SpectreRF Workshop

Mixer Design Using SpectreRF

MMSIM6.0USR2

November 2005

Contents

Mixer Design Measurements 3
Purpose 3
Audience 3
Overview 3
Introduction to Mixers 3
The Design Example: A Differential Input Mixer 4
Testbench 5
Example Measurements Using SpectreRF 6
Lab1: Voltage Conversion Gain Versus LO Signal Power (Swept PSS with PAC) 6
Lab2: Voltage Conversion Gain Versus RF Frequency (PSS and Swept PAC) 16
Lab3: Voltage Conversion Gain Versus RF Frequency (PSS and Swept PXF) 21
Lab4: Power Conversion Gain Versus RF Frequency (QPSS) 26
Lab5: Periodic S-Parameters (PSS and PSP) 31
Lab6: Noise, Noise Figures and Noise Summary (PSS and Pnoise) 38
Lab7: Port-to-Port Isolation Among RF, IF and LO Ports (PSS and Swept PAC) 44
Lab8: Mixer Performance with a Blocking Signal (QPSS, QPAC, and QPNoise) 55
Lab9: IP3 Calculation (Swept QPSS and QPAC) 66
Lab10: IP3 Calculation (QPSS with Shooting Engine or Flexible Balance Engine) 74
Lab11: Rapid IP3 (PAC) 84
Lab12: Compression Distortion Summary (PAC) 88
Lab13: Rapid IP2 (PAC) 92
Lab14: IM2 Distortion Summary (PAC) 97
Conclusion 100
Reference 101

Mixer Design Measurements

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

Purpose

This workshop describes how to use SpectreRF in the Analog Design Environment to measure parameters which are important in design verification of mixers. New features of MMSIM6.0USR2 are included.

Audience

Users of SpectreRF in the Analog Design Environment.

Overview

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

Introduction to Mixers

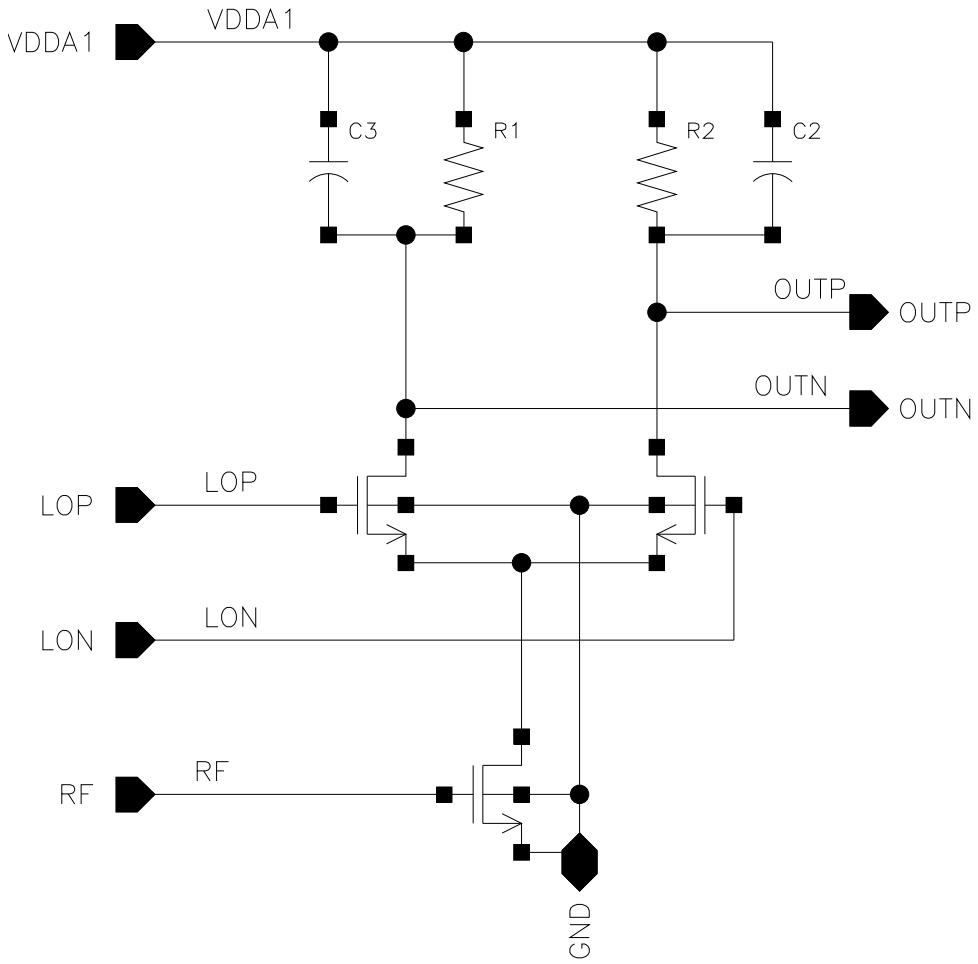
Mixers are key components in both receivers and transmitters. Mixers translate signals from one frequency band to another. The output of the mixer consists of multiple images of the mixers input signal where each image is shifted up or down by multiples of the local oscillator (LO) frequency. The most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

In an ideal situation, the mixer output would be an exact replica of the input signal. In reality mixer output is distorted due to non-linearity in the mixer. In addition, the mixer components and a non-ideal LO signal introduce more noise to the output. Bad design might also cause leakage effects, complicating the design of the complete system.

Noise performance and rejection of out-of-band interferers are both critical to the receiver system because they both limit the receiver system's sensitivity. Linearity is important to transmitter performance, where you want an error-free output signal.

The Design Example: A Differential Input Mixer

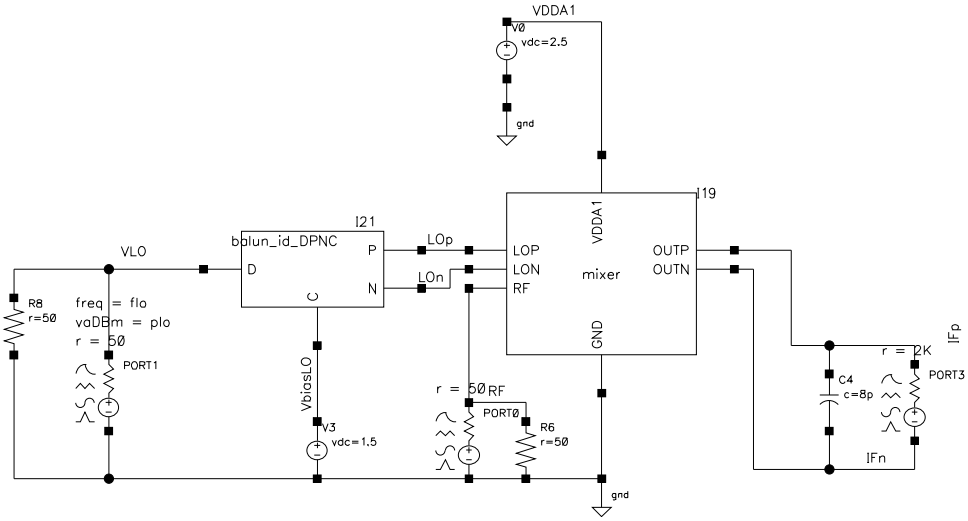
The mixer measurements described in this workshop are calculated using SpectreRF in the Analog Design Environment. The design investigated is the differential input mixer shown in below:



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 baseband output noise from 1 kHz to 10 MHz. The RF signal frequency used for the simulation is around 5001 MHz.

Testbench

Use the mixer measurements testbench shown in below to measure typical mixer characteristics. Use a PORT component and match impedance for each of the inputs and the output.



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

- To represent the RF input to the mixer, use a port (PORT0) which is matched with the mixer input.

- To use the differential output for measurements, match 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 the mixer supply of 2.5 V.

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

- Set the RF port to either dc or 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 SpectreRF

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

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

Change directory to…

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

Action: Invoke tool **icfb&**

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

Lab1: 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:

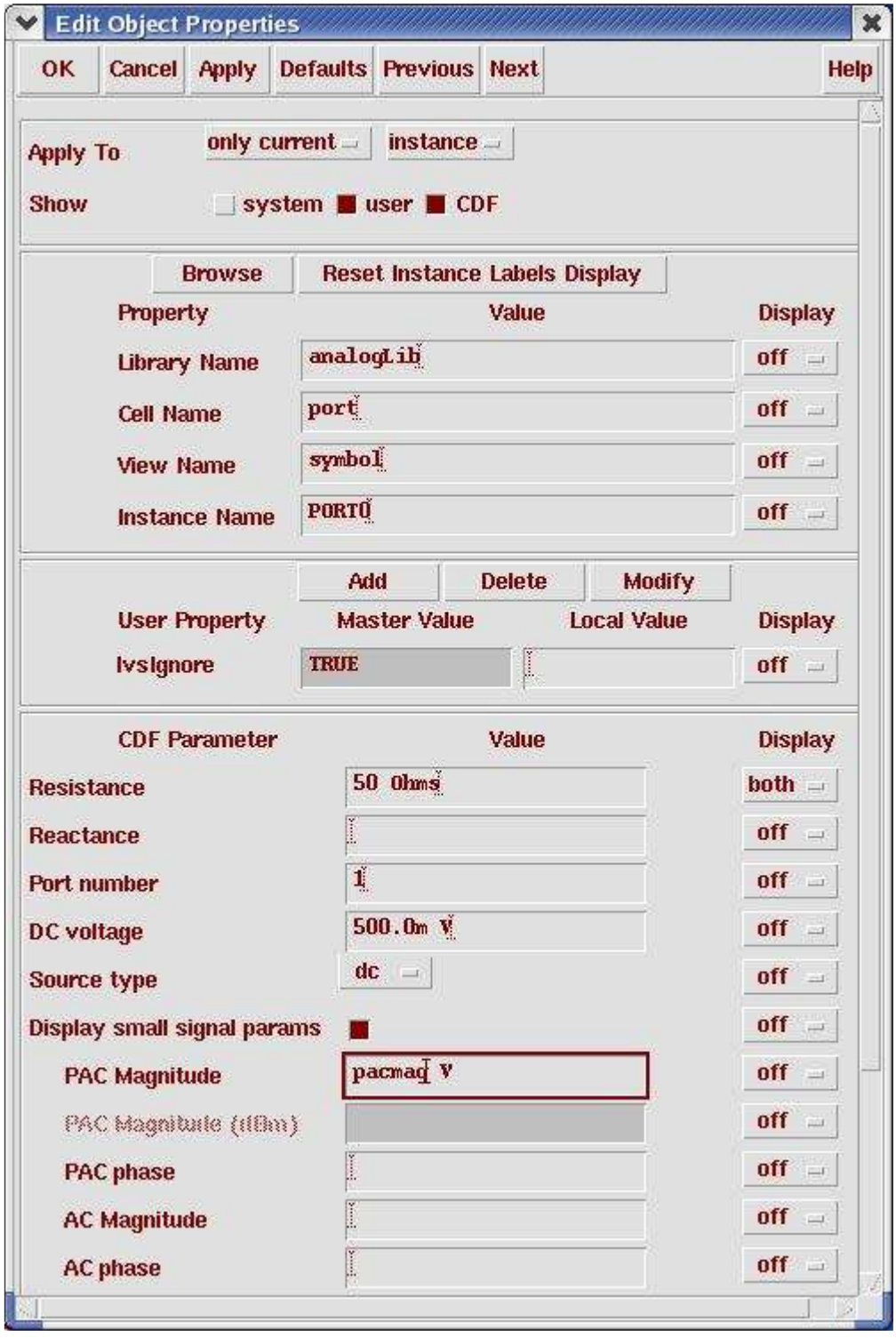
- Using a small-signal analysis, like PSS with PAC or PXF. The PSS with PAC or PXF analyses supply the small-signal gain information. You can use either PAC or PXF analysis to compute the voltage gain.

- 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.

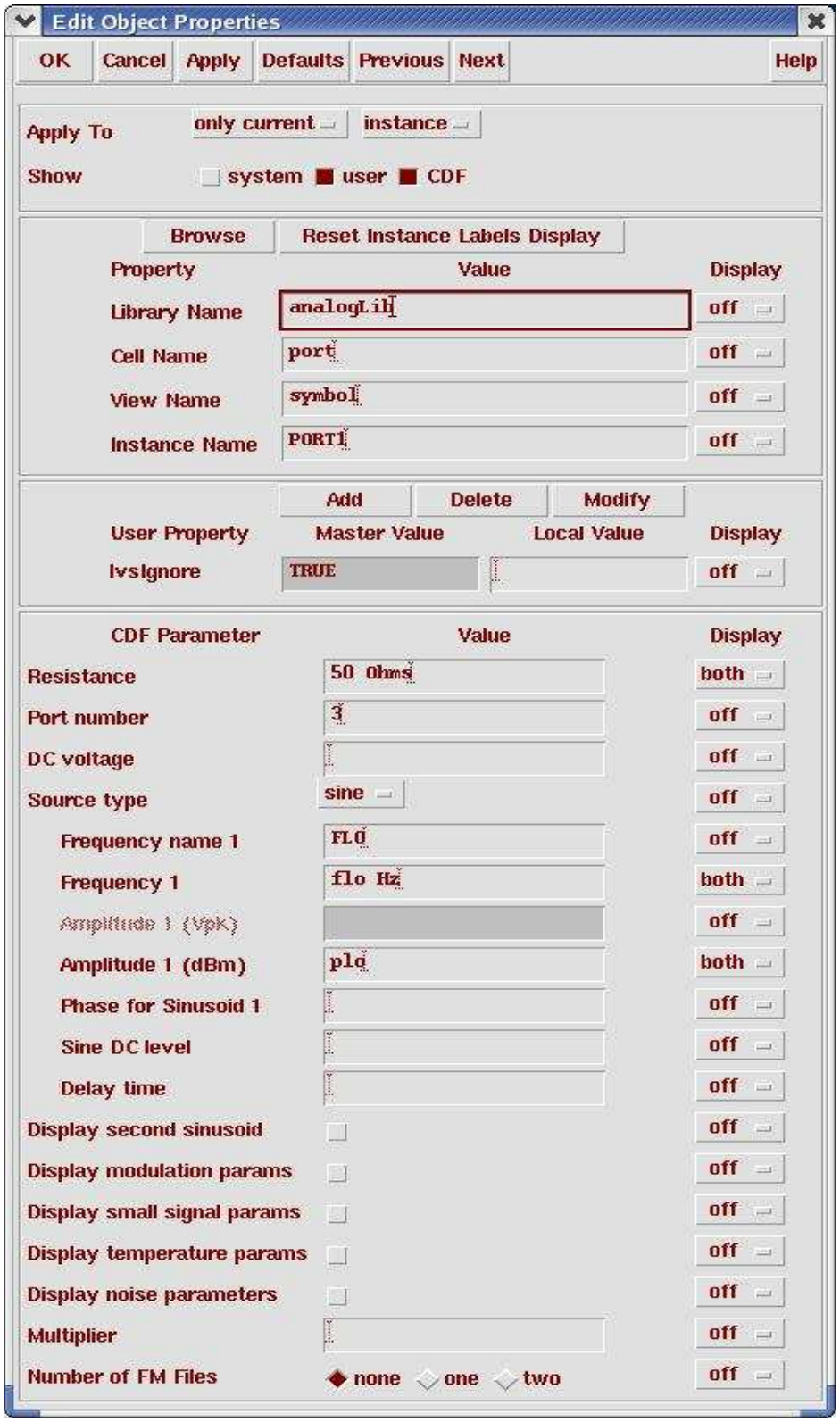
This example measures the variation of conversion gain with the power of the LO signal.

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

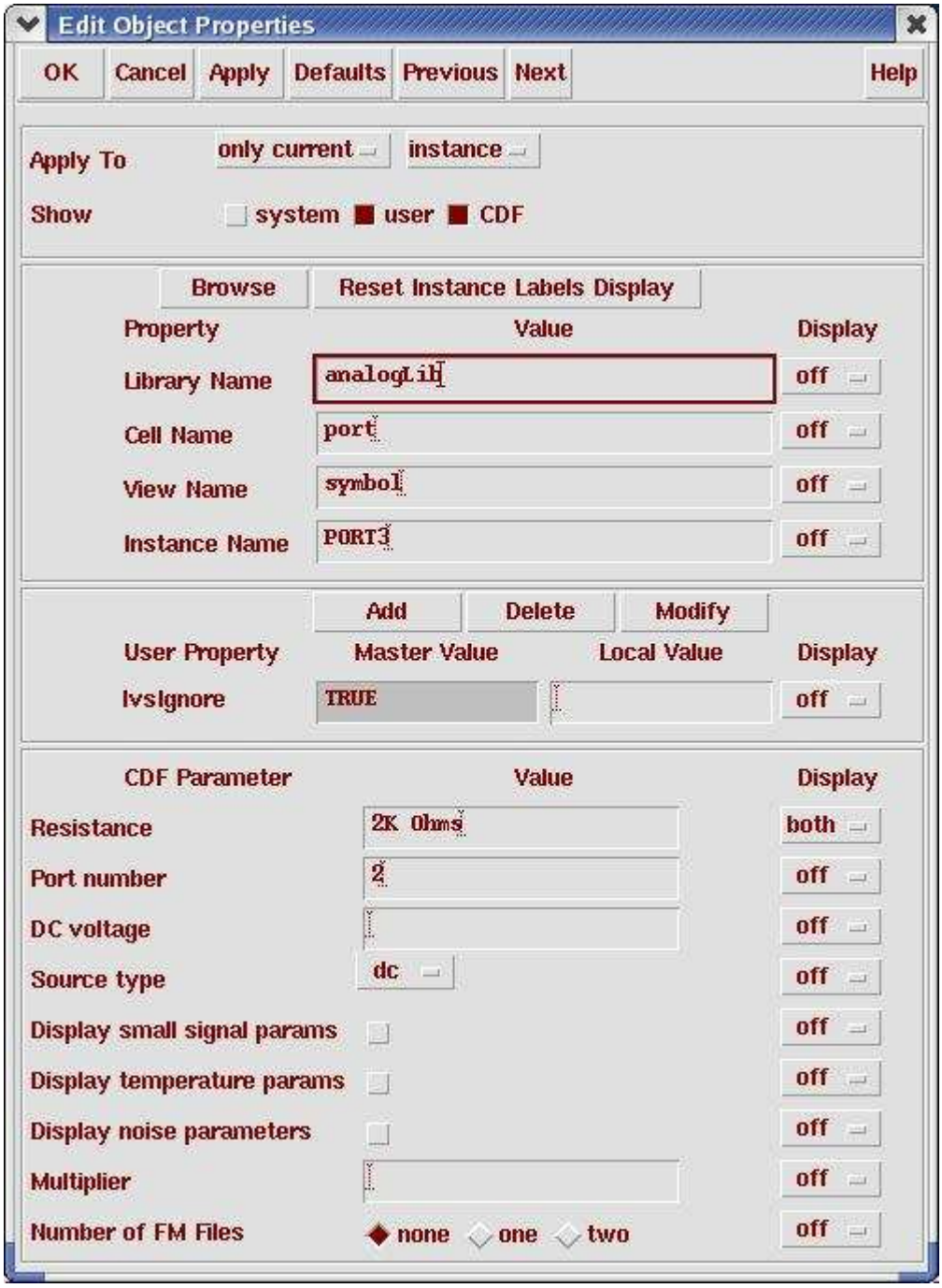
Action 1-2: Select the **PORT0** source by placing the mouse cursor over it and clicking the left mouse button. Then in the Virtuoso Schematic Editor select **Edit—Properties—Objects…** The Edit Object Properties window for the port cell should come up…


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

Action1-3: Click **ok** on the Edit Object Properties window to close it, Select PORT1 and show properties:



Action1-4: Make sure PORT3 is set to DC type.



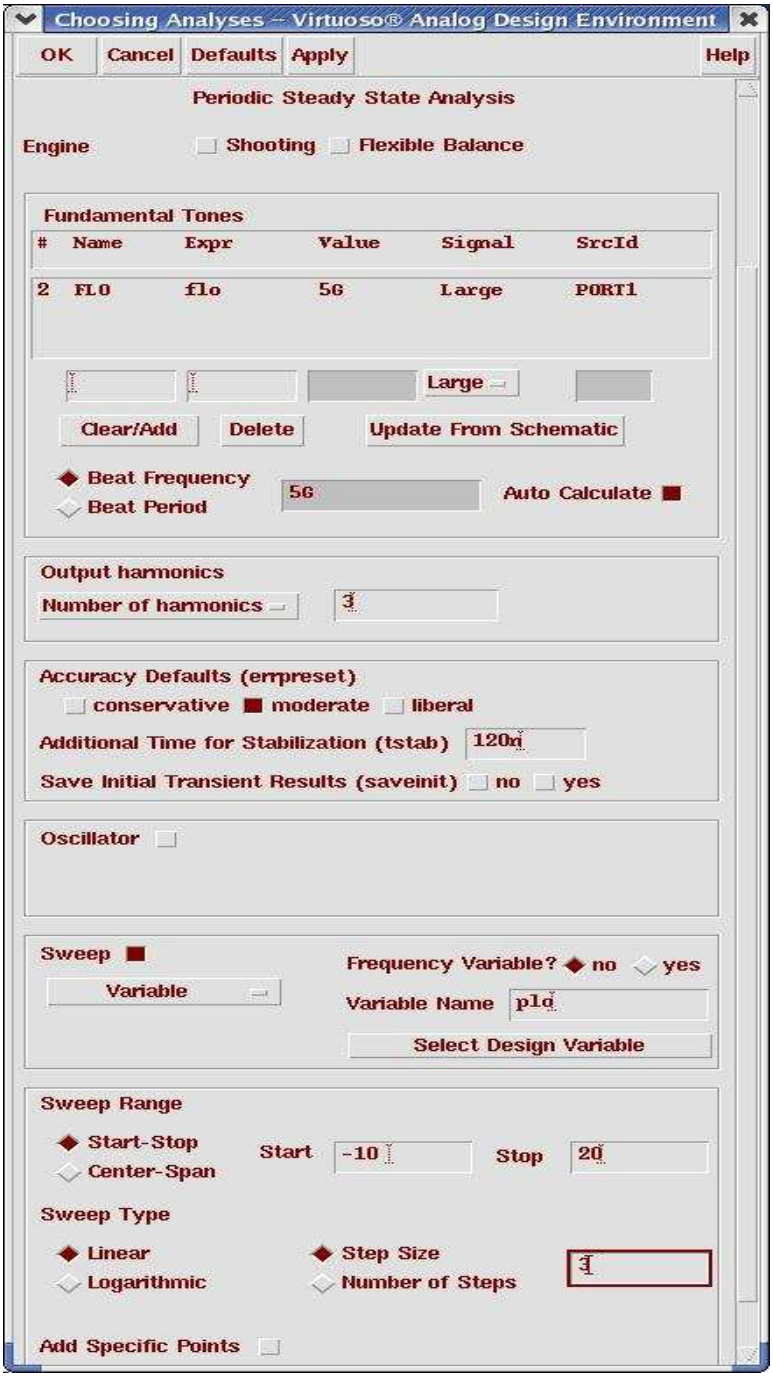
Action1-5: Check and save the schematic.

Action1-6: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog Environment** command.

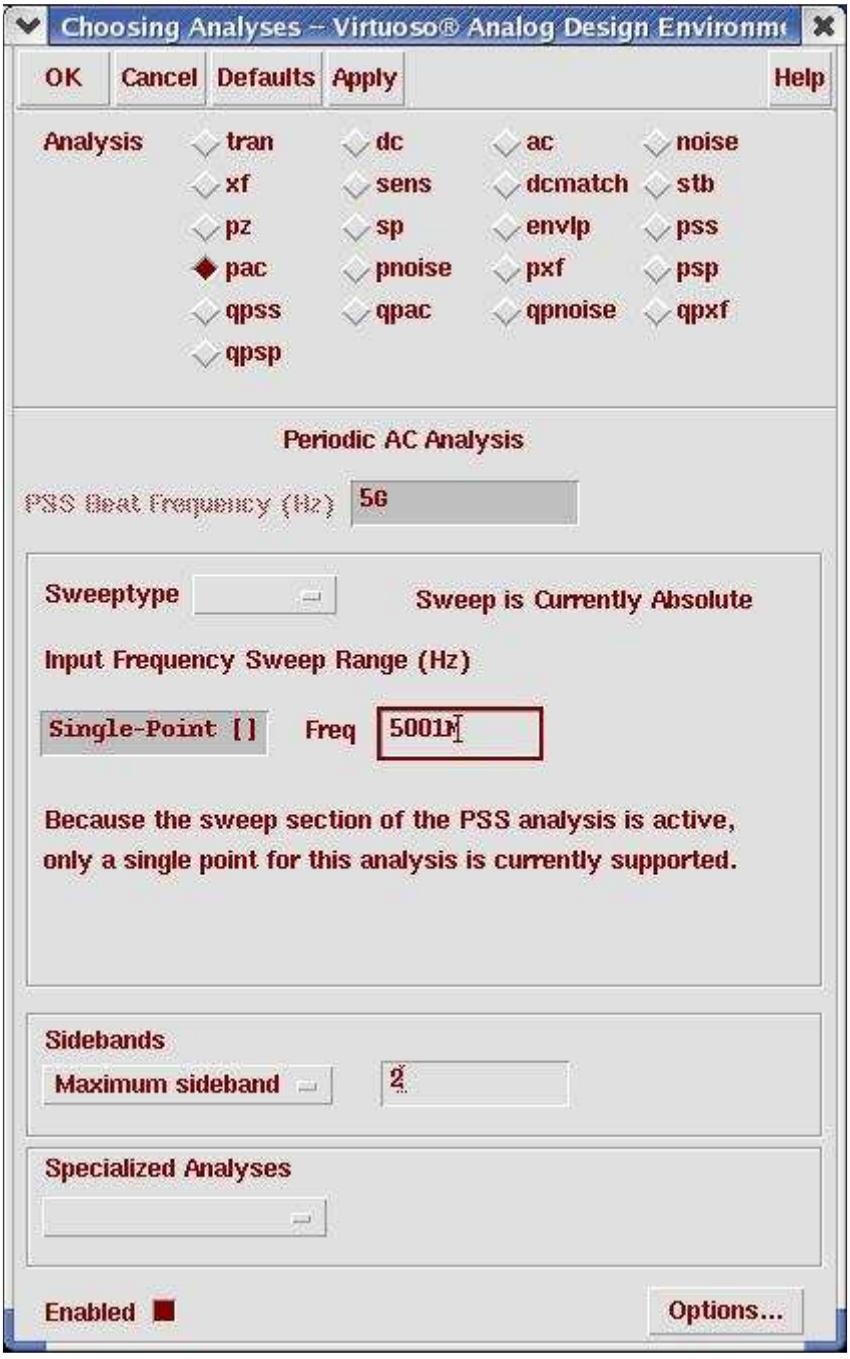
Action1-7: You can choose **Session—Load State**, load state "**Lab1_VCGvsLO_PSSPAC**" and skip to Action1-12 or …

Action1-8: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

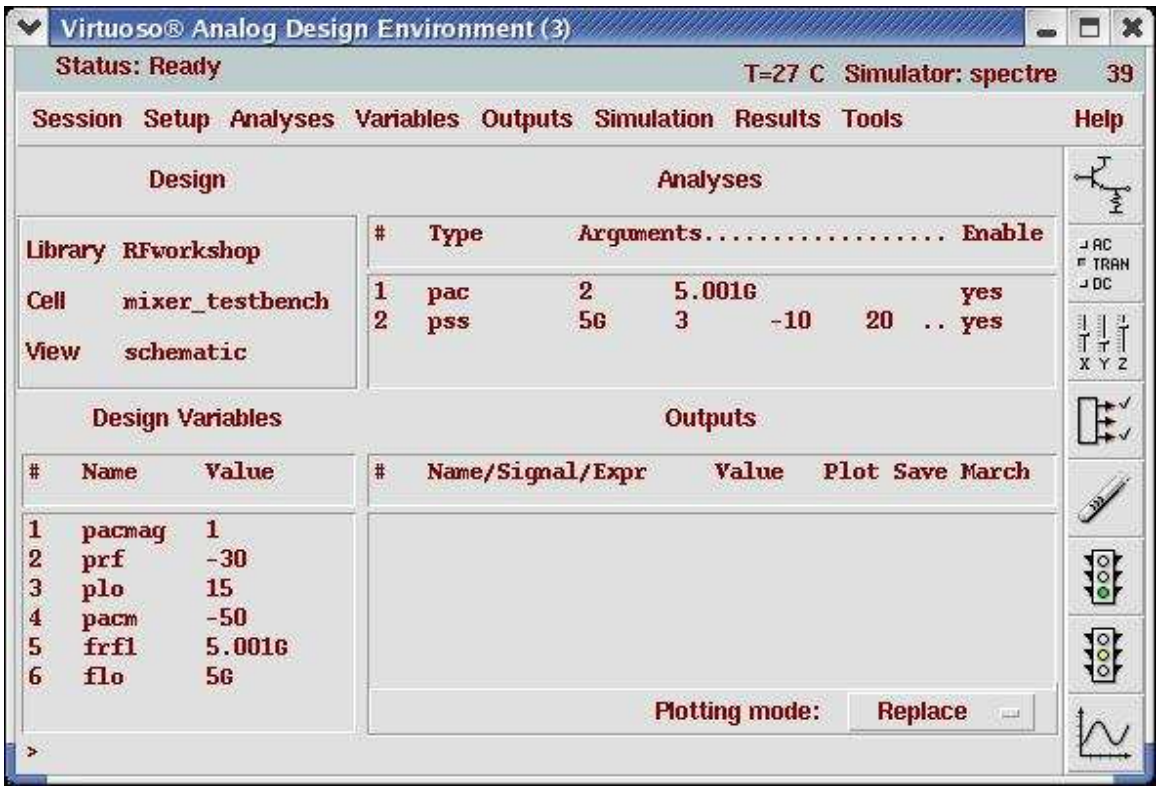
Action1-9: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window. Set the fundamental frequency parameter, flo = 5 GHz. Set errpreset = moderate. Click Sweep and enter plo as the Variable Name parameter to sweep LO power. Click Sweep Range and set Start = -10 dBm and Stop = 20 dBm. This sweeps LO power from a small value to a value above the expected gain saturation. The form should look this:



Action1-10: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window. Set fixed input frequency point to the RF signal frequency, 5001 MHz. Select sidebands either by specifying Maximum sidebands = 2 or using Select from range. Set the maximum sideband to 2 as you are only interested in the first harmonics of LO. The form should look like:



