or loss provided by the mixer must be known.

There are various ways of characterizing gain and all are derived from the mixer's S-parameters. As such, it must be easy to calculate the various S-parameters of the circuit and apply the various gain metrics.

Action 5-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework*.

Action 5-2: Use the mouse to select the `PORT0` source. Then in the Virtuoso Schematic Editor select **Edit — Properties — Objects**. The Edit Object Properties window for the port cell appears. Change the *Source type* to *dc*. Blank out all the Second Sinusoid fields before setting the *Source type* to *dc*.

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	500 mV
Source type	*dc*
PAC Magnitude	(blank)

For small-signal analysis, it is often sufficient to treat the RF input as a small signal (for example, by setting the *Source type* to *dc*). However, sometimes it is important to analyze additional noise folding terms induced by the RF input (larger signal interferer). In those cases, the RF source is a large signal and the *Source type* is *sine*.

Action 5-3: Set **lo** port to *sine* type and output port to *dc* type.

Action 5-4: Check and save the schematic.

Action 5-5: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment.**

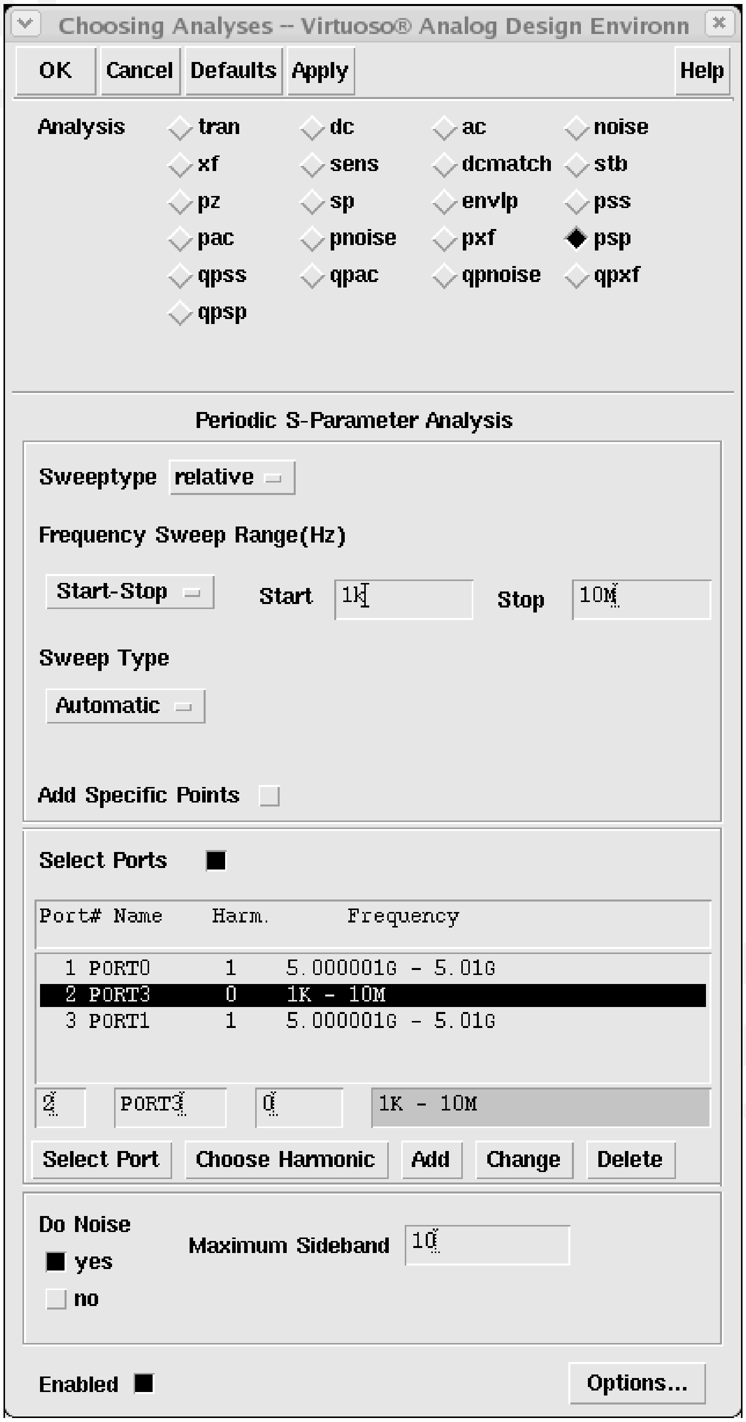
The Virtuoso Analog Design Environment window appears.

Action 5-6: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab5_SParameter_PSSPSP**" and skip to Action 5-11 or …

Action 5-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

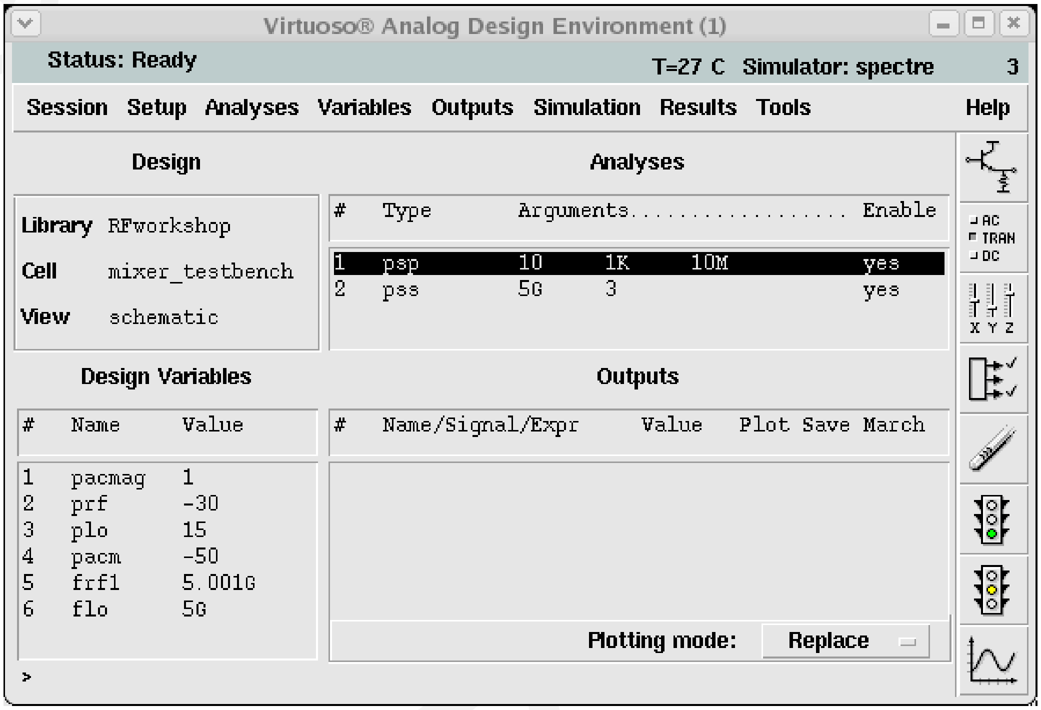
Action 5-8: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window. Set the form as you did in Action 2-6.

Action 5-9: In the Choosing Analyses window, select the *psp* button in the *Analysis* field of the window and set the form as follows:



Action 5-10: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

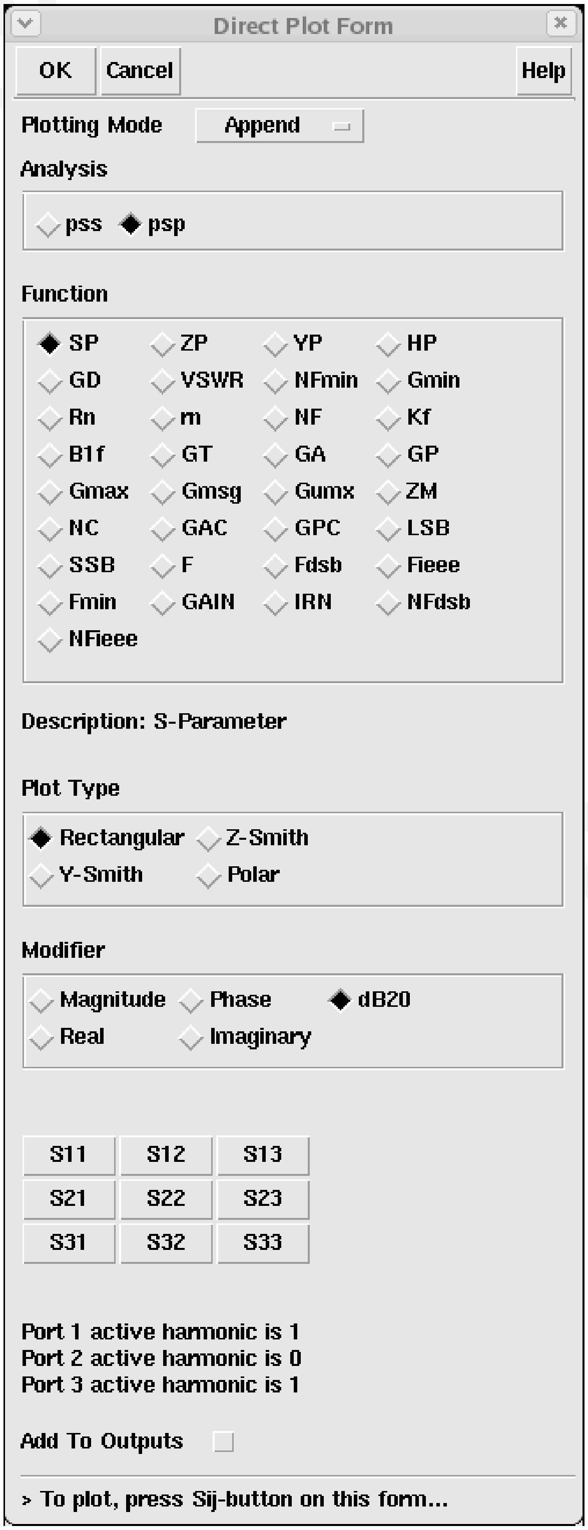
The Virtuoso Analog Design Environment window looks like this.



Action 5-11: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

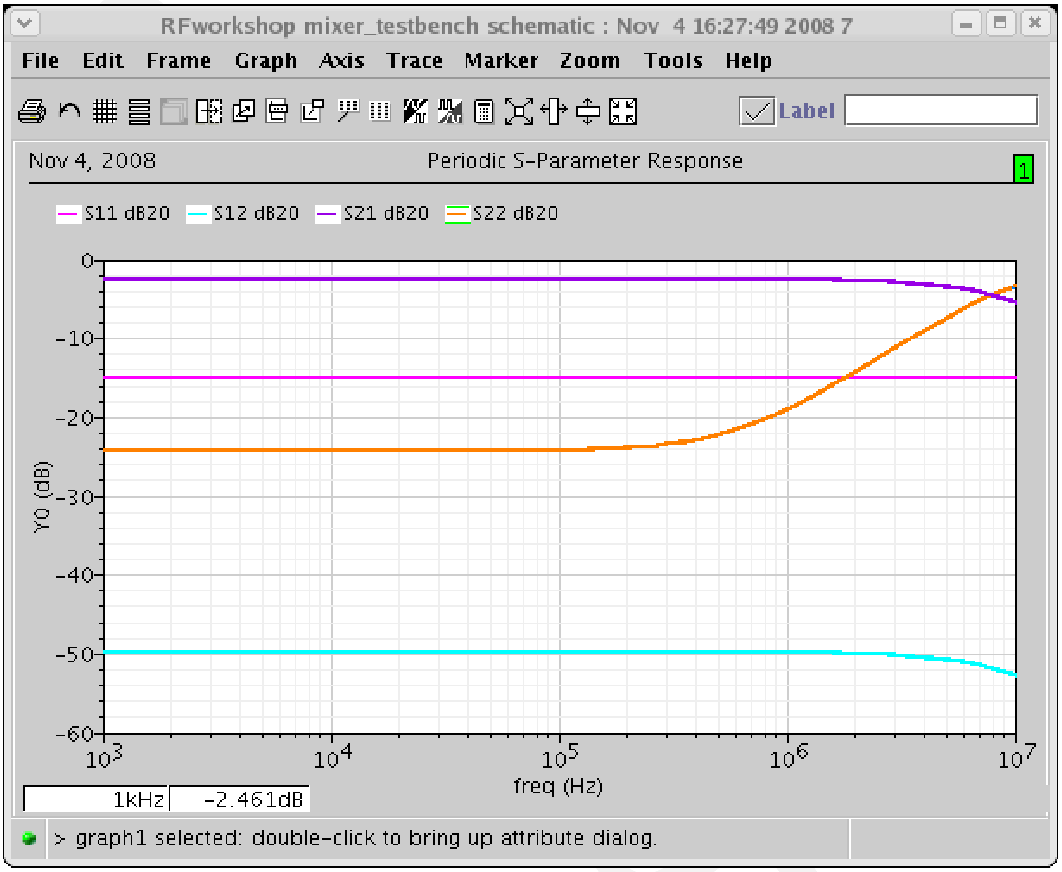
Action 5-12: After the simulation completes, in the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 5-13: In the Direct Plot Form, select the *psp* button, and configure the form as follows:


Direct Plot Form
OK
Cancel
Help
Plotting Mode
Append
Analysis
pss
psp
Function
SP
ZP
YP
HP
GD
VSWR
NFmin
Gmin
Rn
rn
NF
Kf
B1f
GT
GA
GP
Gmax
Gmsg
Gumx
ZM
NC
GAC
GPC
LSB
SSB
F
Fdsb
Fieee
Fmin
GAIN
IRN
NFdsb
NFieee
Description: S-Parameter
Plot Type
Rectangular
Z-Smith
Y-Smith
Polar
Modifier
Magnitude
Phase
dB20
Real
Imaginary
S11
S12
S13
S21
S22
S23
S31
S32
S33
Port 1 active harmonic is 1
Port 2 active harmonic is 0
Port 3 active harmonic is 1
Add To Outputs
> To plot, press Sij-button on this form...

Action 5-14: In the Direct Plot Form, click S11, S21, S12, and S22.

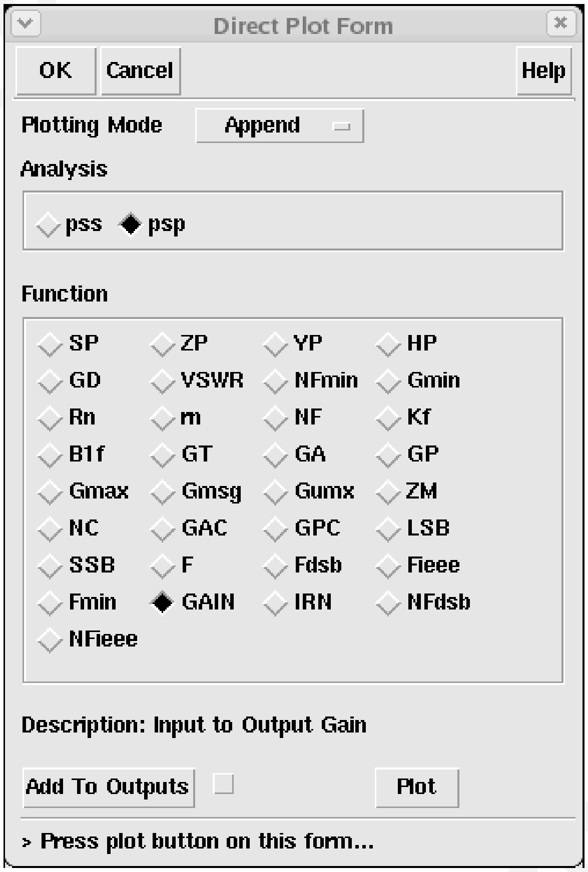
The waveform window appears:



Action 5-15: Close the waveform window.

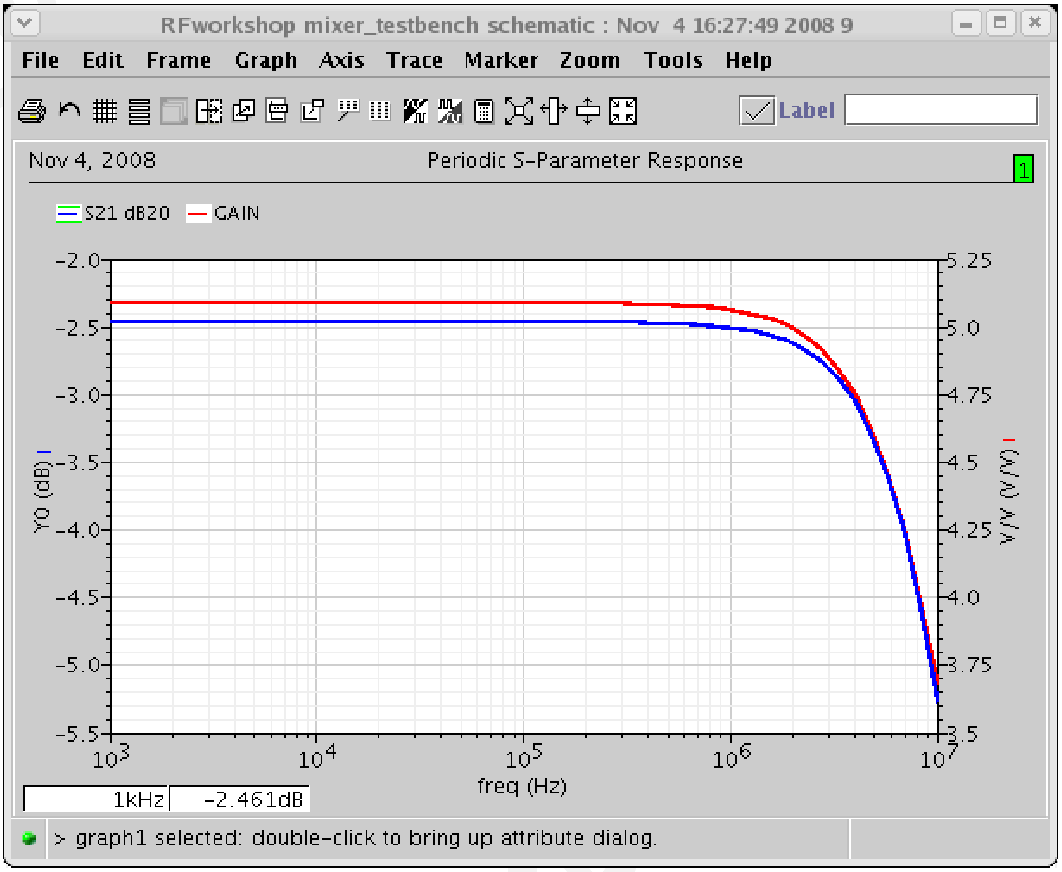
Action 5-16: In the Direct Plot Form, choose the *sp* function, set the *Plot Type* to *Rectangular* and *Modifier* to *dB20*. Plot S21.

Action 5-17: In the Direct Plot Form, choose *GAIN*.



Action 5-18: Click *OK*.

The waveform window appears with the following plot:



Action 5-18: Close the waveform window. Click *Cancel* in the Direct Plot Form.

PSP GAIN differs from the PSP S21 gain and the Pnoise gain because PSP GAIN is independent of input match (determined by the impedance of the RF port). PSP S21 and Pnoise gains vary depending on the input match. PSP GAIN is the voltage gain from the internal port voltage source to the output. For more information, see the Spectre user guide Appendix L (using psp and pnoise analysis).

Lab 6: Noise, Noise Summary and Noise Separation (PSS and Pnoise)

Because the noise from the mixer is moderated by the LNA's gain, it places a limit on how small a signal can be resolved. The sensitivity of the receiver is then adversely affected. Noise is measured using the noise figure (NF), which is a measure of how much noise the mixer adds to the signal relative to the noise that is already present in the signal. An NF of 0 dB is ideal, meaning that the mixer adds no noise. An NF of 3 dB implies that the mixer adds an amount of noise equal to that already present in the signal. For a mixer alone, an NF of 15 dB is typical.

Running the PSS and PNoise analyses produces all the needed information, including the total output noise and the noise figure.

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

Action 6-2: Select the `PORT0` source and set the *Source type* to *dc*. Select `PORT3` and set the *Source type* to *dc*. Set `PORT1` and set the *Source type* to *sine*.

Action 6-3: Check and save the schematic.

Action 6-4: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment.**

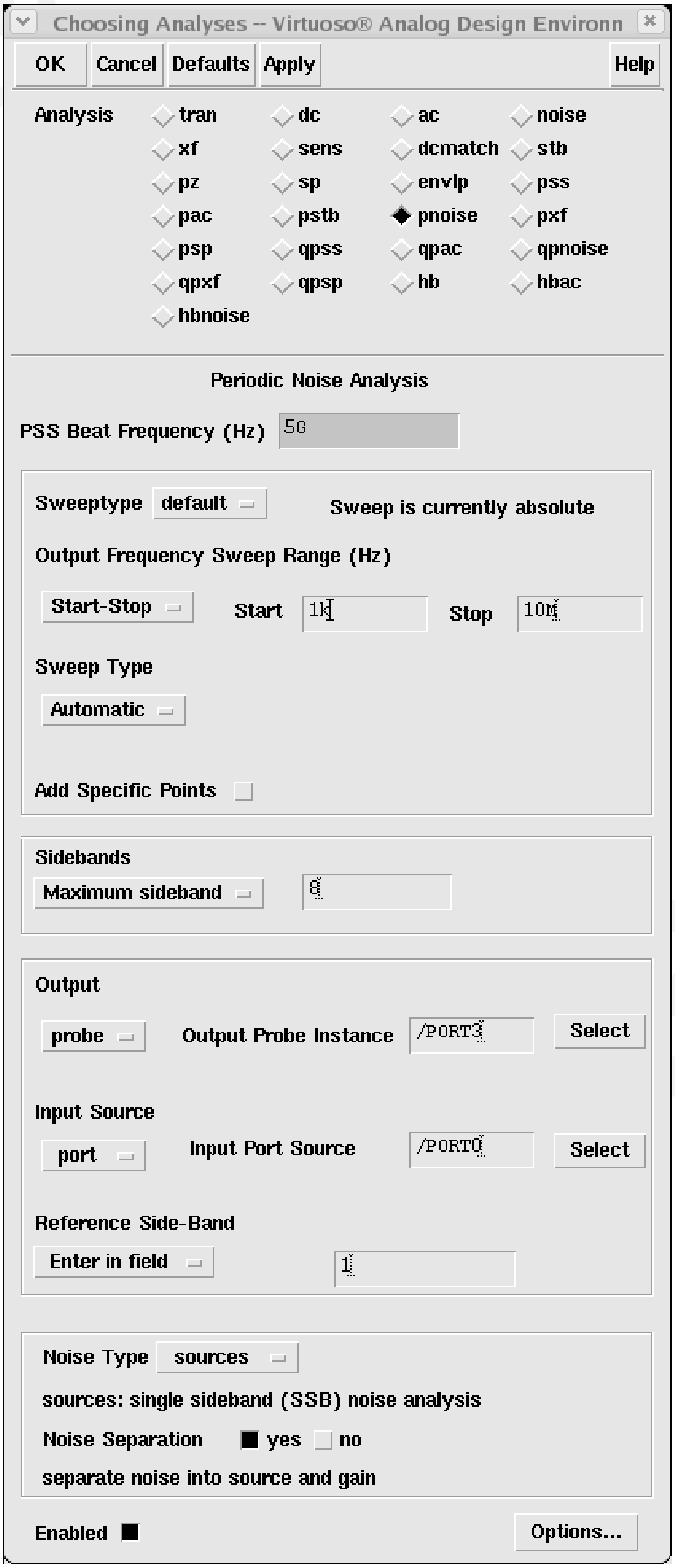
The Virtuoso Analog Design Environment window appears.

Action 6-5: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab6_PNOISE**" and skip to Action 6-10 or …

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

Action 6-7: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window and set the form as you did in Action 2-6.

Action 6-8: In the Choosing Analyses window, select the *pnoise* button in the *Analysis* field of the window and set up 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 Noise Analysis
PSS Beat Frequency (Hz)
5G
Sweeptype
default
Sweep is currently absolute
Output Frequency Sweep Range (Hz)
Start-Stop
Start
1k
Stop
10M
Sweep Type
Automatic
Add Specific Points
Sidebands
Maximum sideband
8
Output
probe
Output Probe Instance
/PORT3
Select
Input Source
port
Input Port Source
/PORT0
Select
Reference Side-Band
Enter in field
1
Noise Type
sources
sources: single sideband (SSB) noise analysis
Noise Separation
yes
no
separate noise into source and gain
Enabled
Options...

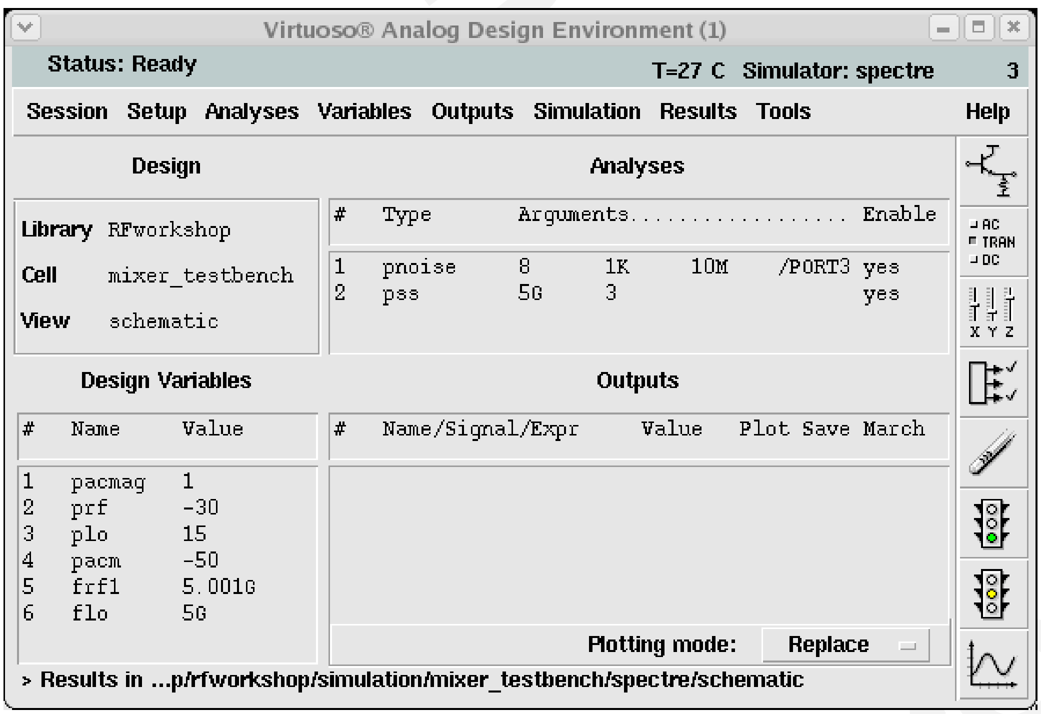
Note: You must specify the load as an output or oprobe. The load can be a resistor or a port. All noise from the source is included in the denominator of the noise factor fraction, including excess noise, so do not specify excess noise on the input port. (Excess noise is specified with the *noisefile* or *noisevec* option.)

In rare cases, there might not be a load, in which case you can specify the output using a pair of nodes. However, if there is a load in the circuit, and a pair of nodes is specified as an output, you obtain different results than if a load is specified as output. This is because the load contributes some noise to the total output noise that must be subtracted out before using the equation above to compute the noise figure. If only a pair of nodes is used, Spectre has no way to determine which of the elements in the circuit is the load (for example, there could be multiple resistors connected to the output nodes) and so cannot determine the amount of output noise due to the load.

Also note that *Noise Separation* is set to *yes*. When the Pnoise simulation runs, the Pnoise separation feature is included during the simulation and the corresponding results are saved. Noise Separation is a new feature for MMSIM61.

Action 6-9: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

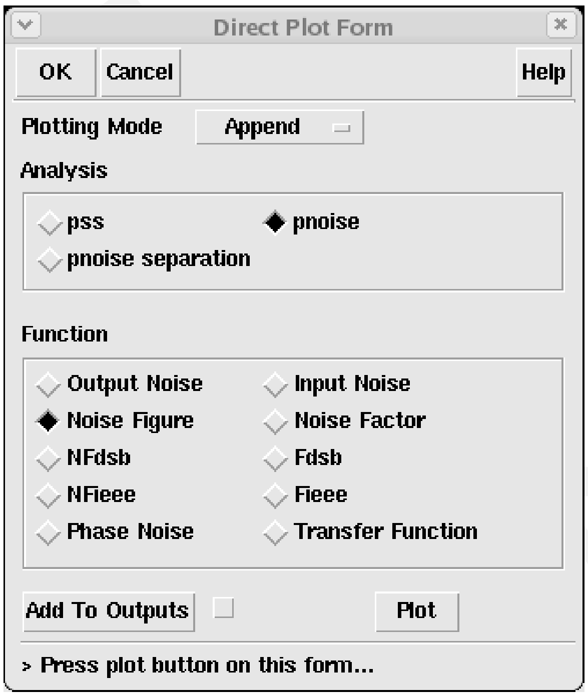
The Virtuoso Analog Design Environment window looks like this.