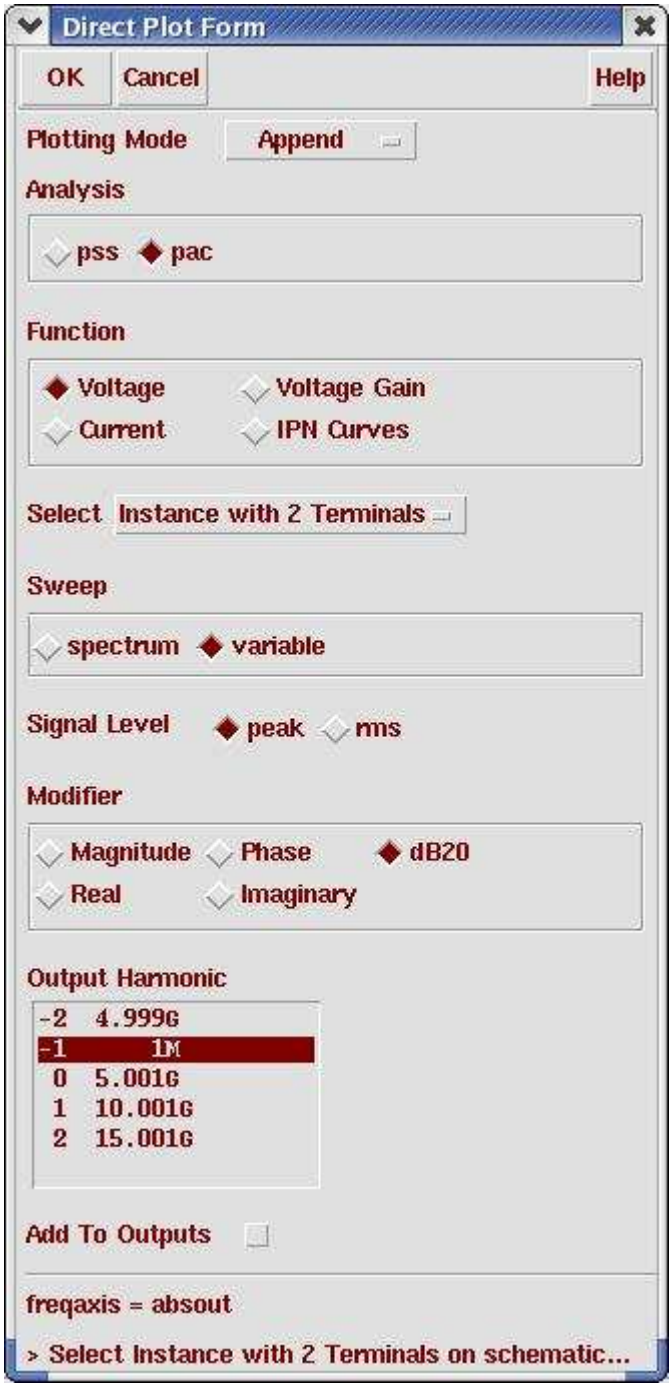
Action1-11: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button. Now your Virtuoso Analog Design Environment will look like:



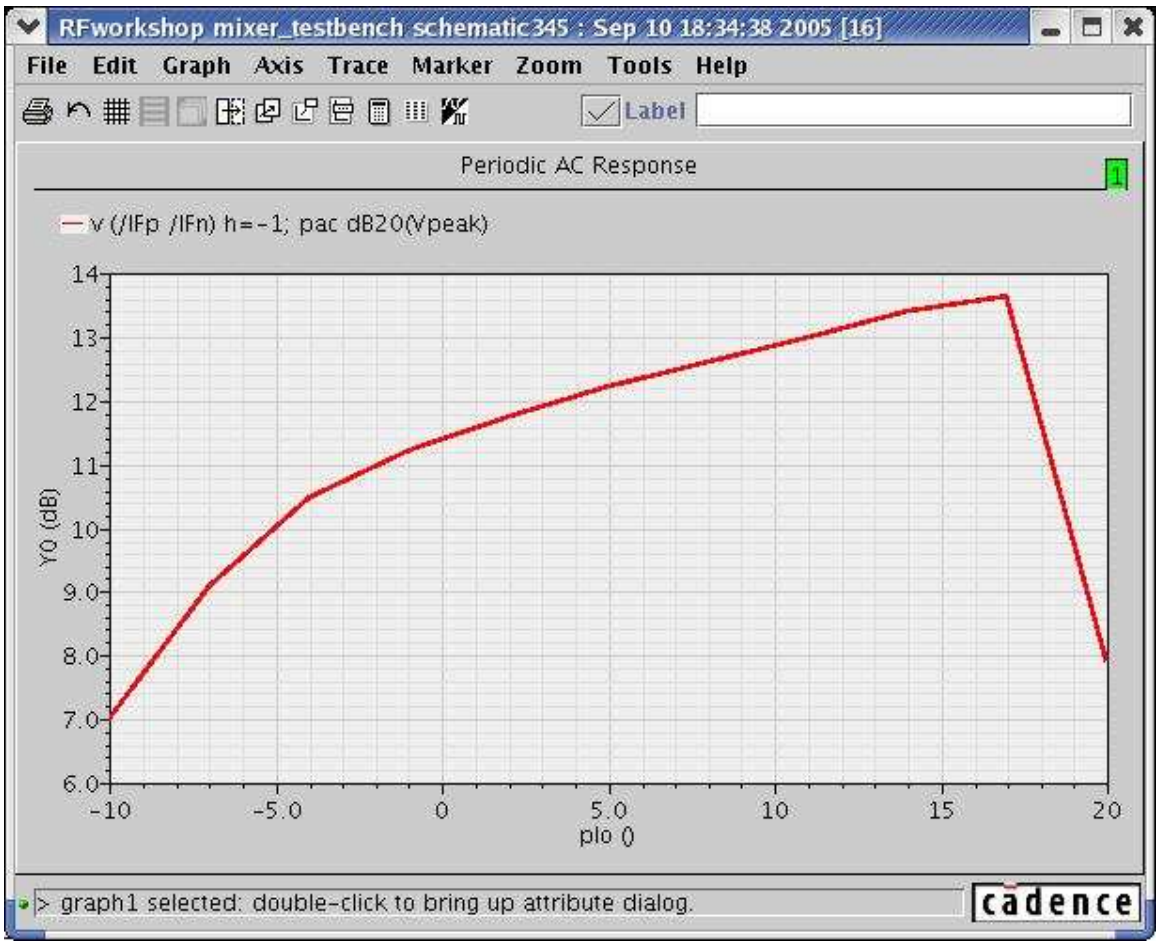
Action1-12: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action1-13: After the simulation completes, in the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action1-14: In the Direct Plot Form, select the **pac** button, and configure the form as follows:



Action1-15: Select **port3** on schematic. You should 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.

Action1-16: After viewing the waveforms, click **Cancel** in the Direct Plot form. Close the waveform window.

Lab2: Voltage Conversion Gain Versus RF Frequency (PSS and Swept PAC)

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

Action2-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

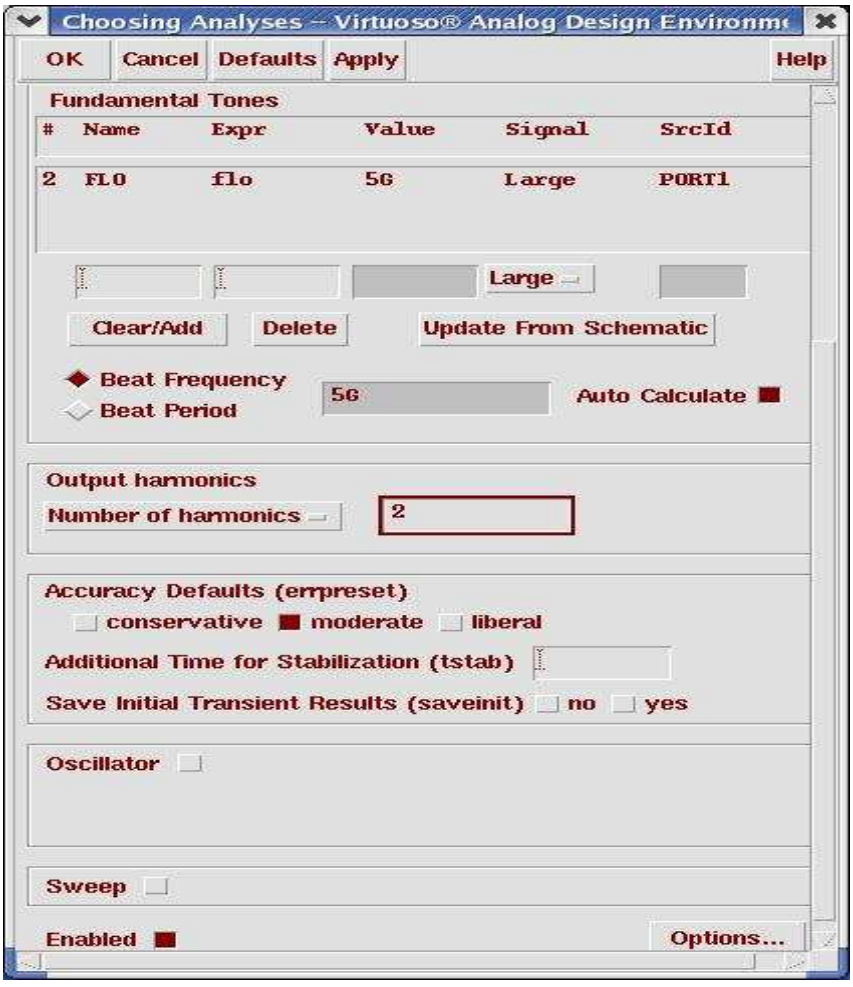
Action2-2: Set port as you did in Lab1.

Action2-3: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

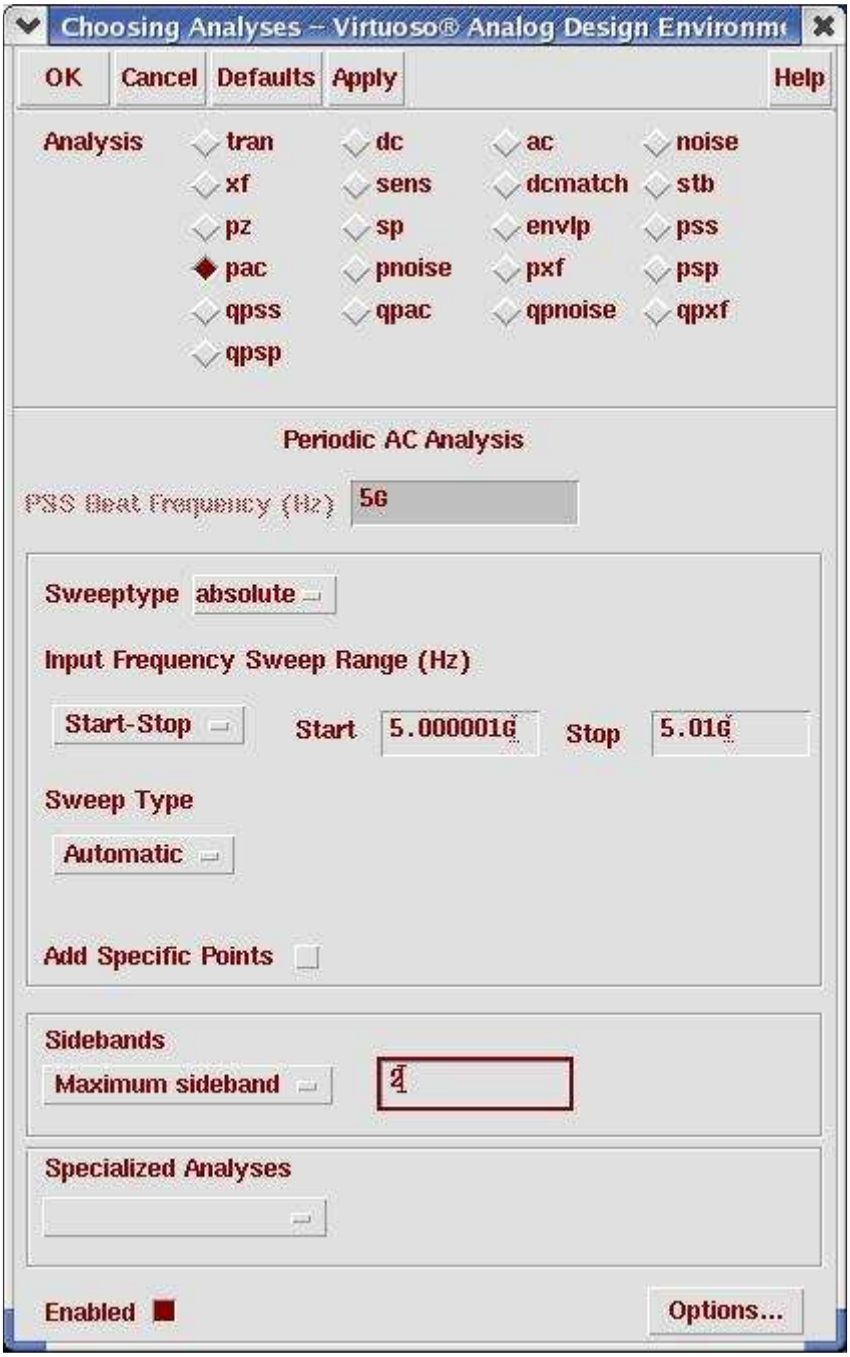
Action2-4: You can choose **Session—Load State**, load state "**Lab2_VCGvsRF_PSSPAC**" and skip to Action2-9 or …

Action2-5: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action2-6: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set the form as follows:

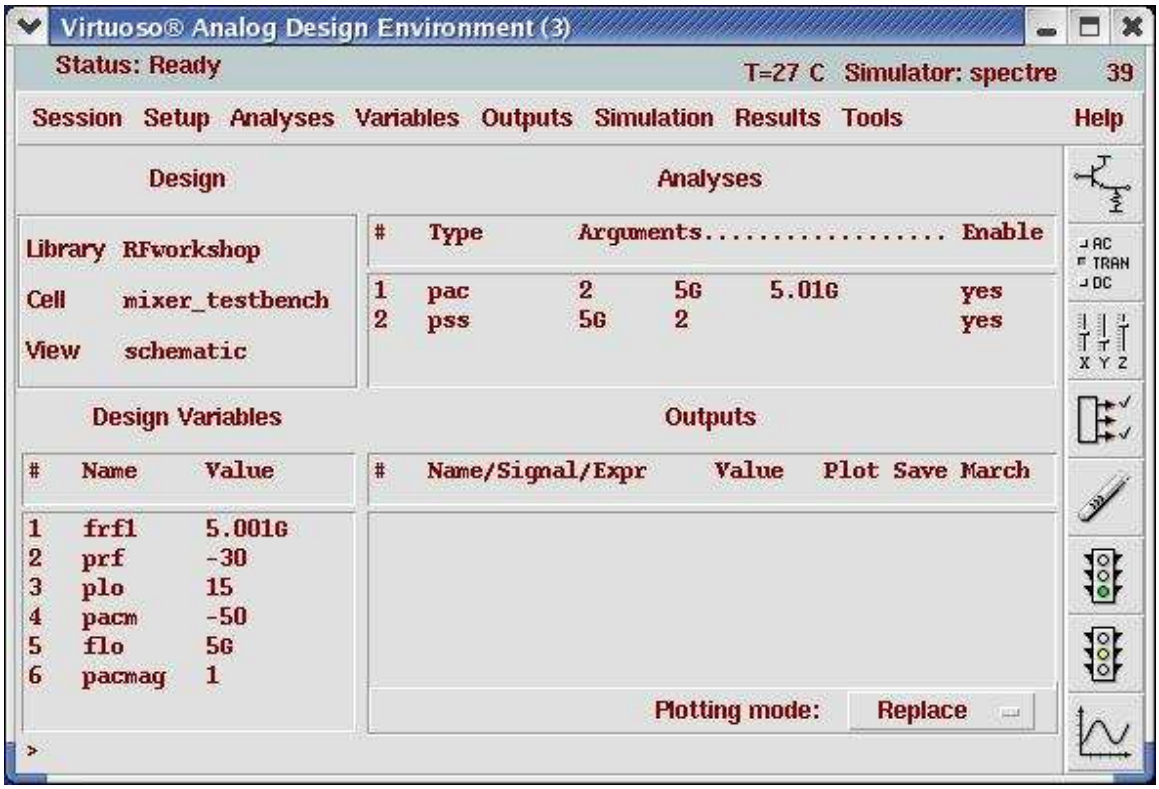


Action2-7: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window. Set the RF input frequency to sweep from 5 G + 1 kHz to 5 G + 10 MHz. Select sidebands either by specifying Maximum sidebands = 2 or using **Select from range**. The form looks like:



Action2-8: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

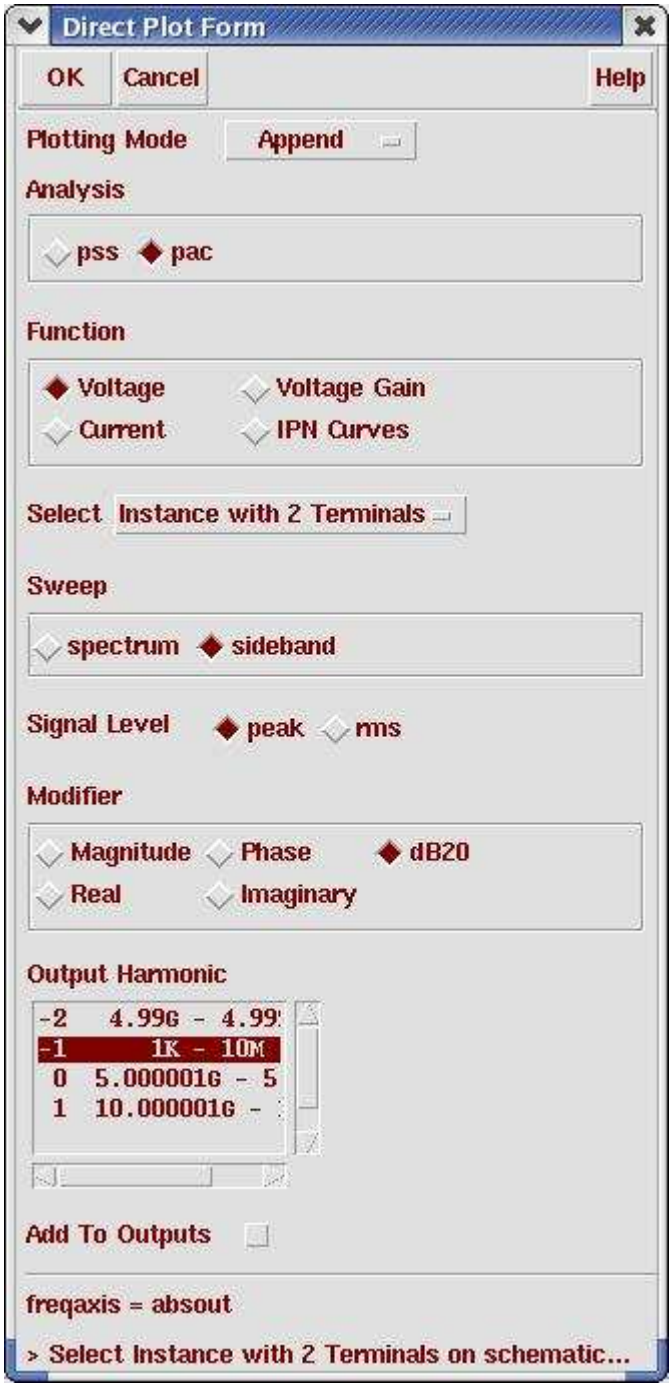
Now your Virtuoso Analog Design Environment will look like:



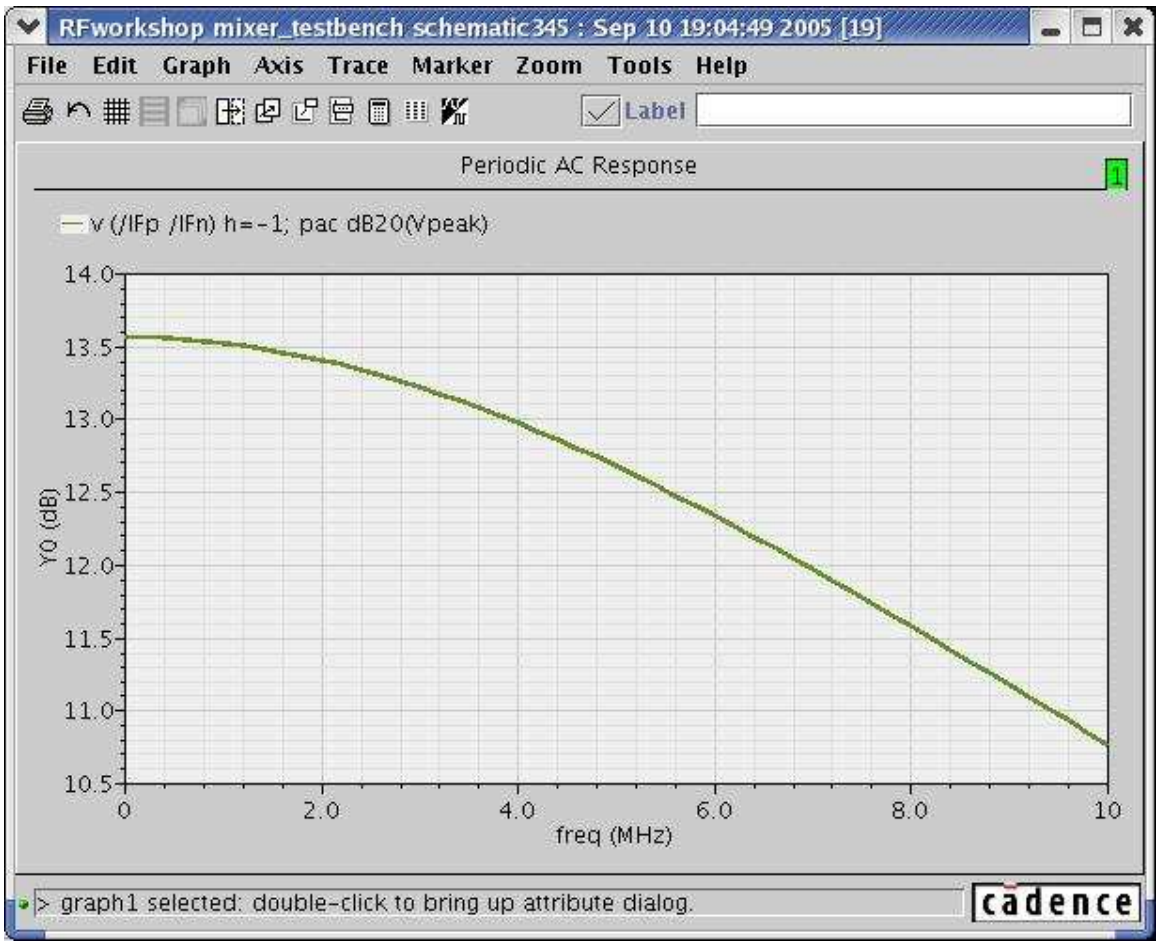
Action2-9: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action2-10: After the simulation completes, in the Virtuoso Analog Design Environment, choose **Results—Direct Plot—Main Form**.

Action2-11: In the Direct Plot Form, select the **pac** button, and configure the form as follows:



Action2-12: Select **port3** on schematic. The waveform window shows up:



Since the sweep type in the analysis was 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 as it is by default.

The same PAC analysis generates results you can use to measure RF to LO isolation and will be used in measurements that follow.

Action2-13: Close the waveform window and click **Cancel** in the Direct Plot form.

Lab3: Voltage Conversion Gain Versus RF Frequency (PSS and Swept PXF)

This example measures the small-signal voltage conversion gain using the PXF analysis.

Action3-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action3-2: Set ports as you did in lab1.

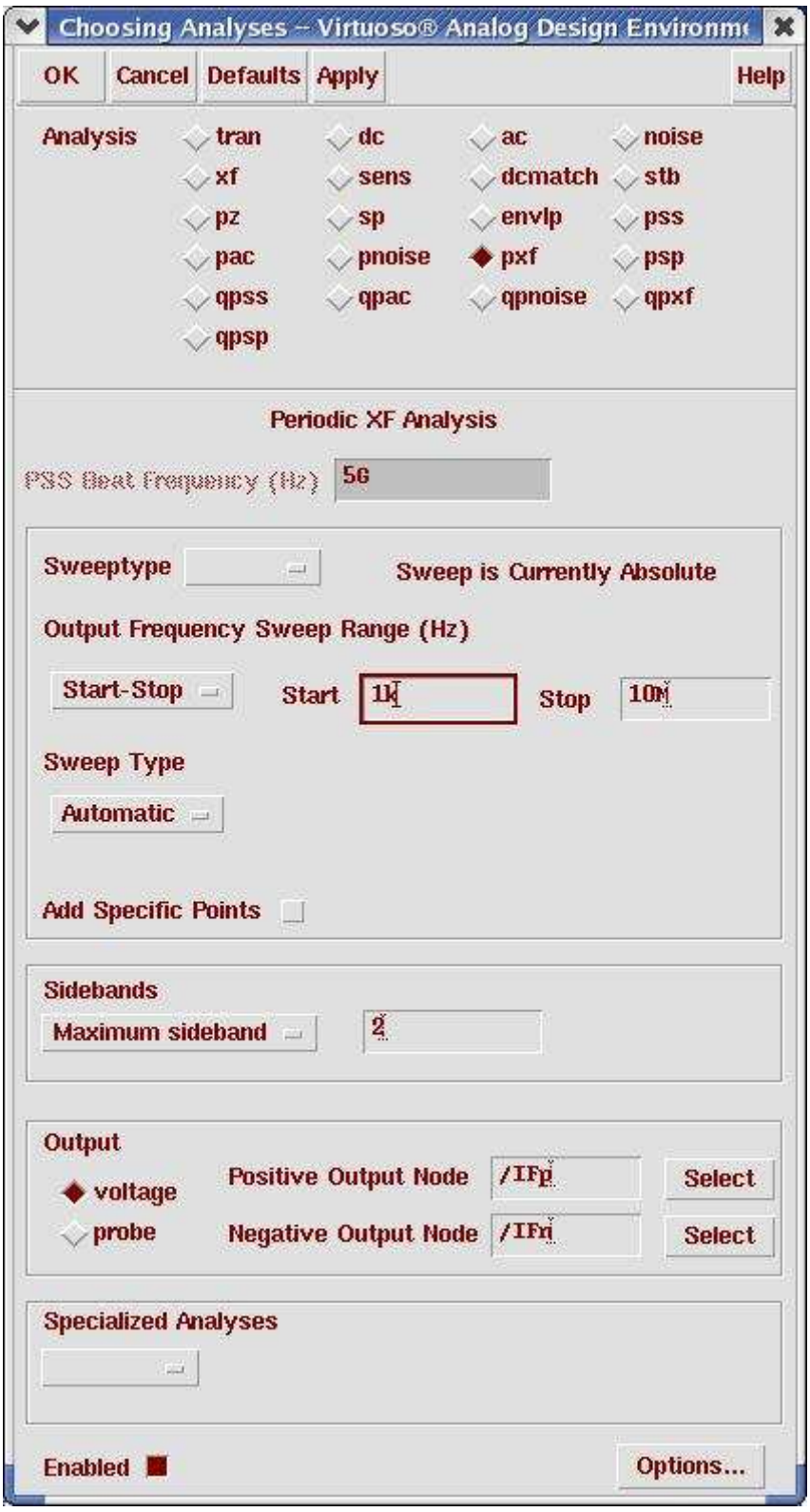
Action3-3: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog Environment** command.

Action3-4: You can choose **Session—Load State**, load state "**Lab3_VCGvsRF_PSSPXF**" and skip to Action3-9 or …

Action3-5: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action3-6: 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 **Auto Calculate** button. Set errpreset = moderate. The form will the same as in Action2-6.

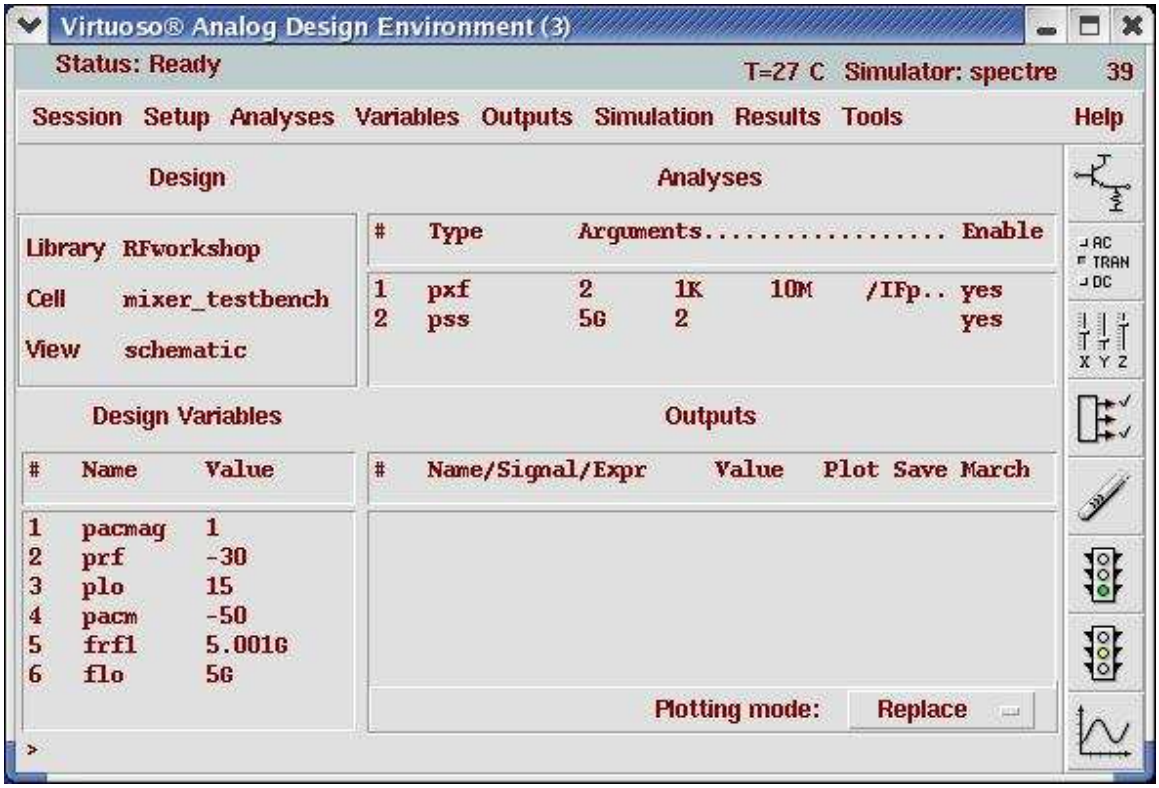
Action3-7: In the Choosing Analyses window, select the **pxf** button in the **Analysis** field of the window. Sweep the output frequency from 1 kHz to 10 MHz. Select sidebands either by specifying Maximum sidebands = 2 or using Select from range. The form looks like:



Note that Maximum sidebands = 2 is good for this example, 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.