Action 6-10: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action 6-11: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 6-12: In the Direct Plot Form, select the *pnoise* button, and configure the form as follows:

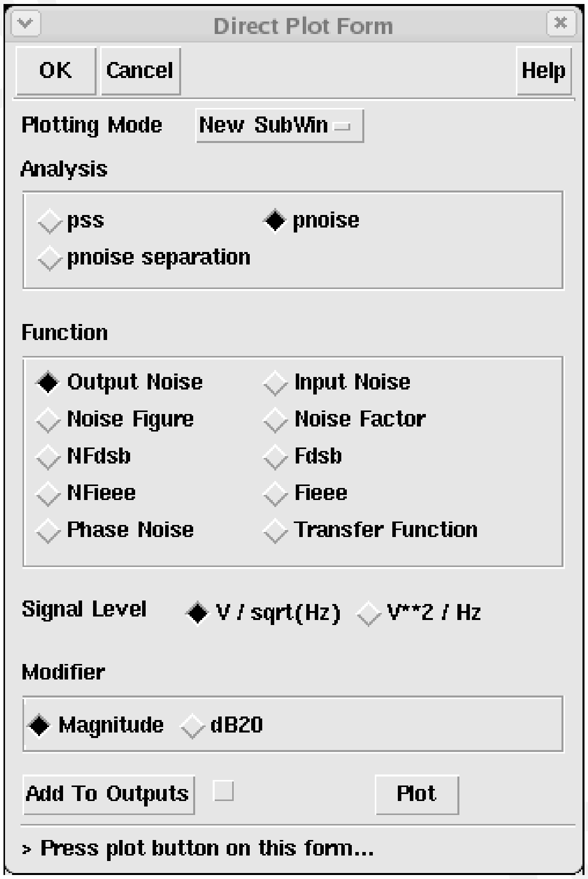


Action 6-13: Click **Plot**.

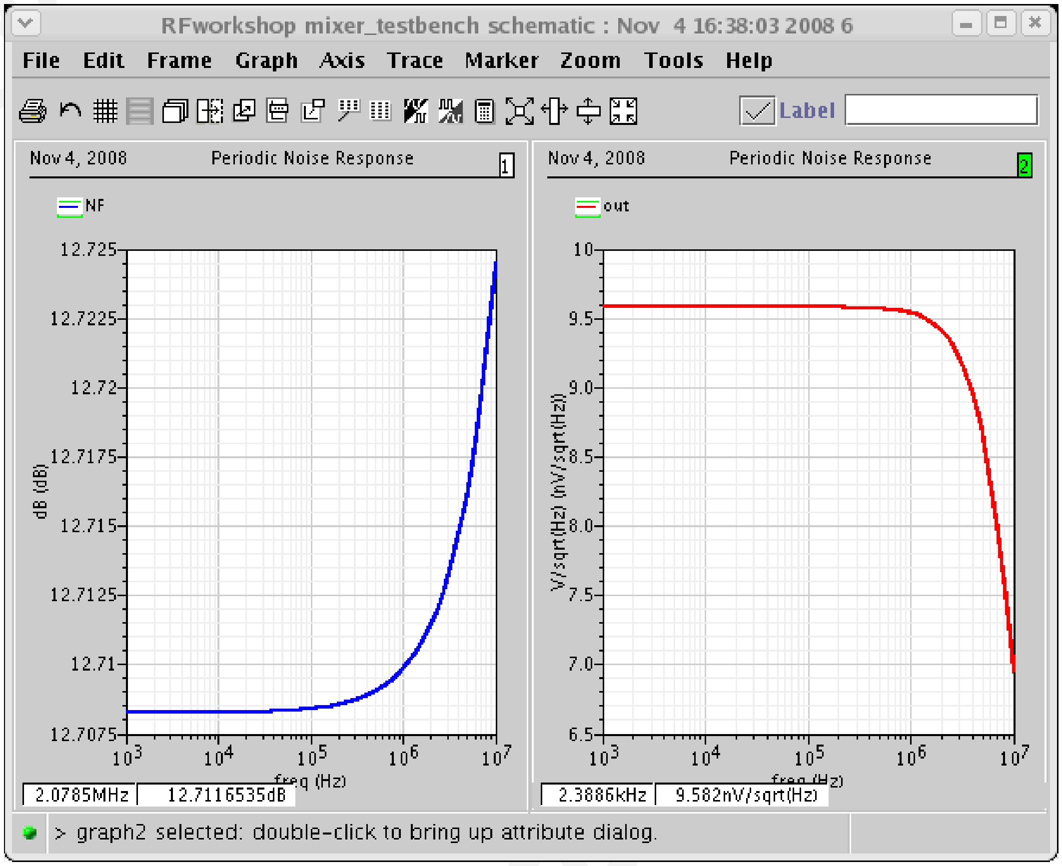
The waveform window appears.

Action 6-14: In the waveform window, click **New Subwindow**.

Action 6-15: In the Direct Plot Form, set the *Function* to *Output Noise* and configure the rest of the form as follows:



Action 6-16: Click *Plot*. The waveform window appears.



Action 6-17: After viewing the waveforms, click *Cancel* in the Direct Plot Form.

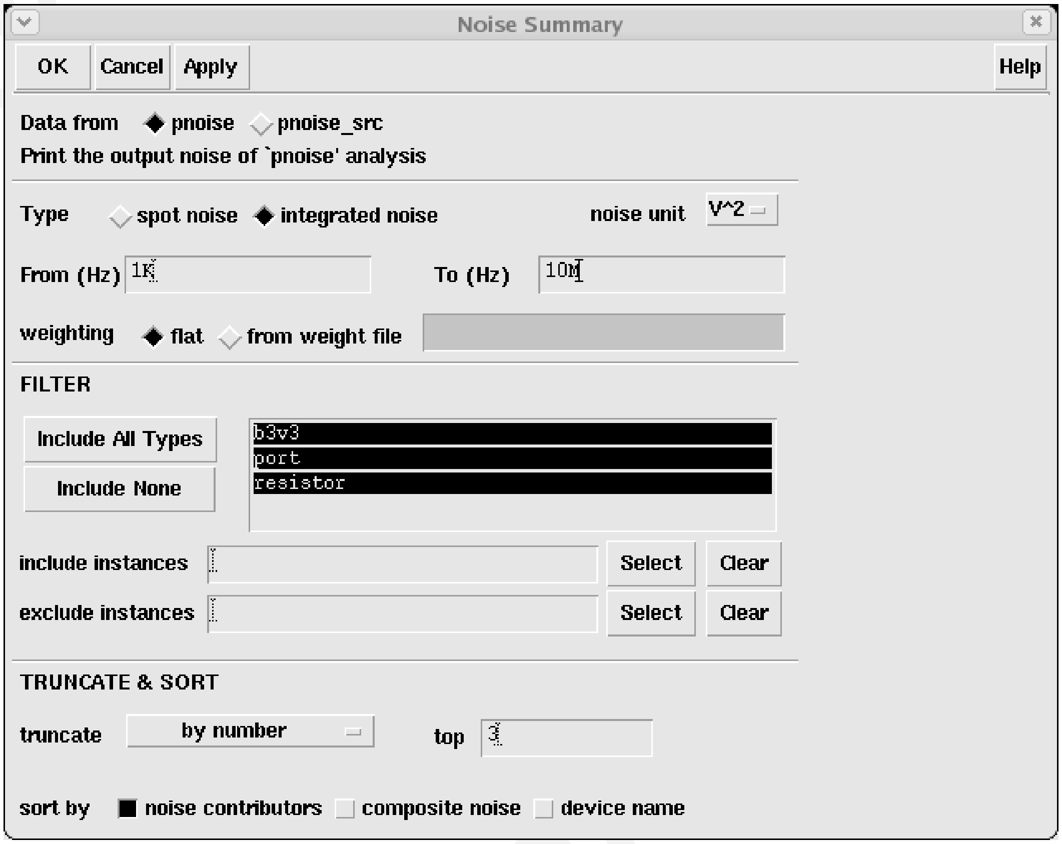
It is valuable to know the main contributors of noise in a system, after the noise performance of the circuit is calculated. This information is readily available from a Pnoise simulation.

Action 6-18: In the Virtuoso Analog Design Environment window, choose **Results — Print — Noise Summary**.

The Noise Summary form appears.

Action 6-19: Fill in the form as shown here.

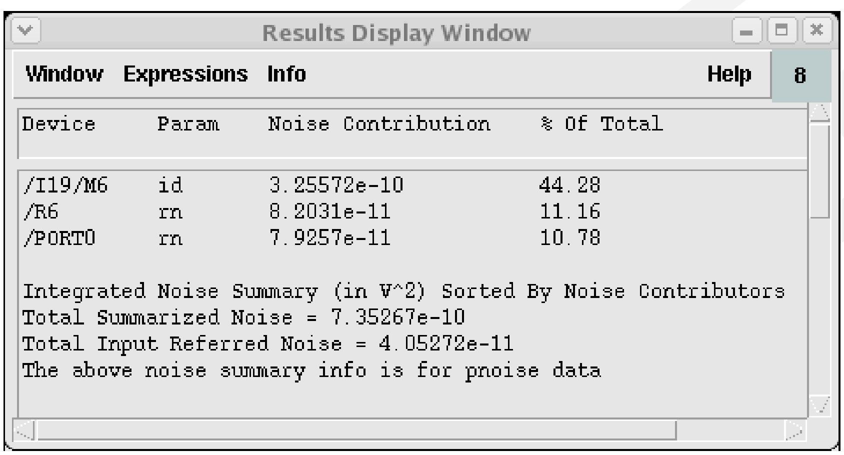
Note: The Noise Summary form includes both a *pnoise* and a *pnoise_src* choice. The *pnoise_src* form and the fill, filter, and truncate methods are the same as those used in the *pnoise* form.



Action 6-20: Click the *Include All Types* button in the FILTER section.

Action 6-21: Click *OK* in the Noise Summary form.

The Results Display Window appears.



Action 6-22: After viewing the results, choose **Window — Close** to close the window.

You can use Spectre to quickly locate the noise sources that cause the most noise to the output. The basic flow is:

[1] Direct Plot Form

[2] Pnoise/Qpnoise Separation

[3] Sideband Output

Here you decide which sidebands contribute more output noise.

[4] Instance Output

Here you decide which instances contribute more output noise to the selected sideband.

[5] Source Output

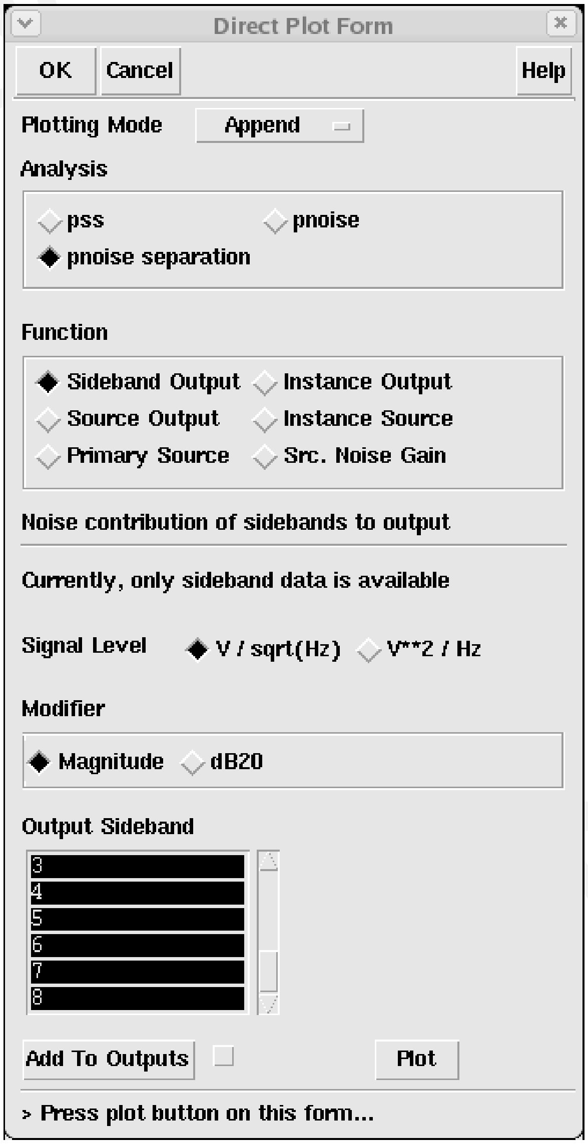
Here you decide which primary sources contribute more output noise to the filtered and truncated instances.

[6] Instance Source/Primary Source/Src. Noise gain

Here you determine which primary noise sources or gains have more effect on the output noise.

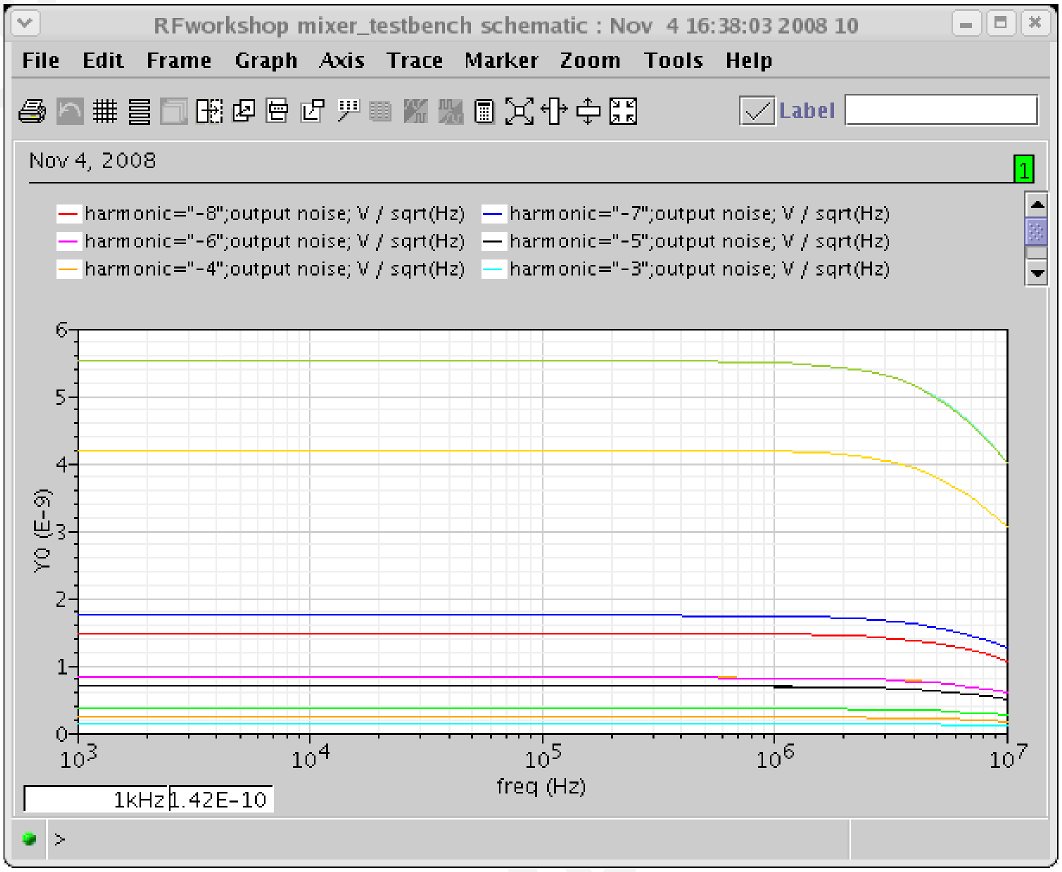
The steps of this flow are illustrated in the following actions.

Action 6-23: In the Direct Plot Form, select **pnoise separation**, choose **Sideband Output**, and. select all the sidebands (from -8 to 8).



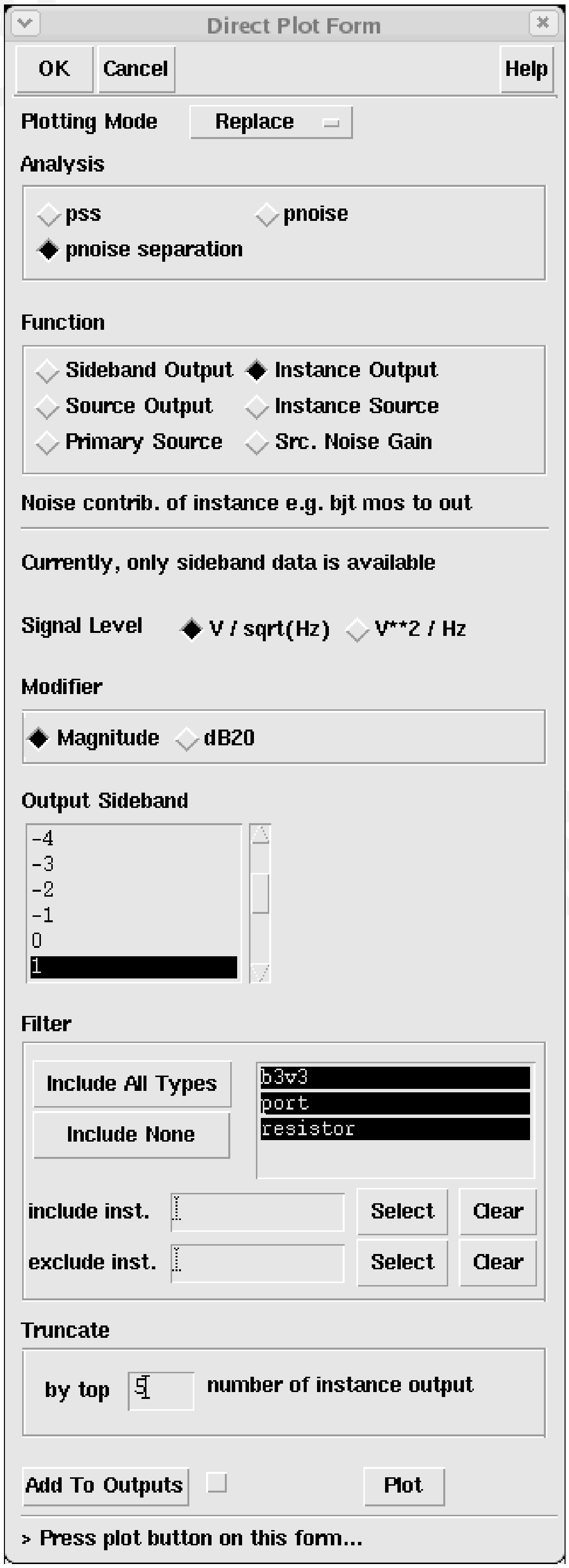
Action 6-24: Click **Plot**.

The waveform window appears with results as shown below:



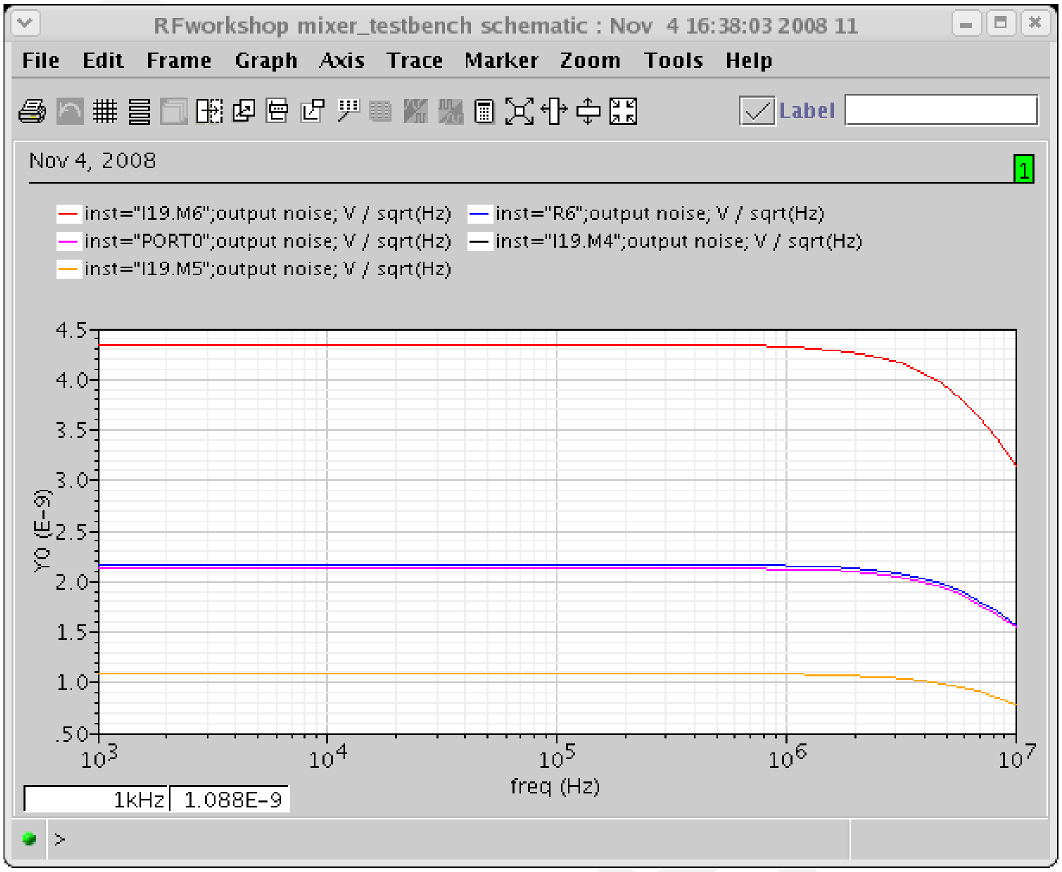
In this example, sidebands 1 and -1 contribute more output noise than the other sidebands, so next you check the instances in sideband 1.

Action 6-25: In the Direct Plot Form, set the *Plotting Mode* to *Replace*. Choose *Instance Output***.** Set *Output Sideband* as 1. Click *Include All Types*, and set *by top* to 5.


Direct Plot Form
OK
Cancel
Help
Plotting Mode
Replace
Analysis
pss
pnoise
pnoise separation
Function
Sideband Output
Instance Output
Source Output
Instance Source
Primary Source
Src. Noise Gain
Noise contrib. of instance e.g. bjt mos to out
Currently, only sideband data is available
Signal Level
V / sqrt(Hz)
V**2 / Hz
Modifier
Magnitude
dB20
Output Sideband
-4
-3
-2
-1
0
1
Filter
Include All Types
Include None
b3v3
port
resistor
include inst.
Select
Clear
exclude inst.
Select
Clear
Truncate
by top
5
number of instance output
Add To Outputs
Plot
> Press plot button on this form...

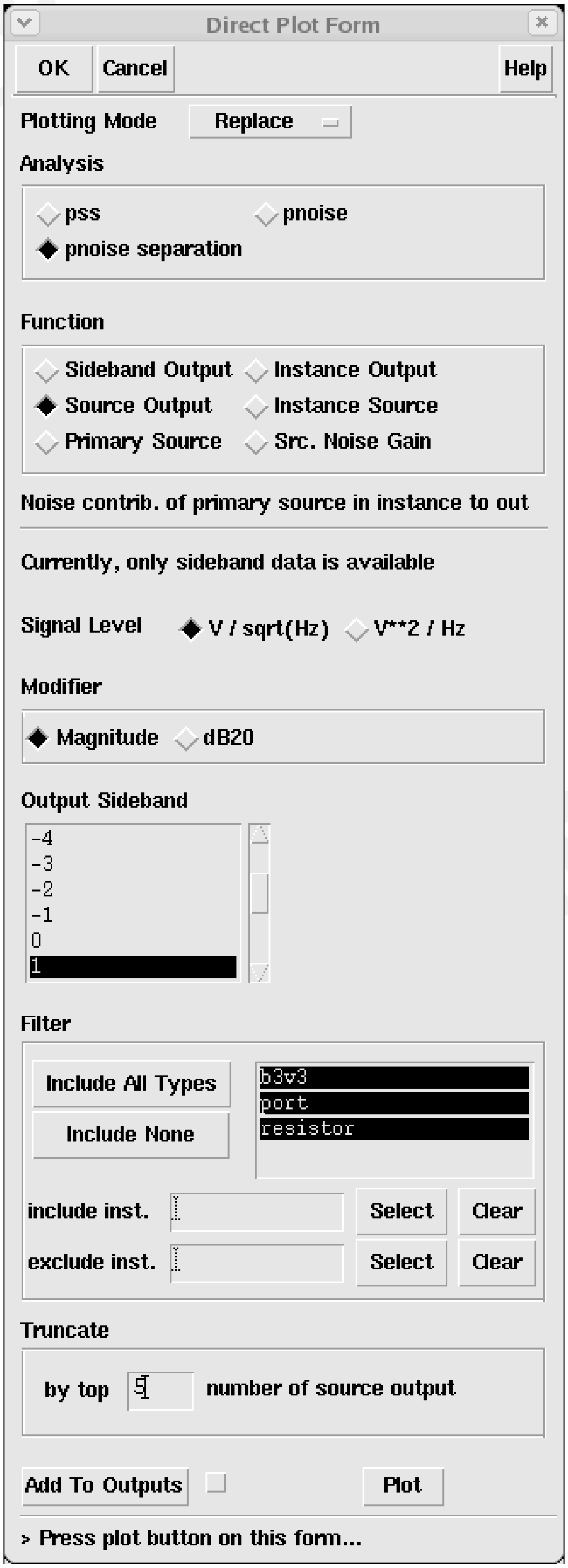
Action 6-26: Click **Plot**.

The result is shown below:



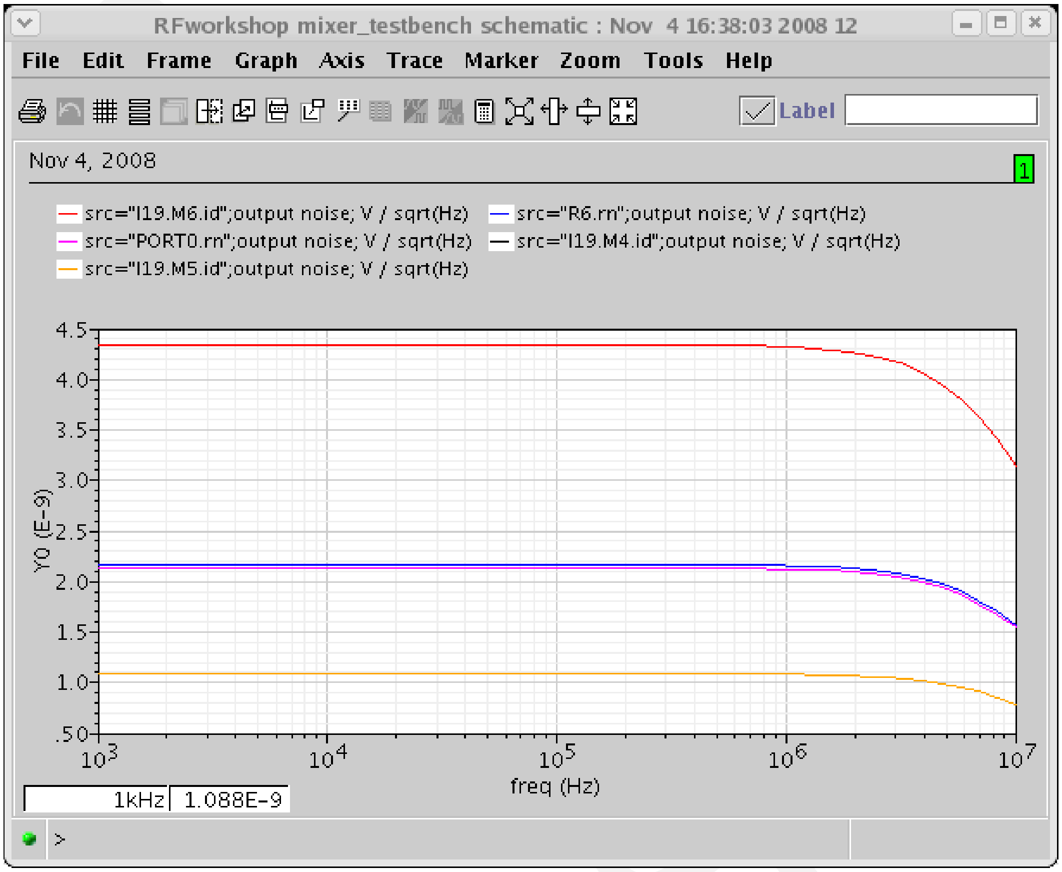
In this example, “I19.M6” contributes more output noise than the other instances.

Action 6-27: In the Direct Plot Form, set *Plotting Mode* to *Replace*. Choose *Source Output*. Set *Output Sideband* to 1, select *Include All Types*, and set *by top* to 5.


Direct Plot Form
OK
Cancel
Help
Plotting Mode
Replace
Analysis
pss
pnoise
pnoise separation
Function
Sideband Output
Instance Output
Source Output
Instance Source
Primary Source
Src. Noise Gain
Noise contrib. of primary source in instance to out
Currently, only sideband data is available
Signal Level
V / sqrt(Hz)
V**2 / Hz
Modifier
Magnitude
dB20
Output Sideband
-4
-3
-2
-1
0
1
Filter
Include All Types
Include None
b3v3
port
resistor
include inst.
Select
Clear
exclude inst.
Select
Clear
Truncate
by top
5
number of source output
Add To Outputs
Plot
> Press plot button on this form...

Action 6-28: Click **Plot**.