Action3-8: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

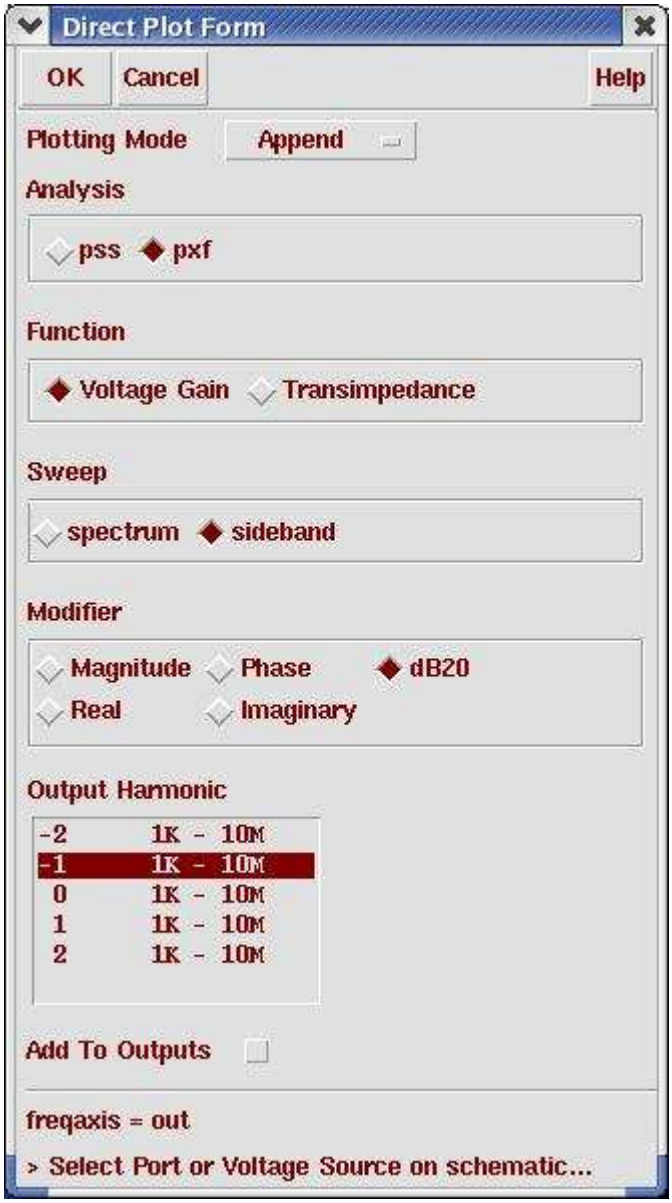
Now your Virtuoso Analog Design Environment will look like:



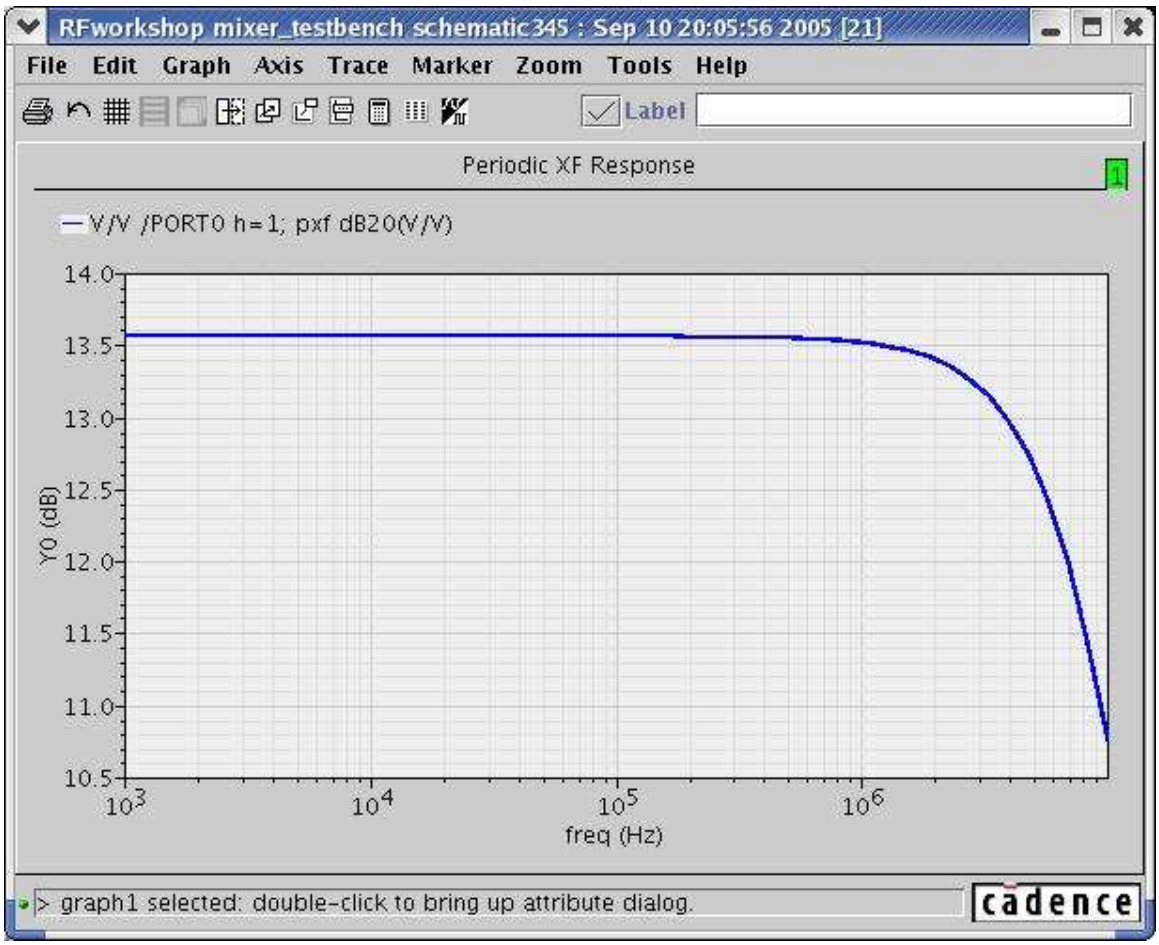
Action3-9: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action3-10: After the simulation completes, in the Virtuoso Analog Design Environment, choose **Results—Direct Plot—Main Form**.

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



Action3-12: Select output **port0** on schematic. The waveform shows up:



Action3-13: After viewing the waveforms, click **Cancel** in the Direct Plot form

The sweep type in the analysis is logarithmic by default. The large frequency range in above figure requires a logarithmic X-axis.

Yet 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. You may want to refer to SpectreRF user guide Appendix L (using psp and pnoise analysis) for more details.

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.

Lab4: 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.

Note: Some users might not be fully aware of the notion of the port and they might get wrong results by relying on a 50 Ohm resistance for the port. To get the correct results for unmatched ports, you need to save the currents on the power supply terminals.

Action4-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action4-2: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action4-3: Select the **PORT0** source by placing the mouse cursor over it and clicking the left mouse button. Then in the Virtuoso Schematic Editor select **Edit—Properties—Objects…** The Edit Object Properties window for the port cell should come up. 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

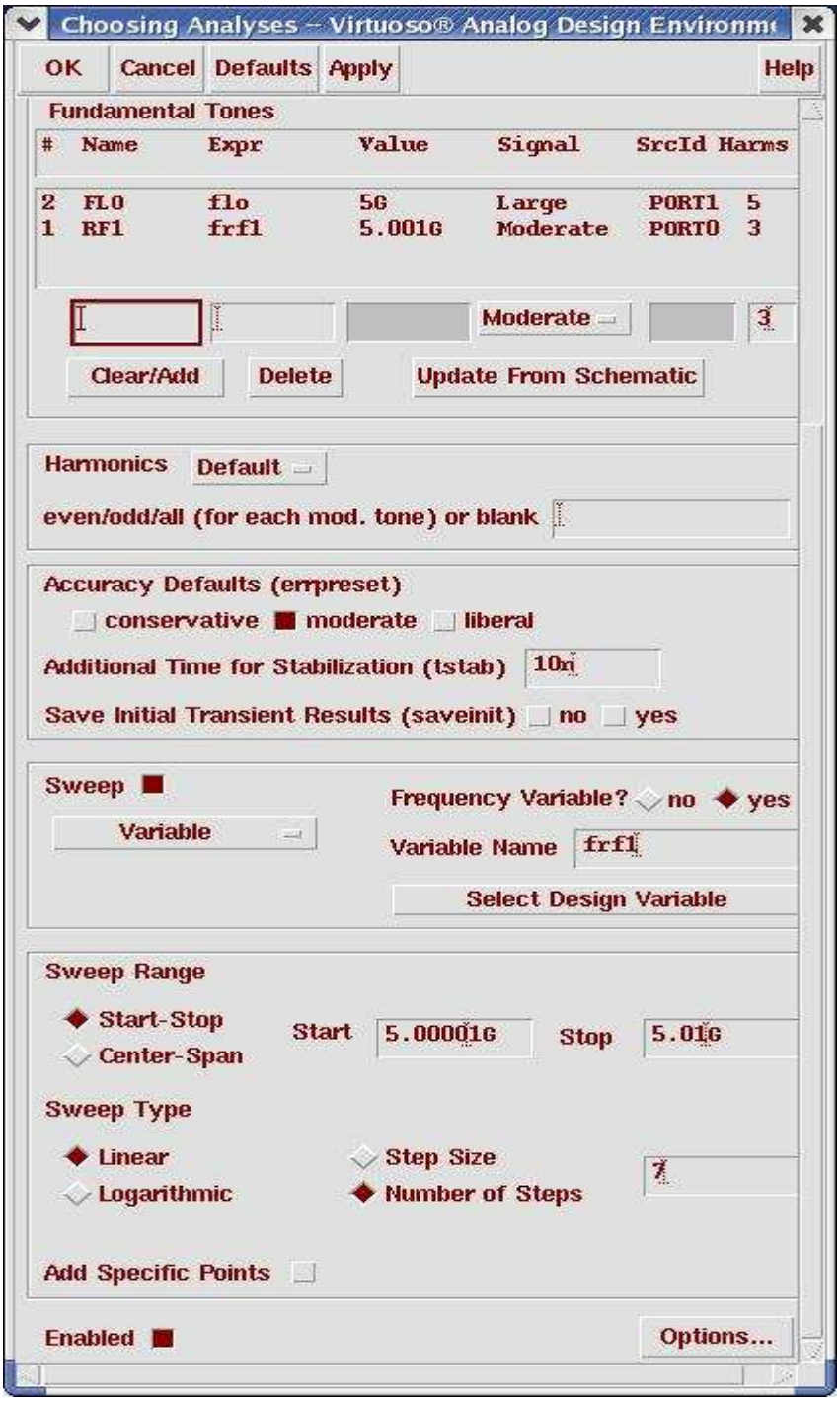
Action4-4: Check and save the schematic.

Action4-5: You can choose **Session—Load State**, load state "**Lab4_PCG_QPSS**" and skip to Action4-12 or …

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

Action4-7: 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 might select any reasonable number of LO harmonics since the large tone is modeled in the time domain and you can analyze up to 40 harmonics without runtime penalty.



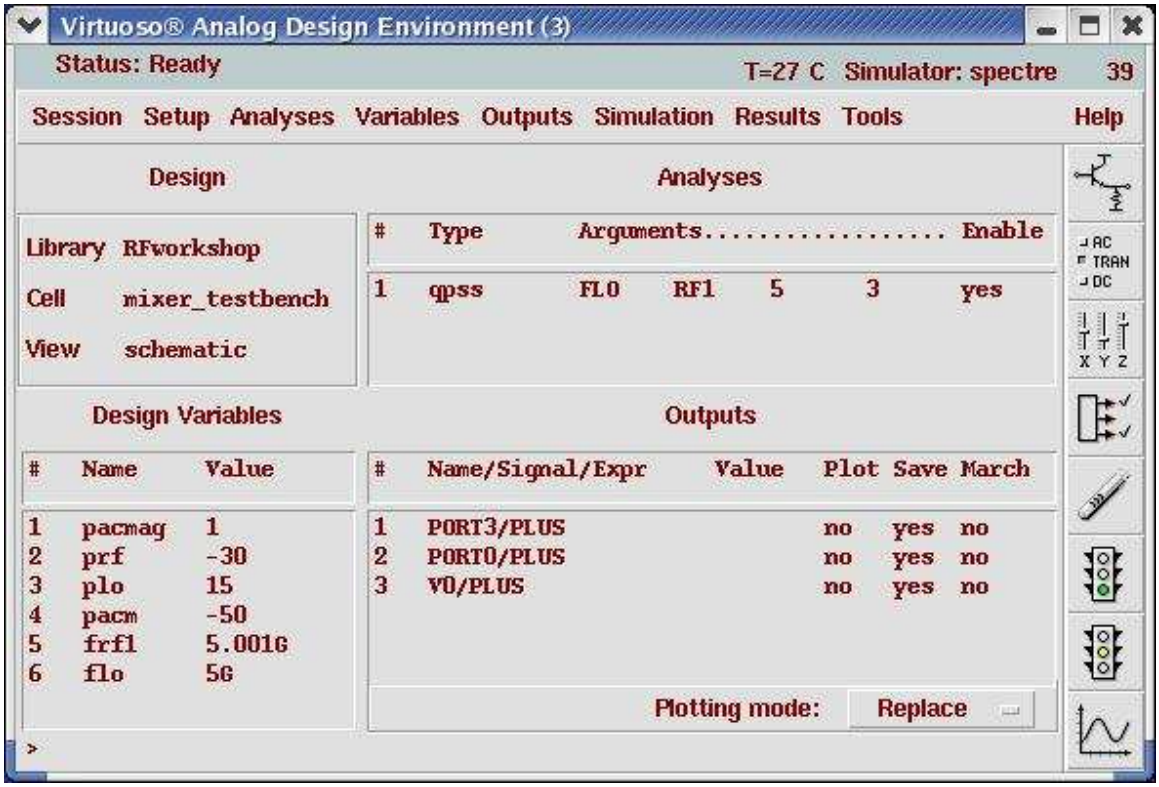
Action4-8: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Action4-9: In the Virtuoso Analog Design Environment window, choose **Outputs—To Be Saved—Select on Schematic**.

Action4-10: In the schematic, select the positive terminals of **port0**, **port3 and V0**. The terminals are circled in the schematic window after you select them, indicate that you will save the currents at these nodes.

Action4-11: press **Esc** with your cursor in the schematic window to end the selections.

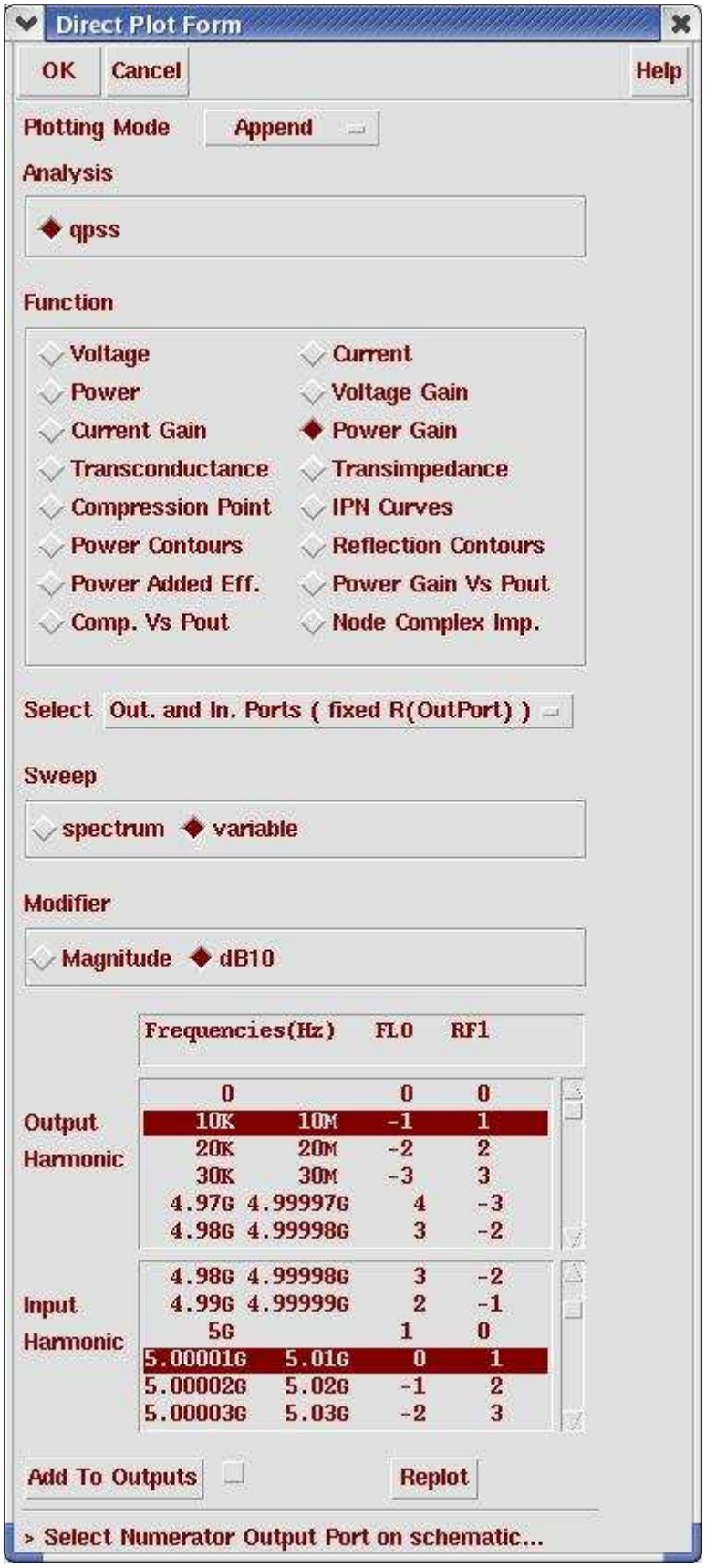
Now your Virtuoso Analog Design Environment will look like:



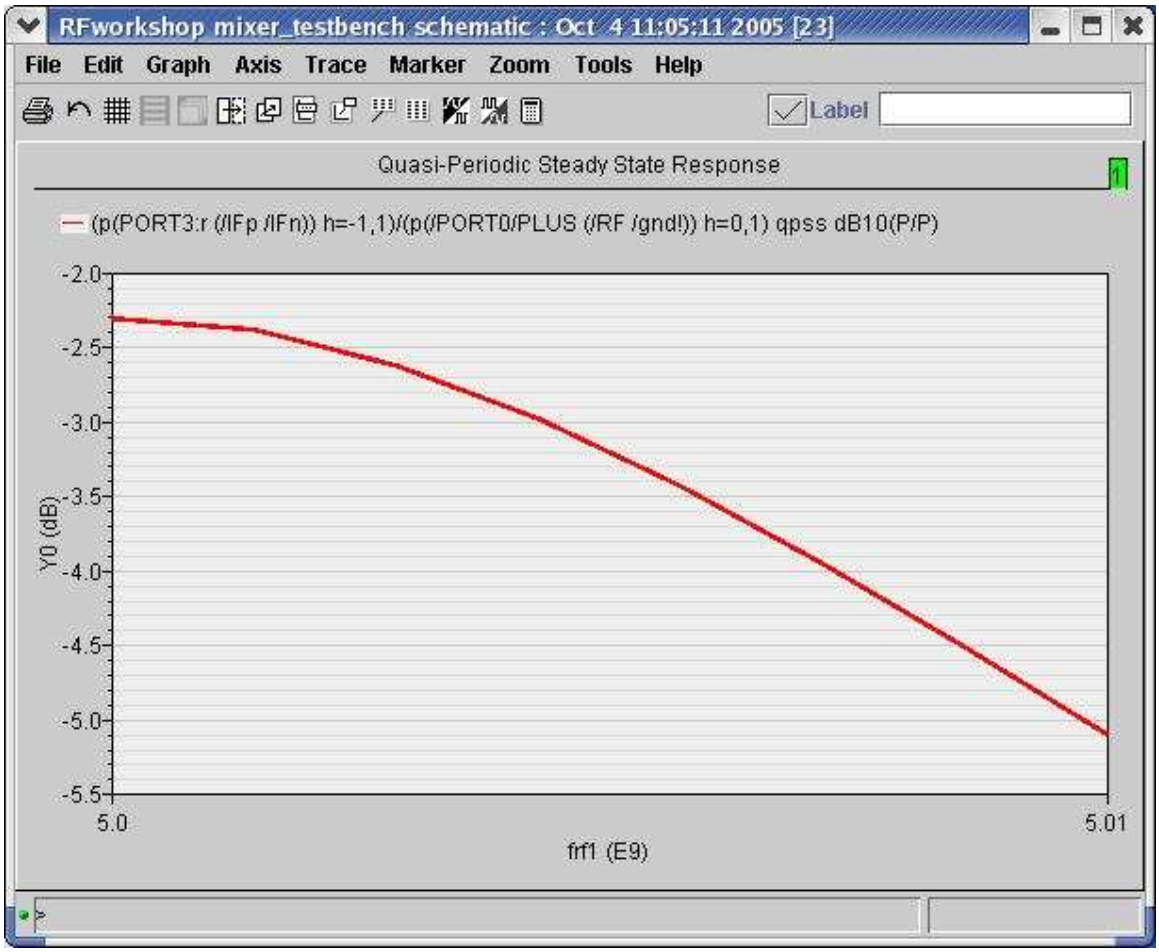
Action4-12: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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

Action4-14: In the Direct Plot Form, select the **qpss** button, and configure the form as follows:


Direct Plot Form
OK
Cancel
Help
Plotting Mode
Append
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
Out. and In. Ports (fixed R(OutPort))
Sweep
spectrum
variable
Modifier
Magnitude
dB10
Frequencies(Hz)
FLO
RF1
Output Harmonic
0 0 0
10K 10M -1 1
20K 20M -2 2
30K 30M -3 3
4.97G 4.99997G 4 -3
4.98G 4.99998G 3 -2
Input Harmonic
4.98G 4.99998G 3 -2
4.99G 4.99999G 2 -1
5G 1 0
5.00001G 5.01G 0 1
5.00002G 5.02G -1 2
5.00003G 5.03G -2 3
Add To Outputs
Replot
> Select Numerator Output Port on schematic...

Action4-15: Select output **port3** and input **port0** on schematic.



Action4-16: Close the waveform window, click **Cancel** in the Direct Plot form

Lab5: 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.

Action5-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action5-2: Select the **PORT0** source by placing the mouse cursor over it and clicking the left mouse button. Then in the Virtuoso Schematic Editor select **Edit—Properties—Objects…** The Edit Object Properties window for the port cell should come up. 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, in many cases it would be sufficient to treat the RF input as 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 interfere). In those cases, the RF source is a large signal and the source type is **sine**.

Action5-3: set **lo** port to **sine** type and output port to **dc** type.

Action5-4: Check and save the schematic.

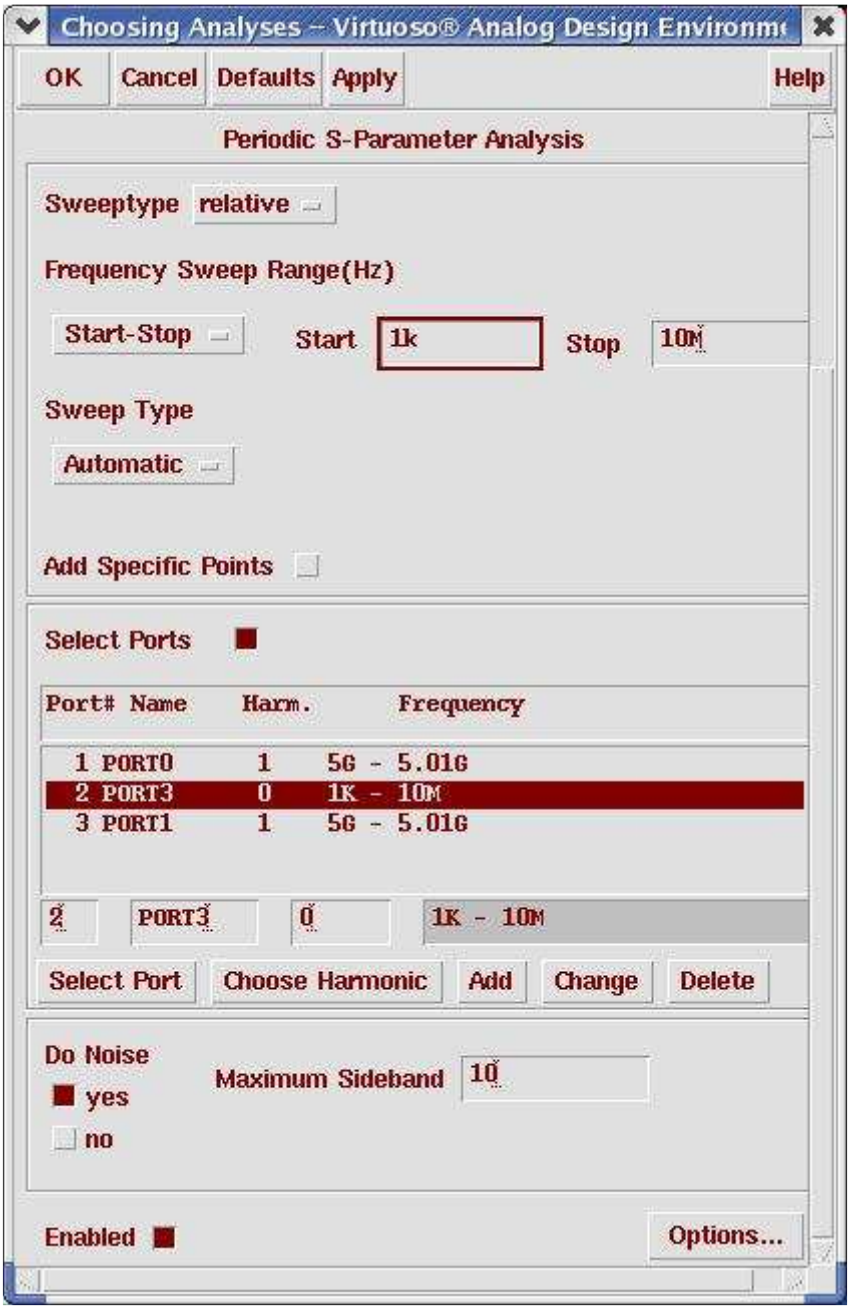
Action5-5: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action5-6: You can choose **Session—Load State**, load state "**Lab5_SParameter_PSSPSP**" and skip to Action5-11 or …

Action5-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

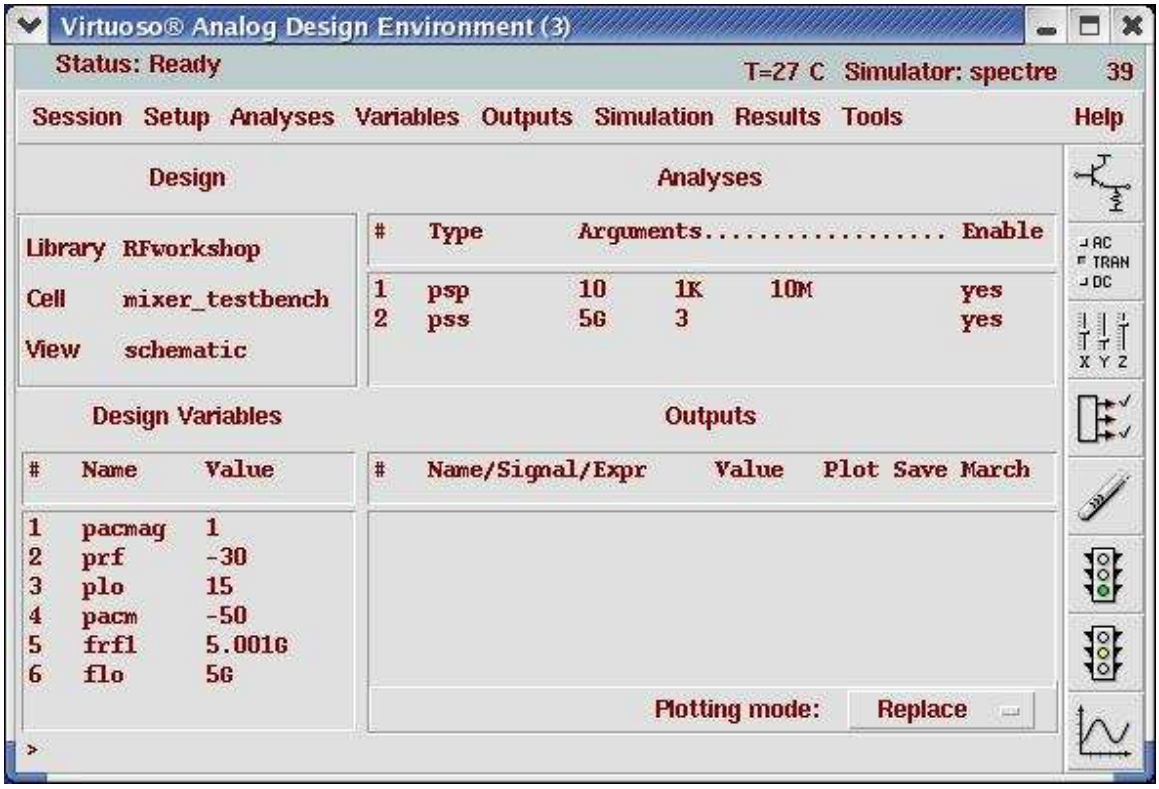
Action5-8: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window. Set the form as you did in Action2-6.