The result is shown below:

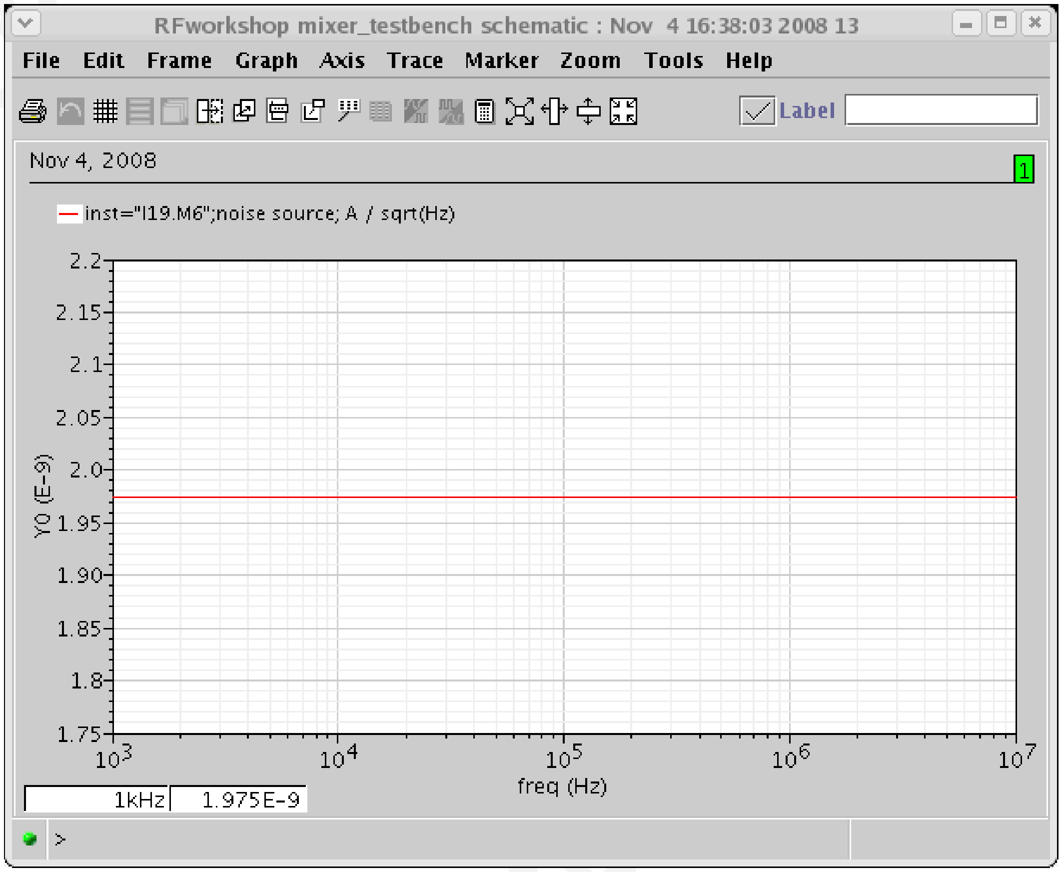


In this example, "I19.M6" contributes more output noise than the other sources. Note that the list order of the instances in this plot is different from that of **instance output**.

By now it is obvious that "I19.M6" is the major contributor to the output noise in this circuit.

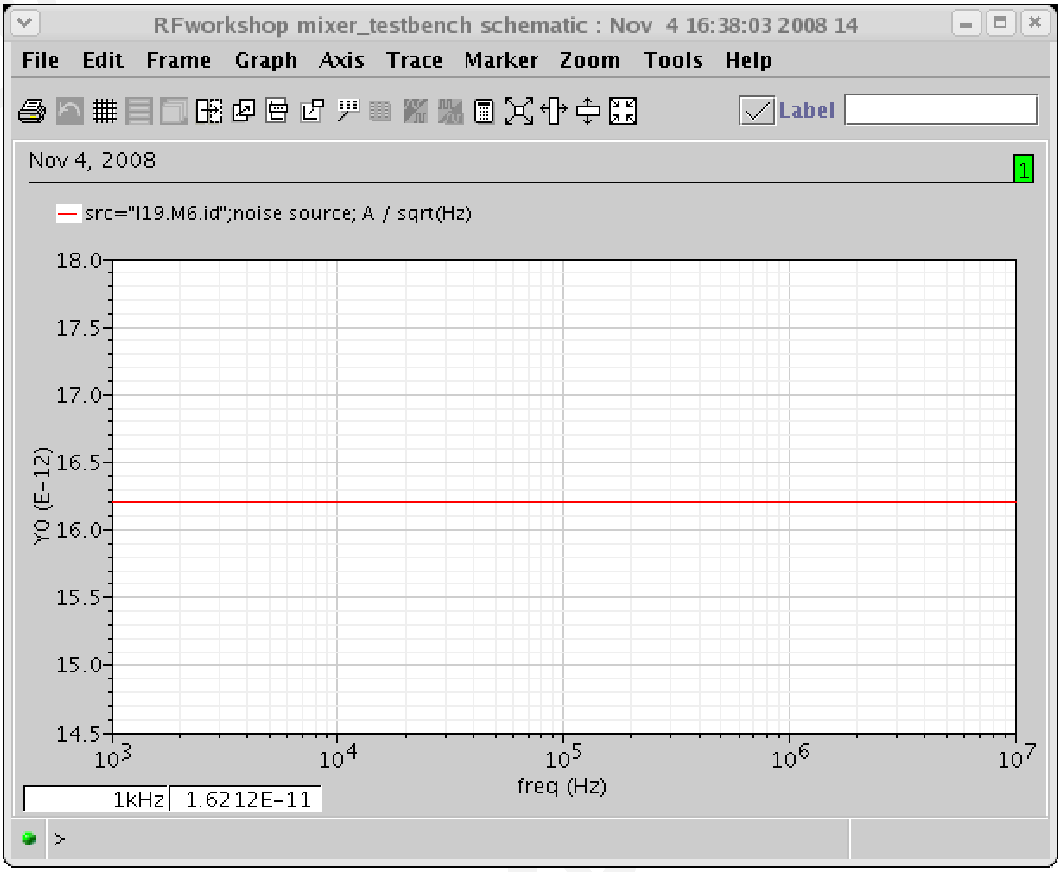
Action 6-29: In the Direct Plot Form, choose *Instance Source*. Set *by top* to 1**.** Click *Plot* to plot the noise source measurement of instance "I19.M6".

The result is shown below:



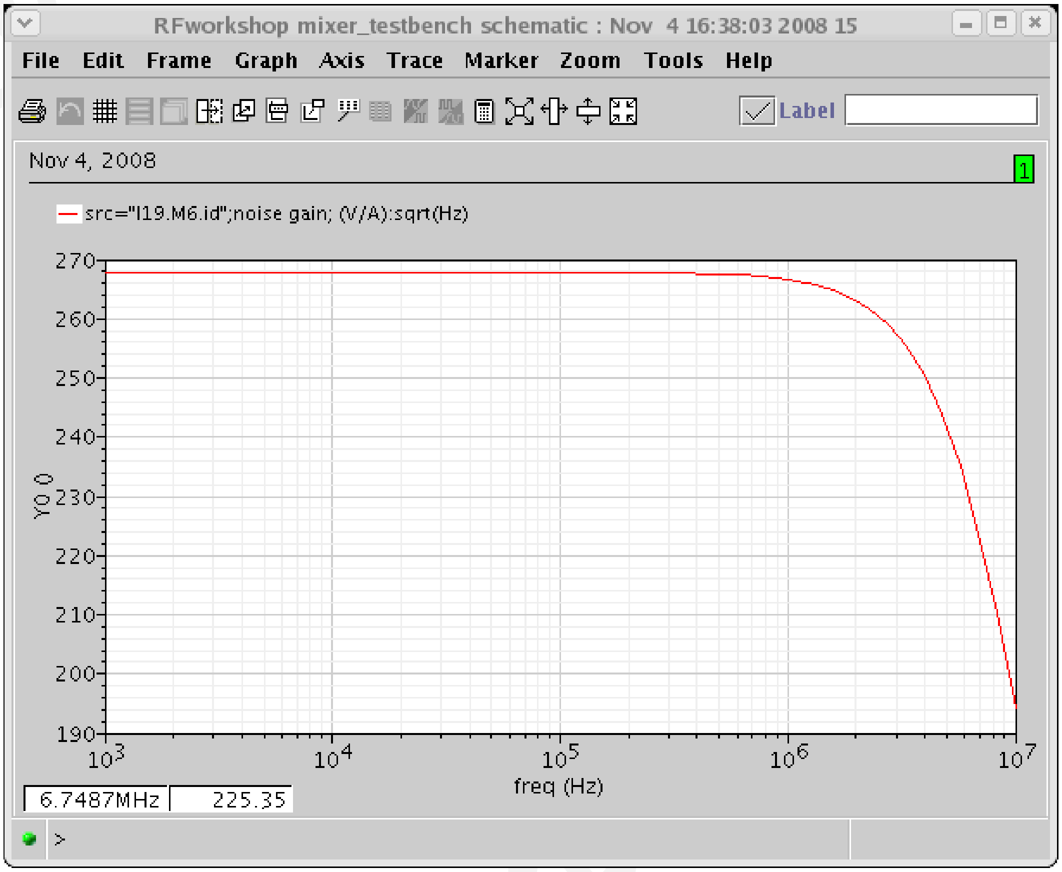
Action 6-30: In the Direct Plot Form, choose *Primary Source*. Set *by top* to 1. Click *Plot* to plot the noise source measurement of primary source in instance "I19.M6".

The result is shown below:



Action 6-31: In the Direct Plot Form, choose *Src. Noise Gain*. Set *by top* to 1. Click *Plot* to plot the noise gain from primary source in instance to output.

The result is shown below:



So to improve the noise performance of this circuit, decreasing the output noise of "I19.M6"is an effective solution. There are two approaches: one is decreasing the magnitude of noise source "I19.M6" by adjusting the device geometric size; the other is decreasing the transfer function of "I19.M6"by adjusting the circuit architecture.

Lab 7: Port-to-Port Isolation among RF, IF and LO Ports (PSS and Swept PAC)

The isolation required between a mixer's ports depends on the circuit and the architecture of the product. Isolation is critical for the mixer to function properly.

You can combine PAC and PXF analyses to produce transfer functions from different ports to each other. One suggested configuration is to set up a PAC analysis with a nonzero pacmag parameter at the signal input (the RF port) and to set up a PXF analysis with the IF port as the output probe. This example uses pacmag = 1 V for simplicity.

Action 7-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework.*

Action 7-2: Use the mouse to select the `PORT0` source. Then, in the Virtuoso Schematic Editor, select **Edit — Properties — Objects**. The Edit Object Properties window for the port cell appears. Change the port properties as follows:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	500 mV
Source type	*dc*
PAC Magnitude	*pacmag*

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

Action 7-4: Set the *Source type* of `PORT1` to *sine*, and the *Source type* of `PORT3` to *dc*.

Action 7-5: Check and save the schematic.

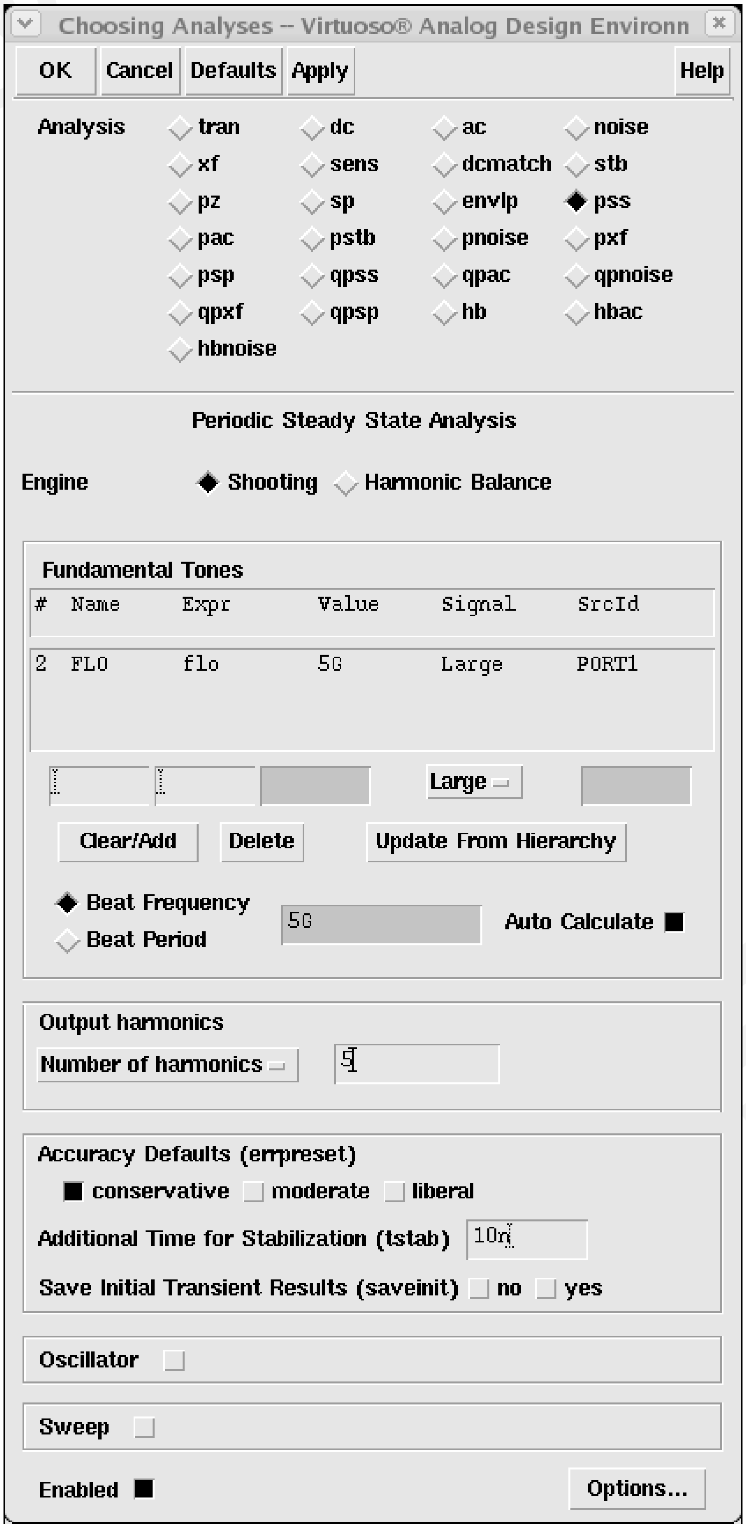
Action 7-6: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

The Virtuoso Analog Design Environment window appears.

Action 7-7: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state “**Lab7_Isolation_PAC**” and skip to Action 7-14 or …

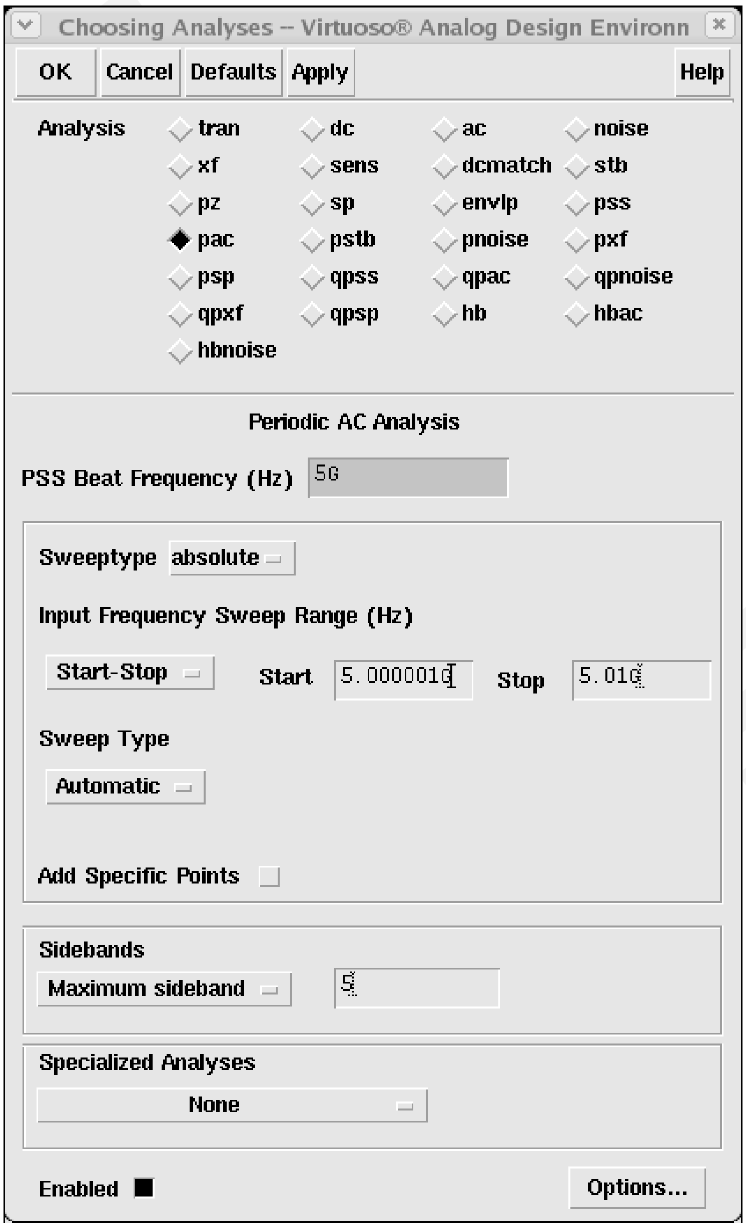
Action 7-8: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

Action 7-9: 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 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
2 FLO flo 5G Large PORT1
Large
Clear/Add
Delete
Update From Hierarchy
Beat Frequency
Beat Period
5G
Auto Calculate
Output harmonics
Number of harmonics
5
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
10n
Save Initial Transient Results (saveinit)
no
yes
Oscillator
Sweep
Enabled
Options...

PSS simulation is set up here to check the LO feedthrough. LO feedthrough is a large-signal effect and should not be measured using a small-signal analysis such as PXF. If you have a small RF conversion product and a large signal (LO) also, the large signal swamps the small-signal (RF) conversion product. Because there is no 1dB/dB relationship between the LO and IF, you can get an incorrect answer. It is therefore not recommended to use PXF to perform this measurement.

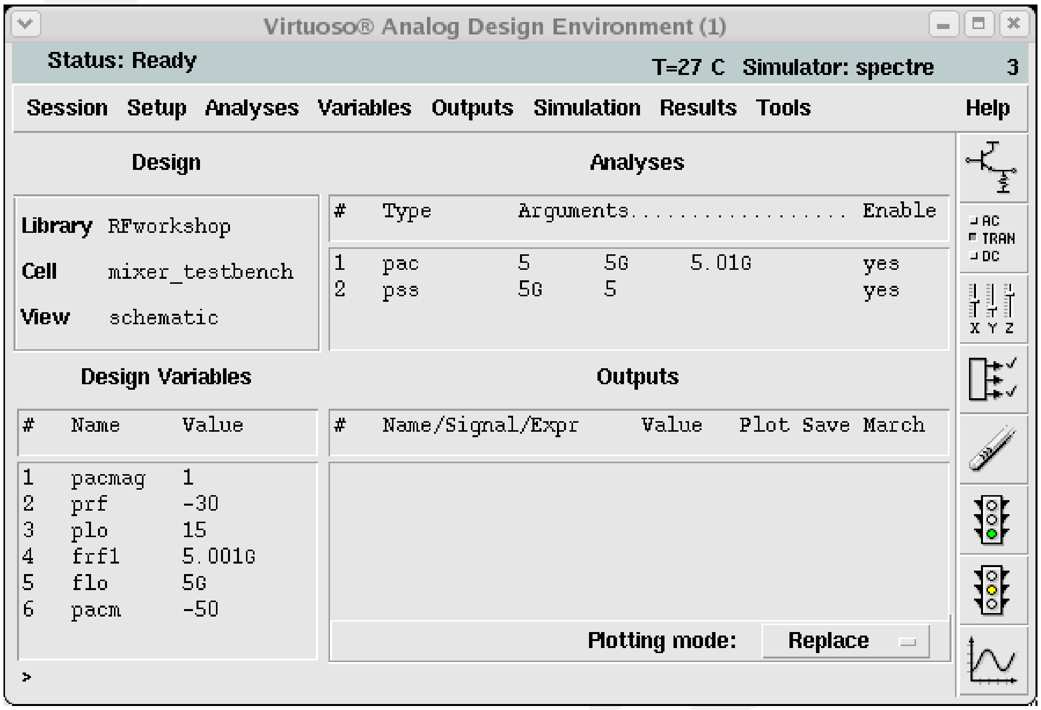
Action 7-10: In the Choosing Analyses window, select the *pac* button in the *Analysis* field of the window. Set up the form as follows:



Action 7-11: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

PAC simulation is set up here to check the RF feed through.

The Virtuoso Analog Design Environment window looks like this.



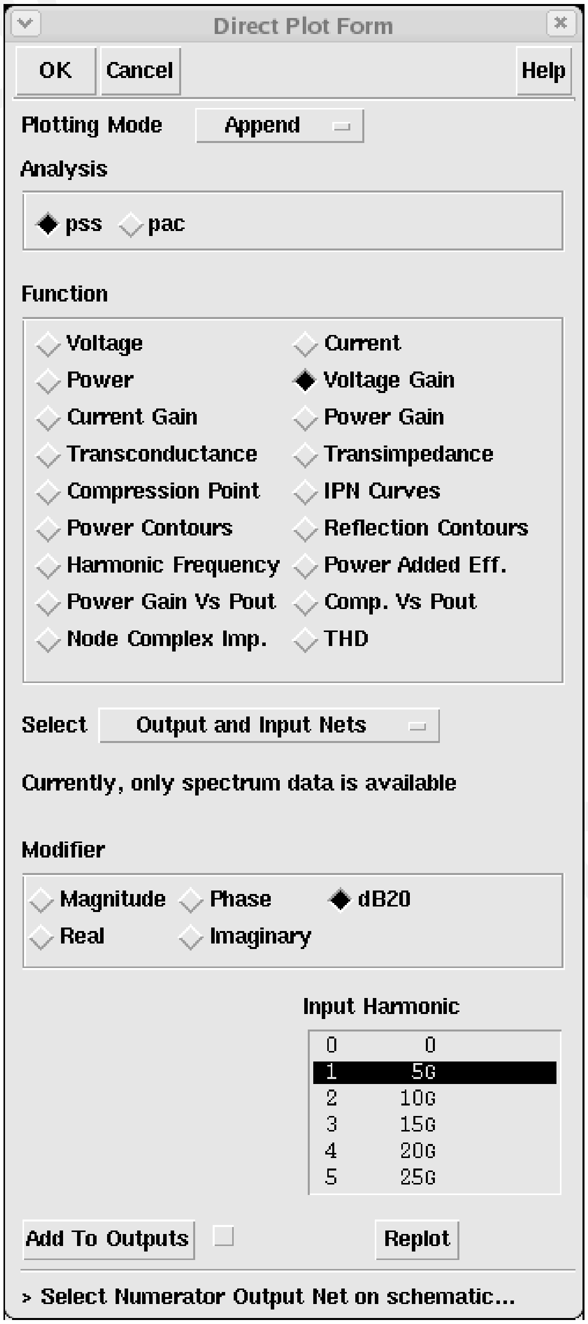
Action 7-12: In the Virtuoso Analog Design Environment, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

After the simulation runs, use the Direct Plot feature to plot the results.

To avoid desensitizing the stage following the mixer with high-level LO signal feedthrough to the output, measure LO-to-IF isolation. Use the results of the PSS analysis with the LO port as input and IF port as output to measure the level of isolation.

Action 7-13: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

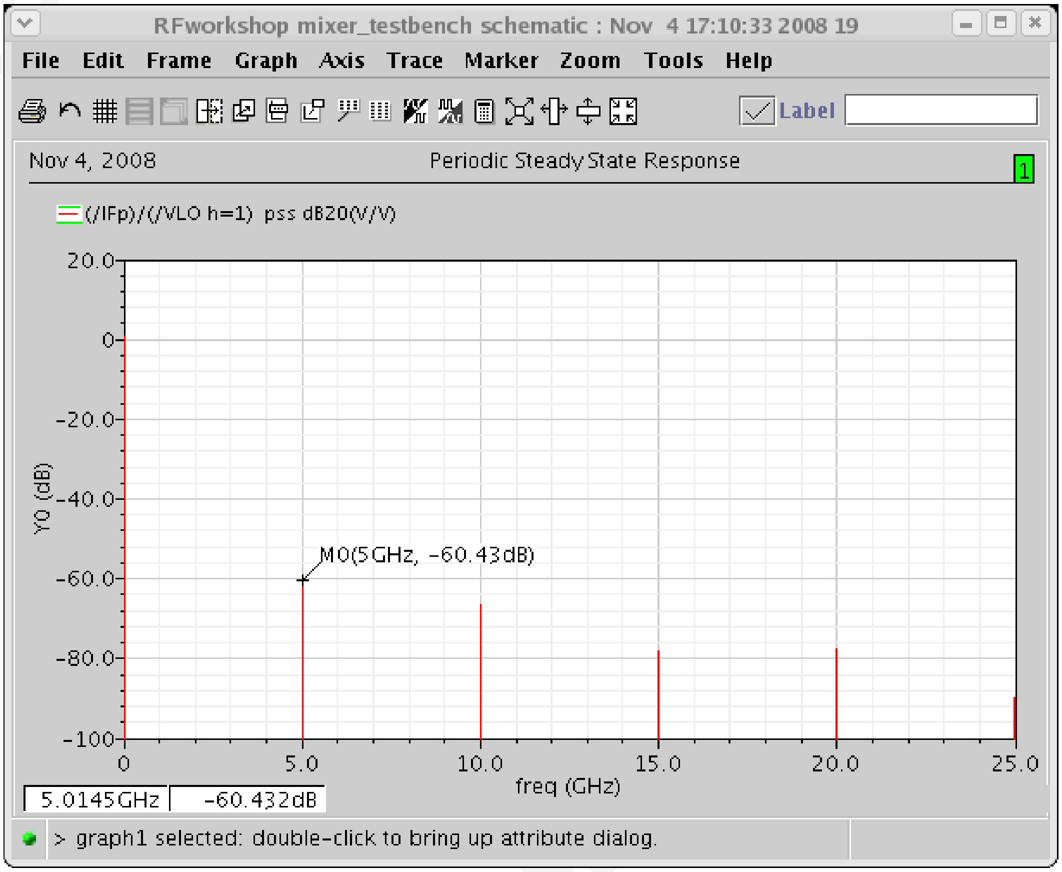
Action 7-14: In the Direct Plot Form, choose the *pss* button, and configure the form as follows:



Action 7-15: In the schematic, select the net IFp as the output and net VLO as input.

Action 7-16: In the waveform window, choose **Marker — Place — Trace Marker** to place a marker at the first harmonic 5 GHz.

The following waveform shows the LO to IF feedthrough:



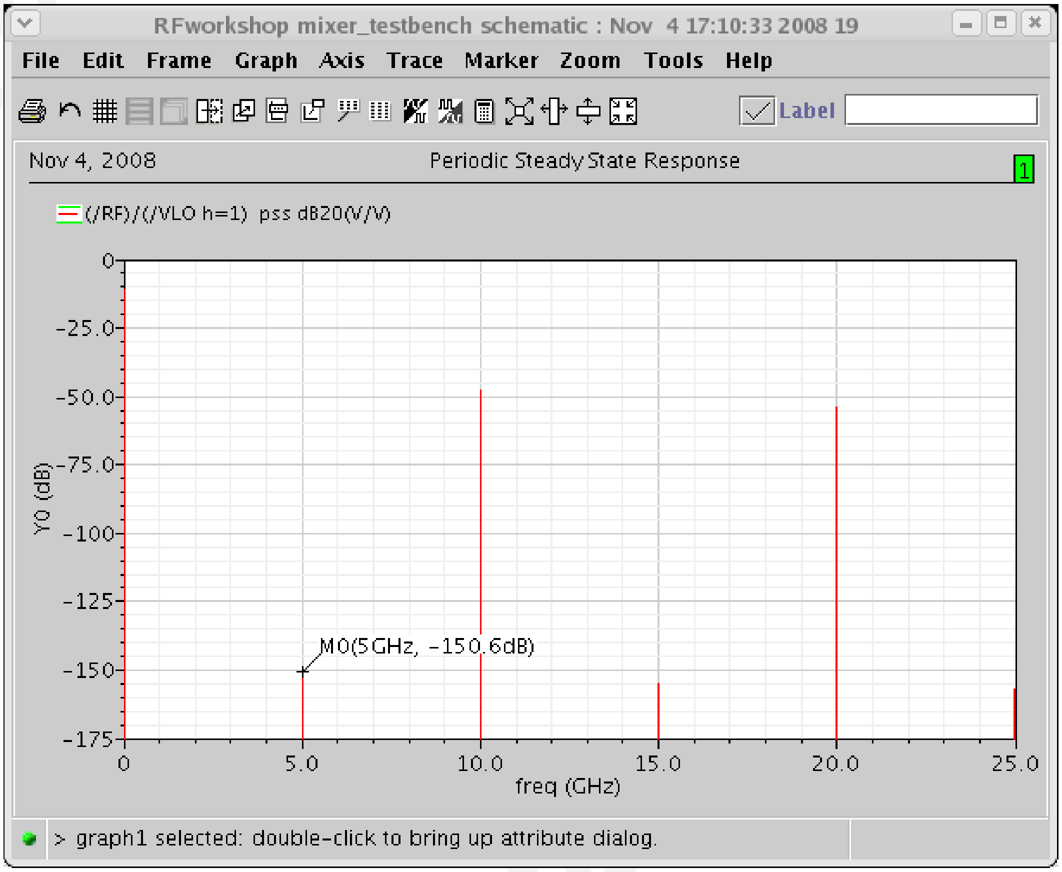
LO-to-RF feedthrough affects the functionality of LNAs and antennas and the self-mixing causes the dc offset. Use the results of the PSS analysis with the LO port as input and the RF port as output to measure the LO-to-RF feedthrough.

Action 7-17: In the Direct Plot Form, change the Plotting Mode to *Replace*.

Action 7-18: In the schematic, select the net RF as the output and net VLO as input.