Action5-9: In the Choosing Analyses window, select the **psp** button in the **Analysis** field of the window and set the form as follows:



Action5-10: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:



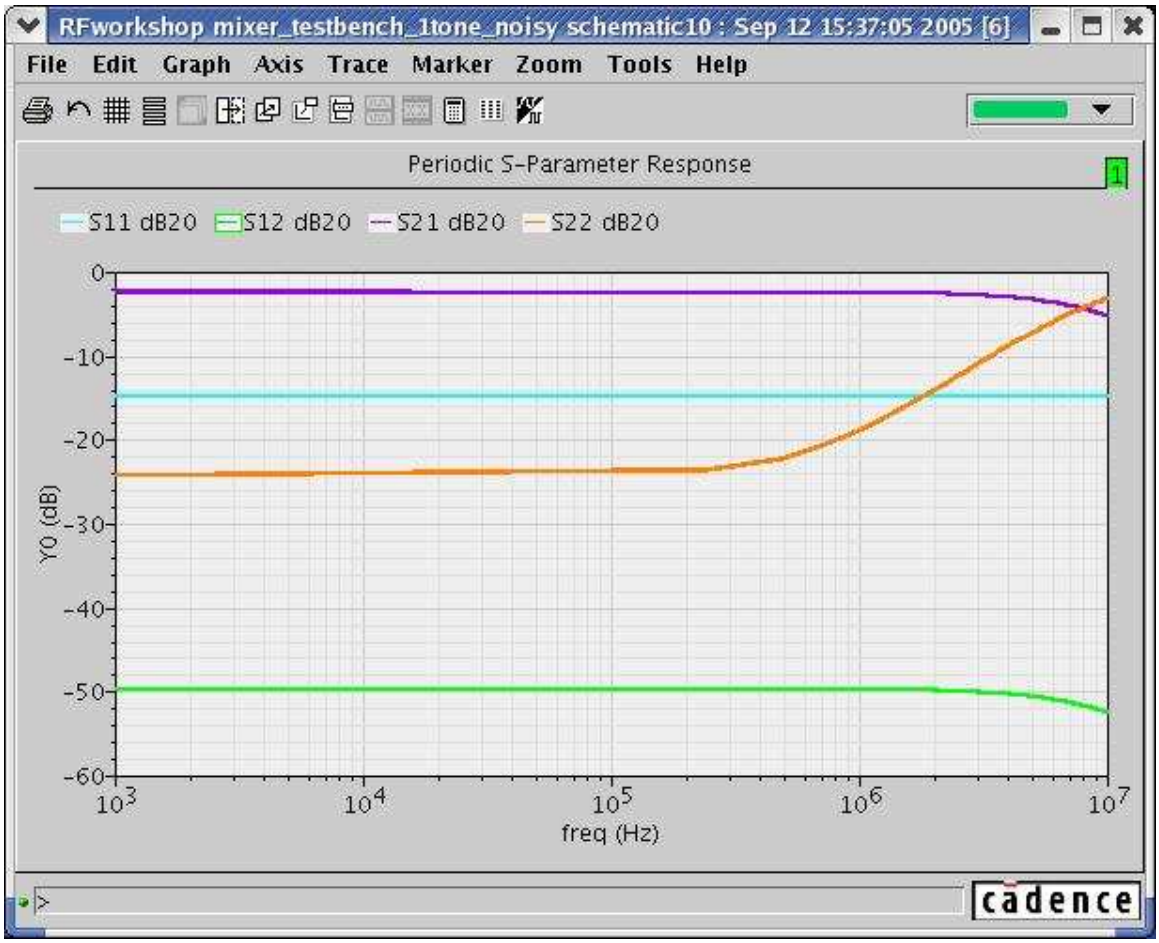
Action5-11: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

Action5-12: After the simulation completes, in the Virtuoso Analog Design Environment, choose **Results—Direct Plot—Main Form**.

Action5-13: 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...

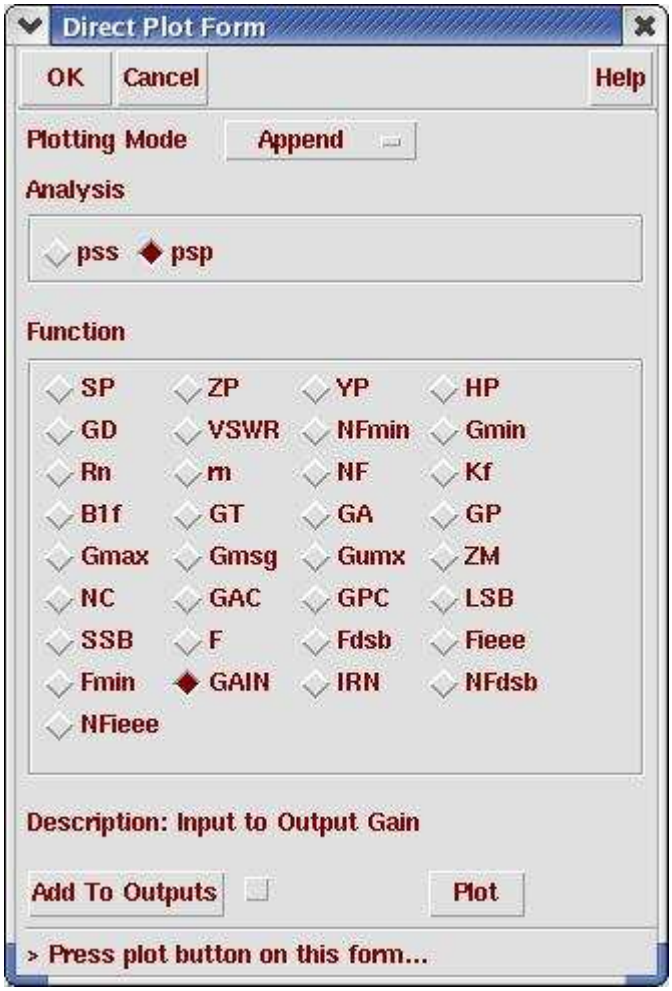
Action5-14: In the Direct Plot form, click on the S11, S21, S12 and S22. Here shows the waveforms:



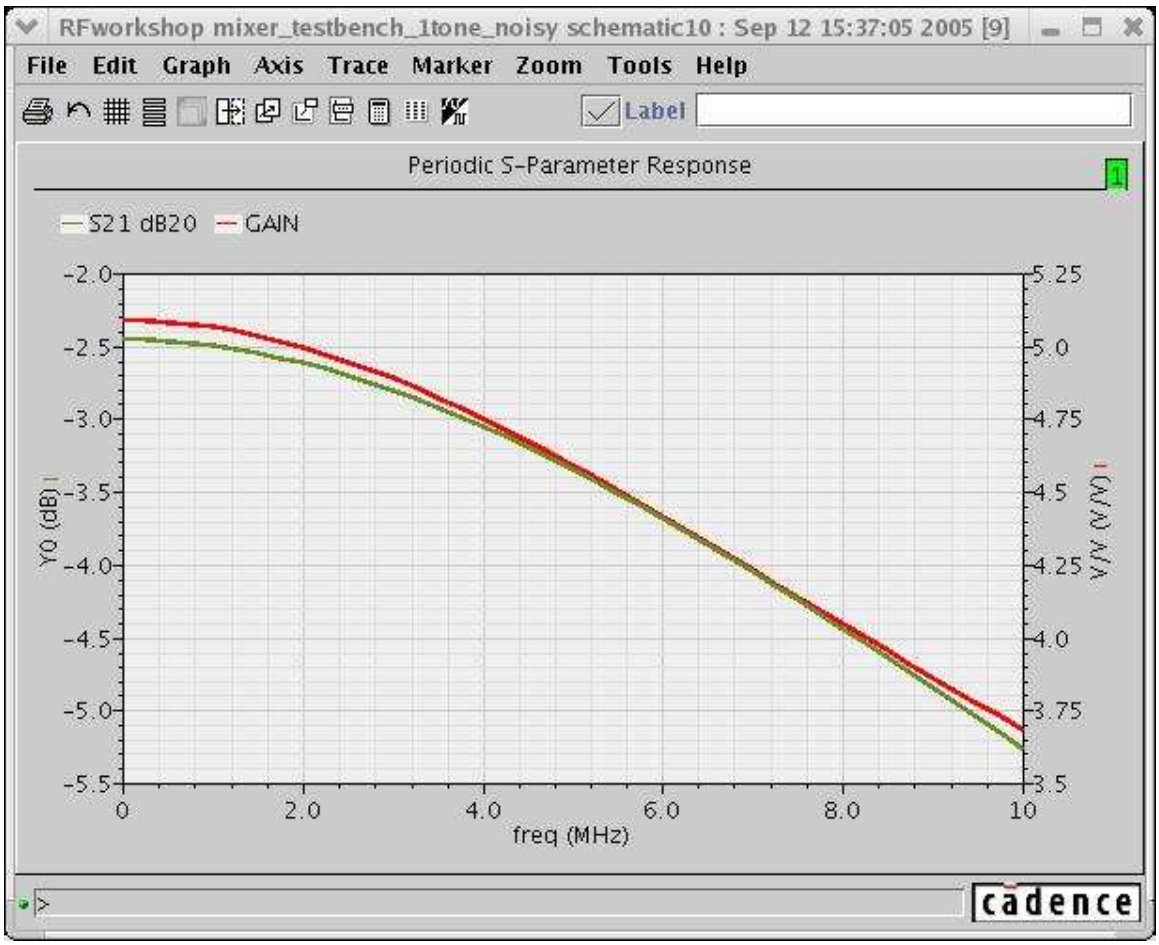
Action5-15: Close the waveform window.

Action5-16: In the Direct Plot form, choose **sp** function, set plot type to **Rectangular** and Modifier to **dB20**. Plot **S21**.

Action5-17: In the Direct Plot form, choose **GAIN** and click **OK** button to plot the input to output gain;



You should get the plot as below:



Action5-18: Close waveform window, click **Cancel** in the Direct Plot form. Close the waveform window.

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 S21and Pnoise gains will vary depending on the input match. PSP Gain is the voltage gain from the internal port voltage source to the output. You can refer to SpectreRF user guide Appendix L (using psp and pnoise analysis) for more details.

Lab6: Noise, Noise Figures and Noise Summary (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 which includes total output noise and noise figure.

Action6-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action6-2: Select the **PORT0** source, set the source type to **dc**. Select the **PORT3**, set the source type to **dc** and set **PORT1** to **sine**.

Action6-3: Check and save the schematic.

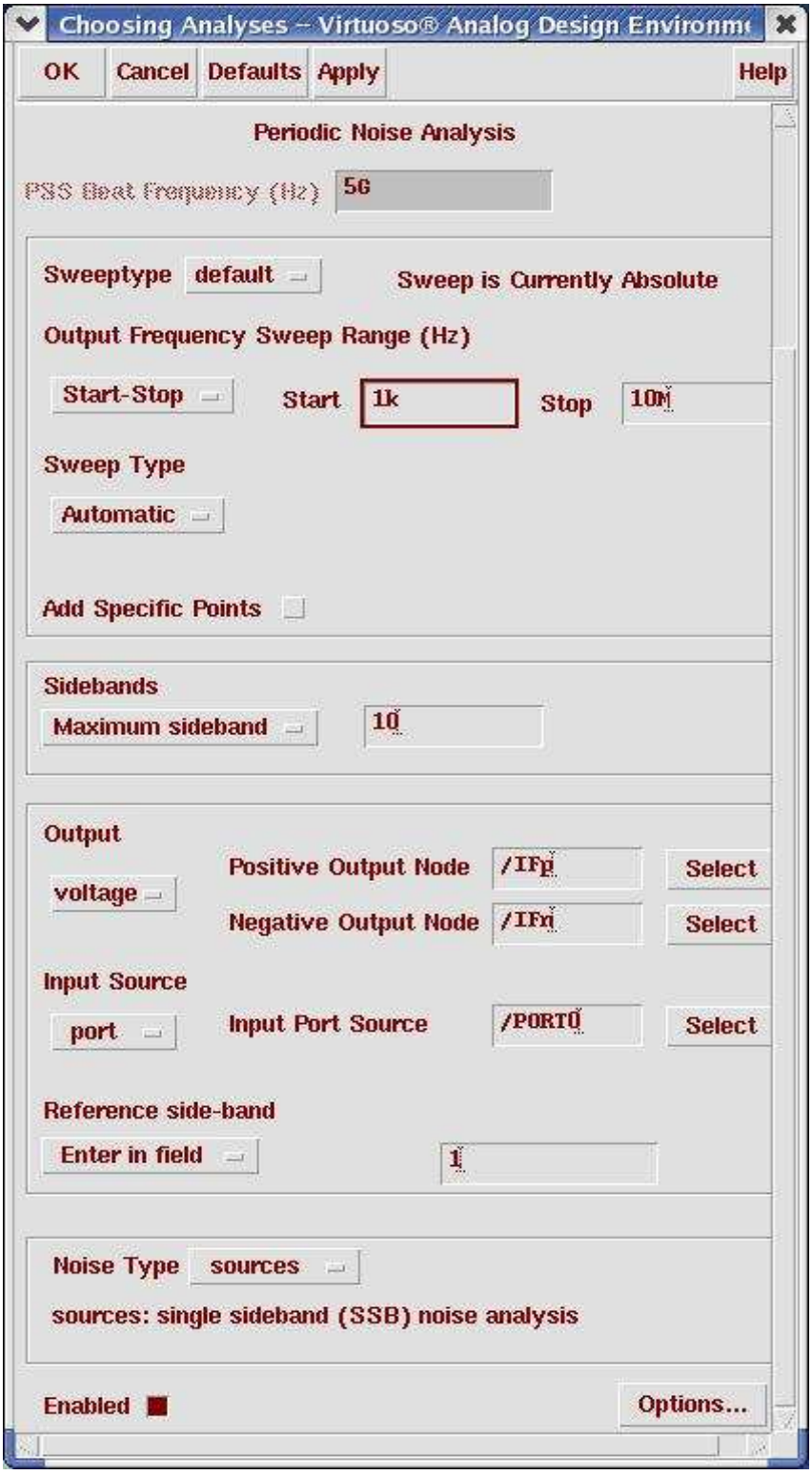
Action6-4: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action6-5: You can choose **Session—Load State**, load state "**Lab6_PNOISE**" and skip to Action6-10 or …

Action6-6: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

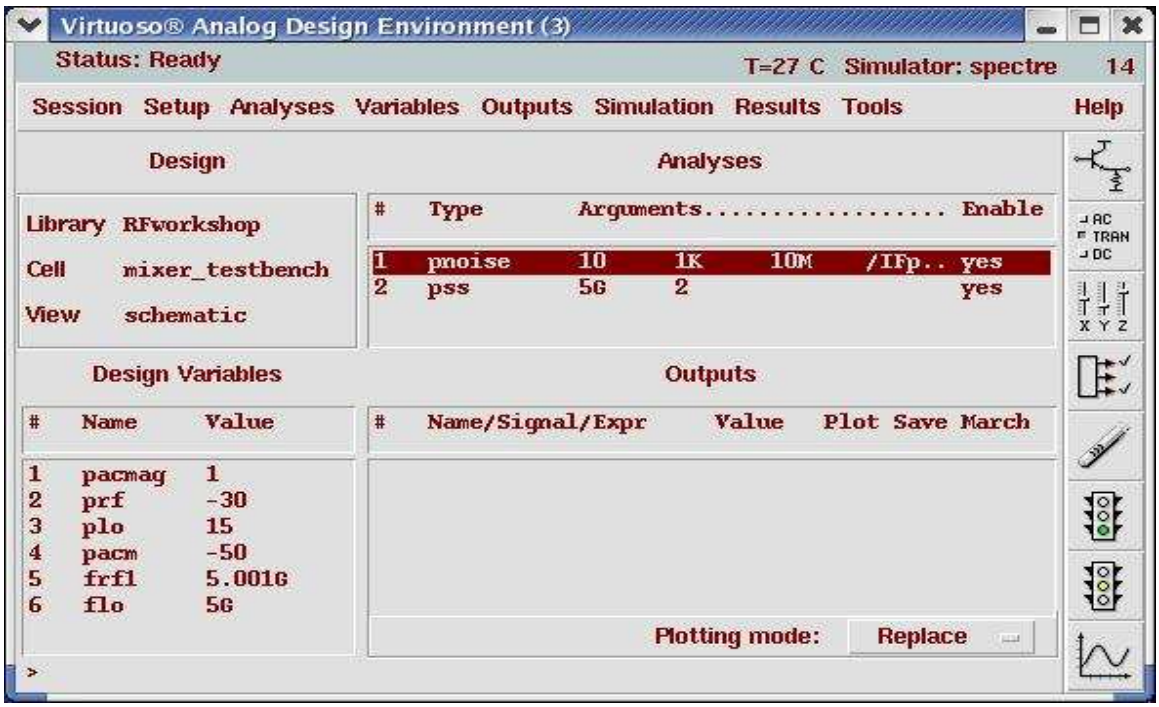
Action6-7: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set the form as you did in Action2-6.

Action6-8: In the Choosing Analyses window, select the **pnoise** button in the **Analysis** field of the window and set up the form as follows:



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

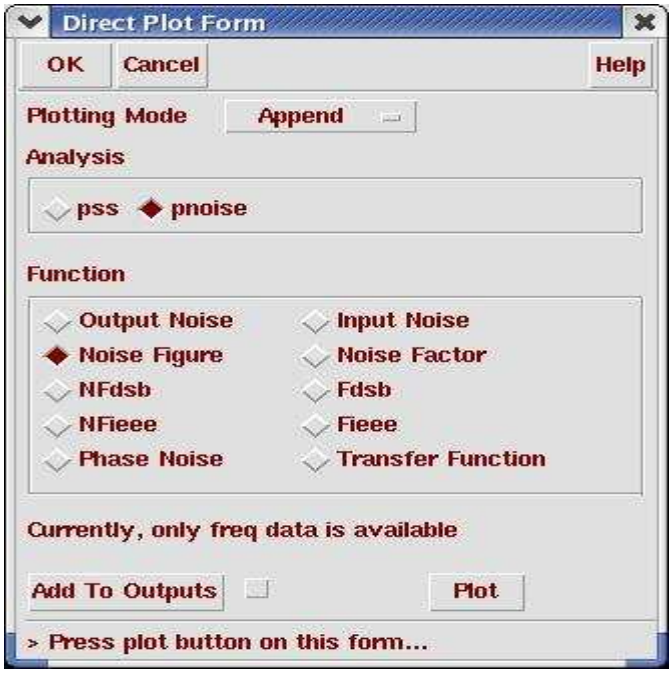
Now your Virtuoso Analog Design Environment will look like:



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

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

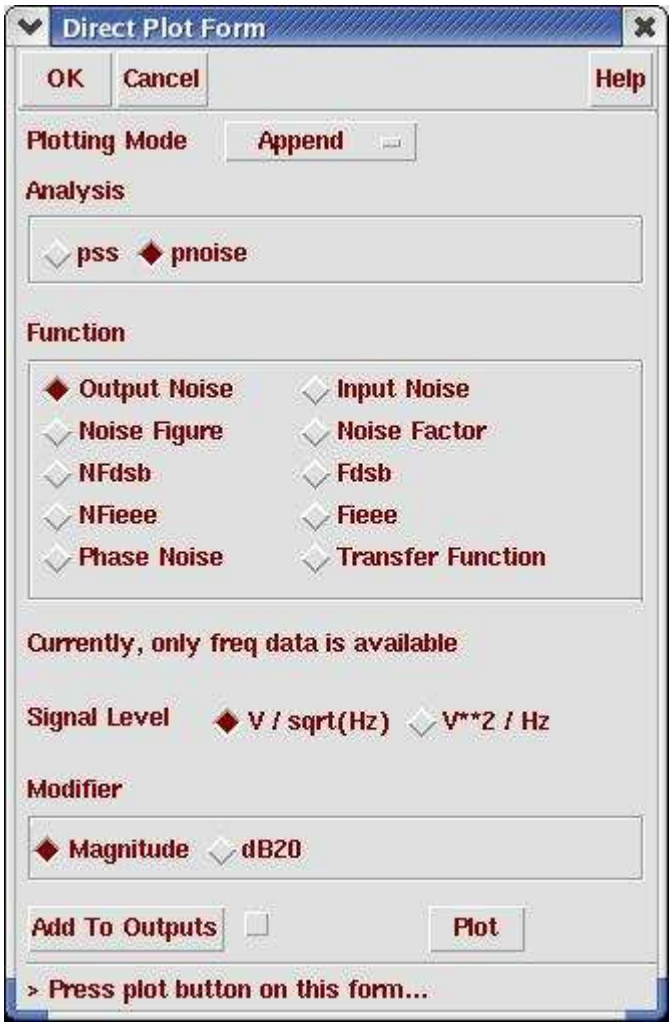
Action6-12: In the Direct Plot Form, select the **pnoise** button, and configure the form as follows:



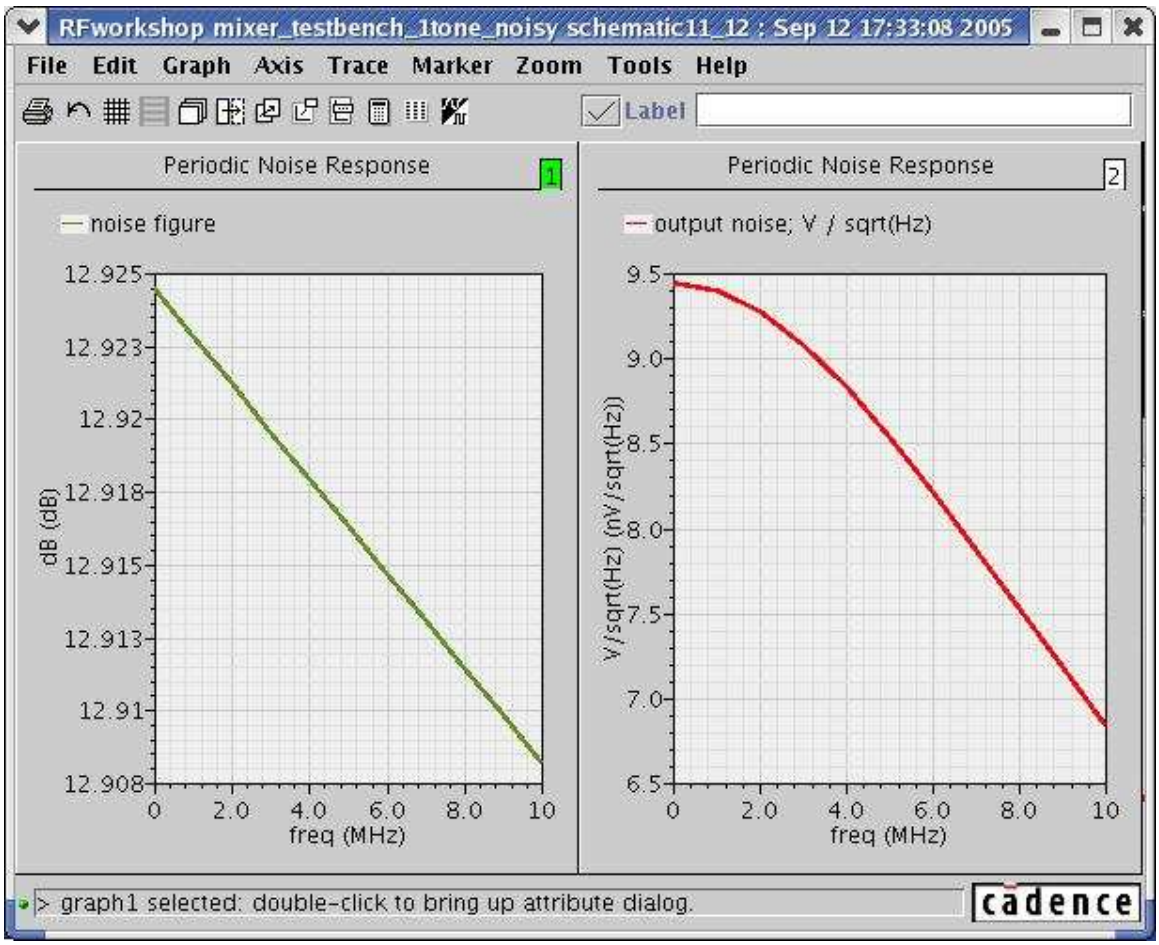
Action6-13: Click the **Plot** Button.

Action6-14: The waveform window will show up. In waveform window, click on **New Subwindow** button.

Action6-15: In the Direct Plot form, set the Function to **Output Noise** and configure the form as follows:



Action6-16: Click the Plot button. The waveform window should look like this:



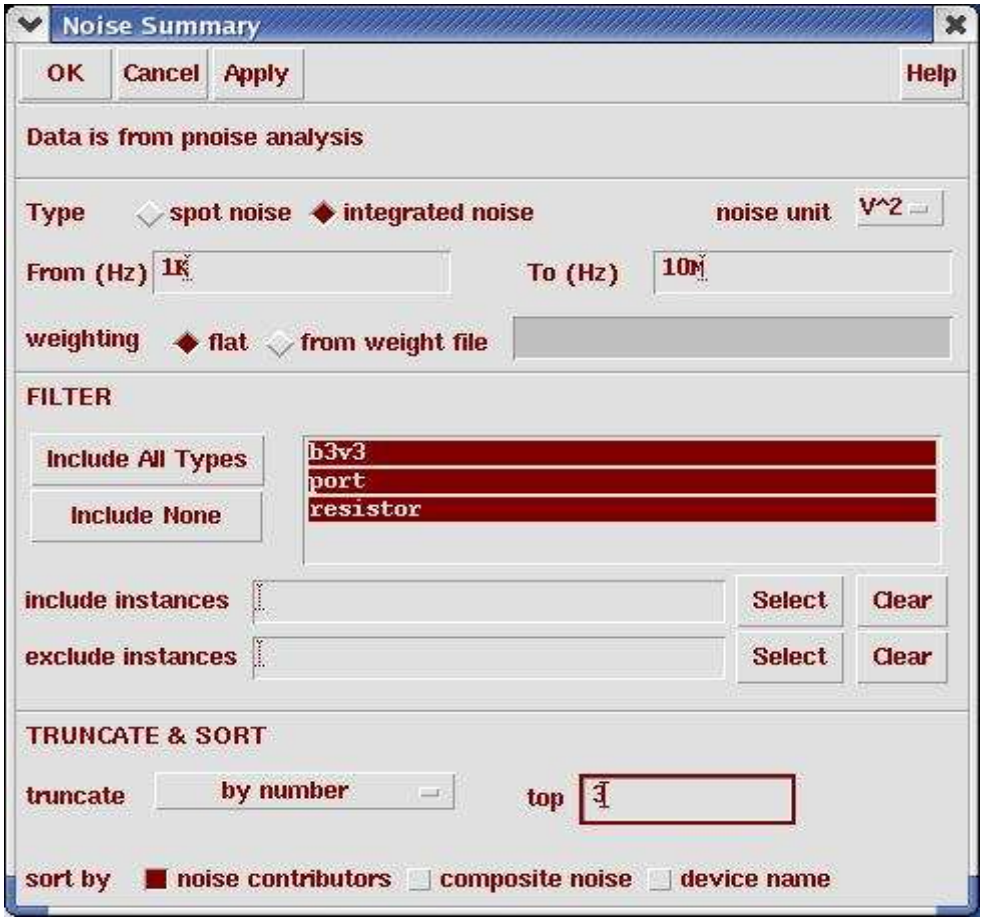
Action6-17: After viewing the waveforms, click **Cancel** in the Direct Plot form

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

Action6-18: In the Virtuoso Analog Design Environment window, choose **Results—Print—Noise Summary**.

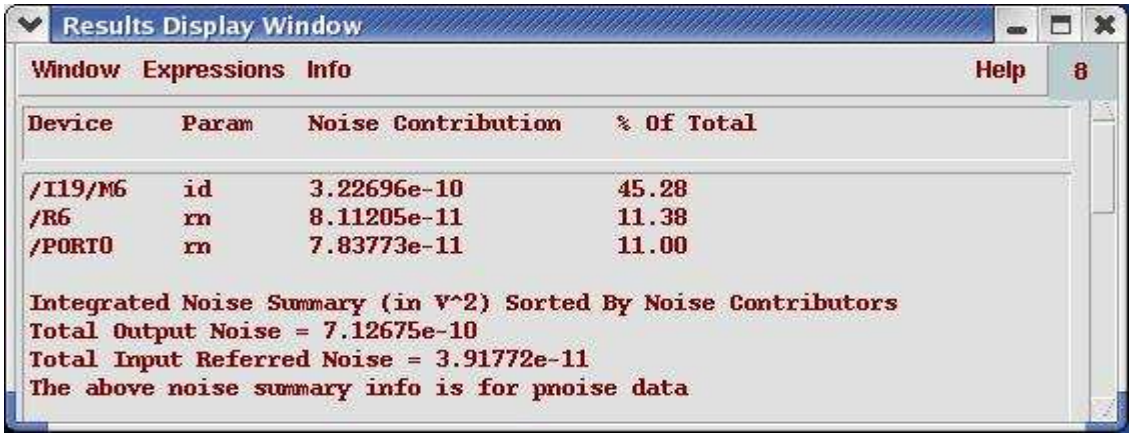
The Noise Summary form appears.

Action6-19: Fill in the form as shown here.



Action6-20: Click the **ALL Types** button in the FILTER section.

Action6-21: Click OK in the Noise Summary form.



Action6-22: After viewing the noise summary, close it by selecting **Window—Close.**

Lab7: 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 might be to set up a PAC analysis with nonzero pacmag parameter at the signal input (the RF port) A PXF analysis with the IF port as the output probe This example uses pacmag = 1 for simplicity.

Action7-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action7-2: Select the **PORT0** source by placing the mouse cursor over it and clicking the left mouse button. Then in the Virtuoso Schematic Editor select **Edit—Properties—Objects…** The Edit Object Properties window for the port cell should come up. Change the port properties as follows:

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

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

Action7-4: Set the source type of **PORT1** to **sine**, and **PORT3** to **dc**.

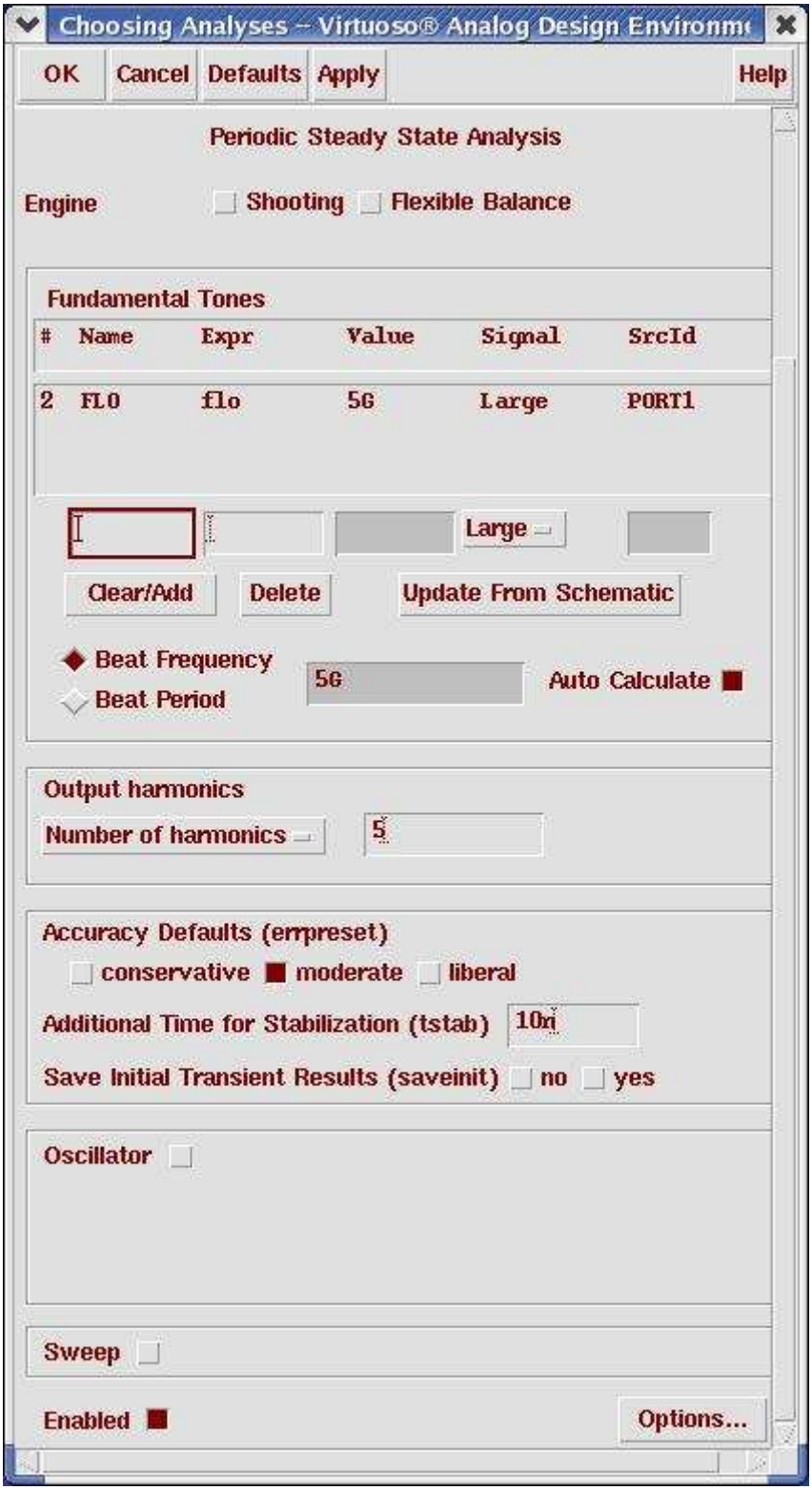
Action7-5: Check and save the schematic.