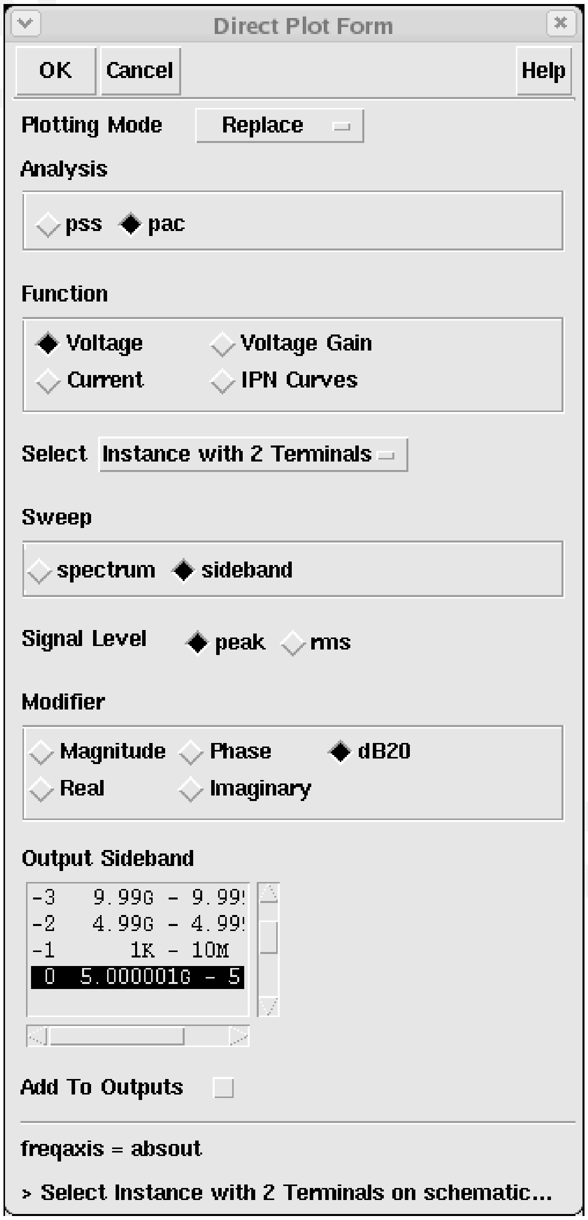
Action 7-19: In the waveform window, select **Marker — Place — Trace Marker** to place a marker at the first harmonic 5 GHz.

The following waveform shows the LO to IF feedthrough:



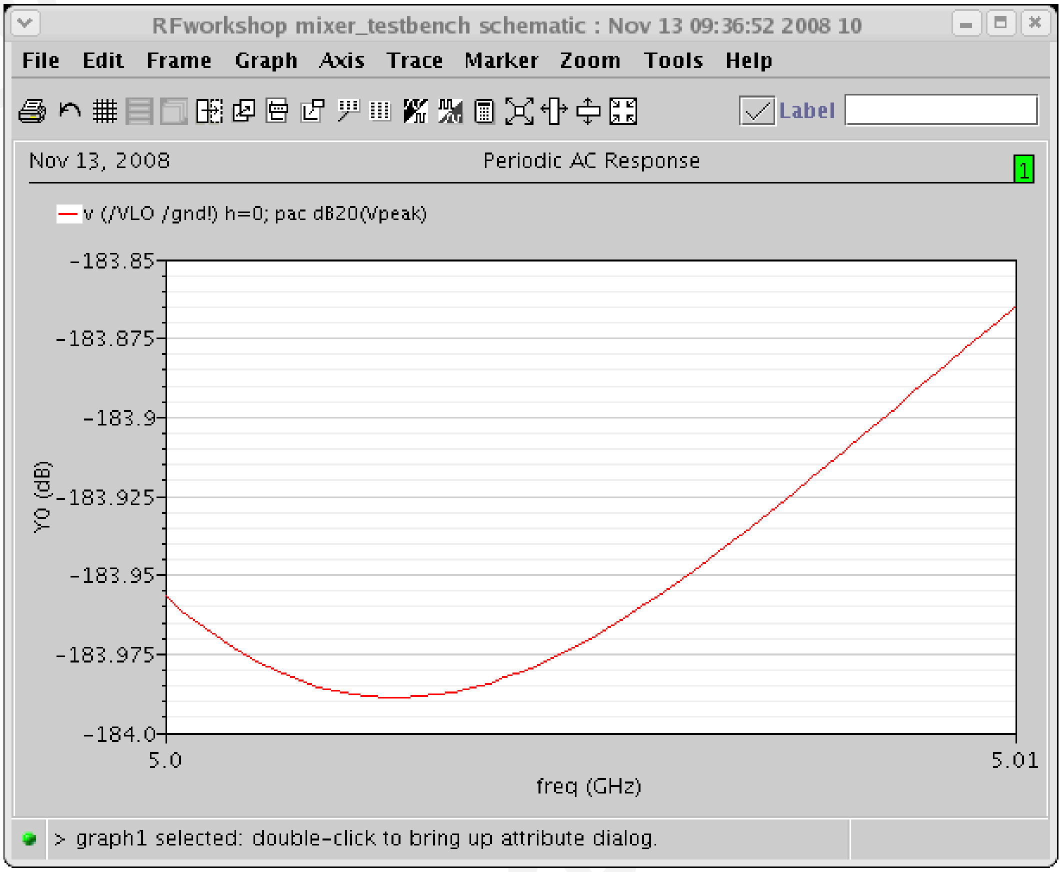
RF-to-LO feedthrough affects the local oscillator by letting strong interferers at the input pass through to the LO. Measure RF-to-LO feedthrough using the PAC analysis results.

Action 7-20: In the Direct Plot Form, select the *pac* button, and configure the form as follows:



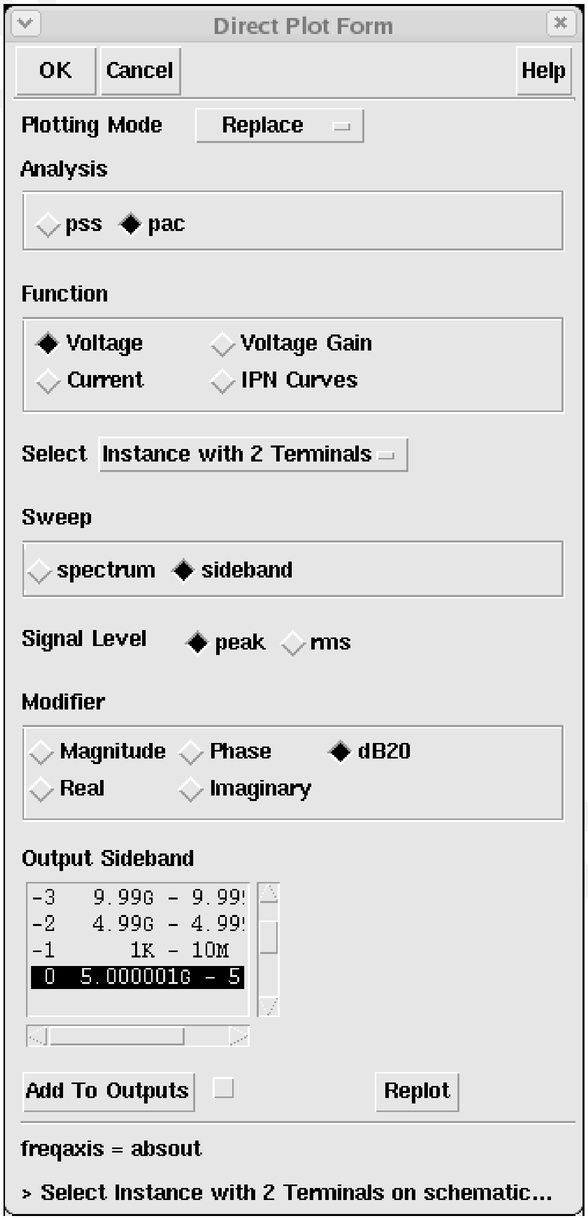
Action 7-21: Select 0 in *Output Sideband* to represent the RF signal and click `PORT1` to select the LO port as the output instance.

The following waveform shows the RF to LO feedthrough:



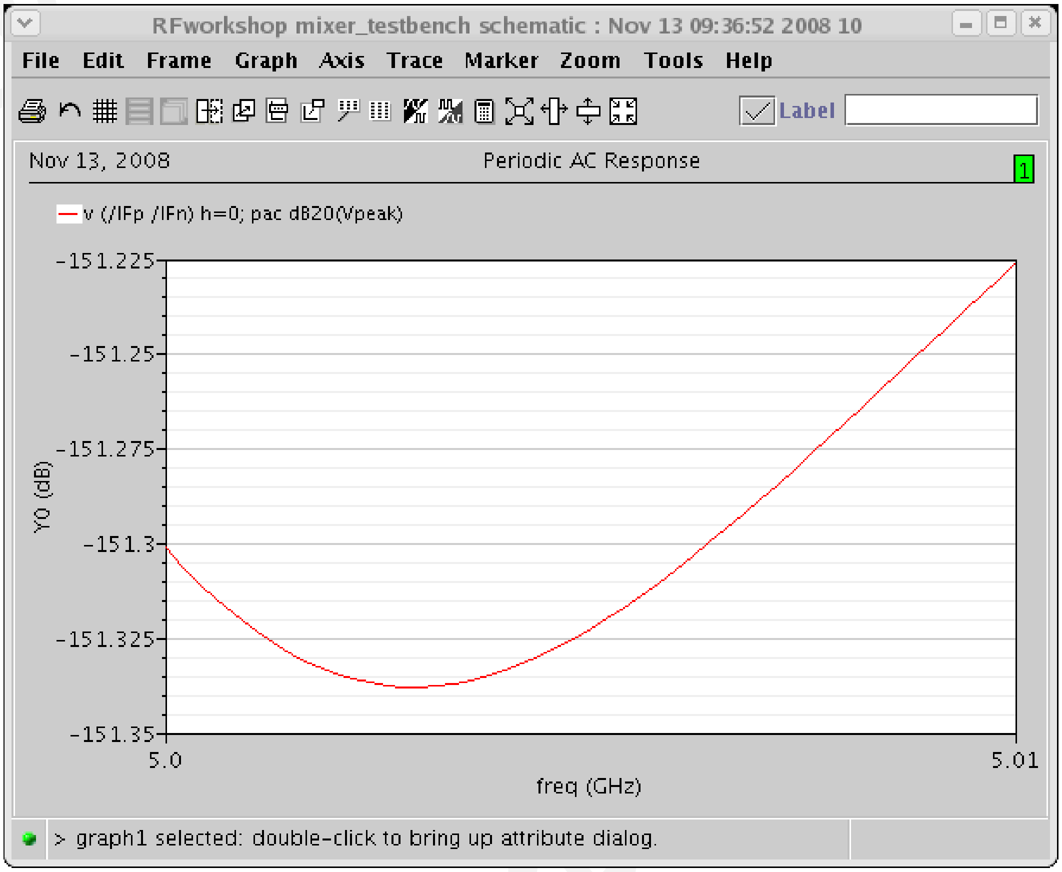
RF-to-IF feedthrough might create an even-order distortion problem for homodyne receivers. Measure RF-to-IF feedthrough using the PAC analysis results with two simple changes.

Action 7-22: In the Direct Plot Form, select the *pac* button, and configure the form as follows:



Action 7-23: Select 0 in *Output Harmonic* and select the IF output `PORT3`.

The following waveform shows the RF to IF feedthrough:



Action 7-24: Click *Cancel* in the Direct Plot Form. Close the waveform window.

Lab 8: Mixer Performance with a Blocking Signal (QPSS, QPAC, and QPNoise)

Large interfering signals are called blockers. Blocking signals reduce the mixer's gain and deteriorate the mixer's noise performance. As such, you need to measure the gain and noise of a mixer in the presence of a blocking signal. All major communication standards include blocking requirements for both mobile and base stations. The requirements use several in-band and multiple out-of-band blocking signals.

Because a mixer has both signal and LO inputs, you should use the multi-tone large signal QPSS analysis for these measurements. Follow the QPSS analysis with QPAC and QPNoise analyses to measure gain and NF variations versus the level of the interfering signal. In the QPSS analysis, model the blocker as a moderate tone.

Action 8-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework.*

Action 8-2: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

The Virtuoso Analog Design Environment window appears.

Action 8-3: Select the `PORT0` 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	RF1
Frequency 1	5.003G
Amplitude 1 (dBm)	prf
PAC magnitude (dBm)	-30

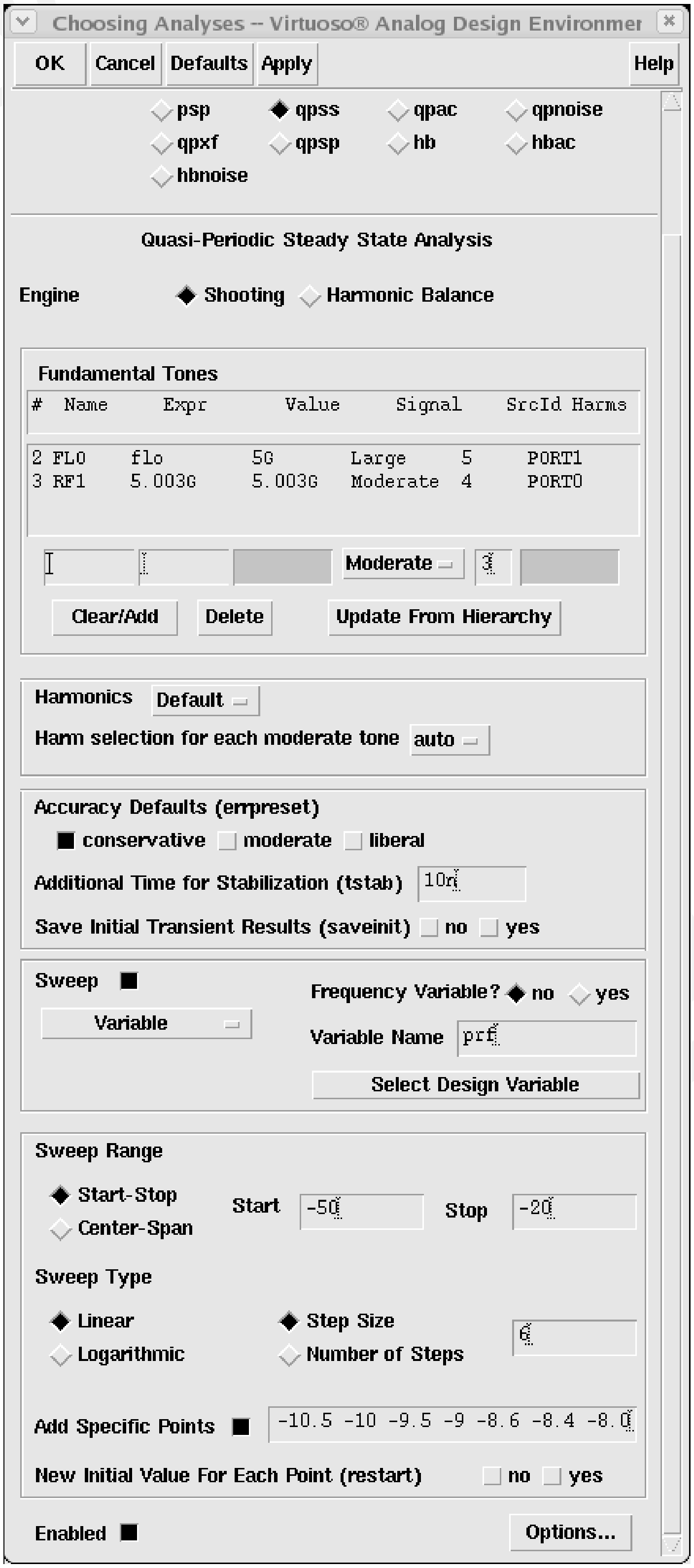
Action 8-4: Make sure the *Source type* of `PORT1` is set to *sine* and the *Source type* of `PORT3` is set to *dc*.

Action 8-5: Check and save the schematic.

Action 8-6: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab8_Blocker_QPSSQPACQPnoise**" and skip to Action 8-14 or …

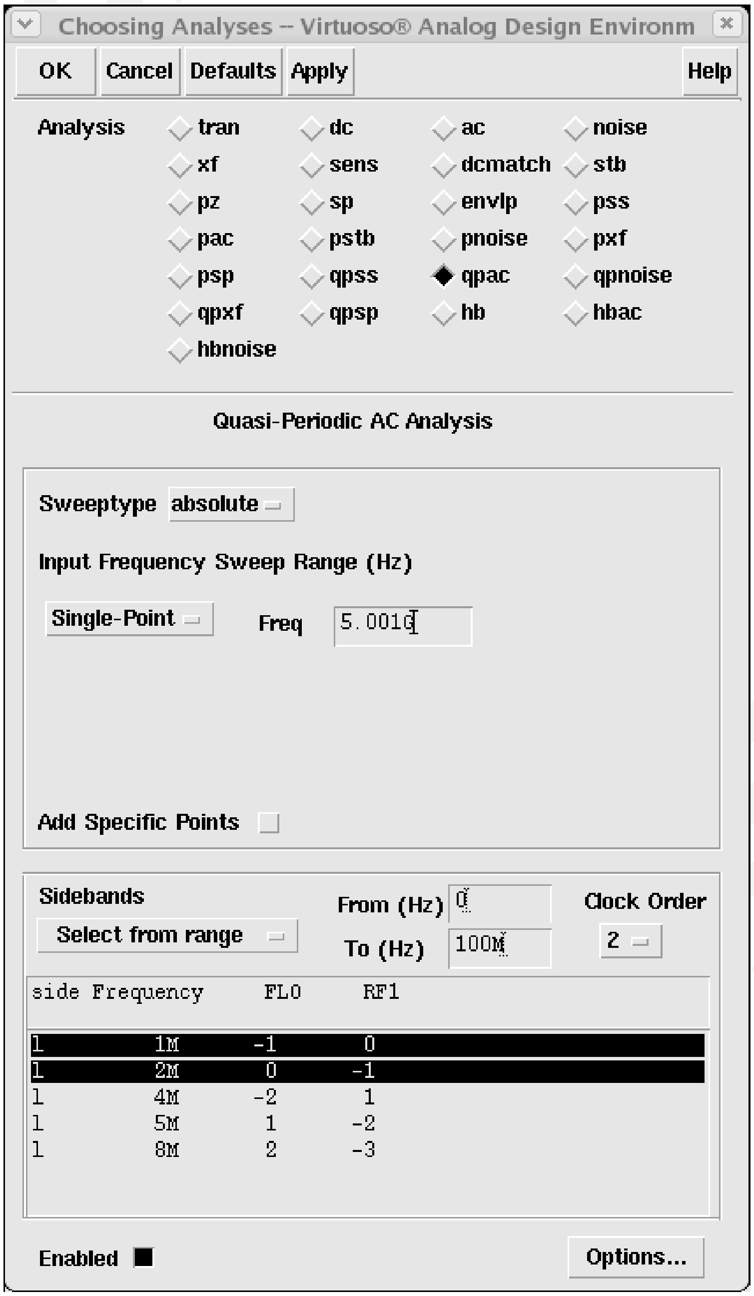
Action 8-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

Action 8-8: In the Choosing Analyses window, select the *qpss* button in the *Analysis* field of the window. Represent the blocking signal by setting the moderate tone frequency frf = 5.003 GHz. Represent a small-signal RF input by setting a fixed value for the pacm parameter. For example, in this example pacm = -30 dB. In the QPSS analysis, sweep the parameter prf from -50 dB to -8 dB. The form looks like this.


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Quasi-Periodic Steady State Analysis
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId Harms
2 FLO flo 5G Large 5 PORT1
3 RF1 5.003G 5.003G Moderate 4 PORT0
Moderate
3
Clear/Add
Delete
Update From Hierarchy
Harmonics
Default
Harm selection for each moderate tone
auto
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
10n
Save Initial Transient Results (saveinit)
no
yes
Sweep
Frequency Variable?
no
yes
Variable
Variable Name
prf
Select Design Variable
Sweep Range
Start-Stop
Center-Span
Start
-50
Stop
-20
Sweep Type
Linear
Logarithmic
Step Size
Number of Steps
6
Add Specific Points
-10.5 -10 -9.5 -9 -8.6 -8.4 -8.0
New Initial Value For Each Point (restart)
no
yes
Enabled
Options...

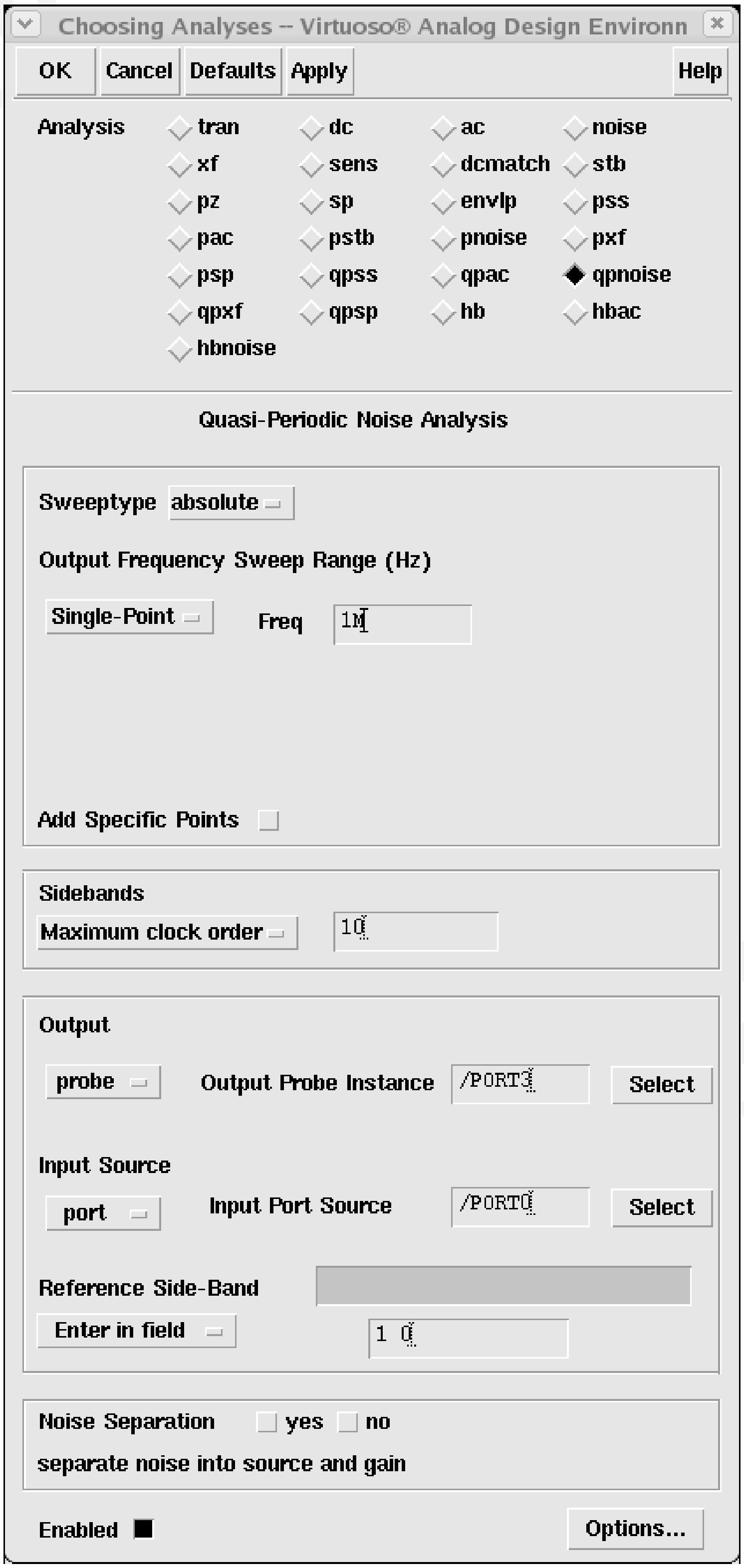
Action 8-9: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *Apply*.

Action 8-10: In the Choosing Analyses window, select the *qpac* button in the *Analysis* field of the window. Set the input frequency = 5.001 GHz and set *Sweeptype* = absolute. The form looks like this.



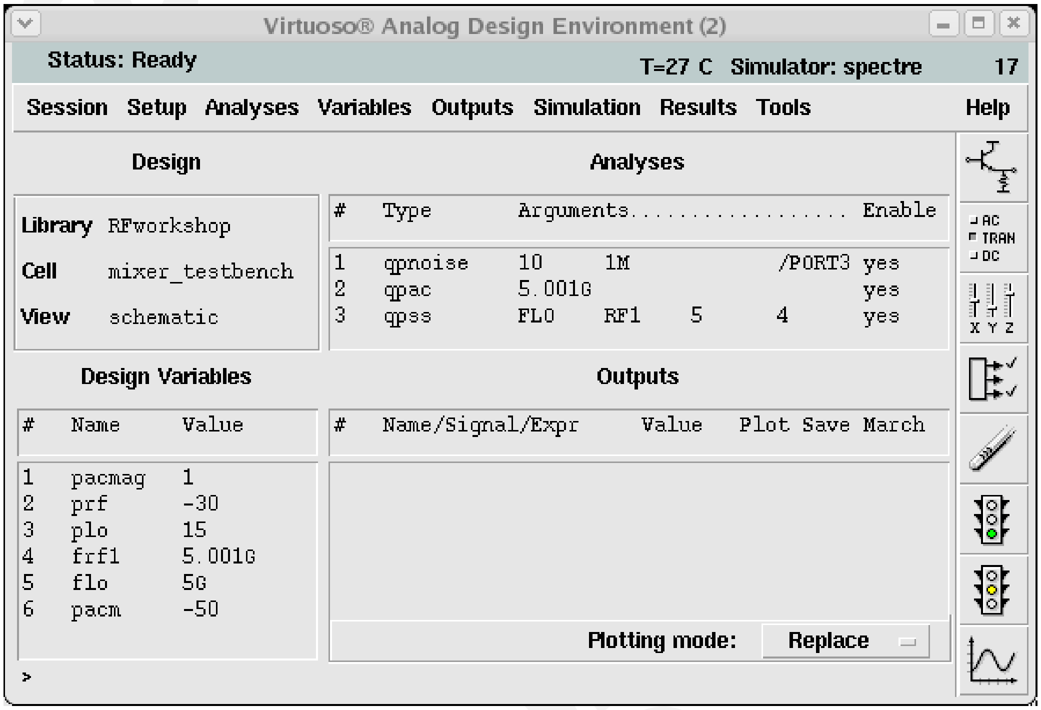
Action 8-11: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *Apply*.

Action 8-12: In the Choosing Analyses window, select the *qpnoise* button in the *Analysis* field of the window. Use a 1 MHz frequency point and *Maximum clock order* = 10. Set Output *probe* as `PORT3` and *Input Source* as `PORT0`. Use the *Reference side-band* as (1 0) to represent a downconverted RF signal relative to the IF output signal, 1 MHz + 1 * f (LO) = f (RF). 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
Quasi-Periodic Noise Analysis
Sweeptype
absolute
Output Frequency Sweep Range (Hz)
Single-Point
Freq
1M
Add Specific Points
Sidebands
Maximum clock order
10
Output
probe
Output Probe Instance
/PORT3
Select
Input Source
port
Input Port Source
/PORT0
Select
Reference Side-Band
Enter in field
1 0
Noise Separation
yes
no
separate noise into source and gain
Enabled
Options...

Action 8-13: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

The Virtuoso Analog Design Environment window looks like:

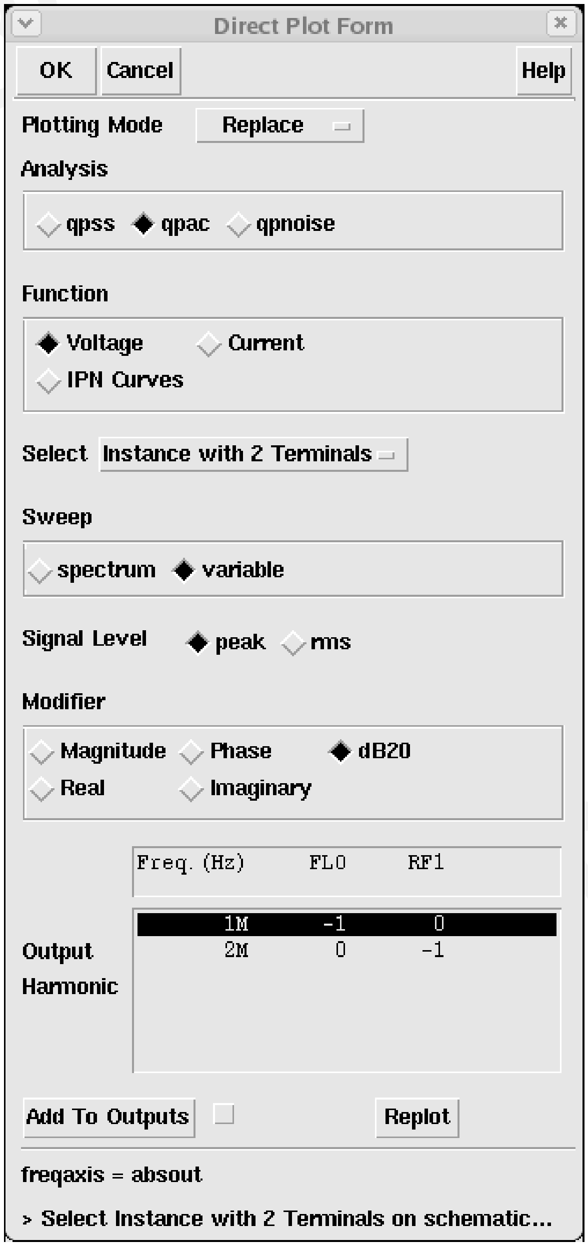


Action 8-14: In the Virtuoso Analog Design Environment, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

When the simulation ends, you can check how the blocking signal affects the performance of the Mixer.

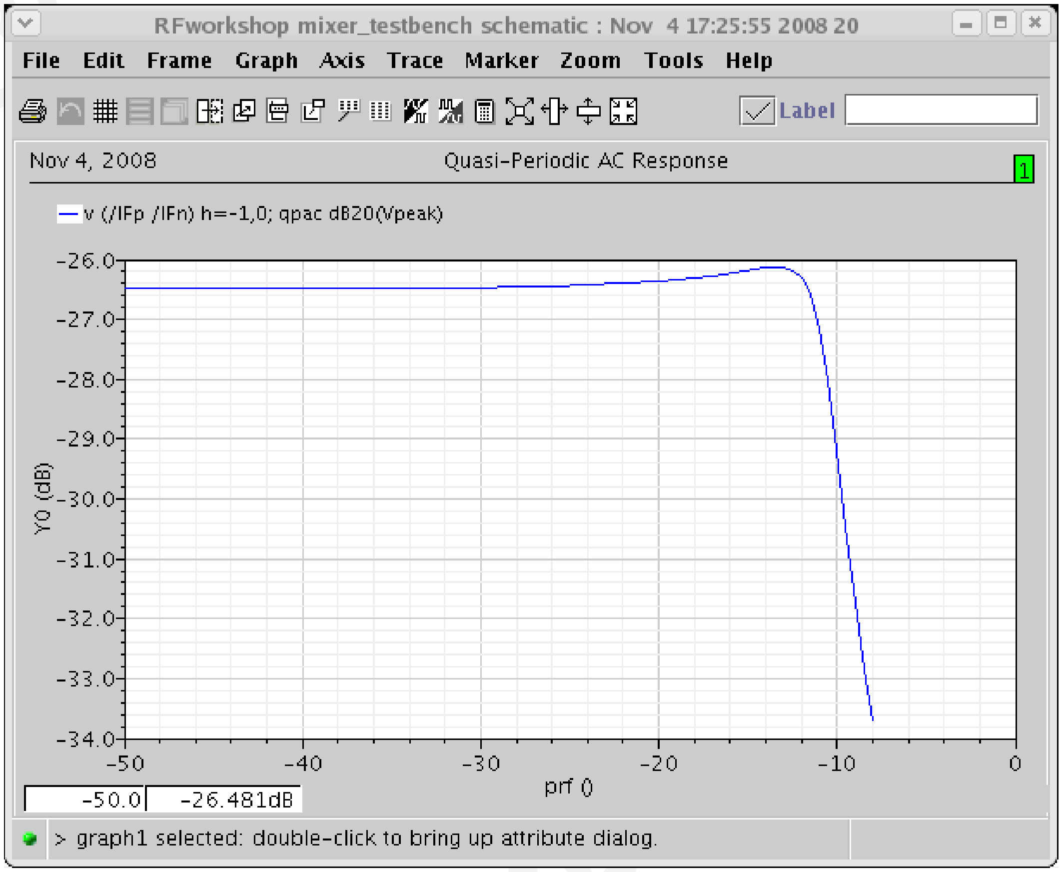
Action 8-15: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 8-16: In the Direct Plot Form, select the *qpac* button, and configure the form as follows:



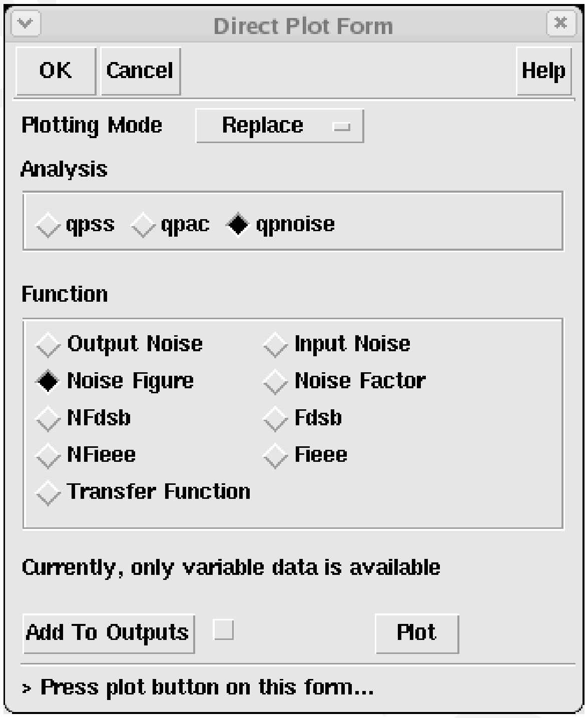
Action 8-17: Select `PORT3`.

The waveform window appears, showing the blocker effect on the voltage gain.



Action 8-18: Close the waveform window.

Action 8-19: In the Direct Plot Form, select the *qpnoise* button, and configure the form as follows:

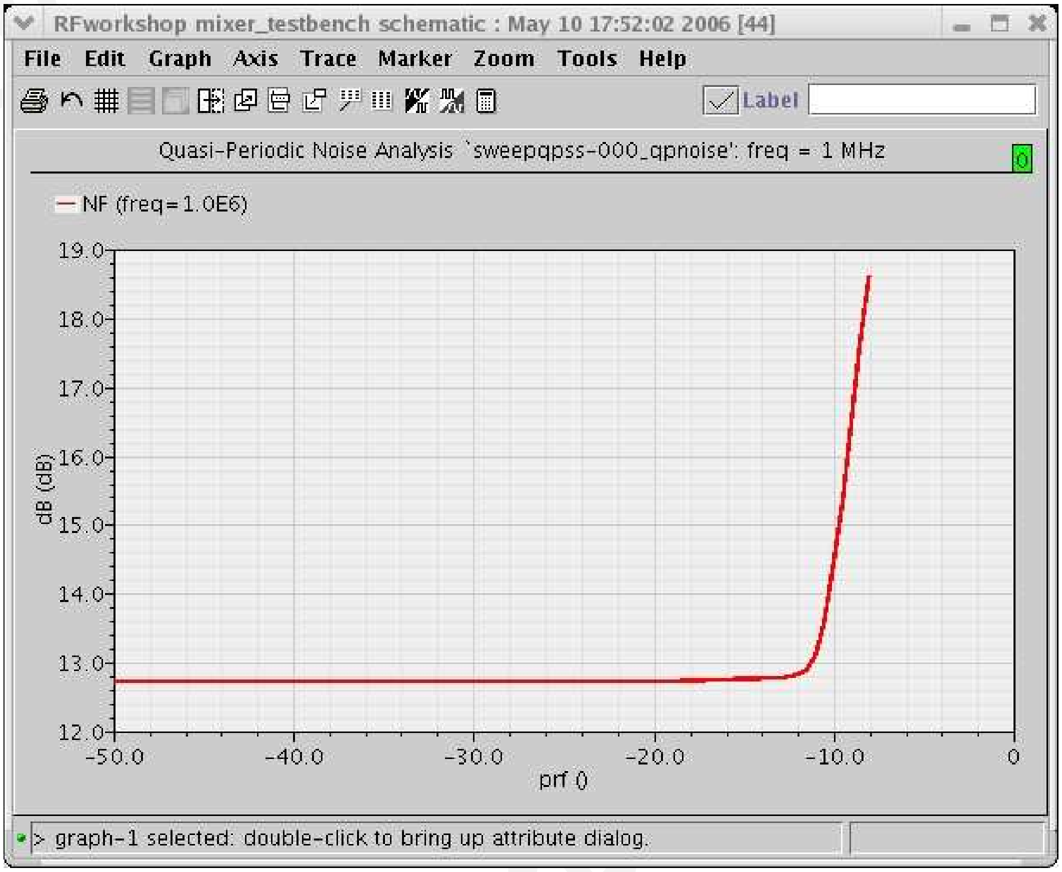


Action 8-20: Click *Plot*.

The waveform window appears.

Action 8-21: Double click the X axis and change the *Sweep Var* field to prf.

The waveform window appears, showing the blocker effect on the noise figure:



Action 8-22: Close the waveform window. Click *Cancel* in the Direct Plot Form.

Intermodulation Distortion and Intercept Points

Mixer distortion limits the sensitivity of a receiver if there is a large interfering signal present that is within the bandwidth of the RF input filter (a characteristic known as selectivity). There are two aspects of distortion that are of concern:

- Compression
- Intermodulation Distortion

The 1 dB compression point (CP1) is the point where the output power of the fundamental crosses the line that represents the output power extrapolated from small-signal conditions minus 1 dB. The 3rd order intercept point (IP3) is the point where the third-order term as extrapolated from small-signal conditions crosses the extrapolated power of the fundamental.

Intermodulation distortion occurs when signals at frequencies f_1 and f_2 mix together to form the response at $2f_1 - f_2$ and $2f_2 - f_1$. If f_1 and f_2 are close enough in frequency, then the intermodulation products $2f_1 - f_2$ and $2f_2 - f_1$ are in-band and interfere with the reception of the input signal. (When choosing f_1 and f_2, perform a PAC analysis to determine the bandwidth of the circuit, and place them in the middle of the bandwidth, close enough in frequency so that their intermodulation terms are well within the bandwidth.) Distortion of the output signal occurs because several of the odd-order intermodulation tones fall within the bandwidth of the circuit.

Intermodulation distortion is typically measured in the form of an intercept point. You determine the 3rd order intercept point (IP3) by plotting the power of the fundamental and the 3rd order intermodulation product versus the input power. Both input and output power should be plotted in some form of dB. Extrapolate both curves from a low power level and identify where they cross—that is the intercept point. To make this determination and to be comfortable with the accuracy of the results, you must have a broad region where both curves follow their asymptotic behavior. When in the asymptotic region, the slope of an nth order distortion product has a slope of n. Thus, when measuring IP3, the fundamental power curve is extrapolated from where the curve has a slope of 1 over a broad region. The 3rd order intermodulation product is extrapolated from a point where its curve has a slope of 3 over a broad region.

Previous versions of Spectre use either qpss-based or qpac-based methods to calculate IP3 in a system that contains a mixer and LO. In the qpss-based method, three-tone qpss analysis with LO, RF1 and RF2 frequencies ω_{LO}, ω_{RF1} and ω_{RF2} is run at a given RF power level. IM3 of harmonic $2\omega_{RF1}-\omega_{RF2}-\omega_{LO}$ is obtained from the solution. Assuming RF power is low enough and IM3 is dominated by leading order V_{RF}^3 terms, $log(V_{IM3})$ is expected to be a linear function of $log(V_{RF})$ with a slope of 3. IP3 is then extrapolated from V_{IM3}. Here V_{IM3} and V_{RF} are amplitudes of the IM3 and RF signals, respectively. This method requires very high accuracy to accommodate the large dynamic range between the RF and LO signals because they are mixed in the same solution vector. For a large circuit, this method also relies on speed and convergence of multi- tone qpss.

In the qpac-based method, a two-tone qpss analysis at frequencies ω_{RF1} and ω_{LO} is run first. Then RF2 input is included as a small signal by qpac analysis to calculate IM3 at $2\omega_{RF1}-\omega_{RF2}-\omega_{LO}$. As in the qpss-based method, this method also has to cover the dynamic range between RF1 and LO and depends on convergence of two-tone qpss.

Compared to the qpss-based approach, the qpac approach reduces computation from three-tone qpss to two-tone qpss plus a qpac by applying first order perturbation to RF2 signal. The amount of computation can be further reduced if we treat both RF signals as perturbation to the steady-state operating point at LO frequency with zero RF input. In this way, leading order intermodulation between RF1 and RF2 in IM3 can be computed directly from third order perturbation.

Starting in the MMSIM60 USR2 release, Spectre provides a perturbative approach to solve weakly nonlinear circuits. This approach does not require explicit high order derivatives from the 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 the RF input source. Parameter ε is introduced to keep track of the order of the perturbation expansion. Under weakly nonlinear condition, 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 or $O(\varepsilon^n)$.

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

Lab 9: IP3 Calculation (Swept QPSS and QPAC)

Action 9-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework*.

Action 9-2: Select the `PORT0` 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	RF1
Frequency 1	frf1
Amplitude 1 (dBm)	prf
PAC magnitude (dBm)	prf

Action 9-3: Click *OK* on the Edit Object Properties window to close.

Action 9-4: Check and save the schematic.

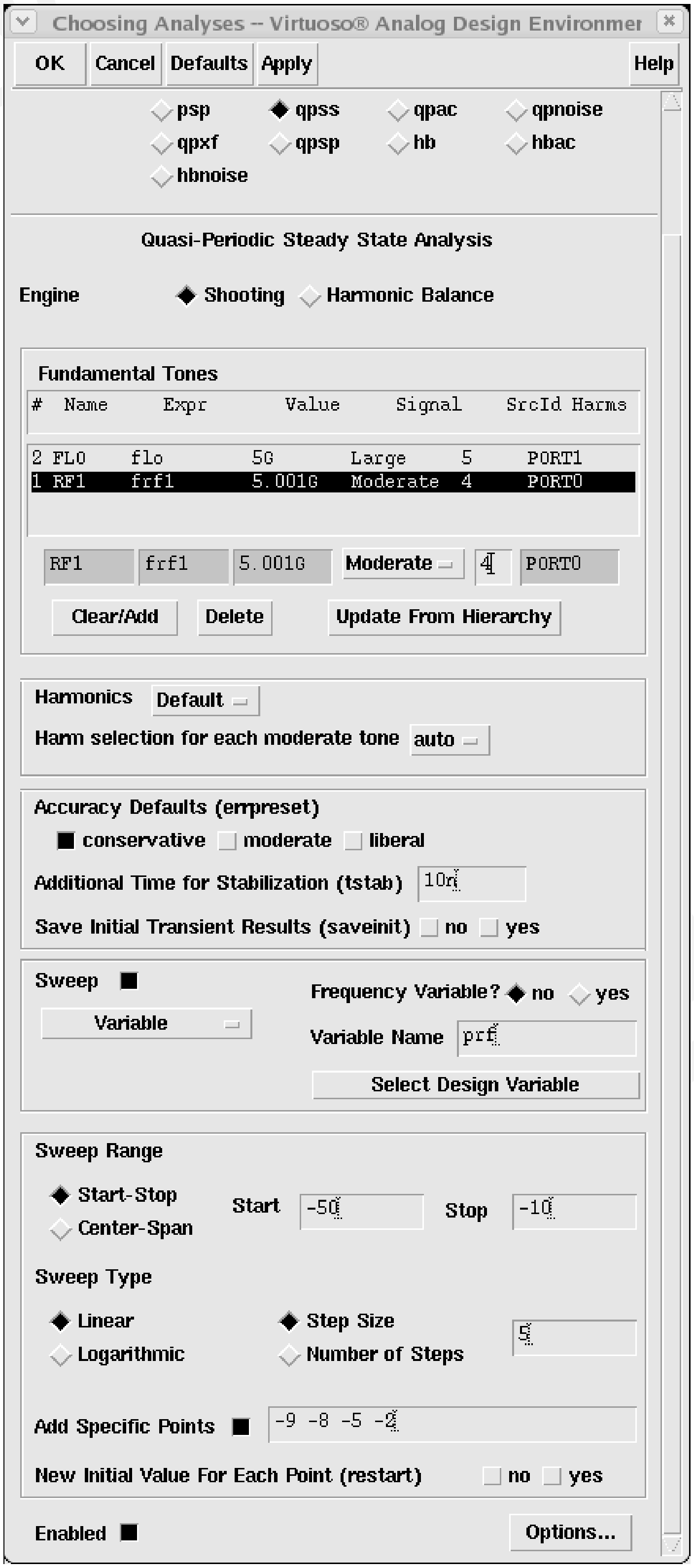
Action 9-5: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

The Virtuoso Analog Design Environment window appears.

Action 9-6: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab9_IP3_QPSSQPAC**" and skip to Action 9-12 or …

Action 9-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**

Action 9-8: In the Choosing Analyses window, select the *qpss* button in the *Analysis* field of the window and set the form as follows:

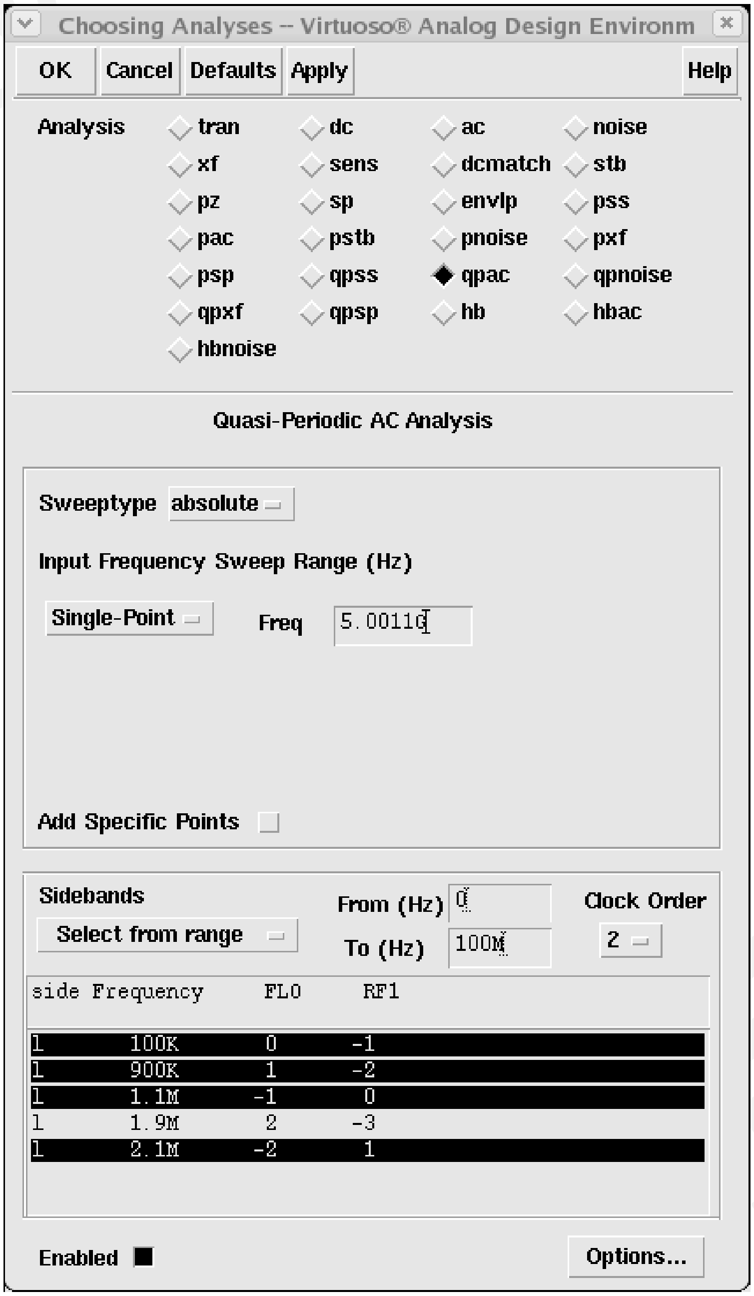

Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Quasi-Periodic Steady State Analysis
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId Harms
2 FLO flo 5G Large 5 PORT1
1 RF1 frf1 5.001G Moderate 4 PORT0
RF1
frf1
5.001G
Moderate
4
PORT0
Clear/Add
Delete
Update From Hierarchy
Harmonics
Default
Harm selection for each moderate tone
auto
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
10n
Save Initial Transient Results (saveinit)
no
yes
Sweep
Variable
Frequency Variable?
no
yes
Variable Name
prf
Select Design Variable
Sweep Range
Start-Stop
Center-Span
Start
-50
Stop
-10
Sweep Type
Linear
Logarithmic
Step Size
Number of Steps
5
Add Specific Points
-9 -8 -5 -2
New Initial Value For Each Point (restart)
no
yes
Enabled
Options...

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

Action 9-10: In the Choosing Analyses window, select the *qpac* button in the ***Analysis*** field of the window.

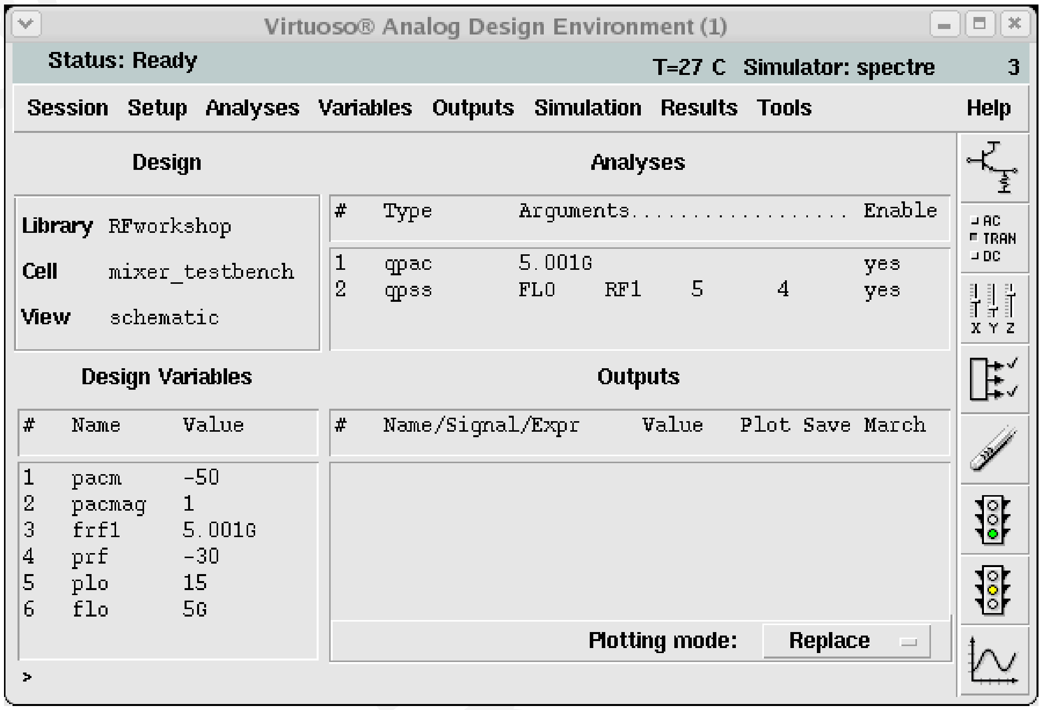
Set the frequency of the small signal very close to f(RF), for example 5.0011 GHz. In the *Select from range* option of the *Sidebands* section, highlight the harmonics of interest. Limit the harmonics to second order in the large tone (Set *Clock Order* = 2), from 0 Hz to 100 MHz. The example does not use the 3rd harmonic of the moderate tone, so remove them from the list.

The form looks like this:



Action 9-11: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

The Virtuoso Analog Design Environment window looks like this.

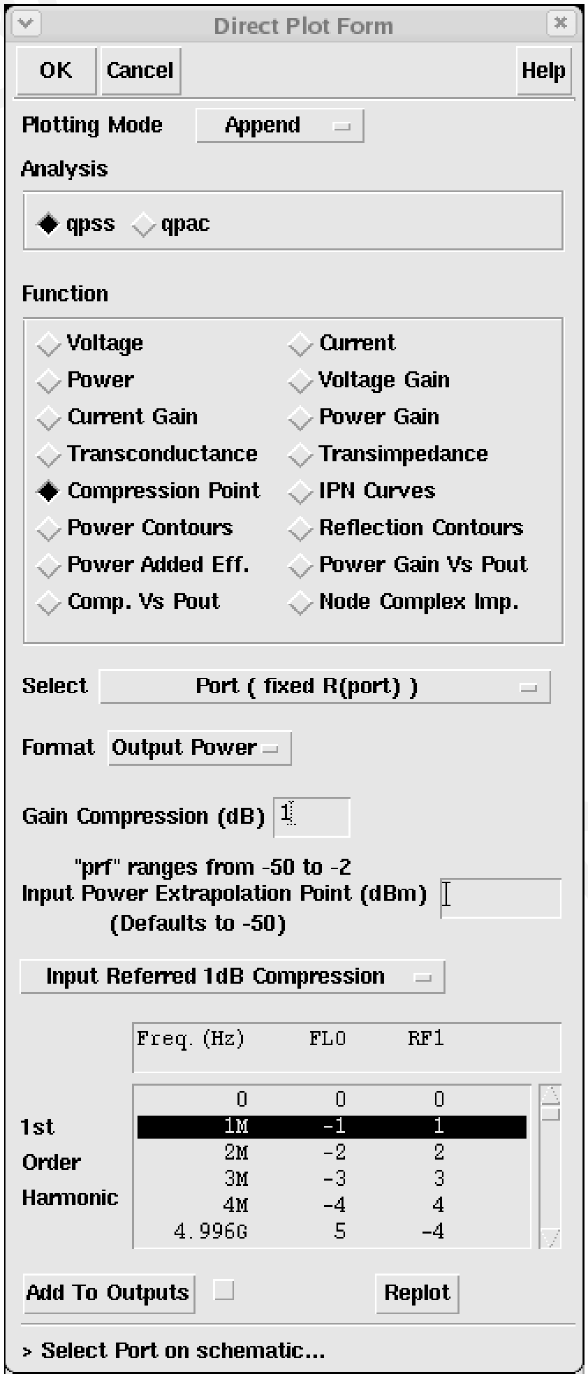


Action 9-12: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

When the simulation completes, use the Direct Plot feature to view the results.

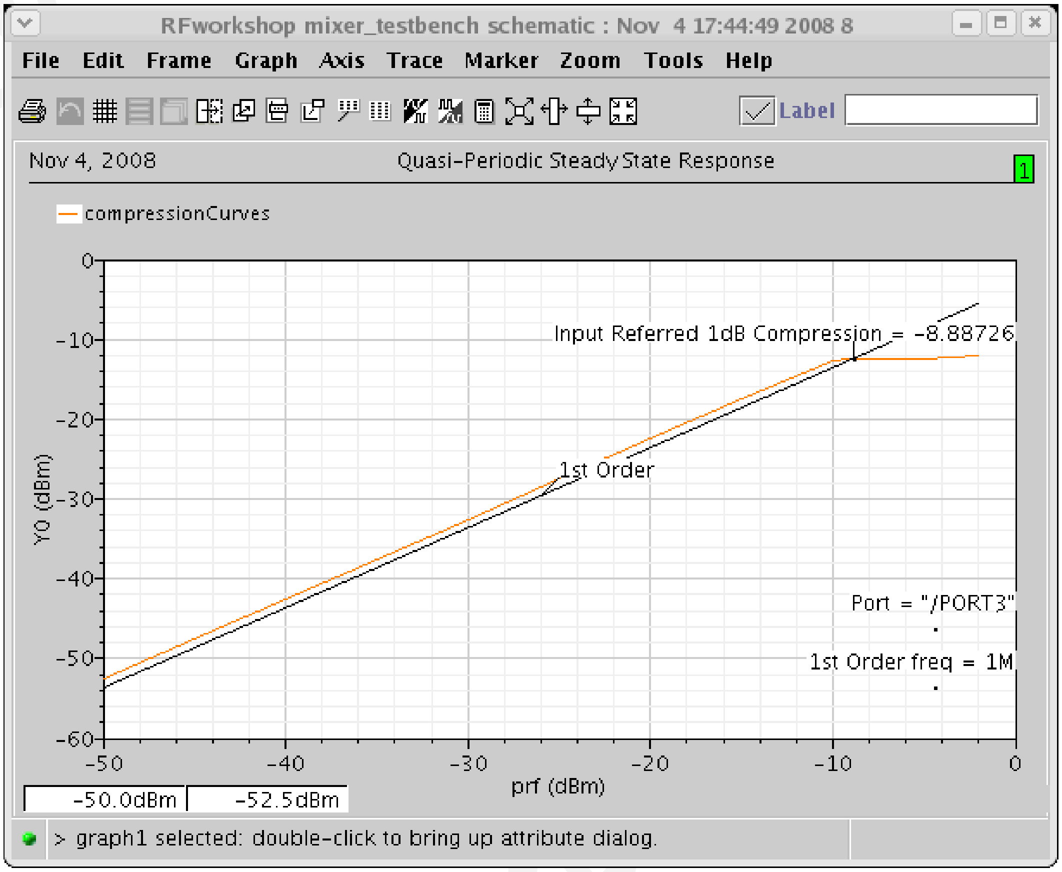
Action 9-13: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 9-14: To plot the 1 dB compression point, click *qpss* analysis in the Direct Plot Form. Select *Compression Point*. Select a point in the linear region for an extrapolation or leave it blank to use the default value. The output harmonic is (-1 1) or 1 MHz.



Action 9-15: Select output `Port3` on schematic.