Action7-6: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action7-7: You can choose **Session—Load State**, load state "**Lab7_Isolation_PAC**" and skip to Action7-14 or …

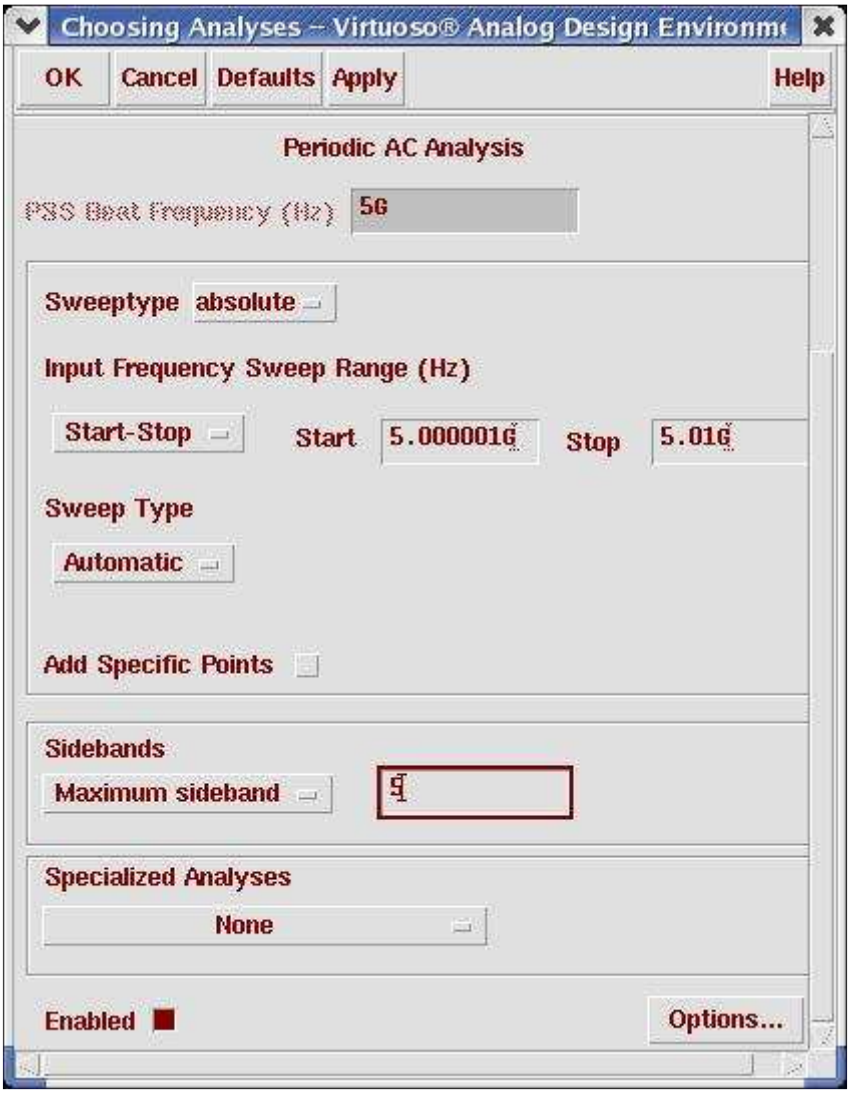
Action7-8: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action7-9: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window. Set up the form as follows:



PSS simulation is set up here to check the LO feed through. PXF is a small signal analysis. LO feedthrough is a large-signal effect and shouldn't be measured using a small-signal analysis.

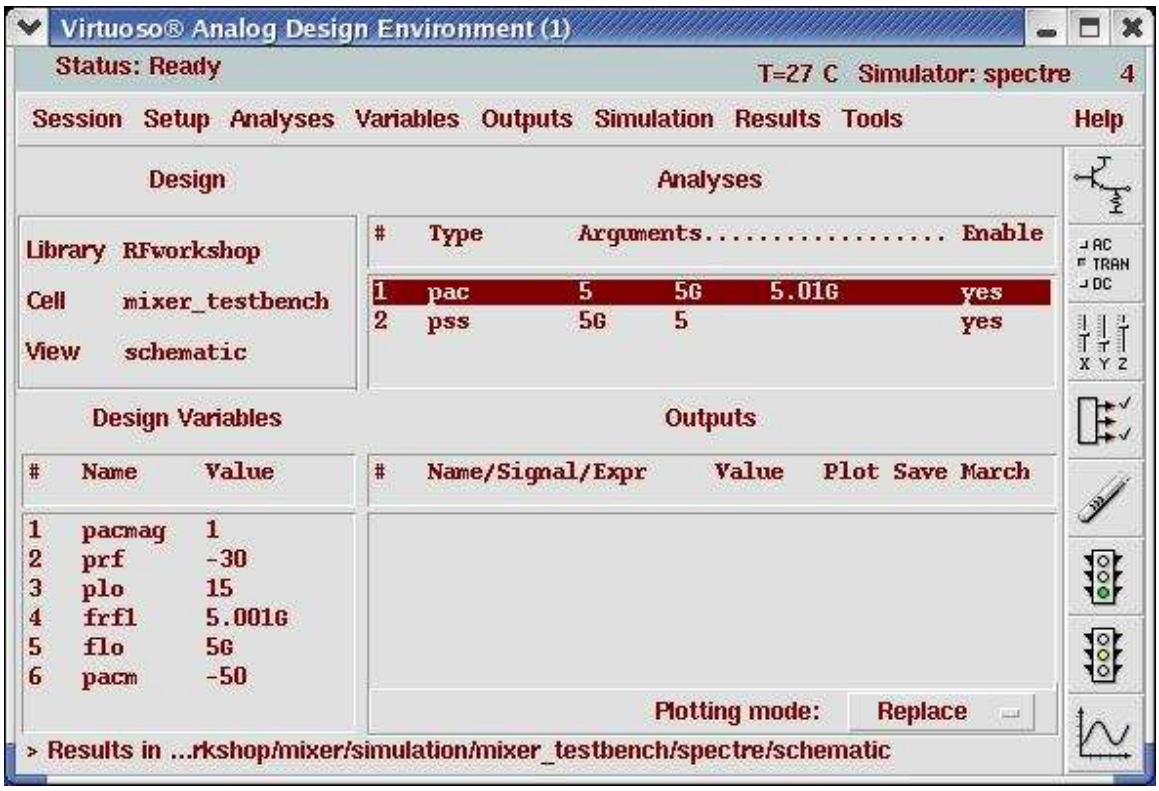
Action7-10: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window. Set up the form as follows:



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

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

Now your Virtuoso Analog Design Environment will look like:



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

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.

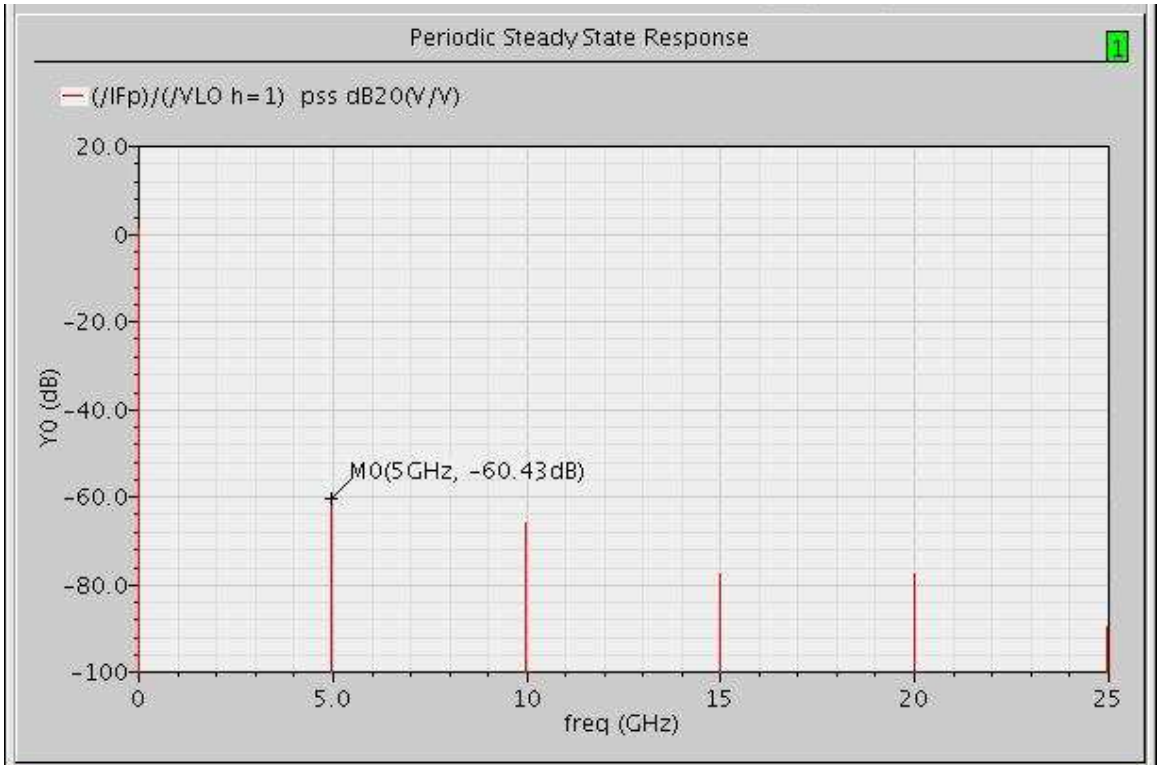
Action7-13: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

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

Action7-15: In the schematic, select the net **IFp** as the output and net **VLO** as input.

Action7-16: In the waveform window, select **Maker—Place—Trace Maker** to place a maker at first harmonic 5 GHz.

The following waveform shows the LO to IF feedthrough:



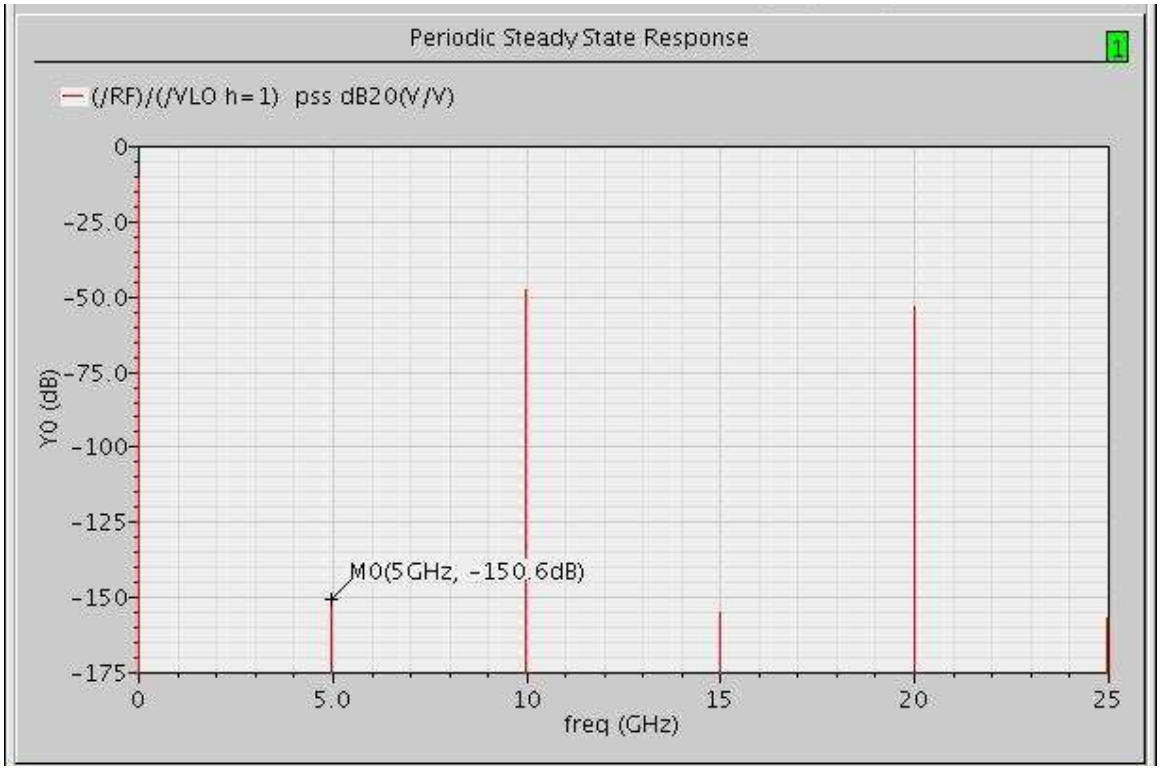
LO-to-RF feedthrough affects the functionality of LNAs and antennas and causes the dc offset because of self-mixing of Mixer. Use the results of the PSS analysis with the LO port as input and RF port as output to measure the LO-to-RF feedthrough.

Action7-17: In the Direct Plot Form, change the Plotting Mode to **Replace**.

Action7-18: In the schematic, select the net **RF** as the output and net **VLO** as input.

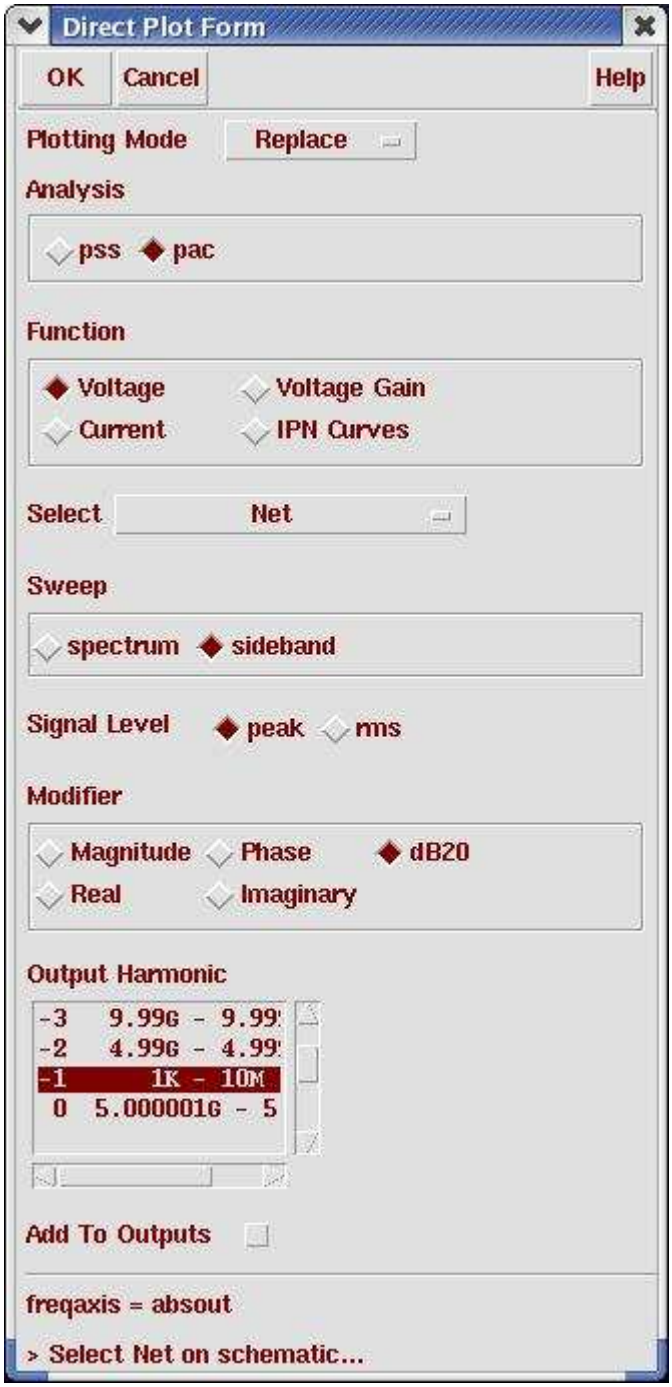
Action7-19: In the waveform window, select **Maker—Place—Trace Maker** to place a maker at first harmonic 5 GHz.

The following waveform shows the LO to IF feedthrough:

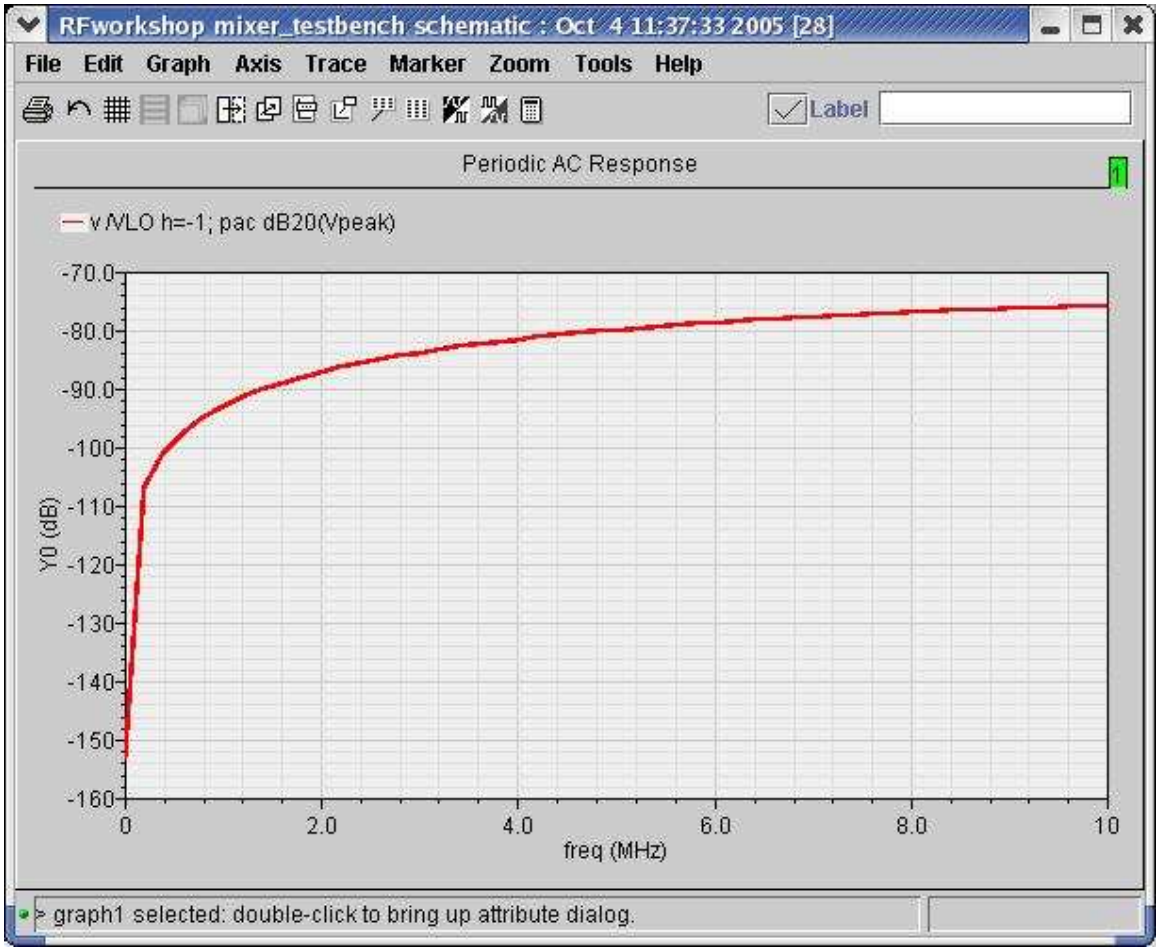


RF-to-LO feedthrough affects the local oscillator by letting strong interferers at the input pass to the LO. Measure RF-to-LO feedthrough using the PAC analysis results.

Action7-20: In the Direct Plot Form, select the **pac** button, and configure the form as follows:

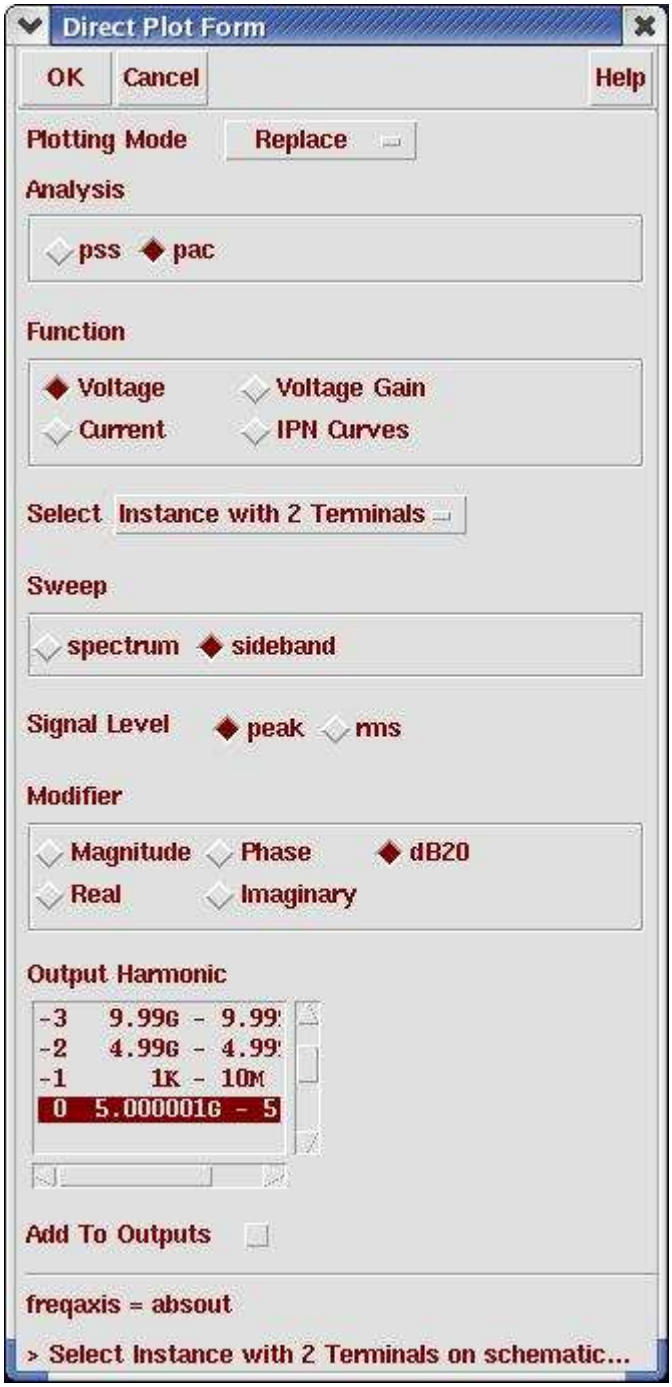


Action7-21: Select -1 in Output Harmonics to represent the down-converted RF signal and click on the **VLO** net to select the LO port as the output instance. The following waveform shows the RF to LO feedthrough:

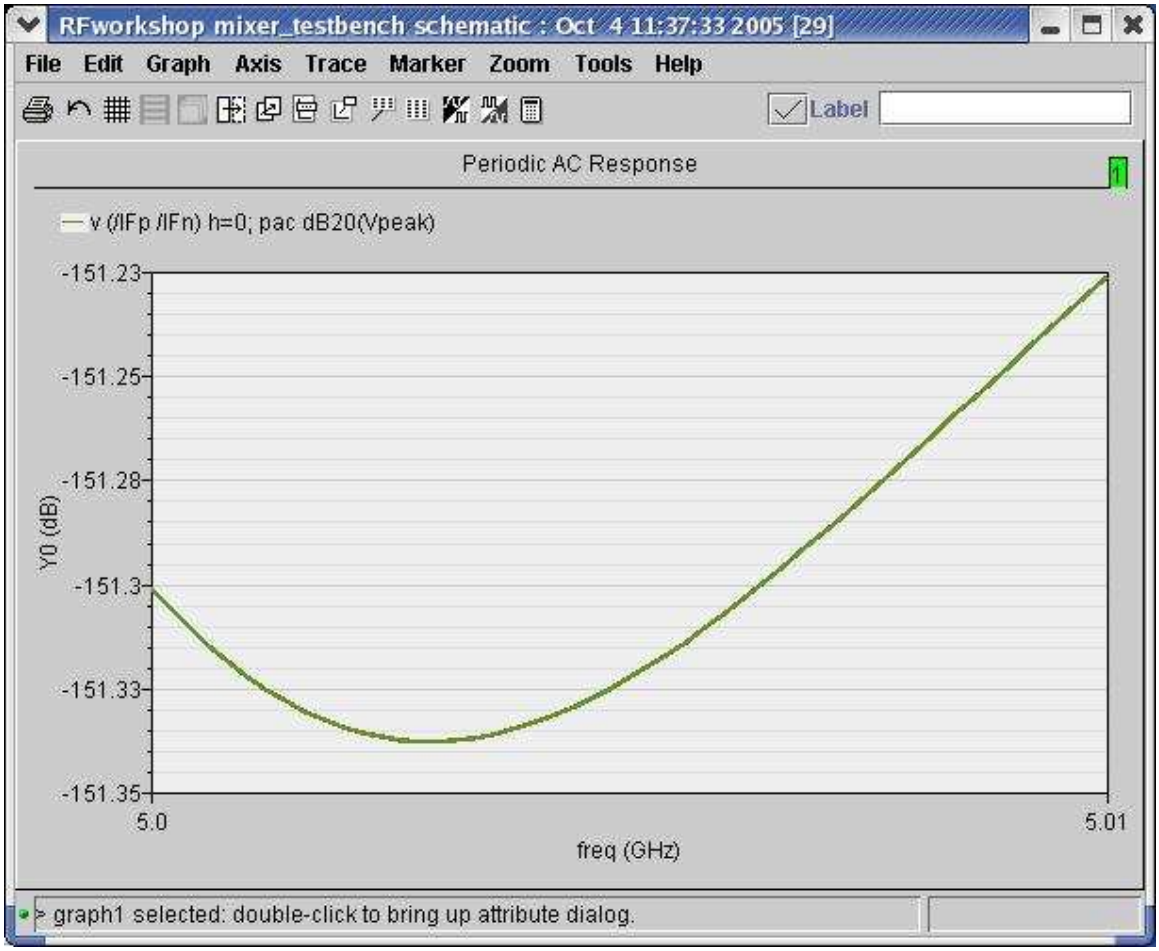


RF-to-IF feedthrough might create an even-order distortion problem for homodyne receivers. Measure RF-to-IF feedthrough using the PAC analysis results with two simple changes.

Action7-22: In the Direct Plot Form, select the **pac** button, and configure the form as follows:



Action7-23: Select 0 in Output Harmonics and select the IF output **PORT3**. The following waveform shows the RF to IF feedthrough:



Action7-24: Click Cancel in the Direct Plot form. Close the waveform window.

Lab8: 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.

Action8-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action8-2: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog Environment** command.

Action8-3: 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

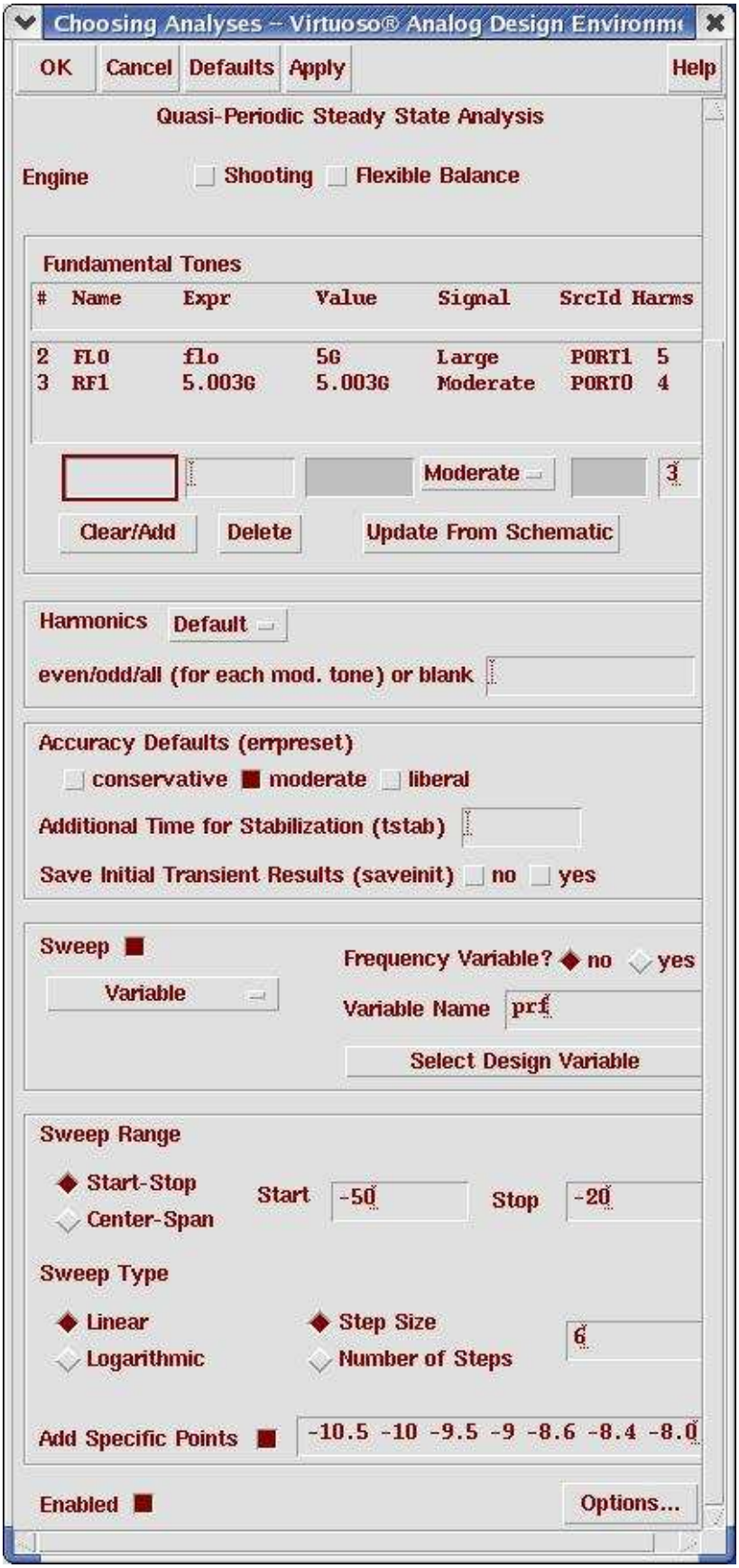
Action8-4: Make sure the source type of **PORT1** is set to sine and **PORT3 dc** type.

Action8-5: Check and save the schematic.

Action8-6: You can choose **Session—Load State**, load state "**Lab8_Blocker_QPSSQPACQPnoise**" and skip to Action8-14 or …

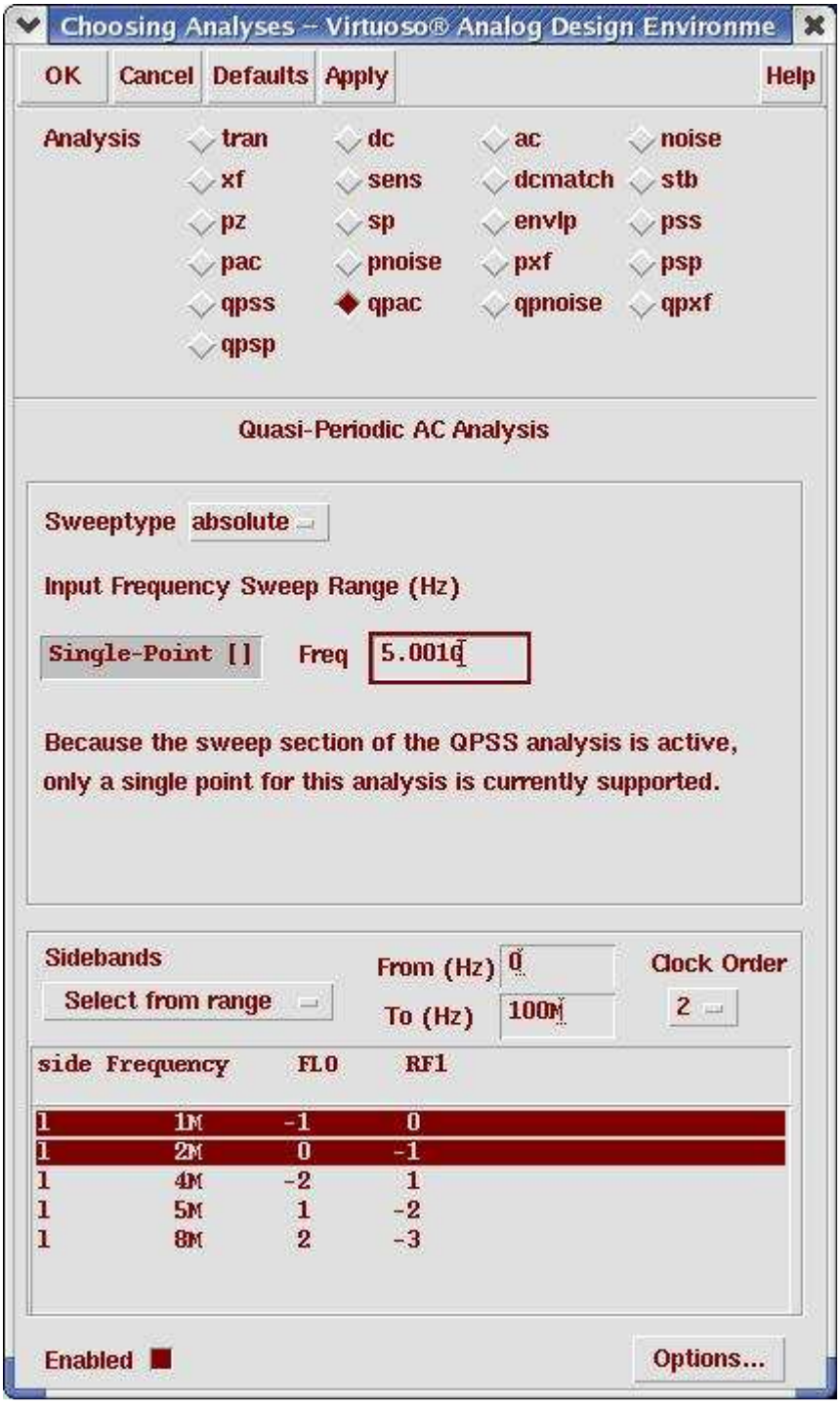
Action8-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action8-8: 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 should look like this:


Choosing Analyses – Virtuoso® Analog Design Environme
OK
Cancel
Defaults
Apply
Help
Quasi-Periodic Steady State Analysis
Engine
Shooting
Flexible Balance
Fundamental Tones
Name Expr Value Signal SrcId Harms
2 FLO flo 5G Large PORT1 5
3 RF1 5.003G 5.003G Moderate PORT0 4
Moderate
3
Clear/Add
Delete
Update From Schematic
Harmonics
Default
even/odd/all (for each mod. tone) or blank
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
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
-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
Enabled
Options...

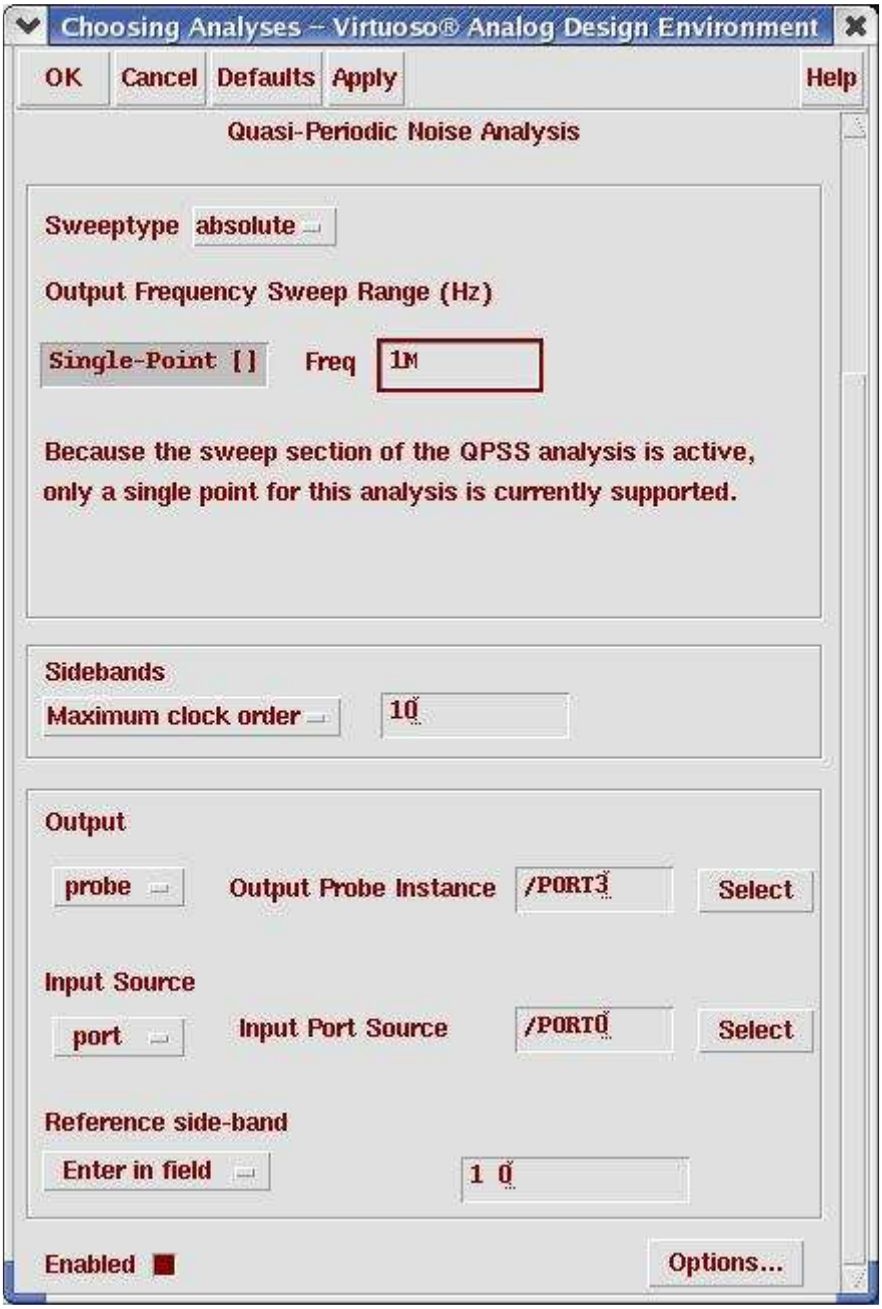
Action8-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **apply** button.

Action8-10: In the Choosing Analyses window, select the **qpac** button in the **Analysis** field of the window. input frequency = 5.001 GHz and Sweep type = absolute. The form should look like this:



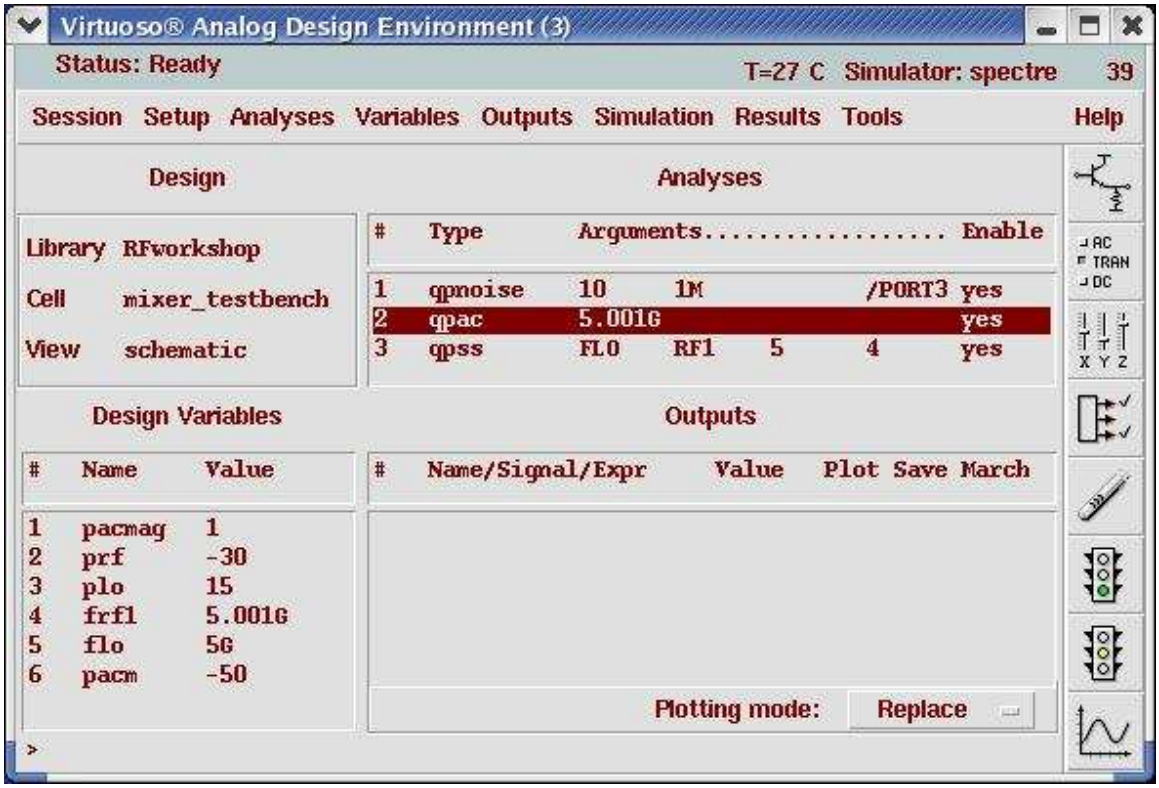
Action8-11: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **apply** button.

Action8-12: 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 sideband as (1 0) to represent a downconverted RF signal relative to the IF output signal, 1 MHz + 1 * f (LO) = f (RF). The form should look like this:



Action8-13: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like

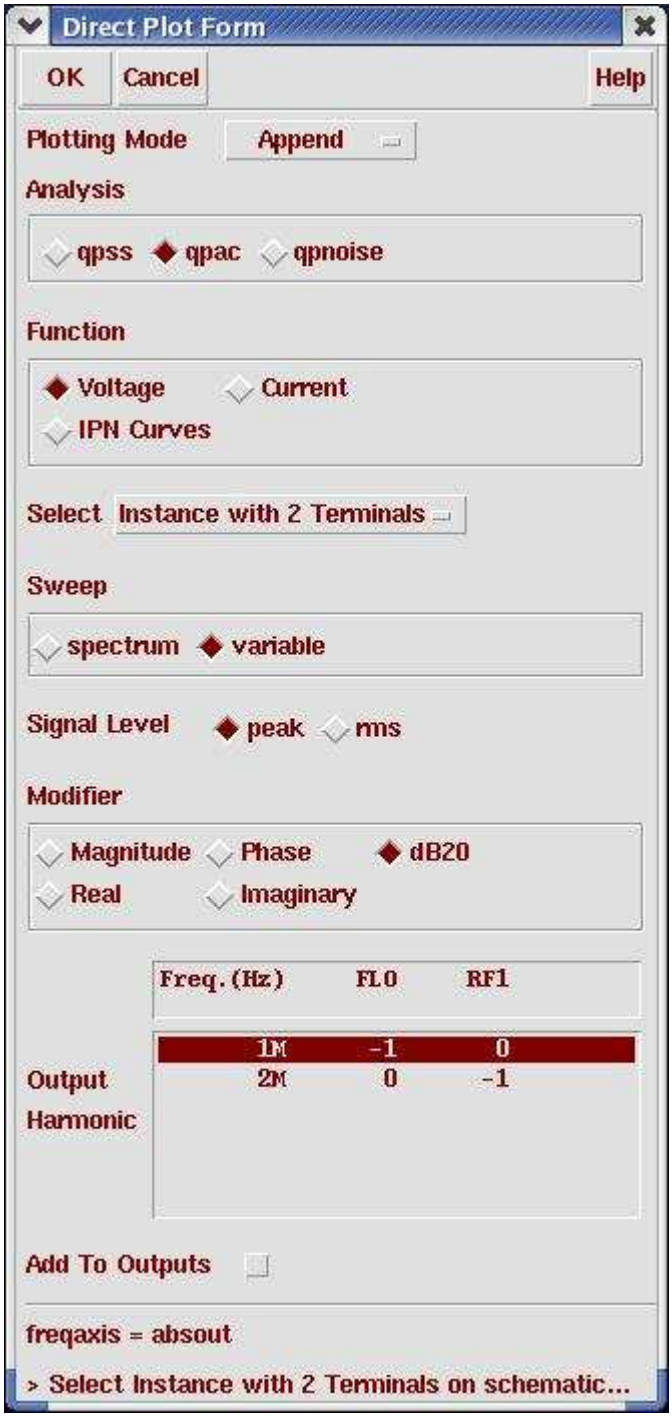


Action8-14: In your Analog Design Environment, choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

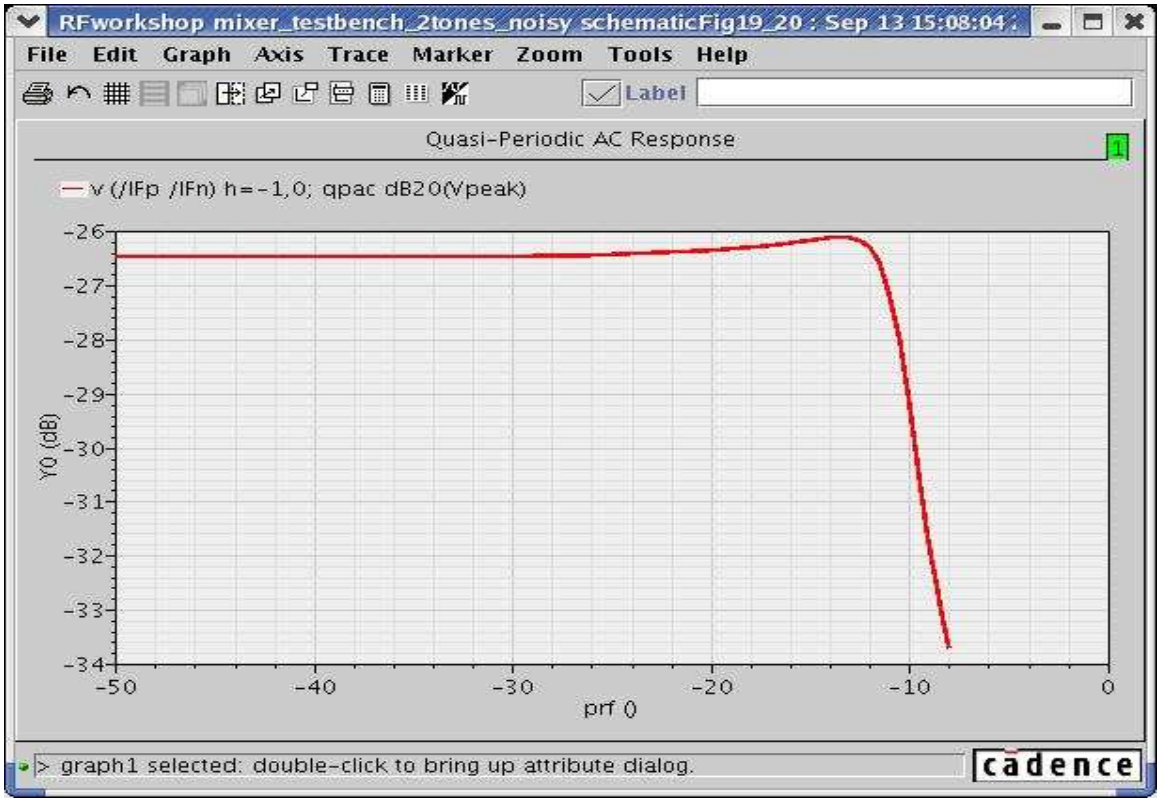
When the simulation ends, we can check how the blocking signal affect the performance of the Mixer.

Action8-15: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action8-16: In the Direct Plot Form, select the **qpac** button, and configure the form as follows:



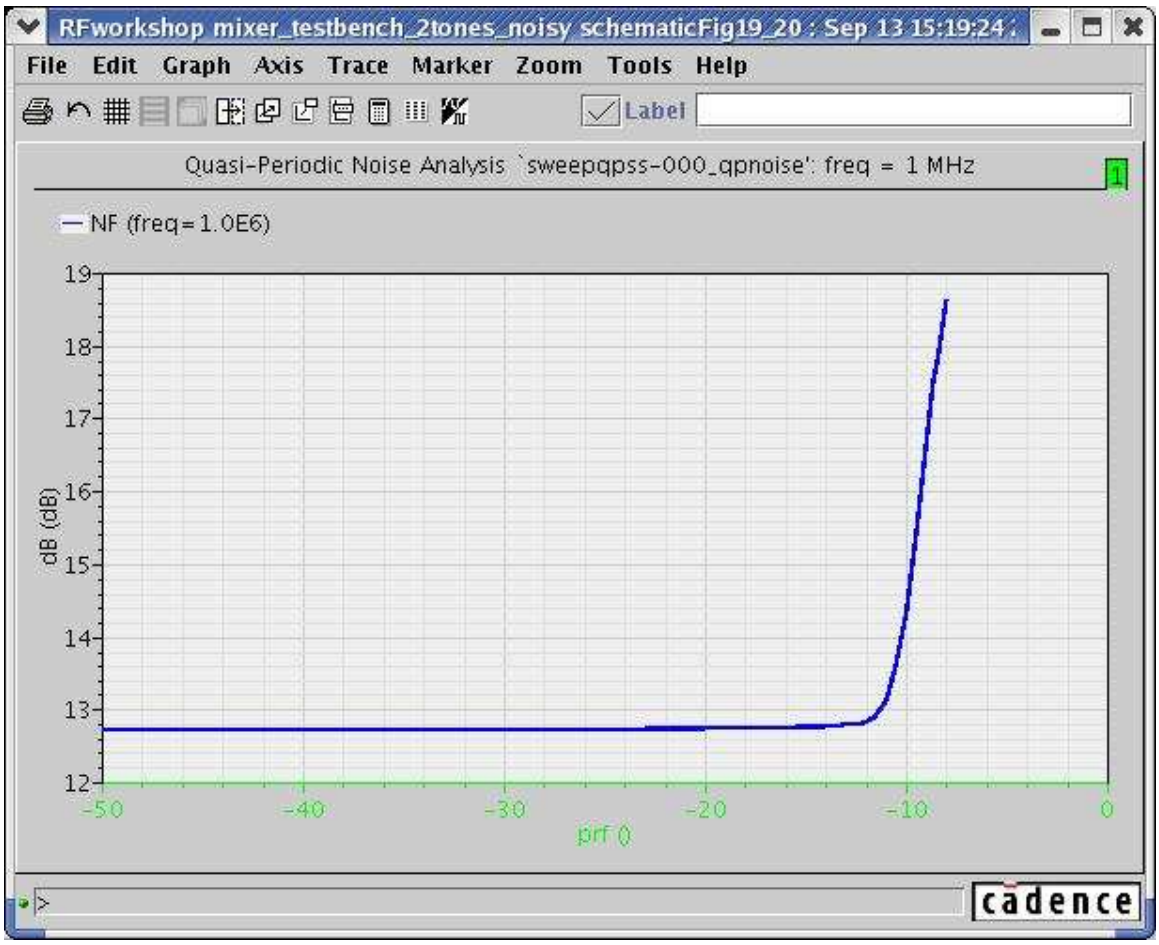
Action8-17: Select PORT3. Here shows the Blocker Effect on Voltage Gain



Action8-18: Close the waveform window.

Action8-19: In the Direct Plot Form, select the **qpnoise** button, and configure the form as follows:

Action8-20: Click on Plot button. The waveform window appears. Double click on X-axis and change the **Sweep Var** filed to **prf**. Here shows the Blocker Effect on Noise Figure:



Action8-21: 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 $2f_2 - f_1$ will be in-band and so will 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 will also be 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 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 will have 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.

In previous version of SpectreRF, user can use either qpss-based or qpac-based method to calculation IP3 in a system contains mixer and LO. In 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 obtain 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 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 IM3 and RF signals, respectively. This method requires very high accuracy to accommodate large dynamic range between RF and LO signals because they are mixed in the same solution vector. For large circuit, it also relies on speed and convergence of multi- tone qpss.

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

Compared to 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.

In MMSIM60USR2, SpectreRF provides perturbative approach to solve weakly nonlinear circuit. The method does not require explicit high order derivatives from device model. All equations are formulated in the form of RF harmonics. They can be implemented in both time and frequency domains.

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

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

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

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

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

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

Lab9: IP3 Calculation (Swept QPSS and QPAC)

Action9-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action9-2: 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

Action9-3: Click **ok** on the Edit Object Properties window to close.

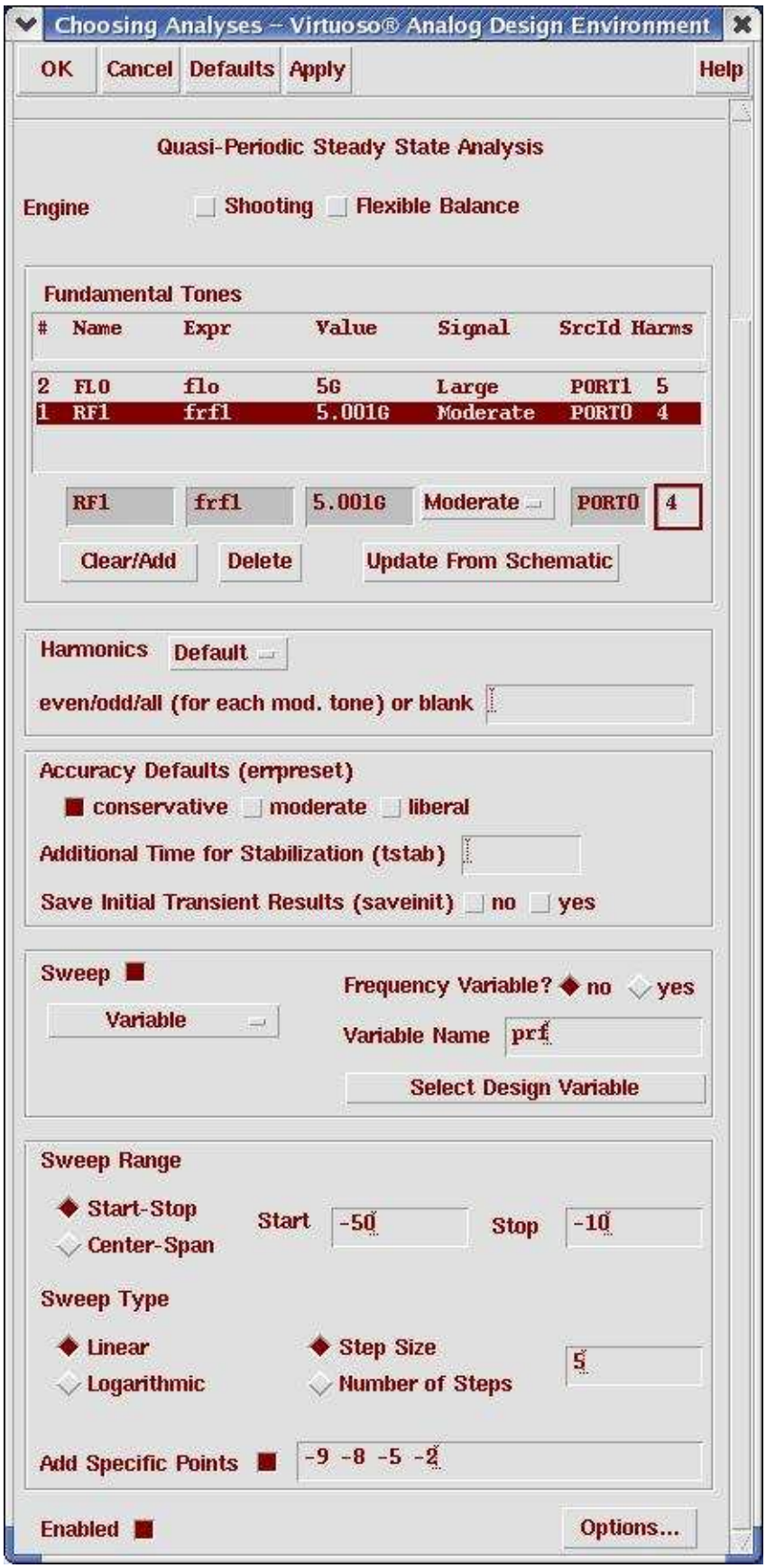
Action9-4: Check and save the schematic.

Action9-5: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog Environment** command.

Action9-6: You can choose **Session—Load State**, load state "**Lab9_IP3_QPSSQPAC**" and skip to Action9-12 or …

Action9-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

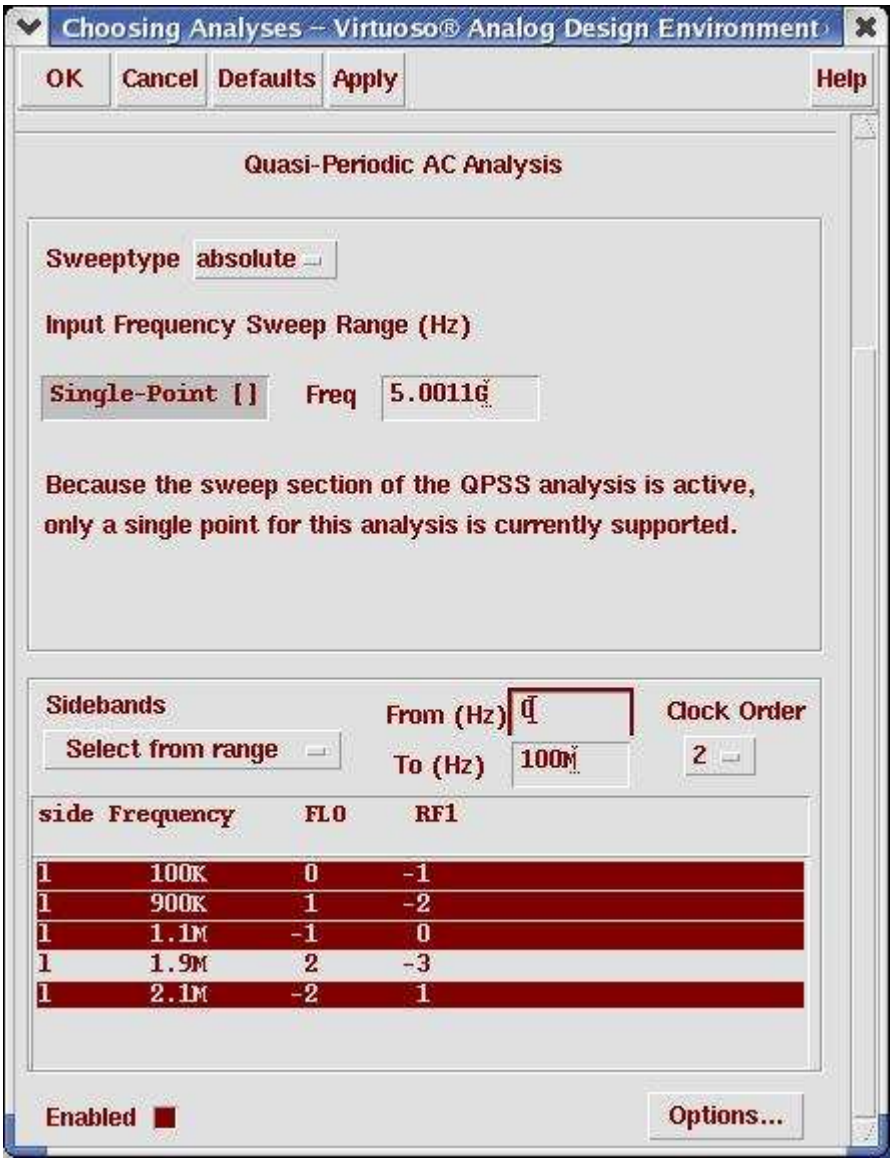
Action9-8: In the Choosing Analyses window, select the **qpss** button in the **Analysis** field of the window and set the form as follows:


Choosing Analyses – Virtuoso® Analog Design Environment
OK
Cancel
Defaults
Apply
Help
Quasi-Periodic Steady State Analysis
Engine
Shooting
Flexible Balance
Fundamental Tones
Name Expr Value Signal SrcId Harms
2 FLO flo 5G Large PORT1 5
1 RF1 frf1 5.001G Moderate PORT0 4
RF1
frf1
5.001G
Moderate
PORT0
4
Clear/Add
Delete
Update From Schematic
Harmonics
Default
even/odd/all (for each mod. tone) or blank
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
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
Enabled
Options...