You see the value of the P1dB value as shown below:

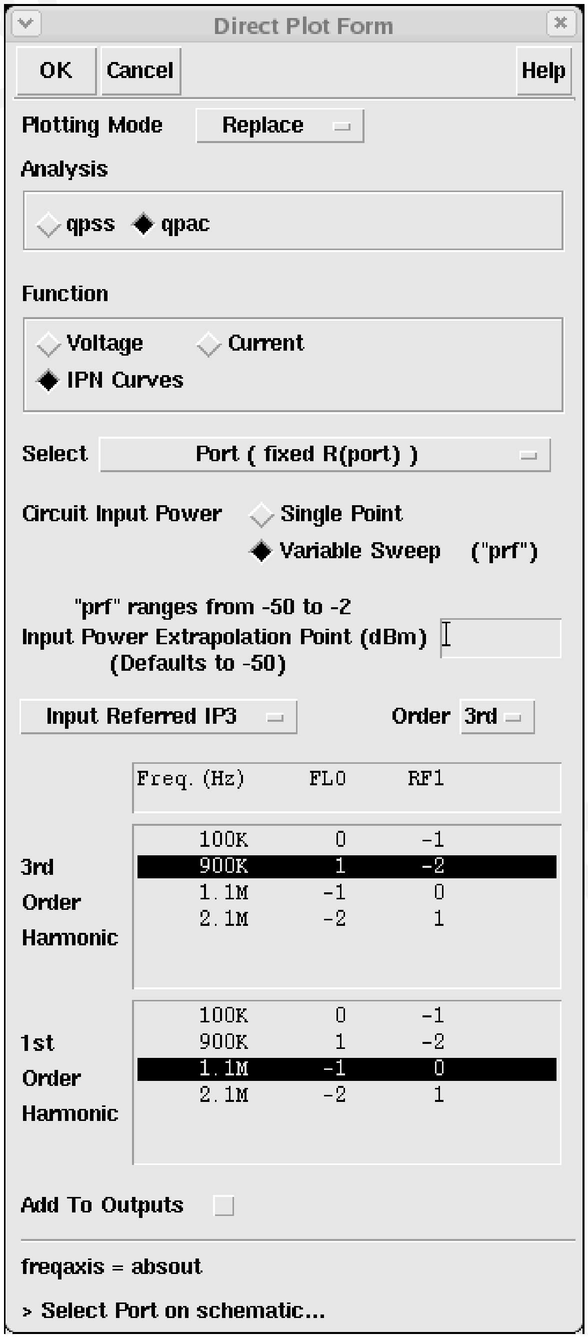


Action 9-16: Close the waveform window.

Action 9-17: To plot IP3, select *qpac* analysis in the Direct Plot Form, select the *IPN Curves* button, select *Variable Sweep* and choose -40 dB for the prf extrapolation. If the first extrapolation point you select is not in the linear range of the IM1 and IM3 curves, you might want to reset the extrapolation point later. To plot the third order input referred intercept point, set the first order harmonic to (-1 0) or 1.1 MHz, and the third order harmonic to (1 -2), or 0.9 MHz. Because the mixer is down-converting to the baseband, the first harmonic is calculated as:

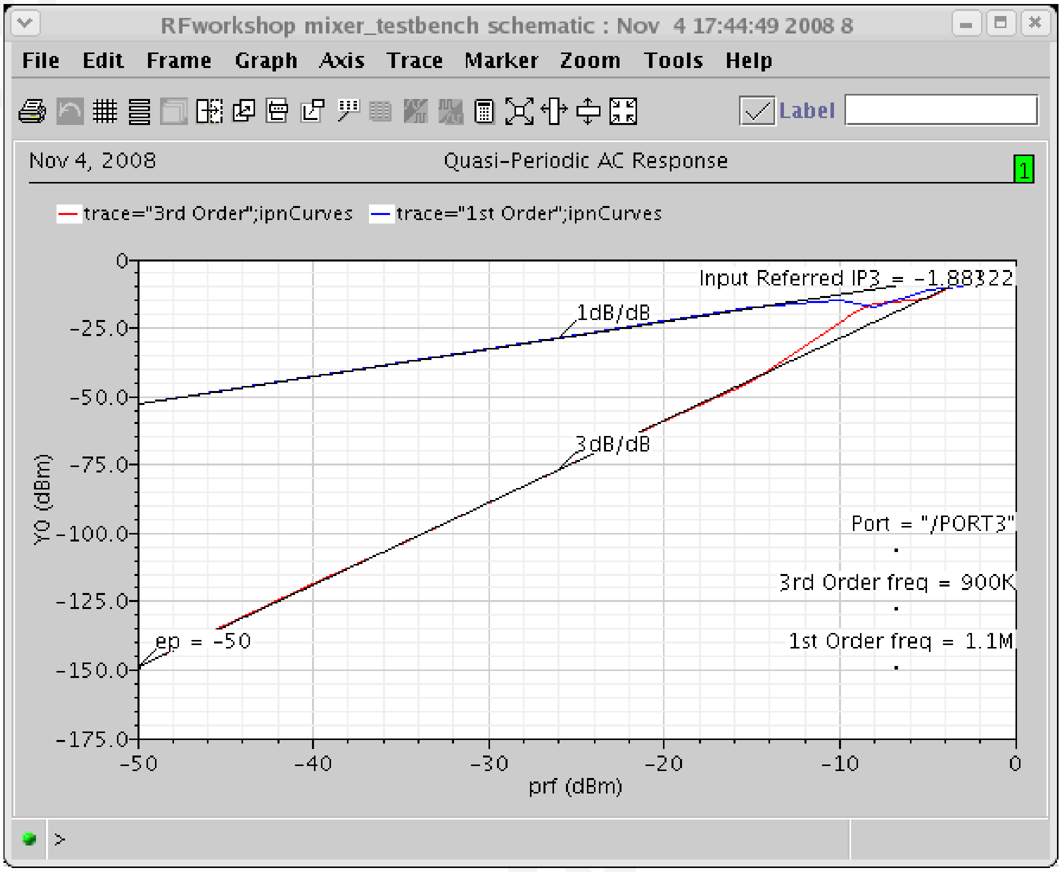
f(small signal) - f(LO) = 5.0011GHz - 5GHz = 1.1MHz

The third harmonic is at 0.9 MHz or -0.9 MHz depending on the *freqaxis* you selected in the Direct Plot Form. The form looks like this.



Action 9-18: Select output `Port3` on the schematic.

The third order input referred intercept point is calculated and curves of harmonics versus prf are presented as shown below:



Action 9-19: After viewing the waveforms, click *Cancel* in the Direct Plot Form.

For more accurate results, you might want to set *errpreset* = *conservative* when setting up the QPSS analysis. Initially, when you do not know the exact location of the linear region for IM3 and IM1, you may use *errpreset* = *moderate* to get a better understanding of your design. When the linear region is known, defining a single point simulation with *errpreset* = *conservative* is typically more accurate and less time-consuming.

Lab 10: IP3 Calculation (QPSS with Shooting Engine or Harmonic Balance Engine)

Another way to calculate IP3 is to apply the LO and two moderate RF input tones in a single QPSS analyses. That approach is illustrated in this lab.

Action 10-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework*

Action 10-2: Select the `PORT0` 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	RF1
Frequency 1	frf1
Amplitude 1 (dBm)	prf
Frequency name 2	RF2
Frequency 2	frf1+0.1M
Amplitude 2 (dBm)	prf

Action 10-3: Click *OK* on the Edit Object Properties window to close.

Action 10-4: Check and save the schematic.

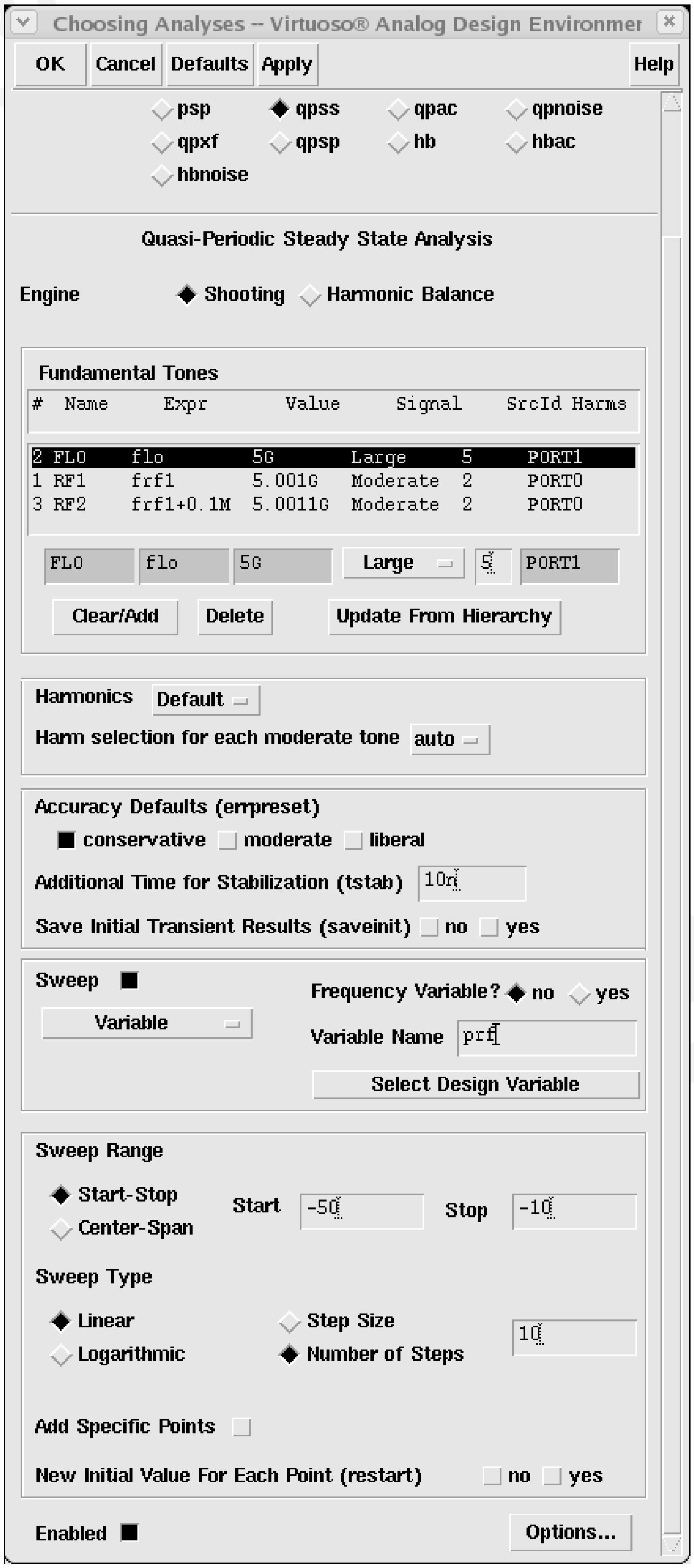
Action 10-5: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

The Virtuoso Analog Design Environment window appears.

Action 10-6: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab10_IP3_QPSS_shooting**" and skip to Action 10-10 or …

Action 10-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

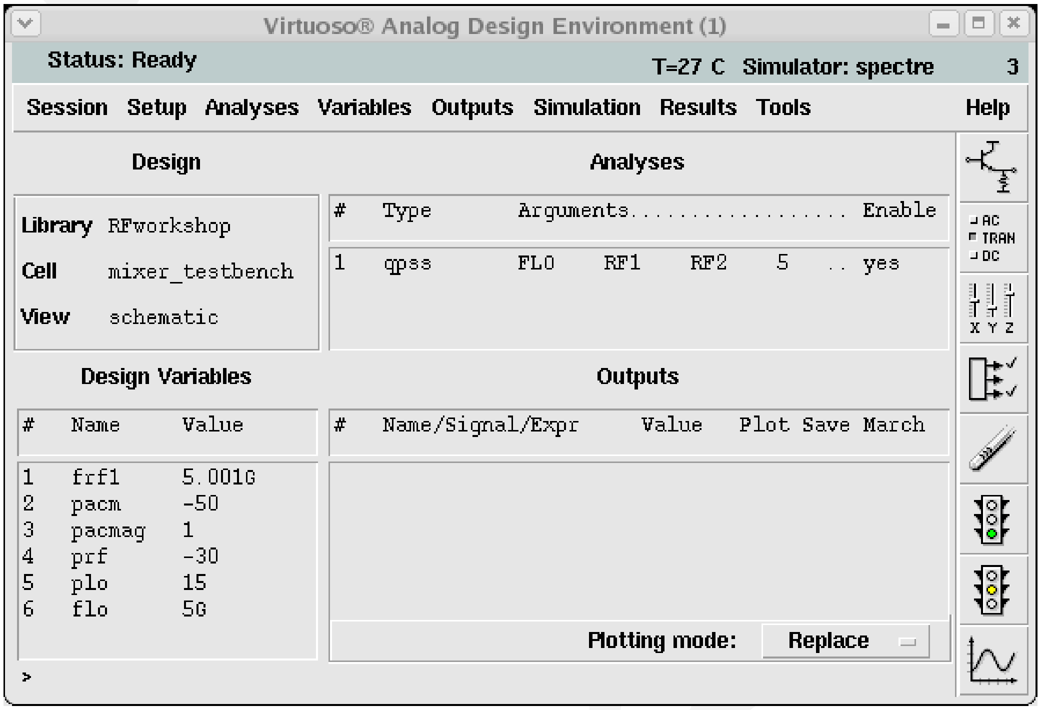
Action 10-8: In the Choosing Analyses window, select the *qpss* button in the *Analysis* field of the window and set the form as follows:


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Quasi-Periodic Steady State Analysis
Engine
Shooting
Harmonic Balance
Fundamental Tones
Name Expr Value Signal SrcId Harms
2 FLO flo 5G Large 5 PORT1
1 RF1 frf1 5.001G Moderate 2 PORT0
3 RF2 frf1+0.1M 5.0011G Moderate 2 PORT0
FLO
flo
5G
Large
5
PORT1
Clear/Add
Delete
Update From Hierarchy
Harmonics
Default
Harm selection for each moderate tone
auto
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
10n
Save Initial Transient Results (saveinit)
no
yes
Sweep
Frequency Variable?
no
yes
Variable
Variable Name
prf
Select Design Variable
Sweep Range
Start-Stop
Center-Span
Start
-50
Stop
-10
Sweep Type
Linear
Logarithmic
Step Size
Number of Steps
10
Add Specific Points
New Initial Value For Each Point (restart)
no
yes
Enabled
Options...

Note: The qpss shooting simulation time increases proportionally with the number of harmonics specified for the moderate signals.

Action 10-9: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

The Virtuoso Analog Design Environment window looks like this:

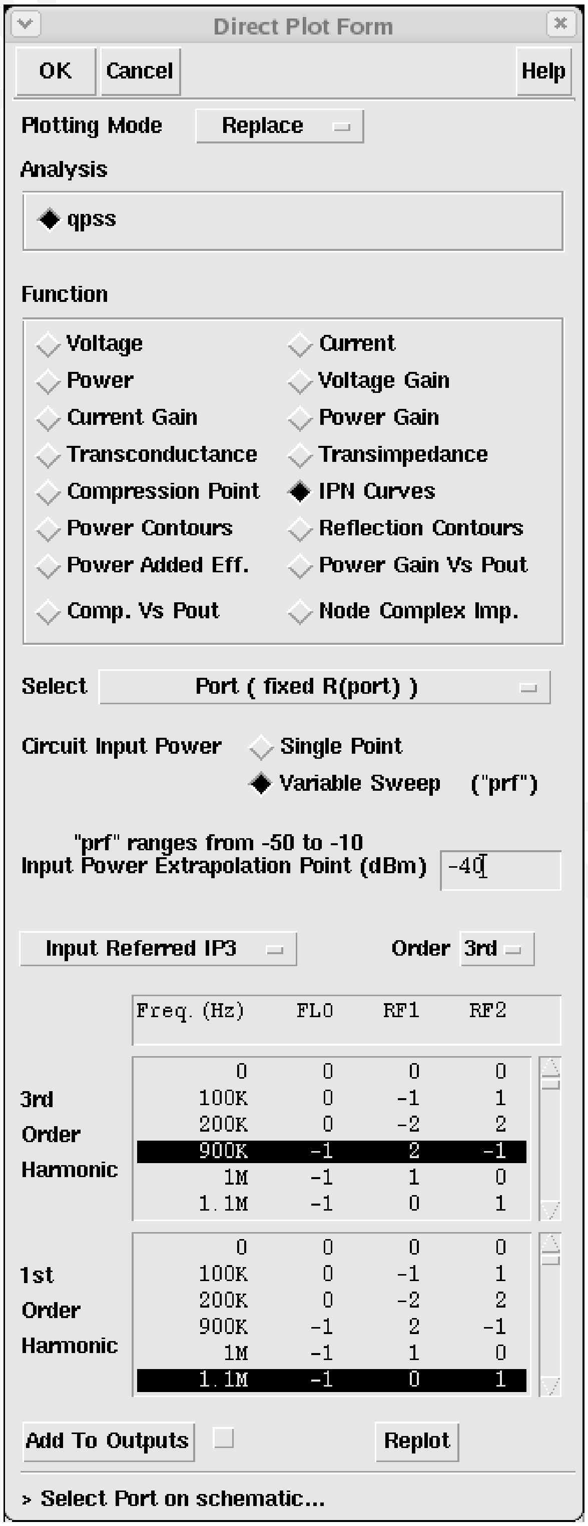


Action 10-10: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

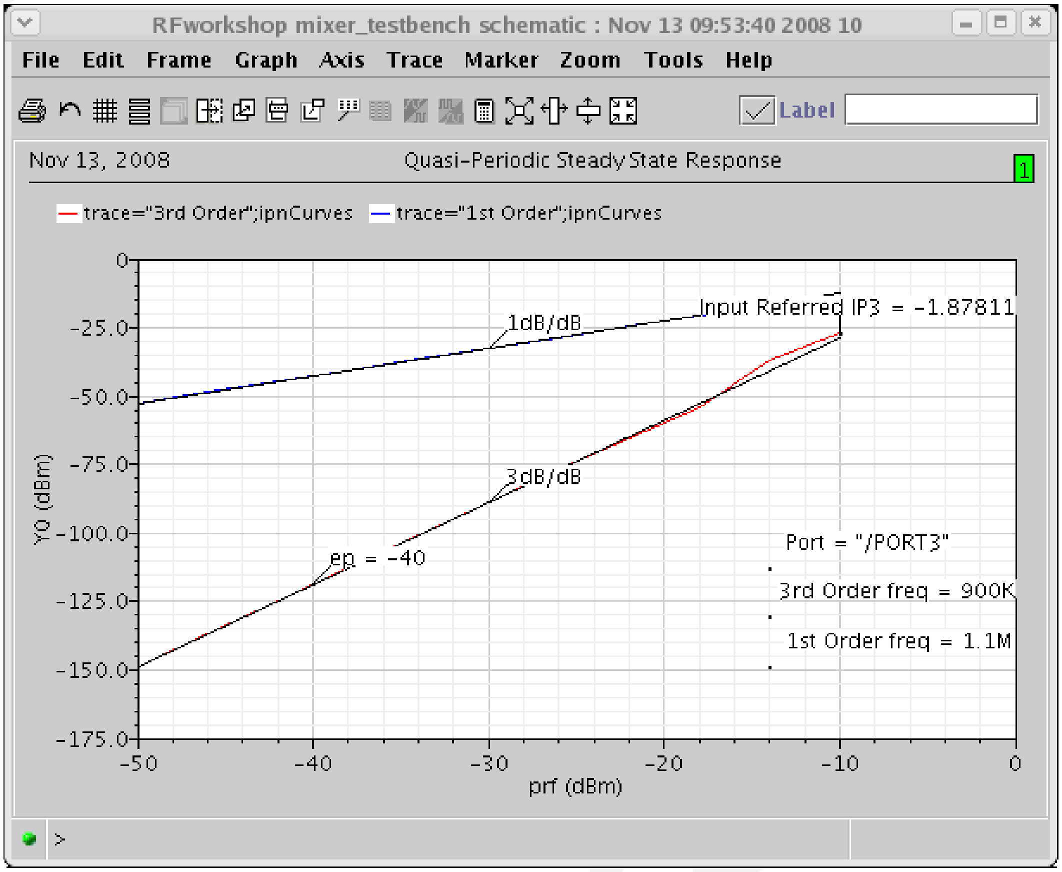
After the simulations finish, plot the IP3 and compare it with the results from Lab 9 (QPSS plus QPAC simulations).

Action 10-11: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 10-12: In the Direct Plot Form, select the *qpss* button, and configure the form as follows:


Direct Plot Form
OK
Cancel
Help
Plotting Mode
Replace
Analysis
qpss
Function
Voltage
Current
Power
Voltage Gain
Current Gain
Power Gain
Transconductance
Transimpedance
Compression Point
IPN Curves
Power Contours
Reflection Contours
Power Added Eff.
Power Gain Vs Pout
Comp. Vs Pout
Node Complex Imp.
Select
Port (fixed R(port))
Circuit Input Power
Single Point
Variable Sweep ("prf")
"prf" ranges from -50 to -10
Input Power Extrapolation Point (dBm)
-40
Input Referred IP3
Order 3rd
Freq. (Hz) FLO RF1 RF2
3rd Order Harmonic
0 0 0 0
100K 0 -1 1
200K 0 -2 2
900K -1 2 -1
1M -1 1 0
1.1M -1 0 1
1st Order Harmonic
0 0 0 0
100K 0 -1 1
200K 0 -2 2
900K -1 2 -1
1M -1 1 0
1.1M -1 0 1
Add To Outputs
Replot
> Select Port on schematic...

Action 10-13: Select output `Port3` on schematic. The IP3 calculation results looks like this.



Action 10-14: After viewing the waveforms, click *Cancel* in the Direct Plot Form

Action 10-15: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab10_IP3_QPSS_FB**" and skip to Action 10-22 or …

Action 10-16: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**

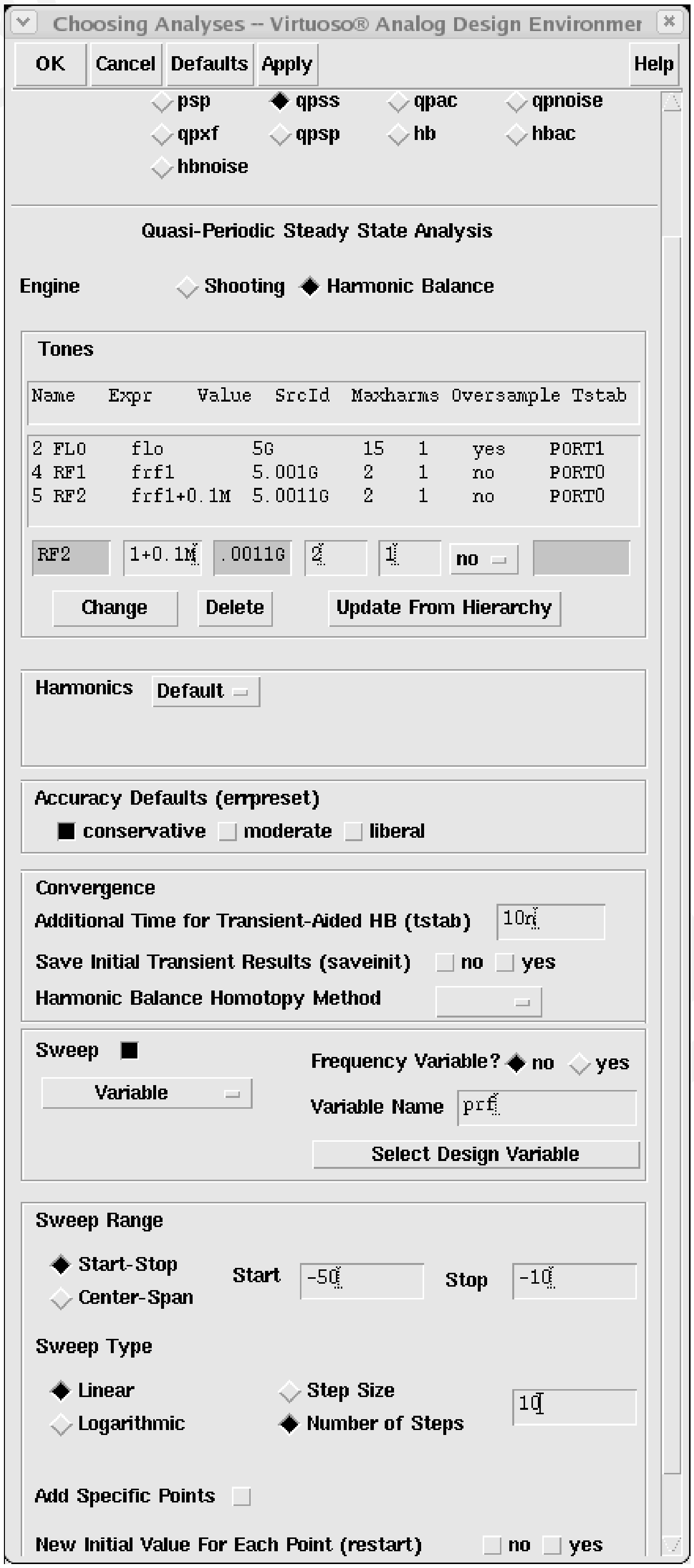
Action 10-17: In the Choosing Analyses window, select the *qpss* button in the *Analysis* field of the window.

Action 10-18: In the *Engine* field, choose *Harmonic Balance*.

Action 10-19: In the *Tones* field, choose FLO. Change the *Maxharms* to 15, because the harmonic balance algorithm needs more harmonics to calculate. Click *Update*.

Action 10-20: Type 10n in the *Additional Time for Stabilization (stab)* field.

The form looks like this.


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Quasi-Periodic Steady State Analysis
Engine
Shooting
Harmonic Balance
Tones
Name Expr Value SrcId Maxharms Oversample Tstab
2 FLO flo 5G 15 1 yes PORT1
4 RF1 frf1 5.001G 2 1 no PORT0
5 RF2 frf1+0.1M 5.0011G 2 1 no PORT0
RF2
1+0.1M
.0011G
2
1
no
Change
Delete
Update From Hierarchy
Harmonics
Default
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Convergence
Additional Time for Transient-Aided HB (tstab)
10n
Save Initial Transient Results (saveinit)
no
yes
Harmonic Balance Homotopy Method
Sweep
Frequency Variable?
no
yes
Variable
Variable Name
prf
Select Design Variable
Sweep Range
Start-Stop
Center-Span
Start
-50
Stop
-10
Sweep Type
Linear
Logarithmic
Step Size
Number of Steps
10
Add Specific Points
New Initial Value For Each Point (restart)
no
yes

Action 10-21: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

Note: The following parameters are required by the HB method:

1. **Harmonic Balance Flag:** The harmonic balance engine shares the same PSS/QPSS statement with time-domain engine and is run by setting the flag flexbalance=yes in the analysis statement. A toggle button is provided in the ADE PSS and QPSS set up forms to switch between time domain shooting and HB.

2. **Maximum Harmonic:** In PSS, the maximum harmonic is specified by **harms**. In QPSS, it is specified by **maxharms** for each tone. It is important to note that harms in PSS and the first maxharms value in QPSS are *output* parameters in the time-domain method. However, they are *input* parameters in the HB method and have direct impact on the accuracy and performance of the simulations.

The best value for the maximum harmonic depends on signal waveform and circuit nonlinearity. The faster the signal varies with time, or the more nonlinear the circuit is, the more harmonics are needed to represent the solution accurately. In multi-tone mixer cases, because the large LO tone has a higher power level than moderate RF tones and causes more nonlinear effects, usually more harmonics are used for the large tone than for the moderate tones.

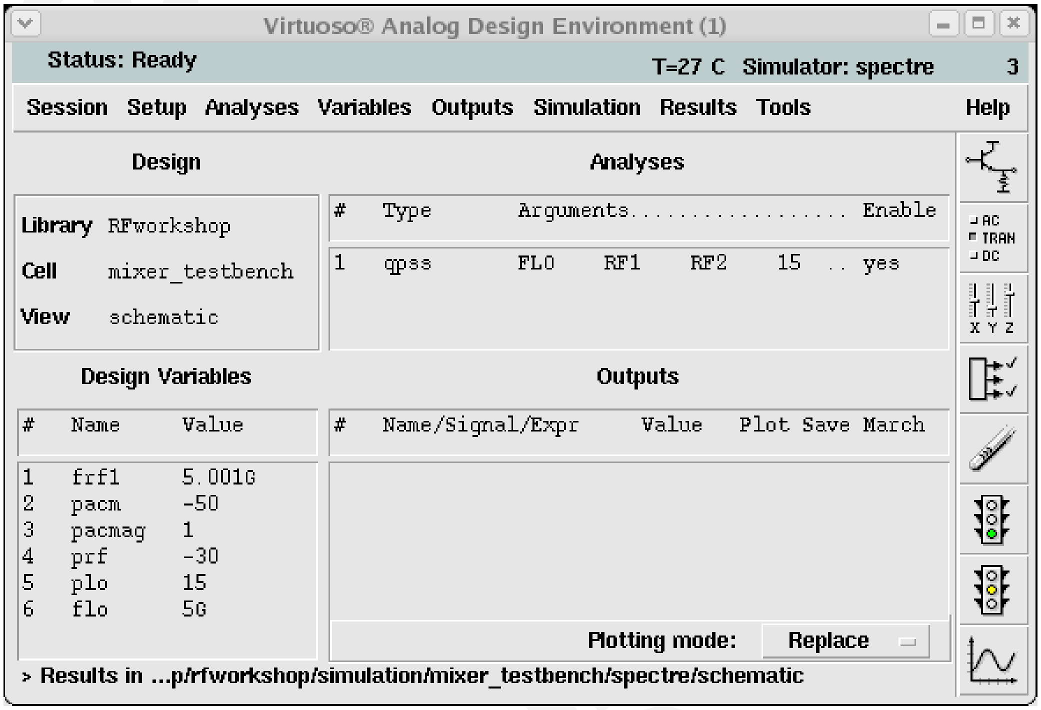
The maximum harmonic also depends on the order of nonlinear effect that you want to study. For example, for a mixer IP3 measurement, the maxharms of moderate tones must be set to at least 2 in order to capture the IM3 mode at frequency $2\omega_1 - \omega_2 - \omega_{LO}$.