Action9-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **apply** button.

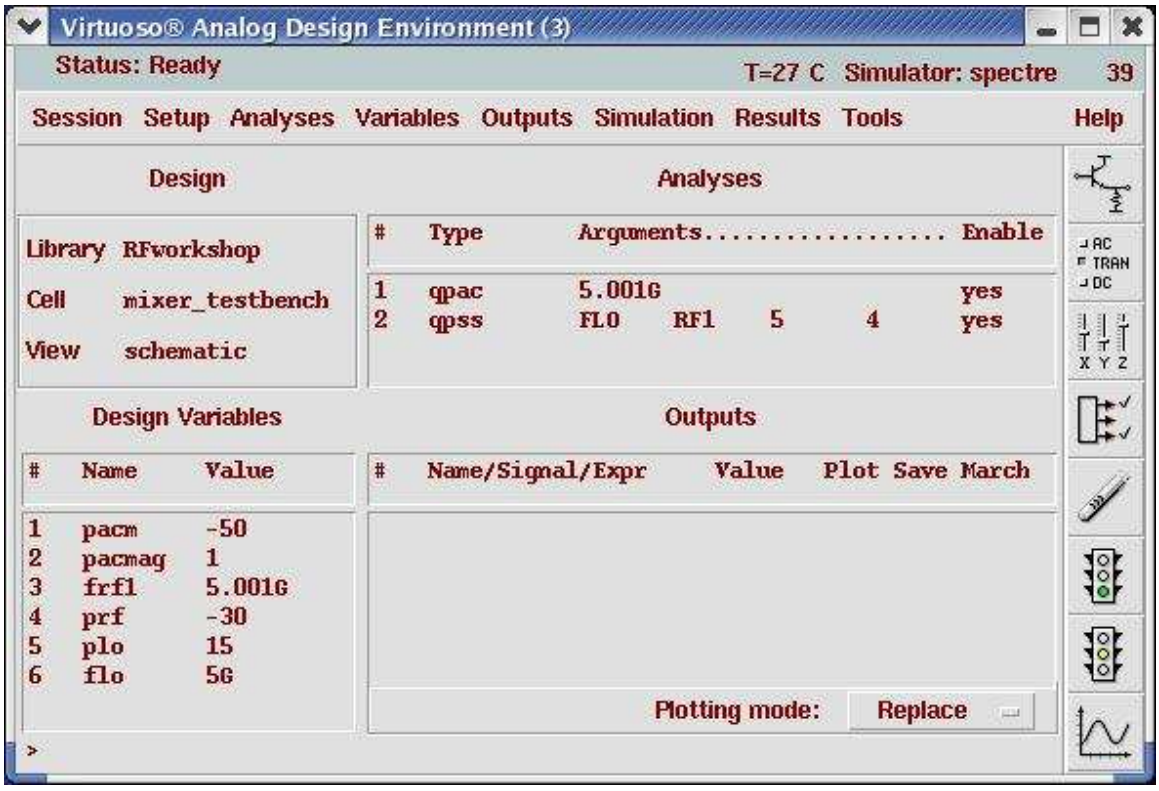
Action9-10: In the Choosing Analyses window, select the **qpac** button in the **Analysis** field of the window. Set the frequency of the small signal very close to f(RF), for example 5.0011 GHz. In the Select from range option of the Sidebands section, highlight the harmonics of interest. Limit the harmonics to second order in the large tone (Set Clock order = 2), from 0 Hz to 6 GHz. The example does not use the 3rd harmonic of the moderate tone, so remove them from the list.

The form should look like this:



Action9-11: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:

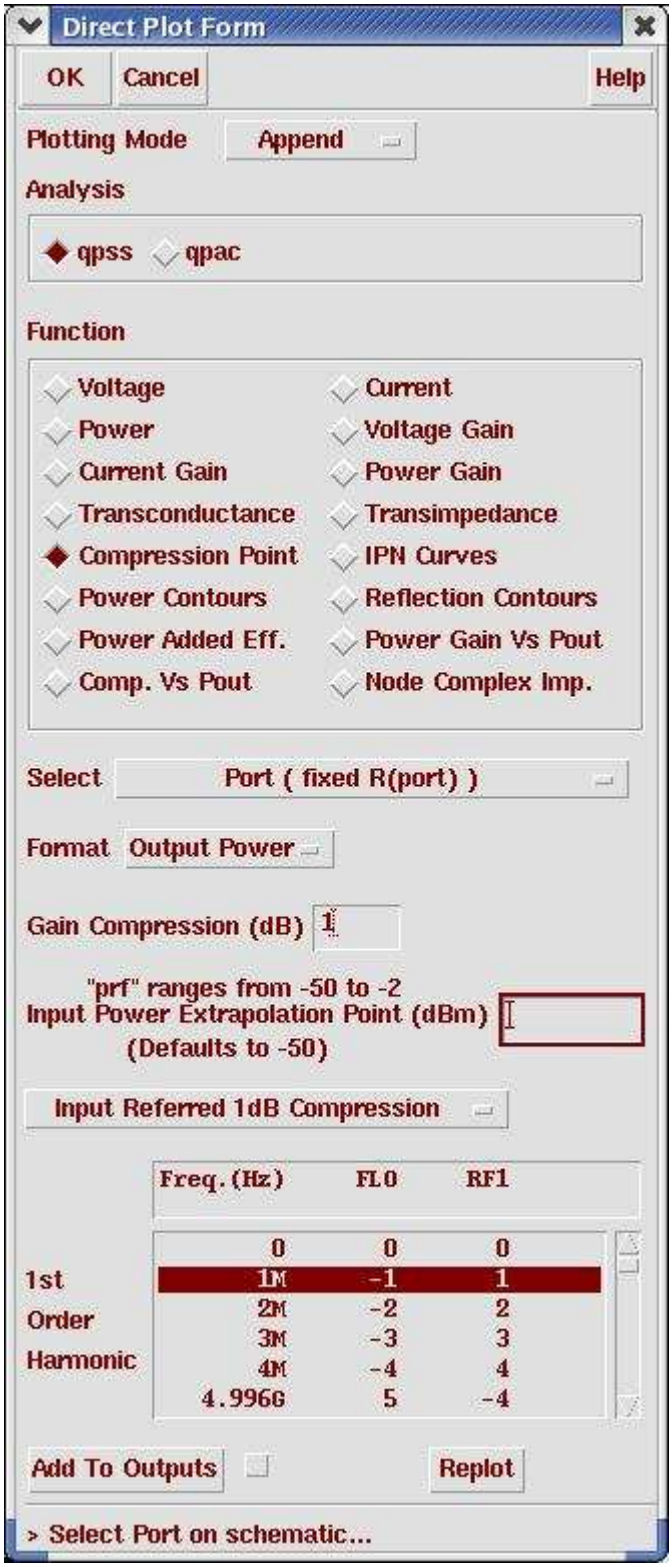


Action9-12: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

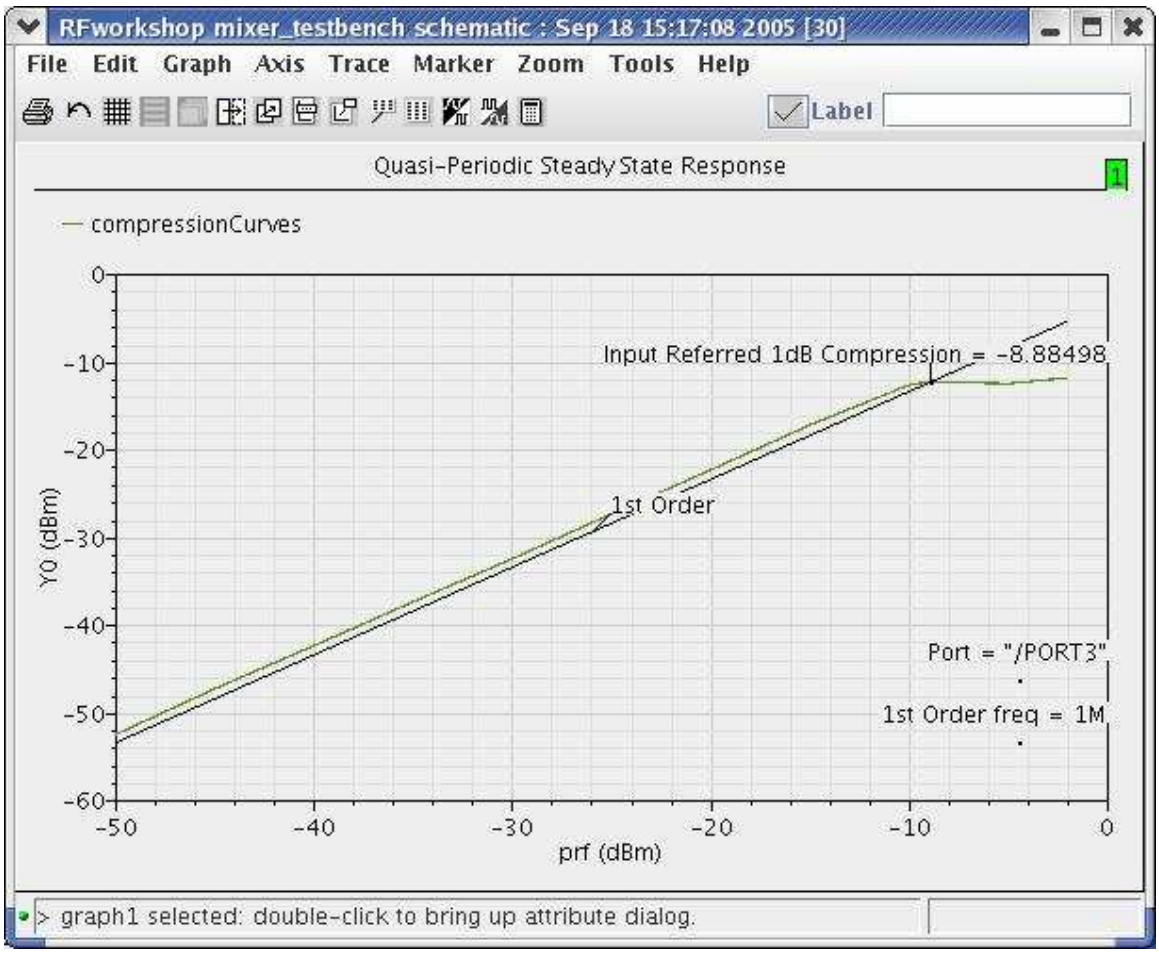
When the simulation completes, use the Direct Plot feature to view the results.

Action9-13: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action9-14: To plot the 1 dB compression point, click **qpss** analysis in Direct Plot form. Select Compression Point. Select a point in the linear region for an extrapolation or leave it blank to use default value. The output harmonic is (-1 1) or 1 MHz.



Action9-15: Select output **Port3** on schematic. You then see the value of P1dB value as shown in follows:

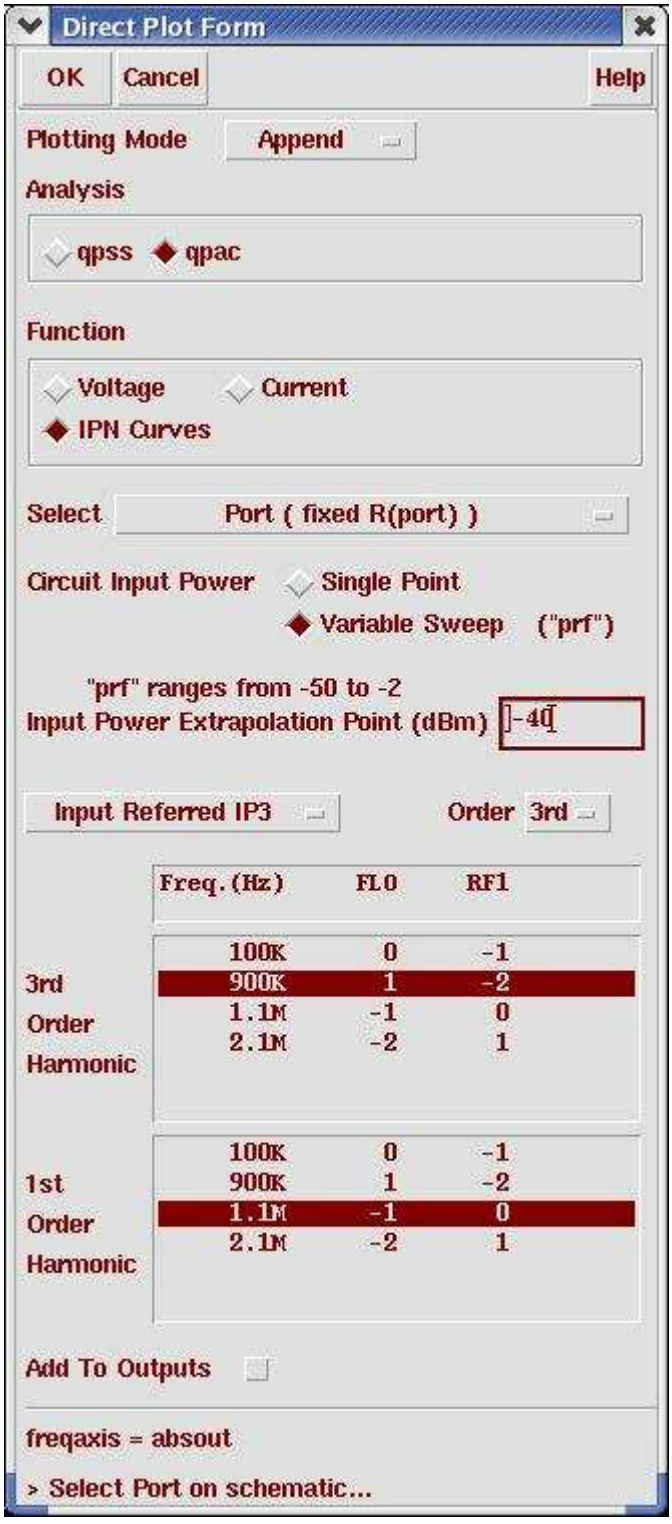


Action9-16: Close the waveform window.

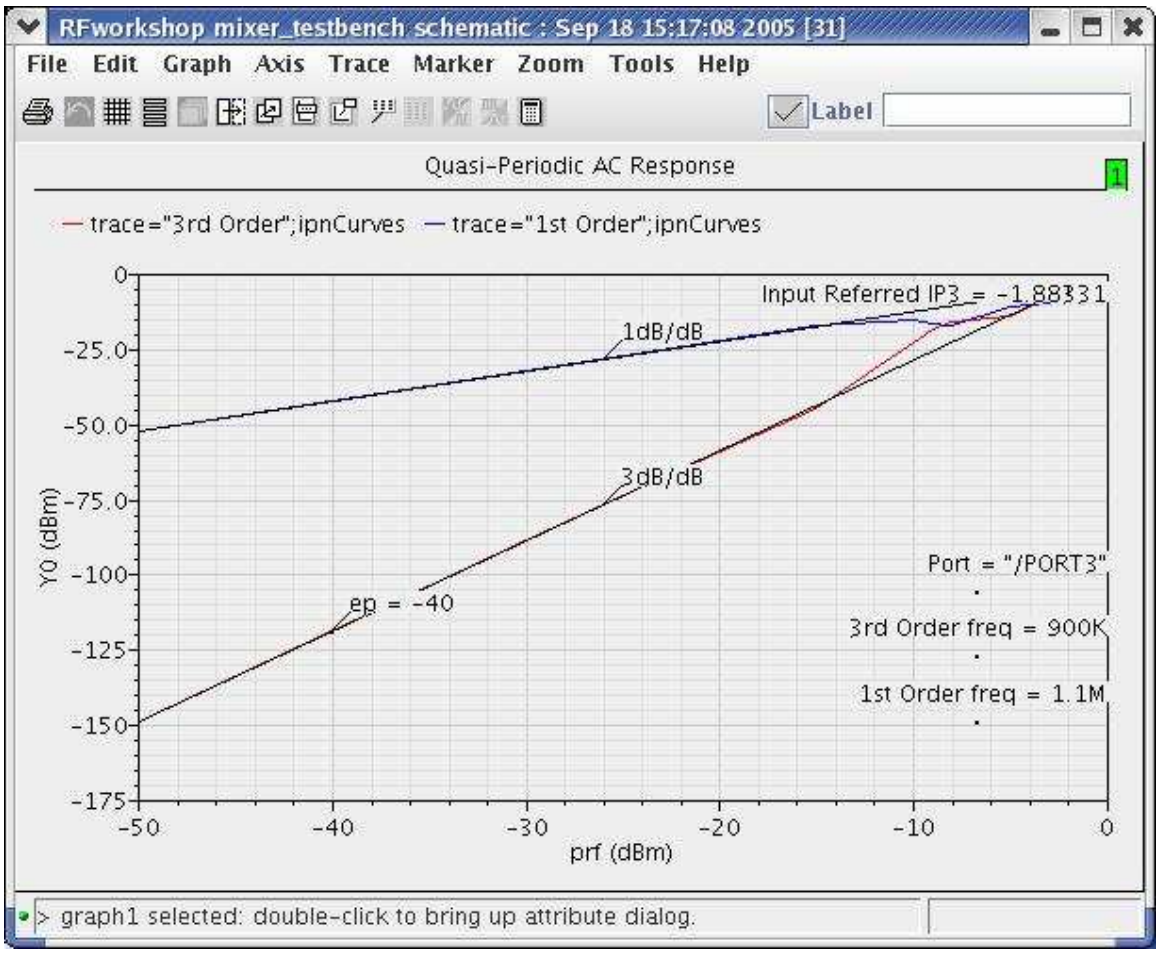
Action9-17: To plot IP3, select **qpac** analysis in the Direct Plot Form, select **IPN Curves** button, select **Variable** sweep and choose **-40** dB for the prf extrapolation. If the first extrapolation point you select is not in the linear range of the IM1 and IM3 curves, you might want to reset 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. Since 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 QPAC Options form.. The form should look like this:



Action9-18: 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 in below:



Action9-19: After viewing the waveforms, click **Cancel** in the Direct Plot form

Note: For more accurate results, you may want to set errpreset = conservative when setting up 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.

Lab10: IP3 Calculation (QPSS with Shooting Engine or Flexible Balance Engine)

To calculate IP3, the other way is to apply the LO and two moderate RF input tones in a single QPSS analyses.

Action10-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action10-2: 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

Action10-3: Click ok on the Edit Object Properties window to close.

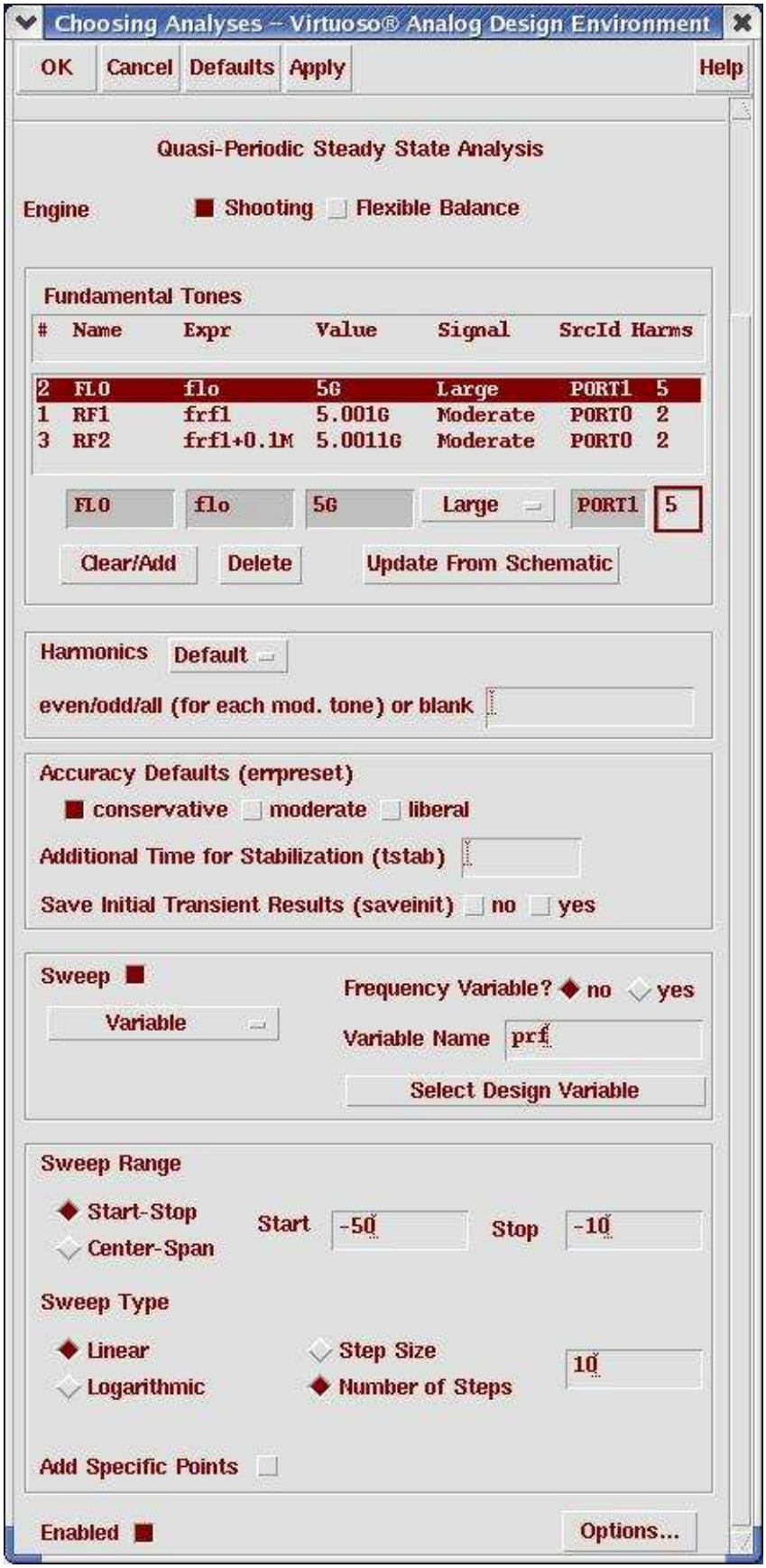
Action10-4: Check and save the schematic.

Action10-5: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action10-6: You can choose **Session—Load State**, load state "**Lab10_IP3_QPSS_shooting**" and skip to Action10-10 or …

Action10-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

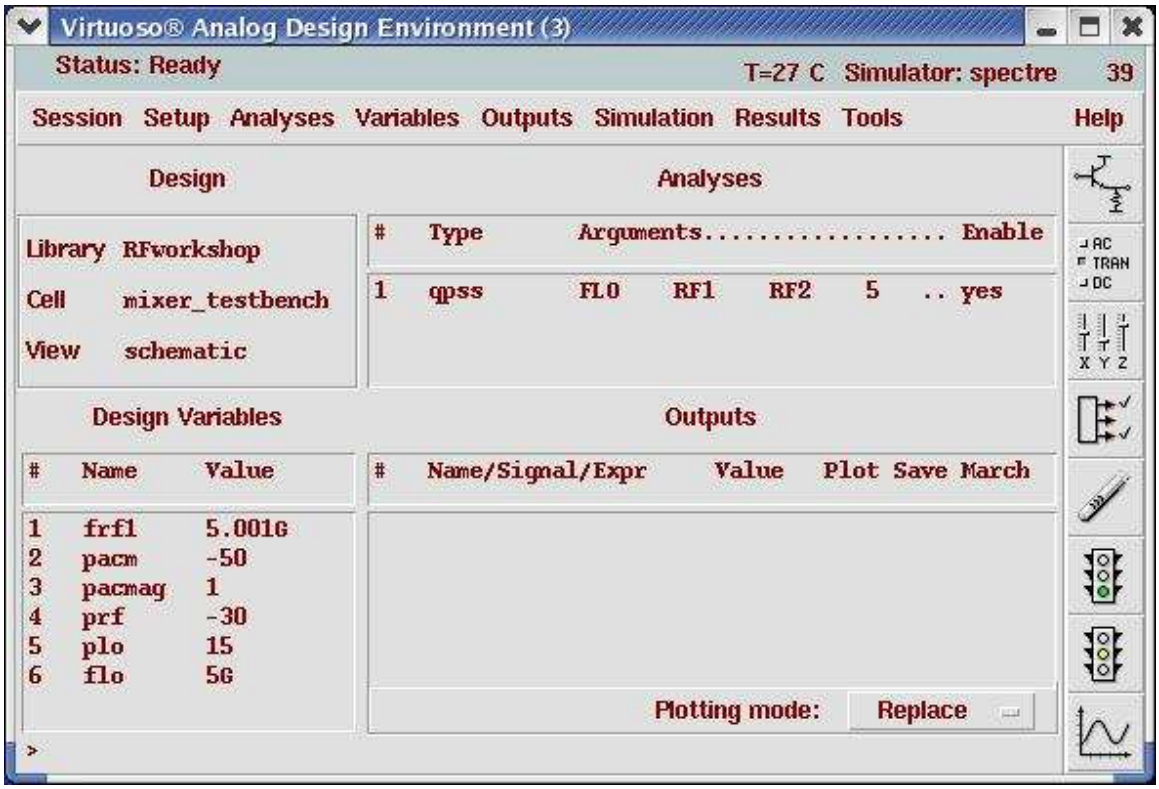
Action10-8: In the Choosing Analyses window, select the **qpss** button in the **Analysis** field of the window and set the form as follows:


Choosing Analyses – Virtuoso® Analog Design Environment
OK
Cancel
Defaults
Apply
Help
Quasi-Periodic Steady State Analysis
Engine
Shooting
Flexible Balance
Fundamental Tones
Name Expr Value Signal SrcId Harms
2 FLO flo 5G Large PORT1 5
1 RF1 frf1 5.001G Moderate PORT0 2
3 RF2 frf1+0.1M 5.0011G Moderate PORT0 2
FLO
flo
5G
Large
PORT1
5
Clear/Add
Delete
Update From Schematic
Harmonics
Default
even/odd/all (for each mod. tone) or blank
Accuracy Defaults (errpreset)
conservative
moderate
liberal
Additional Time for Stabilization (tstab)
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
10
Add Specific Points
Enabled
Options...

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

Action10-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:

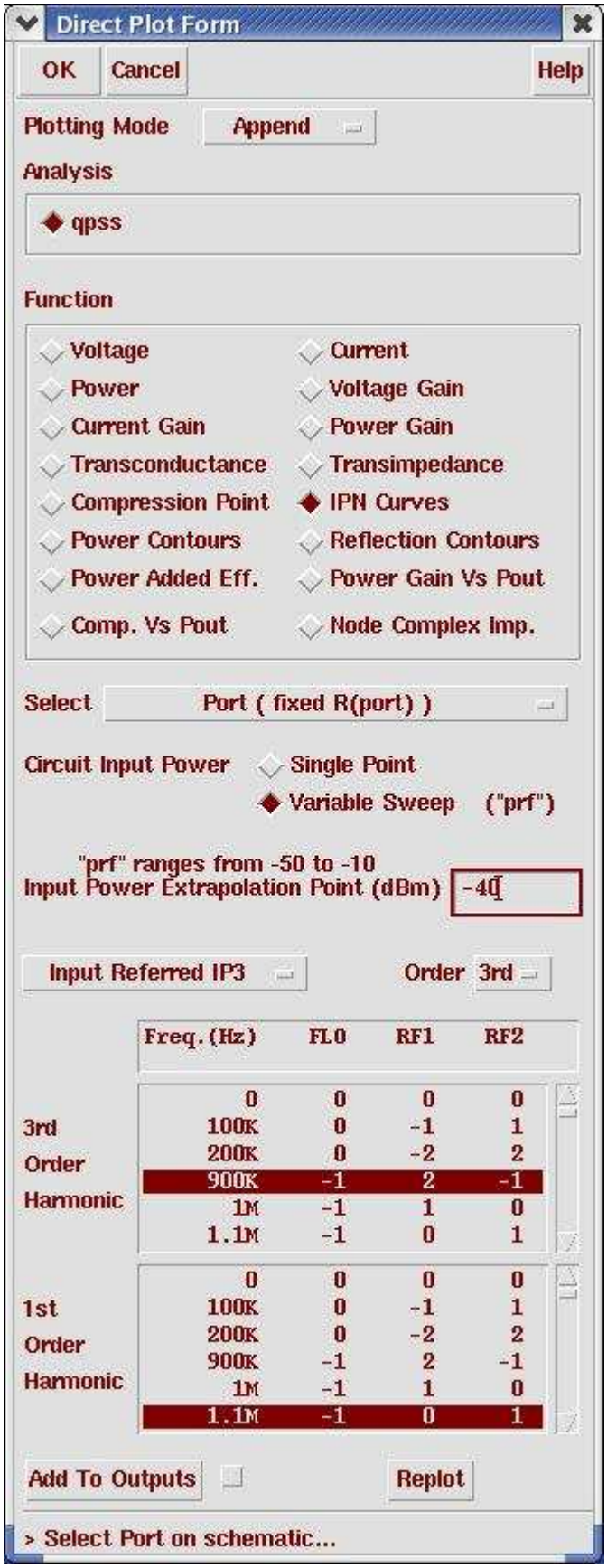


Action10-10: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

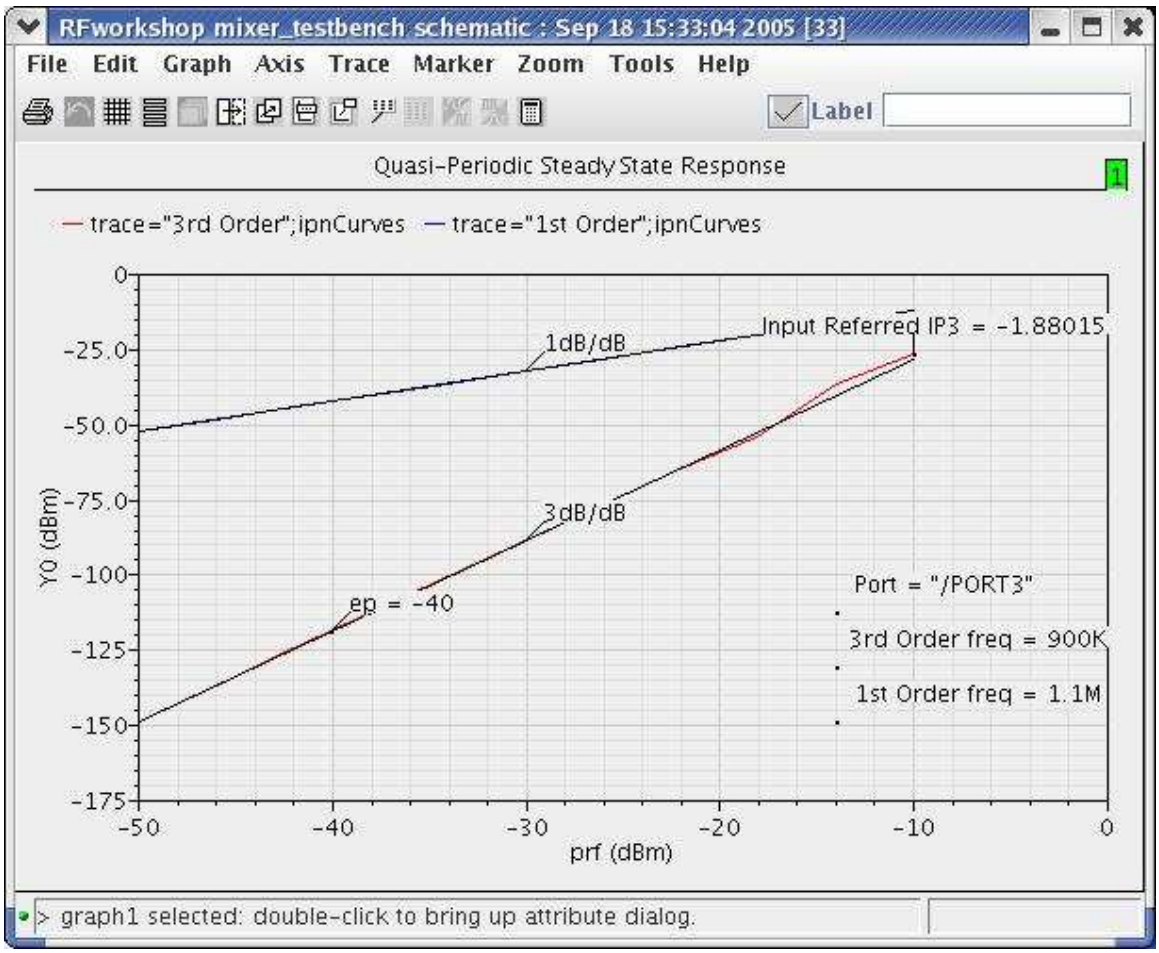
After the simulations finish, plot the IP3 and compare it with the results from Lab9 (QPSS plus QPAC simulations)

Action10-11: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action10-12: In the Direct Plot Form, select the **qpss** button, and configure the form as follows:


Direct Plot Form
OK
Cancel
Help
Plotting Mode
Append
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...