3. **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 the signal period. For QPSS analysis, you can choose the specific tone during the tstab period and only one tone is allowed. One additional cycle is run for FFT. If **tstab** is set to 0, dc results are used as initial condition for HB.

Oversample Factor: (Optional HB parameter). In a PSS/QPSS statement, you can use **oversamplefactor**=m to specify the oversample factor. Spectre oversamples for each tone, and, as a result, the size of the Fourier transform is increased by using an oversample factor. For multi-tone cases, you can also specify **oversample**=[m1 ,m2, m3, ...] to oversample the tones by a specified factor, and again, the size of the Fourier transform is increased by using oversample factors. The default oversample factor is 1.

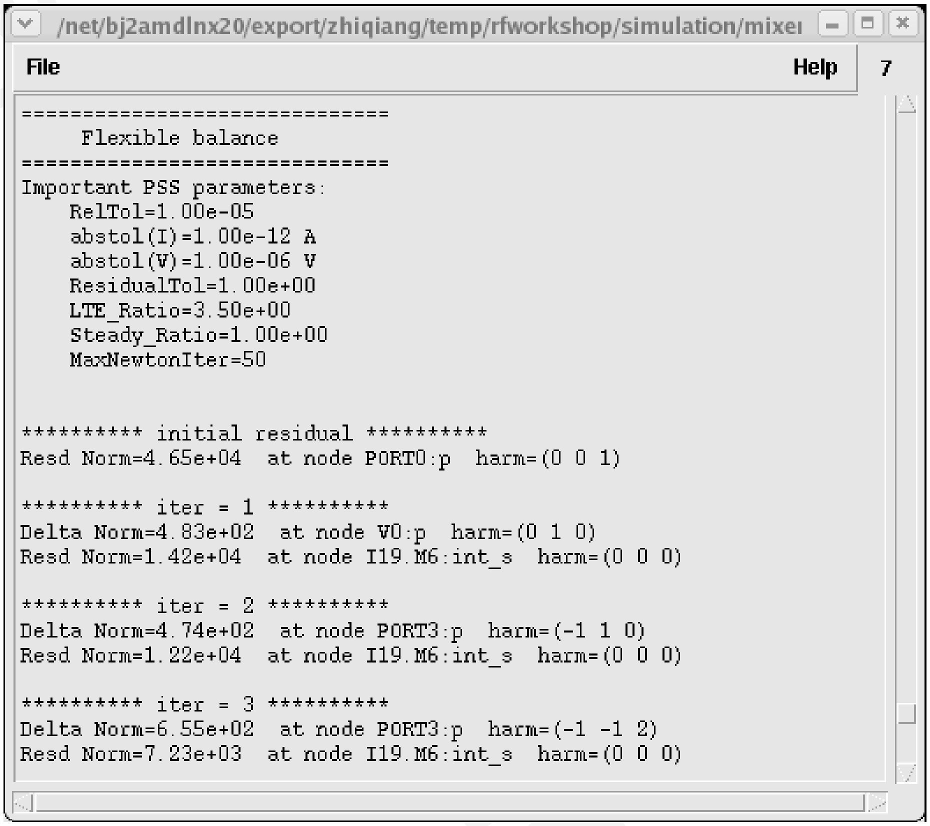
Even with oversampling, you must ensure that the maximum harmonic meets the accuracy requirement for your results of interest.

The Virtuoso Analog Design Environment window looks like this.



Action 10-22: 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. They differ from the messages produced by time domain qpss:



```
/net/bj2amdlnx20/export/zhiqiang/temp/rfworkshop/simulation/mixer
File                                                    Help   7
=============================
     Flexible balance
=============================
Important PSS parameters:
    RelTol=1.00e-05
    abstol(I)=1.00e-12 A
    abstol(V)=1.00e-06 V
    ResidualTol=1.00e+00
    LTE_Ratio=3.50e+00
    Steady_Ratio=1.00e+00
    MaxNewtonIter=50


********** initial residual **********
Resd Norm=4.65e+04  at node PORT0:p  harm=(0 0 1)

********** iter = 1 **********
Delta Norm=4.83e+02  at node V0:p  harm=(0 1 0)
Resd Norm=1.42e+04  at node I19.M6:int_s  harm=(0 0 0)

********** iter = 2 **********
Delta Norm=4.74e+02  at node PORT3:p  harm=(-1 1 0)
Resd Norm=1.22e+04  at node I19.M6:int_s  harm=(0 0 0)

********** iter = 3 **********
Delta Norm=6.55e+02  at node PORT3:p  harm=(-1 -1 2)
Resd Norm=7.23e+03  at node I19.M6:int_s  harm=(0 0 0)
```

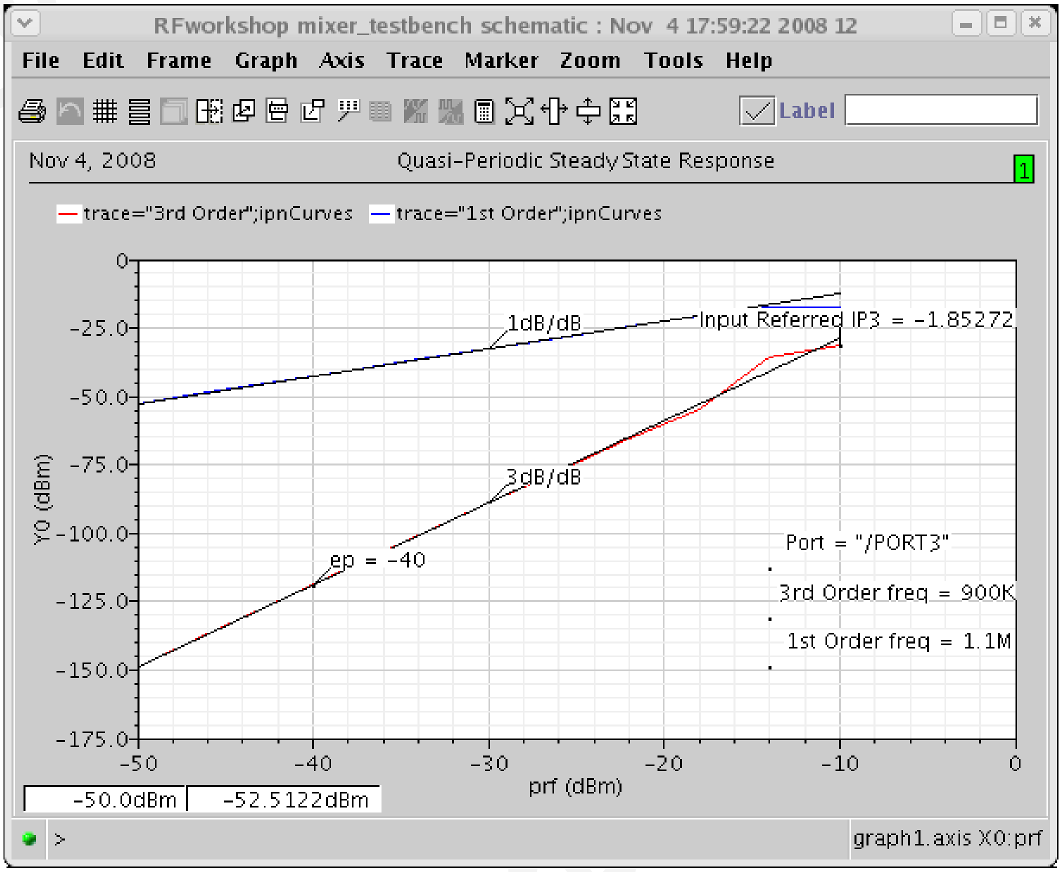
After the simulations finish, plot the IP3 and compare it with the result produced by the shooting engine.

Action 10-23: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 10-24: In the Direct Plot Form, select the *qpss* button, and configure the form the same as for the shooting engine.

Action 10-25: Select output `Port3` on the schematic.

The IP3 calculation results look like this.



Note: In this case, the Harmonic Balance engine produces the same results as the shooting engine but in a much reduced simulation time. For multi-tone cases such as mixers, HB is significantly more efficient than the shooting QPSS method due to its natural representation of circuit equations.

The HB method handles frequency dependent components better than the time domain method. HB also demonstrates better performance than shooting in post-layout circuits with a large number of linear elements. However, PSS HB performs better than time domain for weakly non linear circuits. (For strongly non linear circuits, time domain works best.)

For circuits driven by multi-tone stimulus, HB QPSS is better than HB PSS. Because HB multi-tone simulation does not have the convergence and speed issues encountered in shooting QPSS, HB QPSS should always be used to simulate multi-tone circuits. HB PSS analysis using a beat frequency as a fundamental is very inefficient in handling multi-tone cases. When source frequencies are closely spaced, their common frequency is so low that hundreds or even thousands of harmonics must be used.

Action 10-26: Close the waveform window and click *Cancel* on the Direct Plot Form.

Lab 11: Rapid IP3 (PAC)

Rapid Ip2/Ip3 based on perturbation technology extends both shooting and harmonic balance. Rapid IM (IP2, IP3) calculations are an order of magnitude faster than using harmonic balance or shooting alone.

Action 11-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework*.

Action 11-2: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

The Virtuoso Analog Design Environment window appears.

Action 11-3: Select the `PORT0` 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	*dc*
PAC magnitude (dBm)	pacm

Action 11-4: Click *OK* on the Edit Object Properties window to close it.

Action 11-5: Check and save the schematic.

Action 11-6: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab11_Rapid_IP3_PAC**" and skip to Action 11-11 or …

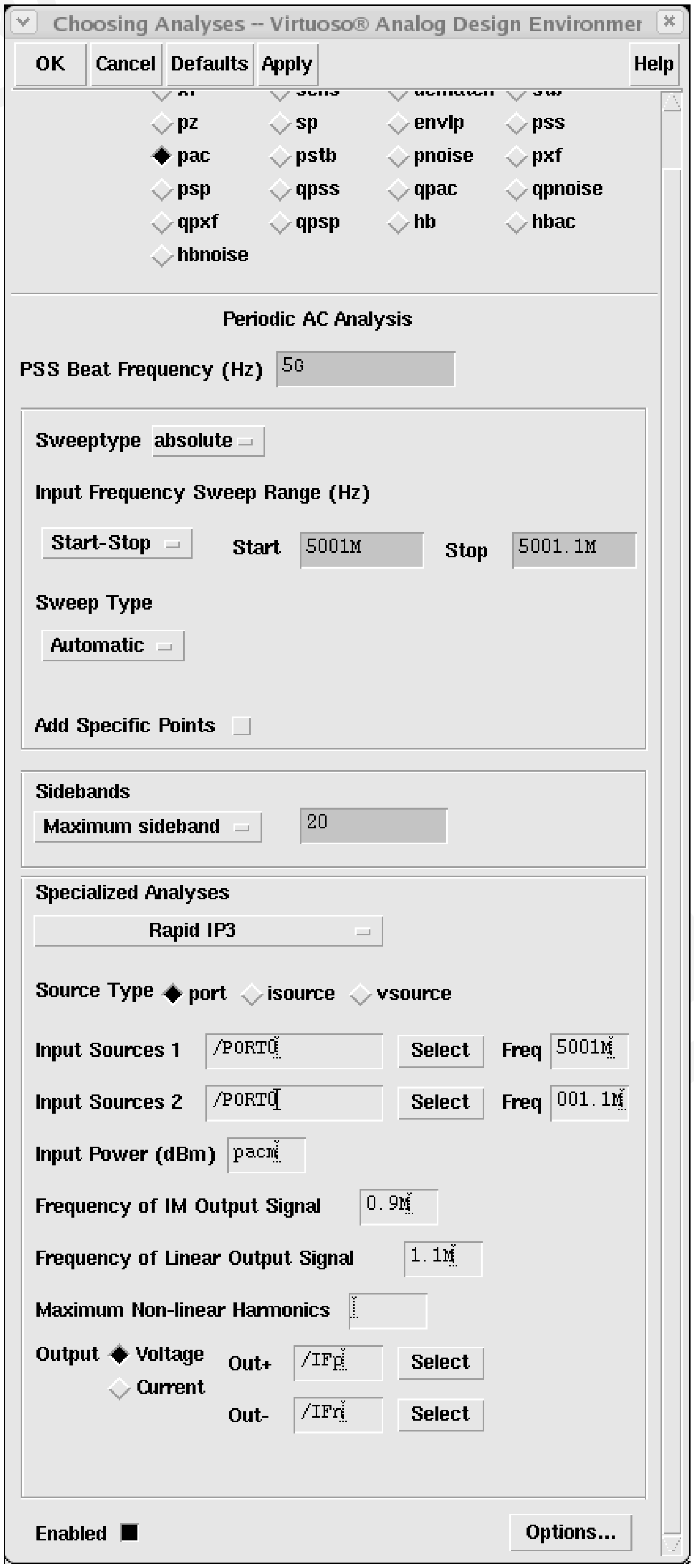
Action 11-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

Action 11-8: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window and set the form as shown in Action 2-6.

Action 11-9: 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 Sources 1* to `/PORT0` by selecting `PORT0` on the schematic.

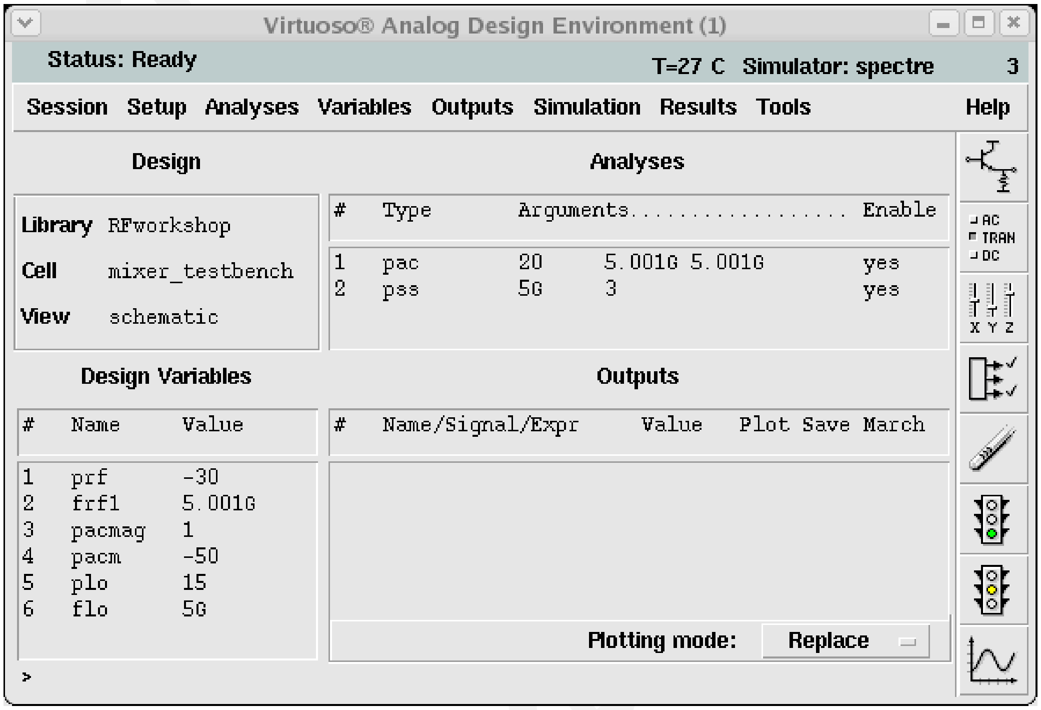
Press ESC to terminate the selection process. Set the *Freq* of source 1 to 5001M and *Freq* of Source 2 to 5001.1M. Set the *Frequency of IM Output Signal* as 0.9M and the *Frequency of Linear Output Signal* as 1.1M.

The form looks like this:


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Periodic AC Analysis
PSS Beat Frequency (Hz)
5G
Sweeptype
absolute
Input Frequency Sweep Range (Hz)
Start-Stop
Start
5001M
Stop
5001.1M
Sweep Type
Automatic
Add Specific Points
Sidebands
Maximum sideband
20
Specialized Analyses
Rapid IP3
Source Type
port
isource
vsource
Input Sources 1
/PORT0
Select
Freq
5001M
Input Sources 2
/PORT0
Select
Freq
001.1M
Input Power (dBm)
pacm
Frequency of IM Output Signal
0.9M
Frequency of Linear Output Signal
1.1M
Maximum Non-linear Harmonics
Output
Voltage
Current
Out+
/IFp
Select
Out-
/IFn
Select
Enabled
Options...

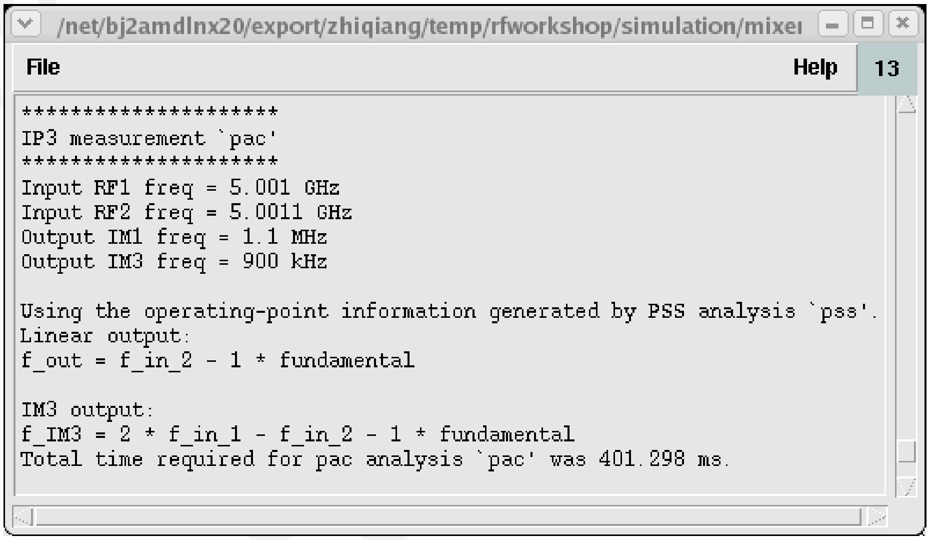
Action 11-10: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

The Virtuoso Analog Design Environment window looks like this.

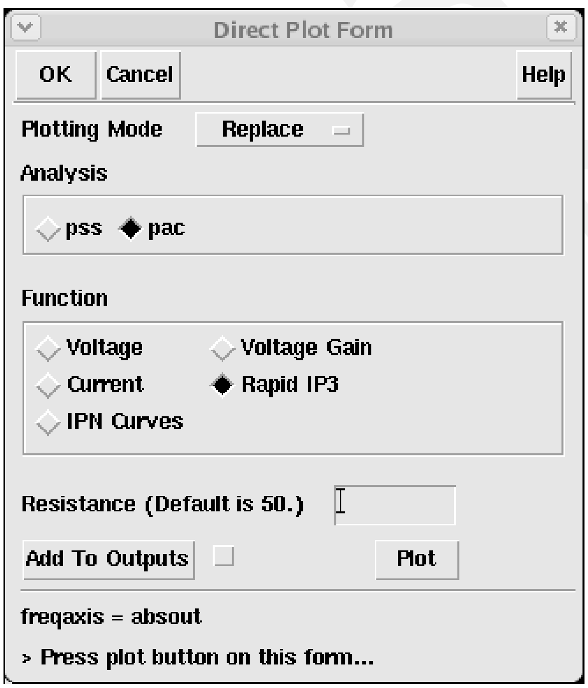


Action 11-11: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

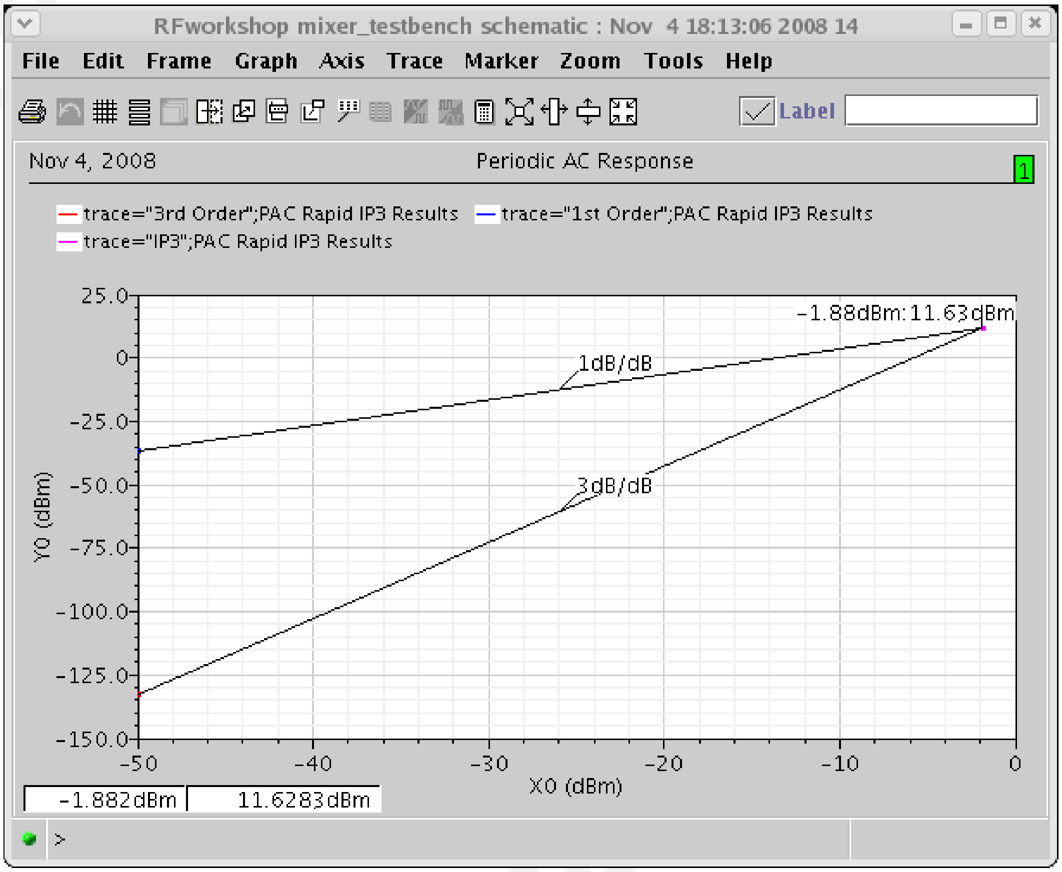
As the simulation progresses, messages similar to the following appear in the simulation output log window:



Action 11-12: In the Direct Plot Form, select the *pac* button, and choose *Rapid IP3*. The form looks like this.



Action 11-13: Click *Plot*. The calculated IP3 appears in the waveform window:



Action 11-14: Close the waveform window and click *Cancel* on the Direct Plot Form..

Lab 12: Compression Distortion Summary (PAC)

Action 12-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework*.

Action 12-2: Select `PORT0`. 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	*dc*
PAC Magnitude (dBm)	pacm

Action 12-3: Click *OK* on the Edit Object Properties window to close it.

Action 12-4: Check and save the schematic.

Action 12-5: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

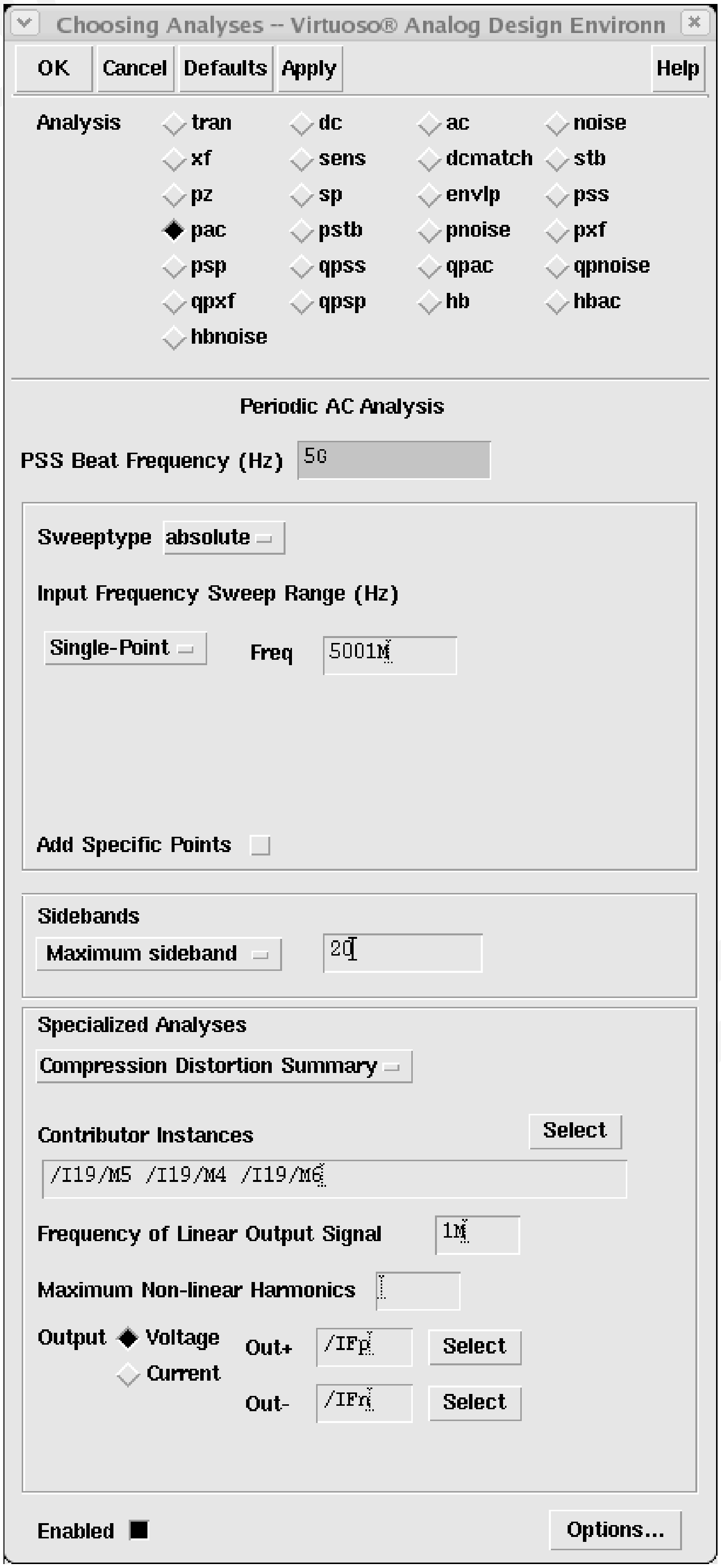
The Virtuoso Analog Design Environment window appears.

Action 12-6: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab12_CompDistorSmry_PAC**" and skip to Action 12-11 or …

Action 12-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**

Action 12-8: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window and set the form as shown in Action 2-6.

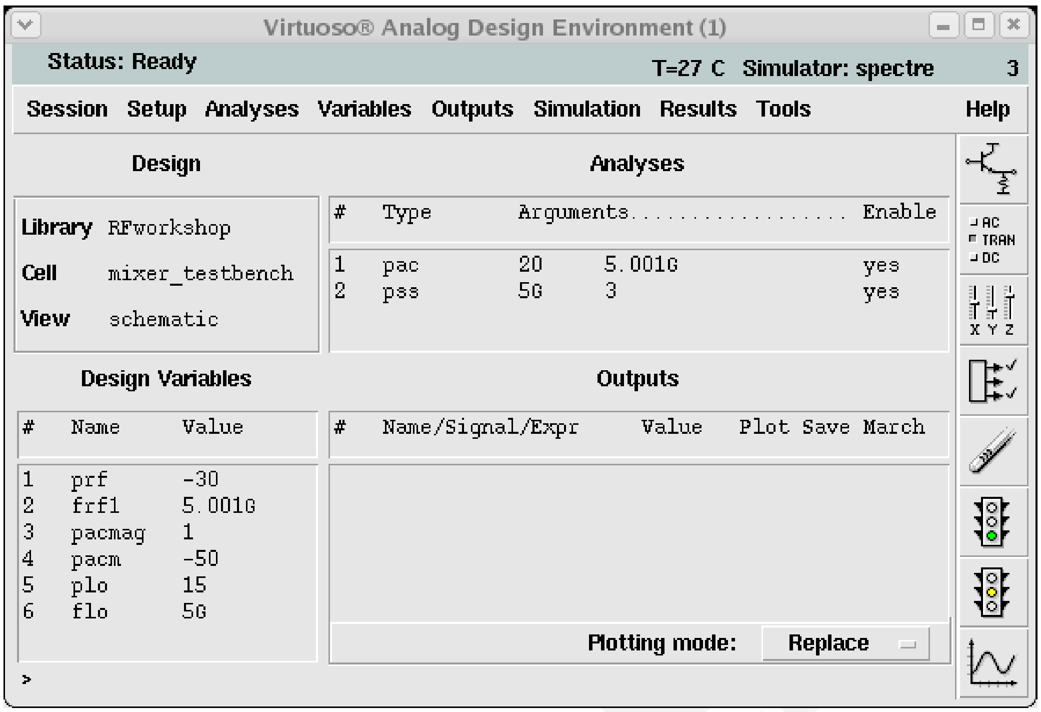
Action 12-9: In the Choosing Analyses window, select the *pac* button in the *Analysis* field of the window and set the form as follows:


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 AC Analysis
PSS Beat Frequency (Hz)
5G
Sweeptype
absolute
Input Frequency Sweep Range (Hz)
Single-Point
Freq
5001M
Add Specific Points
Sidebands
Maximum sideband
20
Specialized Analyses
Compression Distortion Summary
Contributor Instances
Select
/I19/M5 /I19/M4 /I19/M6
Frequency of Linear Output Signal
1M
Maximum Non-linear Harmonics
Output
Voltage
Current
Out+
/IFp
Select
Out-
/IFn
Select
Enabled
Options...

In the above form, the Maximum Non-linear Harmonics is not specified, so the default value 4 is used.

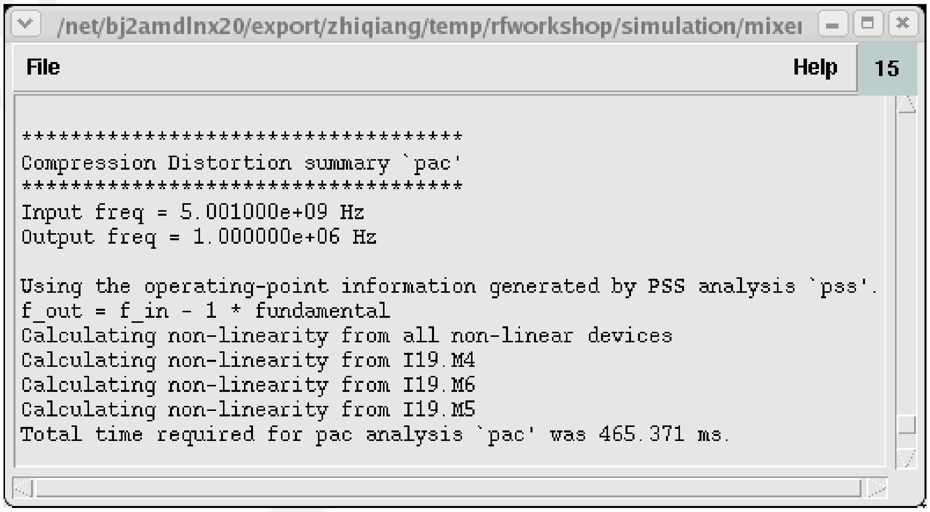
Action 12-10: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

The Virtuoso Analog Design Environment window looks like this.



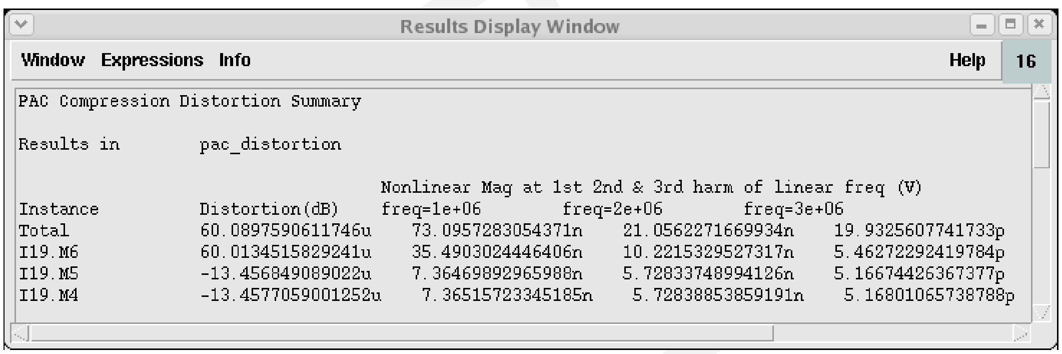
Action 12-11: In the Virtuoso Analog Design Environment, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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



Action 12-12: After the simulation completes, go to the Virtuoso Analog Design Environment window and choose **Results — Print — PAC Distortion Summary.**

The Results Display Window appears:



Action 12-13: After viewing the results, close it by choosing **Window — Close**.

Lab 13: Rapid IP2 (PAC)

Action 13-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework.*

Action 13-2: Use the mouse to select the `PORT0` source. Then in the Virtuoso Schematic Editor select **Edit — Properties — Objects.**

The Edit Object Properties window for the port cell appears. Set the *Source type* to *dc*.

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	500 mV
Source type	*dc*

Action 13-3: Click *OK* on the Edit Object Properties window to close it.

Action 13-4: Check and save the schematic.

Action 13-5: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

The Virtuoso Analog Design Environment window appears.

Action 13-6: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab13_Rapid_IP2_PAC**" and skip to Action 13-12 or …

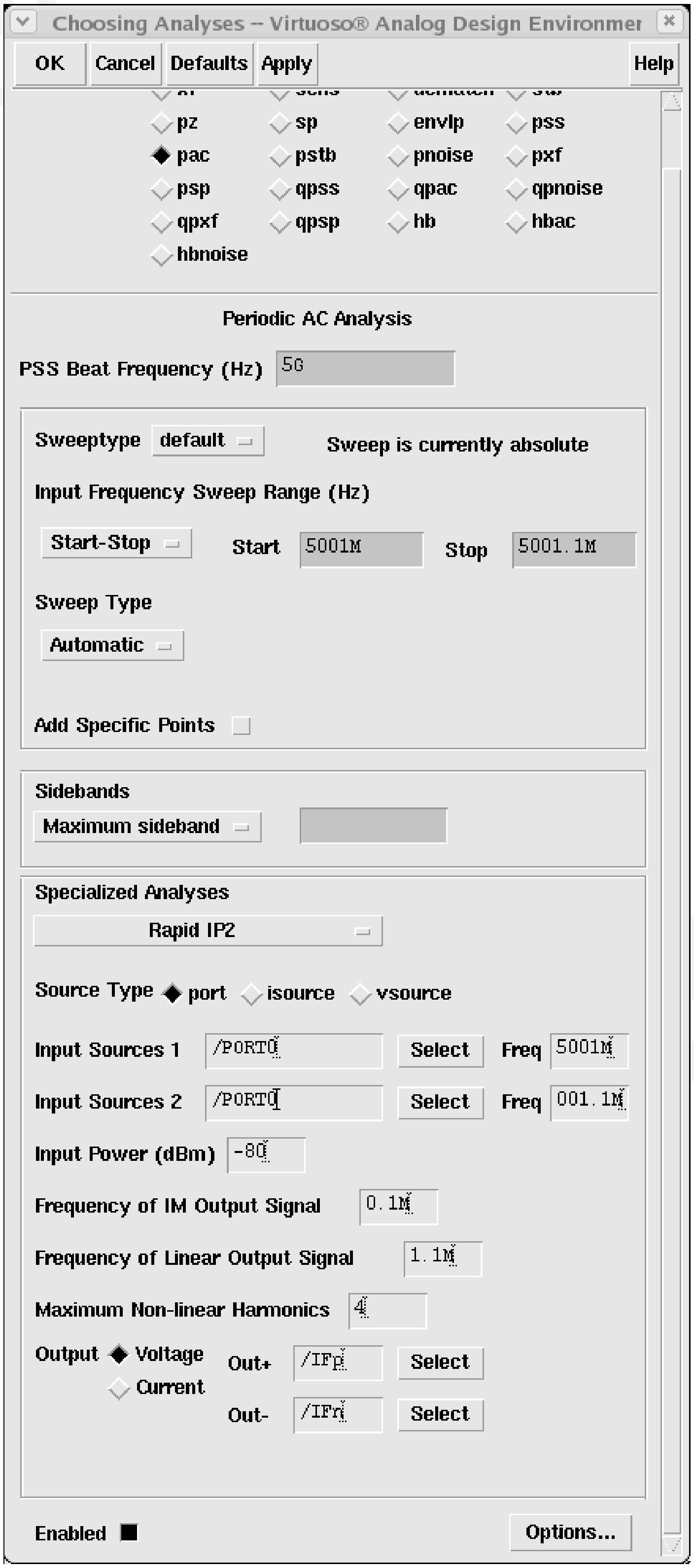
Action 13-7: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

Action 13-8: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window and set the form as shown in Action 2-6.

Action 13-9: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *Apply*.

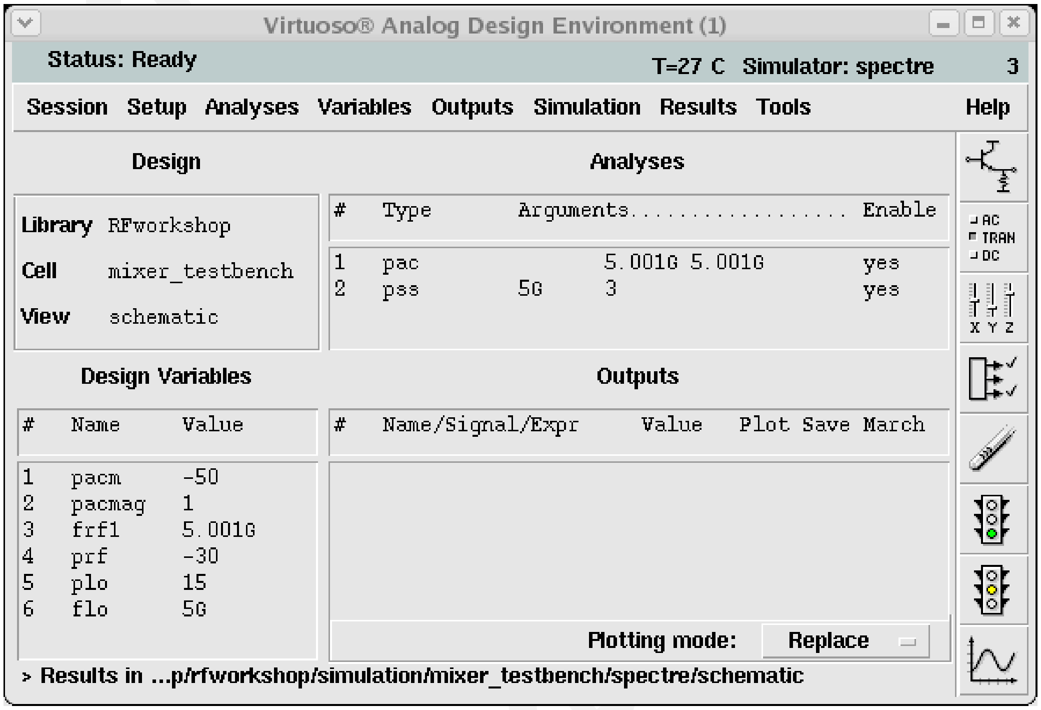
Action 13-10: In the Choosing Analyses window, select the *pac* button in the *Analysis* field of the window. In the *Specialized Analyses* field, choose *Rapid IP2*. Set *Input Sources 1* to `/PORT0` by selecting `PORT0` on the schematic.

Press the ESC key to terminate the selection process. Set the *Freq* of source 1 to 5001M and *Freq* of Source 2 to 5001.1M. Set the *Frequency of IM Output Signal* as 0.1M and the *Frequency of Linear Output Signal* as 1.1M. The form looks like this.


Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Periodic AC Analysis
PSS Beat Frequency (Hz)
5G
Sweeptype
default
Sweep is currently absolute
Input Frequency Sweep Range (Hz)
Start-Stop
Start
5001M
Stop
5001.1M
Sweep Type
Automatic
Add Specific Points
Sidebands
Maximum sideband
Specialized Analyses
Rapid IP2
Source Type
port
isource
vsource
Input Sources 1
/PORT0
Select
Freq
5001M
Input Sources 2
/PORT0
Select
Freq
001.1M
Input Power (dBm)
-80
Frequency of IM Output Signal
0.1M
Frequency of Linear Output Signal
1.1M
Maximum Non-linear Harmonics
4
Output
Voltage
Current
Out+
/IFp
Select
Out-
/IFn
Select
Enabled
Options...

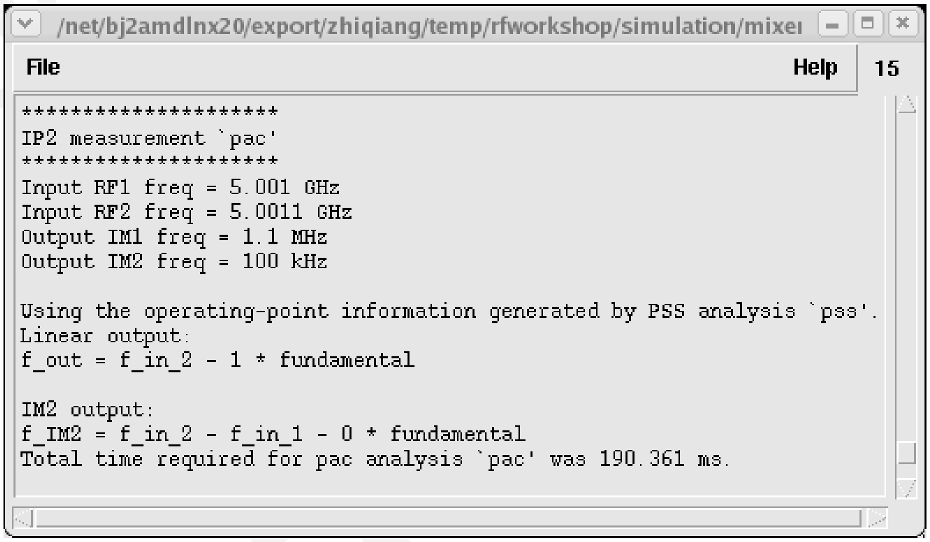
Action 13-11: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.

The Virtuoso Analog Design Environment window looks like this.



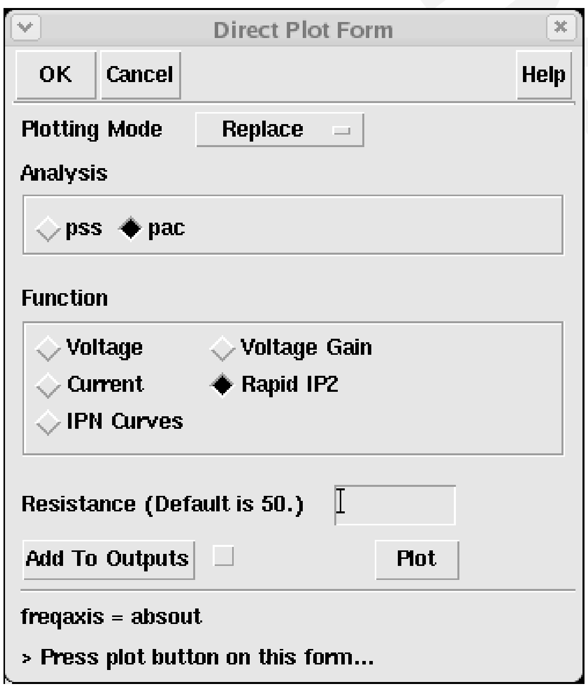
Action 13-12: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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

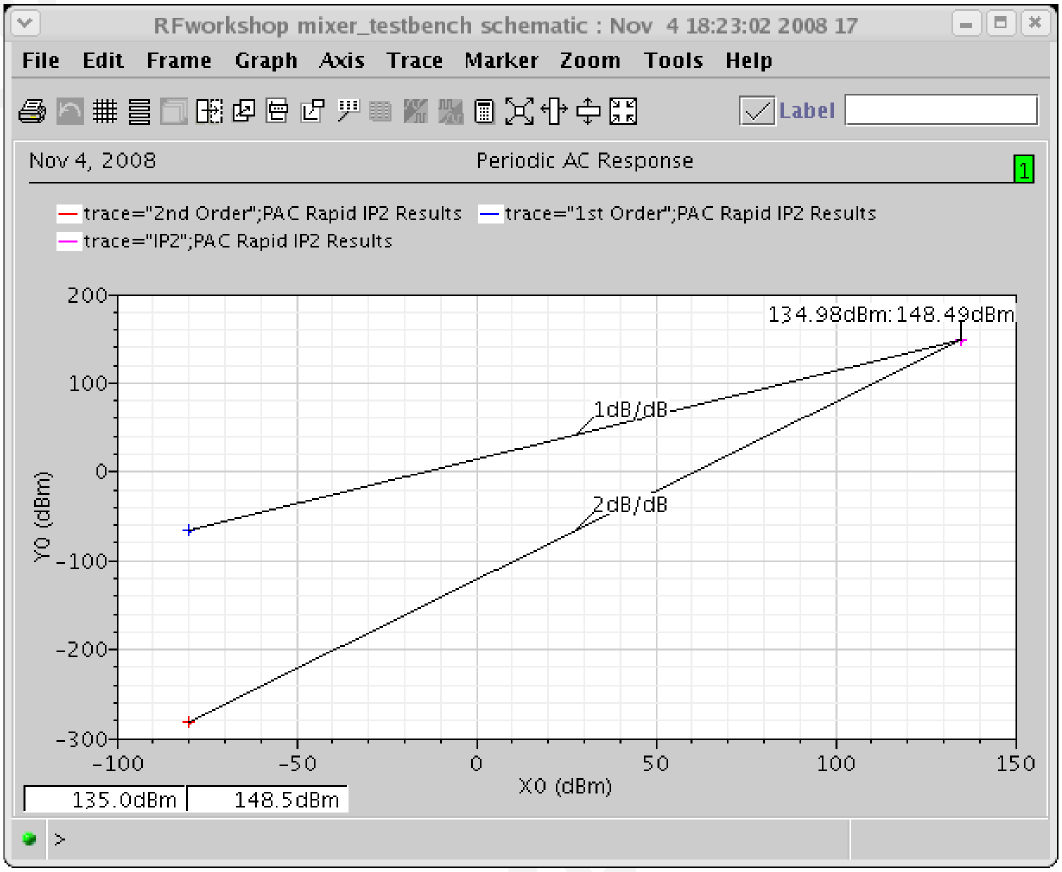


Action 13-13: In the Virtuoso Analog Design Environment window, choose **Results — Direct Plot — Main Form**.

Action 13-14: In the Direct Plot Form, select the *pac* button in analysis field and choose *Rapid IP2* in the *Function* field.



Action 13-15: Click *Plot*.



Action 13-16: Close the waveforms window, and click *Cancel* in the Direct Plot Form.

Lab 14: IM2 Distortion Summary (PAC)

Action 14-1: If it is not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFhomework.*

Action 14-2: Make sure the *Source type* of `PORT0` is *dc*.

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	500 mV
Source Type	*dc*

Action 14-3: Check and save the schematic.

Action 14-4: From the *Mixer_testbench* schematic, choose **Tools — Analog Environment**.

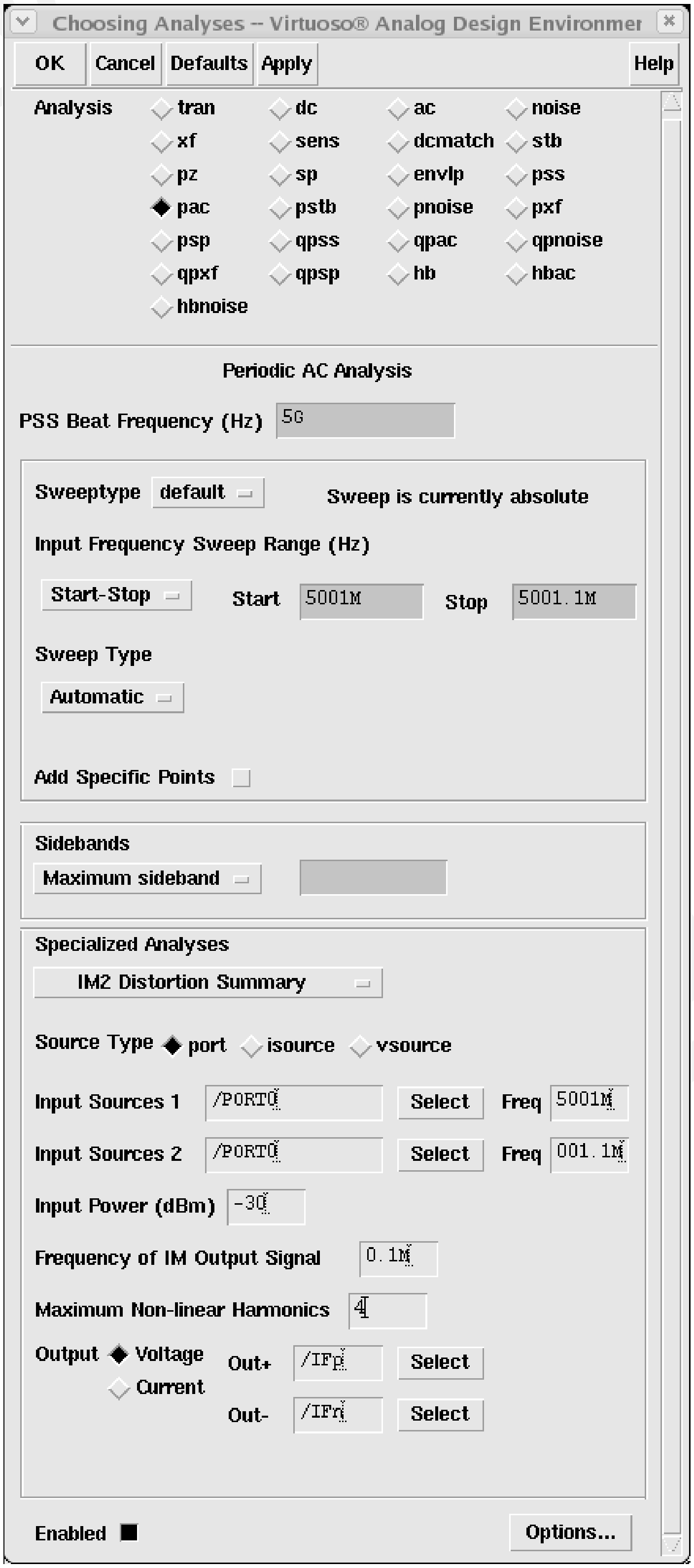
The Virtuoso Analog Design Environment window appears.

Action 14-5: You can choose **Session — Load State**, select **Cellview** in **Load State Option** and load state "**Lab14_IM2DistorSmary_PAC**" and skip to Action 14-10 or …

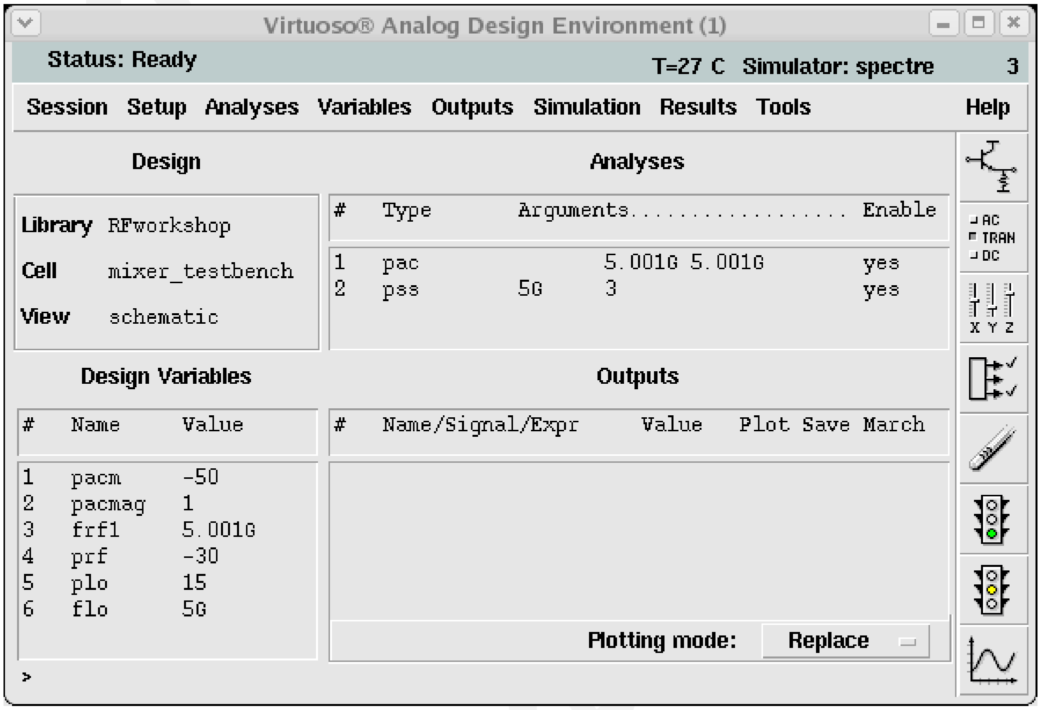
Action 14-6: In the Virtuoso Analog Design Environment window, choose **Analyses — Choose**.

Action 14-7: In the Choosing Analyses window, select the *pss* button in the *Analysis* field of the window and set the form as you did in the Rapid IP2 simulation.

Action 14-8: In the Choosing Analyses window, select the *pac* button in the *Analysis* field of the window. In the *Specialized Analyses* field, choose *IM2 Distortion Summary*. Set *Input Sources 1* to `/PORT0` by selecting `PORT0` on the schematic. Press the ESC key to terminate the selection process. Set the *Freq* of source 1 to 5001M and *Freq* of Source 2 to 5001.1M. Set the *Frequency of IM Output Signal* as 0.1M. The form looks like this.

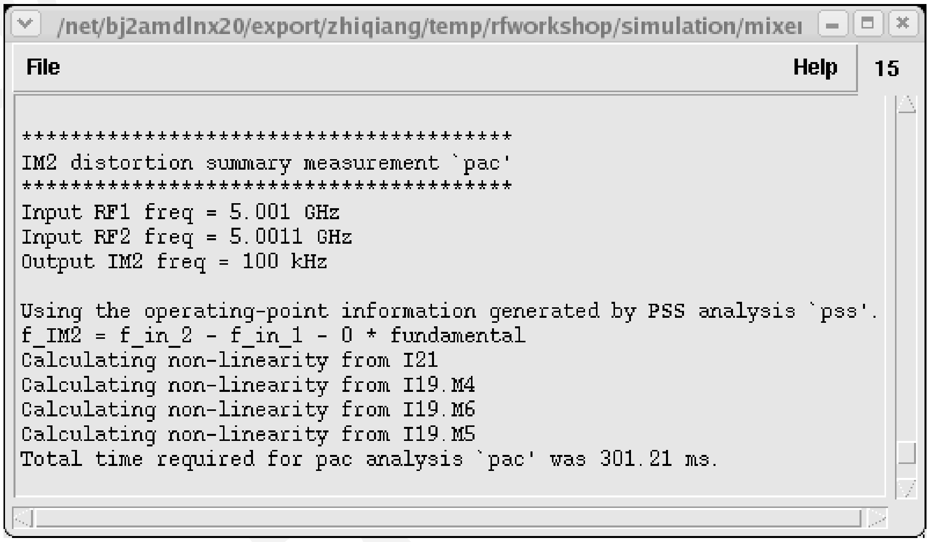

Choosing Analyses -- Virtuoso® Analog Design Environmer
OK
Cancel
Defaults
Apply
Help
Analysis
tran
dc
ac
noise
xf
sens
dcmatch
stb
pz
sp
envlp
pss
pac
pstb
pnoise
pxf
psp
qpss
qpac
qpnoise
qpxf
qpsp
hb
hbac
hbnoise
Periodic AC Analysis
PSS Beat Frequency (Hz)
5G
Sweeptype
default
Sweep is currently absolute
Input Frequency Sweep Range (Hz)
Start-Stop
Start
5001M
Stop
5001.1M
Sweep Type
Automatic
Add Specific Points
Sidebands
Maximum sideband
Specialized Analyses
IM2 Distortion Summary
Source Type
port
isource
vsource
Input Sources 1
/PORT0
Select
Freq
5001M
Input Sources 2
/PORT0
Select
Freq
001.1M
Input Power (dBm)
-30
Frequency of IM Output Signal
0.1M
Maximum Non-linear Harmonics
4
Output
Voltage
Current
Out+
/IFp
Select
Out-
/IFn
Select
Enabled
Options...

Action 14-9: Make sure the *Enabled* button is on. In the Choosing Analyses window, click *OK*.
The Virtuoso Analog Design Environment window looks like this.



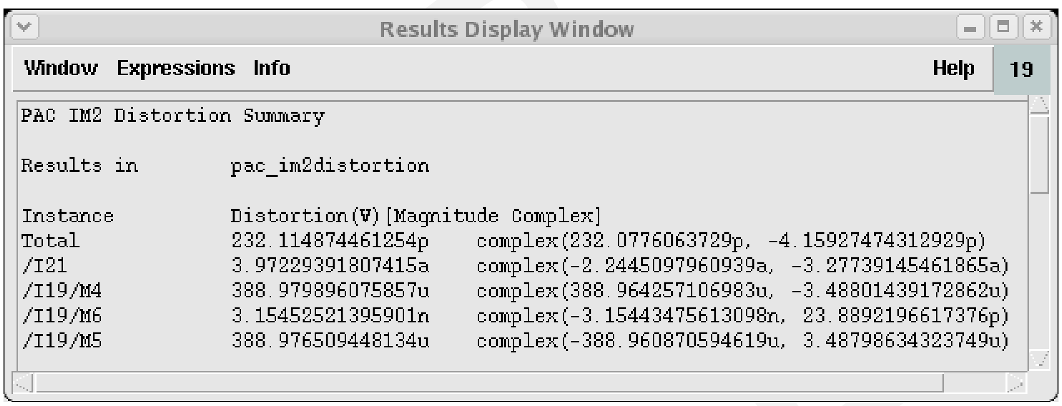
Action 14-10: In the Virtuoso Analog Design Environment window, choose **Simulation — Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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



Action 14-11: In the Virtuoso Analog Design Environment window, choose **Results — Print — PAC Distortion Summary.**

The Results Display Window shows the PAC IM2 Distortion Summary.



The distortion is listed in dB for each instance. Due to the very low RF input power, the distortion is very small.

Action 14-12: After viewing the distortion summary report, close it by choosing **Window — Close**.

Conclusion

This tutorial illustrates how to use Spectre to simulate a mixer and to extract design parameters such as IP3, 1dB compression point, or port-to-port isolation. Various techniques using PSS, Pnoise, PAC, and QPSS analyses are demonstrated. Spectre Harmonic Balance and Time domain algorithms are demonstrated and their accuracies are compared.

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.