Action10-13: Select output **Port3** on schematic. The IP3 calculation results should look like this:



Action10-14: After viewing the waveforms, click **Cancel** in the Direct Plot form

Action10-15: You can choose **Session—Load State**, load state "**Lab10_IP3_QPSS_FB**" and skip to Action10-22 or …

Action10-16: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

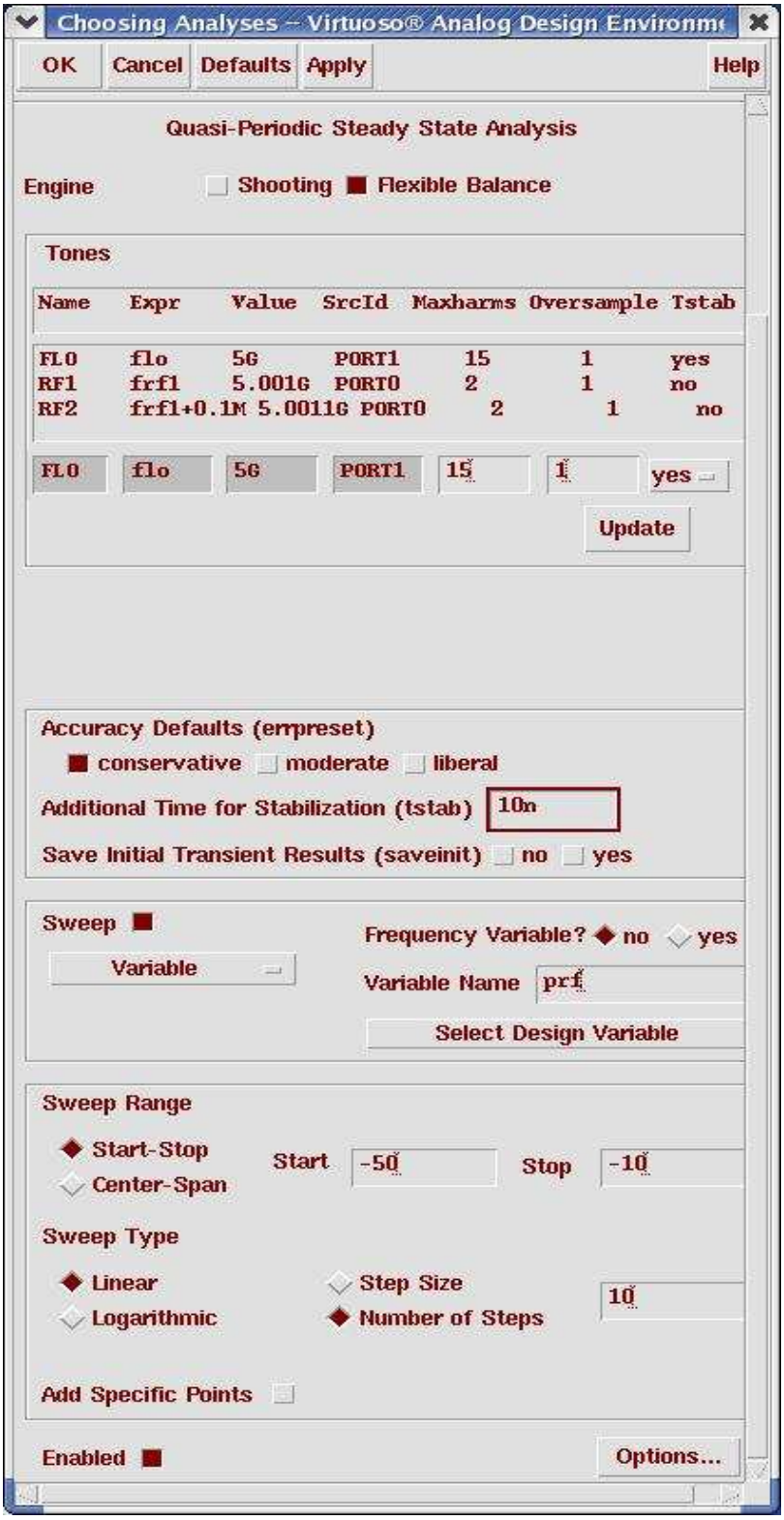
Action10-17: In the Choosing Analyses window, select the **qpss** button in the **Analysis** field of the window.

Action10-18: In **Engine** filed, choose **Flexible Balance**.

Action10-19: In field of Function Tones, choose FLO. Change the Maxharms to 15, because Flexible balance algorithmic need more harmonics to calculate. Click Update.

Action10-20: Put **10n** in the field of **Additional Time for Stabilization (stab)**.

The form should look like this:



Action10-21: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Note: The following parameters are required by HB method:

1. **Flexible Balance Flag:** Harmonic balance engine shares the same PSS/QPSS statement with time-domain engine and is invoked by setting flag **flexbalance**=yes in the analysis statement. A toggle button is available for user to switch between time domain shooting and HB in ADE PSS and QPSS set up form.

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

A proper choice of 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, since the large LO tone has higher power level than moderate RF tones and causes more nonlinear effects, usually more harmonics are used for large tone than moderate tones.

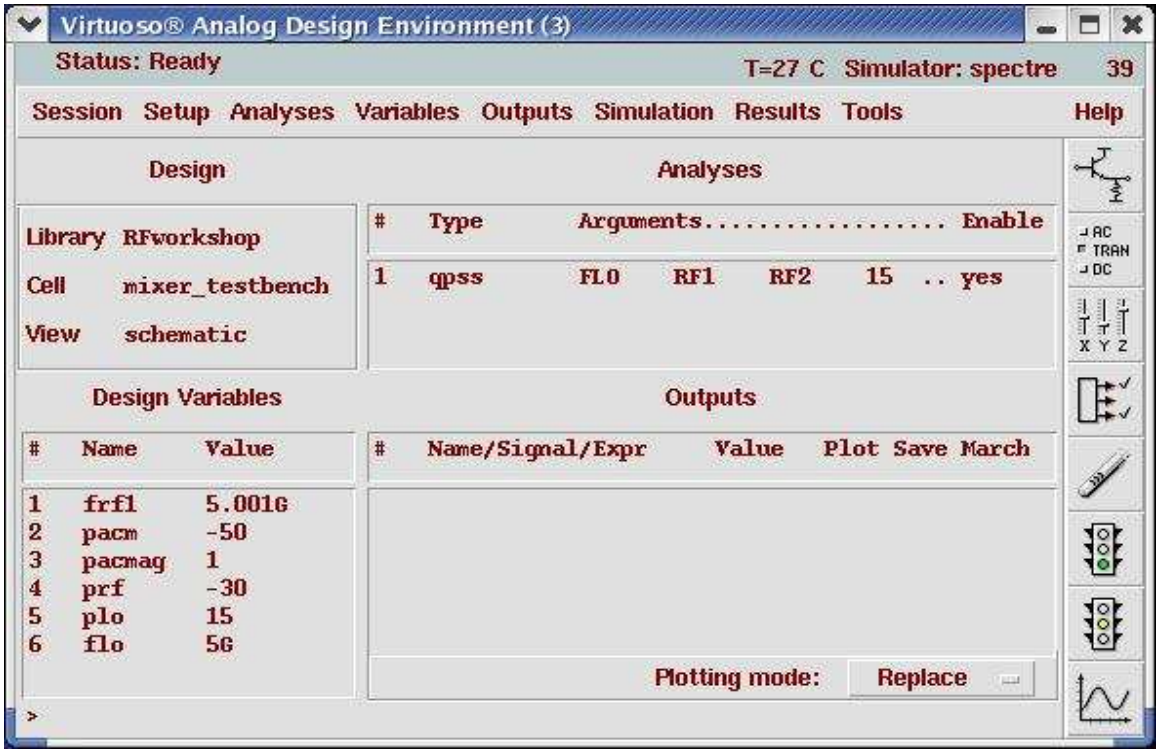
Maximum harmonic also depends on the order of nonlinear effect user wants to study. For example, for mixer IP3 measurement, maxhamrs of moderate tones must be at least set to 2 in order to capture the IM3 mode at frequency $2\omega_1 - \omega_2 - \omega_{LO}$.

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

Oversample factor is optional HB parameter. In PSS/QPSS statement, user can use **oversamplefactor**=m to specify oversample factor. Spectre will oversample and by actor for each tone. As a result, size of Fourier transform is increased by . For multi-tone cases, user can also specify **oversample**=[m1 ,m2, m3, ...] to oversample the tones by factor . Size of Fourier transform is therefore increased by . Default oversample factor is 1.

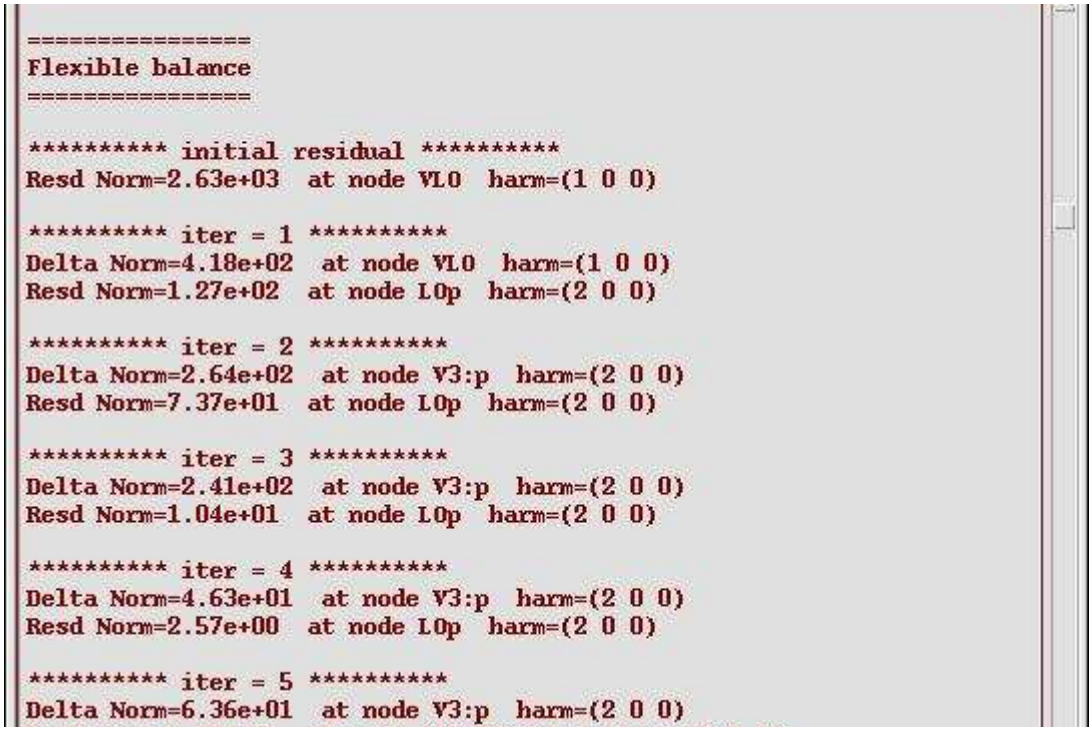
It should be pointed out that even with oversampling, user must make sure maximum harmonic meets the accuracy requirement on results he is interested in.

Now your Virtuoso Analog Design Environment will look like:



Action10-22: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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



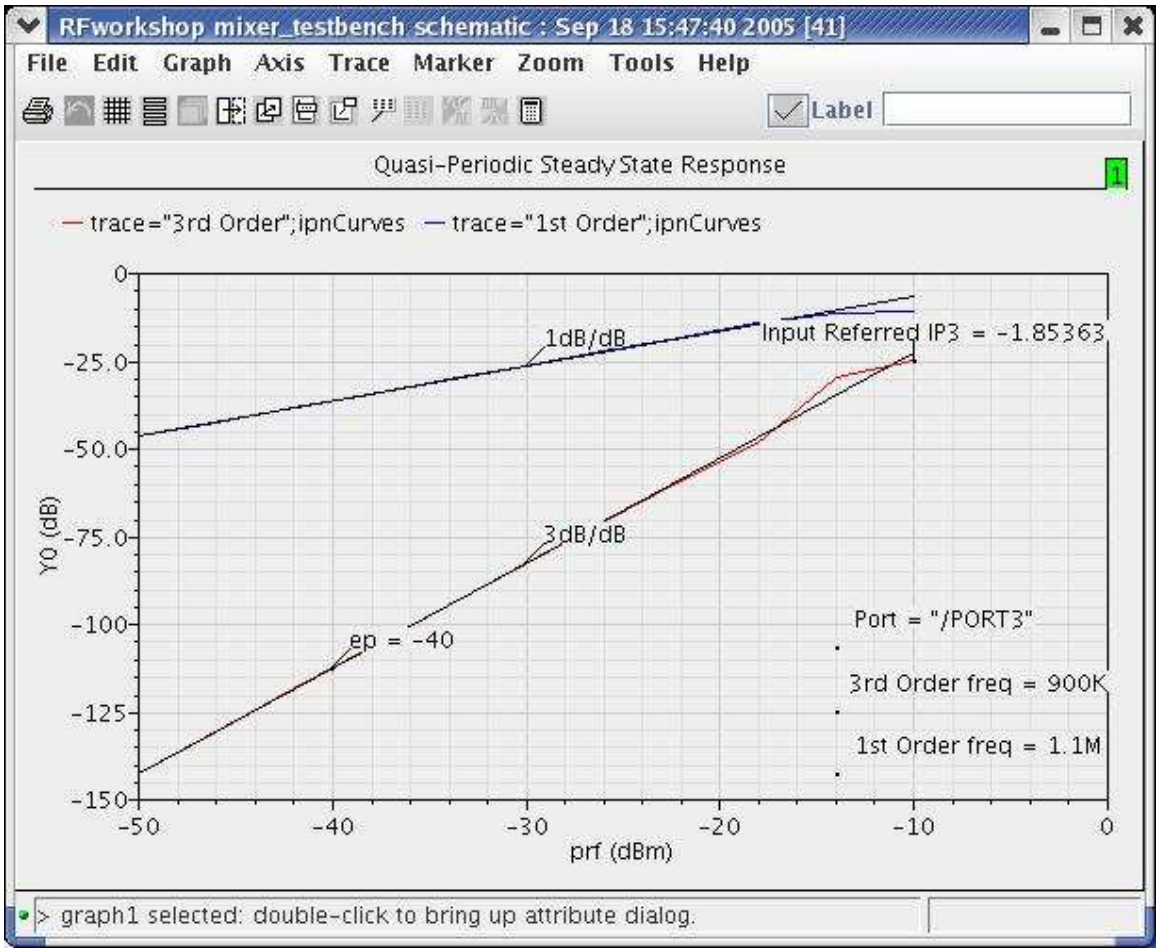
After the simulations finish, plot the IP3 and compare it with the result from the one simulated with shooting engine.

Action10-23: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action10-24: In the Direct Plot Form, select the **qpss** button, and configure the form the same as for shooting engine.

Action10-25: Select output **Port3** on schematic.

The IP3 calculation results should look like this:



Note: The Flexible Balance engine provides same results as with the shooting engine but in this case 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.

HB method is advantageous over time domain method in handling frequency dependent components. In post-layout circuits with large amount of linear elements, HB also demonstrates better performance than shooting. However PSS HB is also advantageous over time domain for weakly non linear circuits (for strongly non linear circuits, time domain prevails).

For circuits driven by multi-tone stimulus, it is better to use HB QPSS other than HB PSS.
As 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 beat frequency as 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.

Action10-26: Close the waveform window and click **Cancel** on the Direct Plot form.

Lab11: Rapid IP3 (PAC)

Rapid Ip2/Ip3 based on perturbation technology extends both shooting and flexible balance. It is new feature of MMSIM6.0USR2. Rapid IM (IP2, IP3) calculations are an order of magnitude faster than using flexible balance or shooting alone.

Action11-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action11-2: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action11-3: 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

Action11-4: Click **ok** on the Edit Object Properties window to close it.

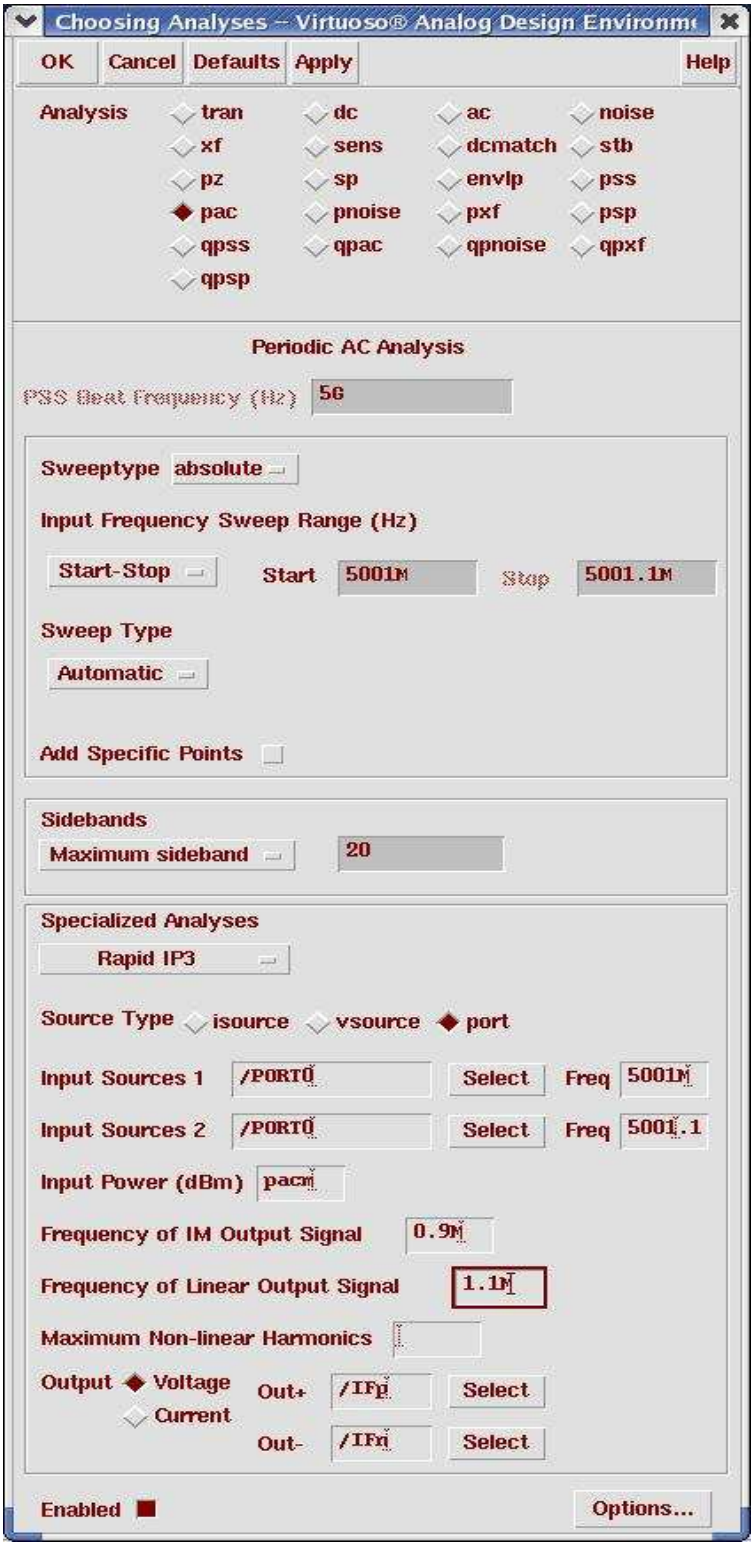
Action11-5: Check and save the schematic.

Action11-6: You can choose **Session—Load State**, load state "**Lab11_Rapid_IP3_PAC**" and skip to Action11-11 or …

Action11-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

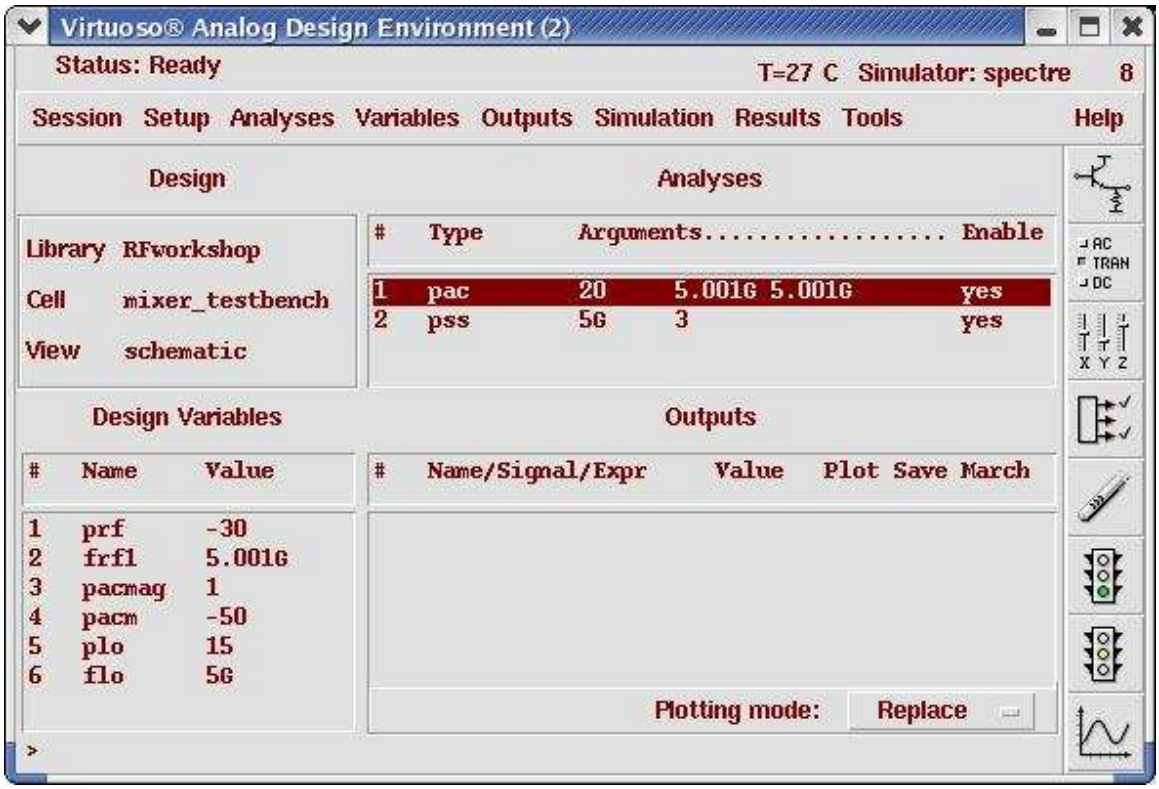
Action11-8: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set the form as shown in Action2-6

Action11-9: 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 Source 1 to /PORT0 by select PORT0 on the schematic. Push the ESC key on your keyboard 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 should look like:



Action11-10: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:

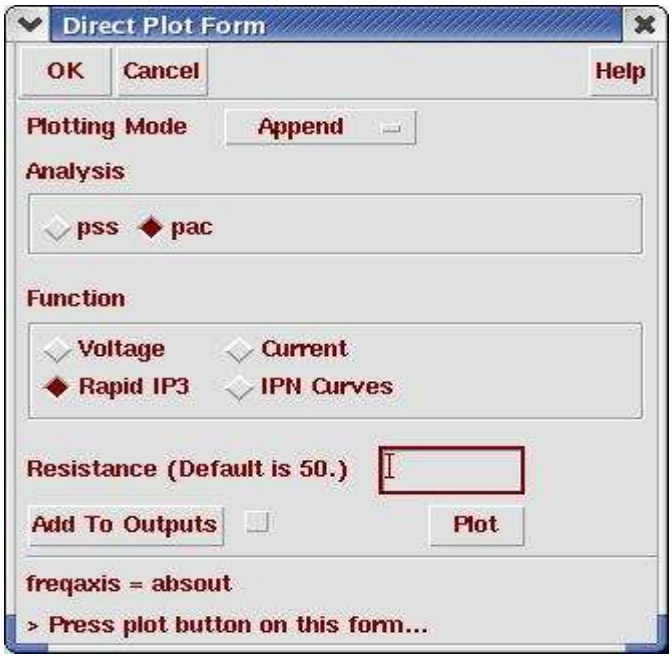


Action11-11: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

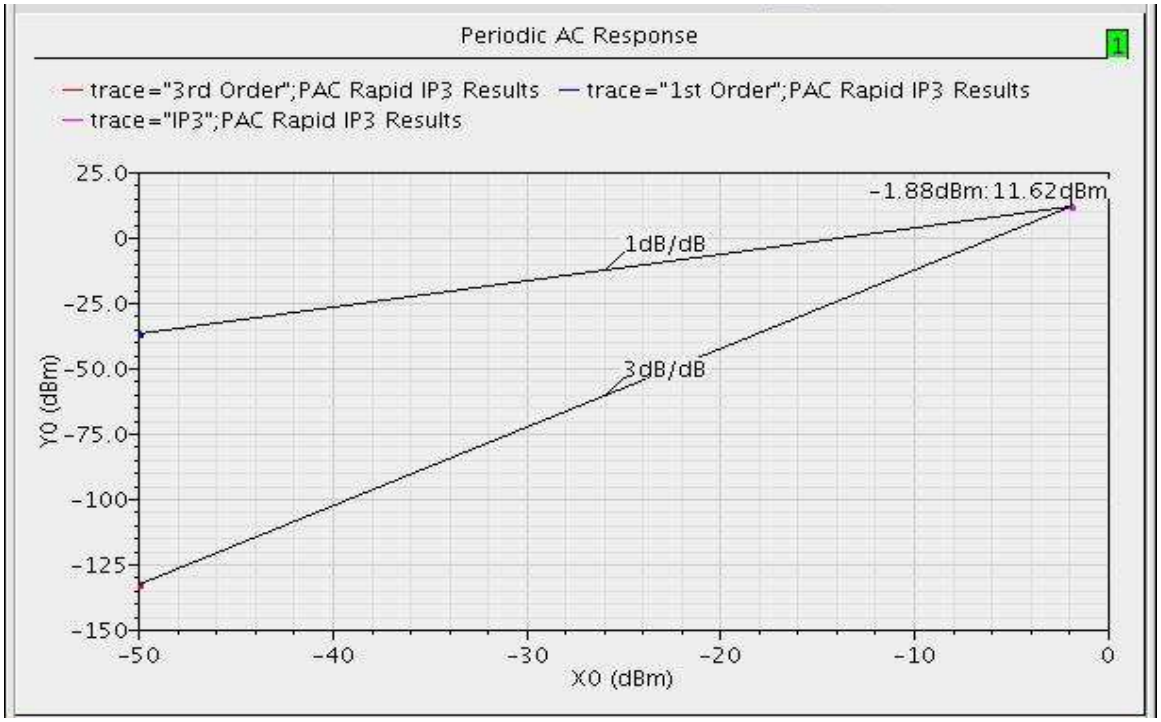
As the simulation progresses, note messages in the simulation output log window that are similar to the following:

```
**********
IP3 measurement `pac'
**********
Input RF1 freq = 5.001000e+09 Hz
Input RF2 freq = 5.001100e+09 Hz
Output IM1 freq = 1.100000e+06 Hz
Output IM3 freq = 9.000000e+05 Hz

Using the operating-point information generated by PSS analysis `pss'.

Notice from spectre during PAC analysis `pac'.
    PORT0: `freq' temporarily changed from 5.001 GHz to 5 GHz to make

Linear output:
f_out = f_in_2 - 1 * fundamental

IM3 output:
f_IM3 = 2 * f_in_1 - f_in_2 - 1 * fundamental
Total time required for pac analysis `pac' was 2.21 s.
```

Action11-12: In the Direct Plot Form, select the **qac** button, and choose **Rapid IP3**. The form looks like:



Action11-13: Click on Plot Button. The calculated IP3 appears in the waveform window:



Action11-14: Close the waveform window and click cancel on the Direct Plot form..

Lab12: Compression Distortion Summary (PAC)

Action12-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action12-2: Select the **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

Action12-3: Click **ok** on the Edit Object Properties window to close it.

Acrion12-4: Check and save the schematic.

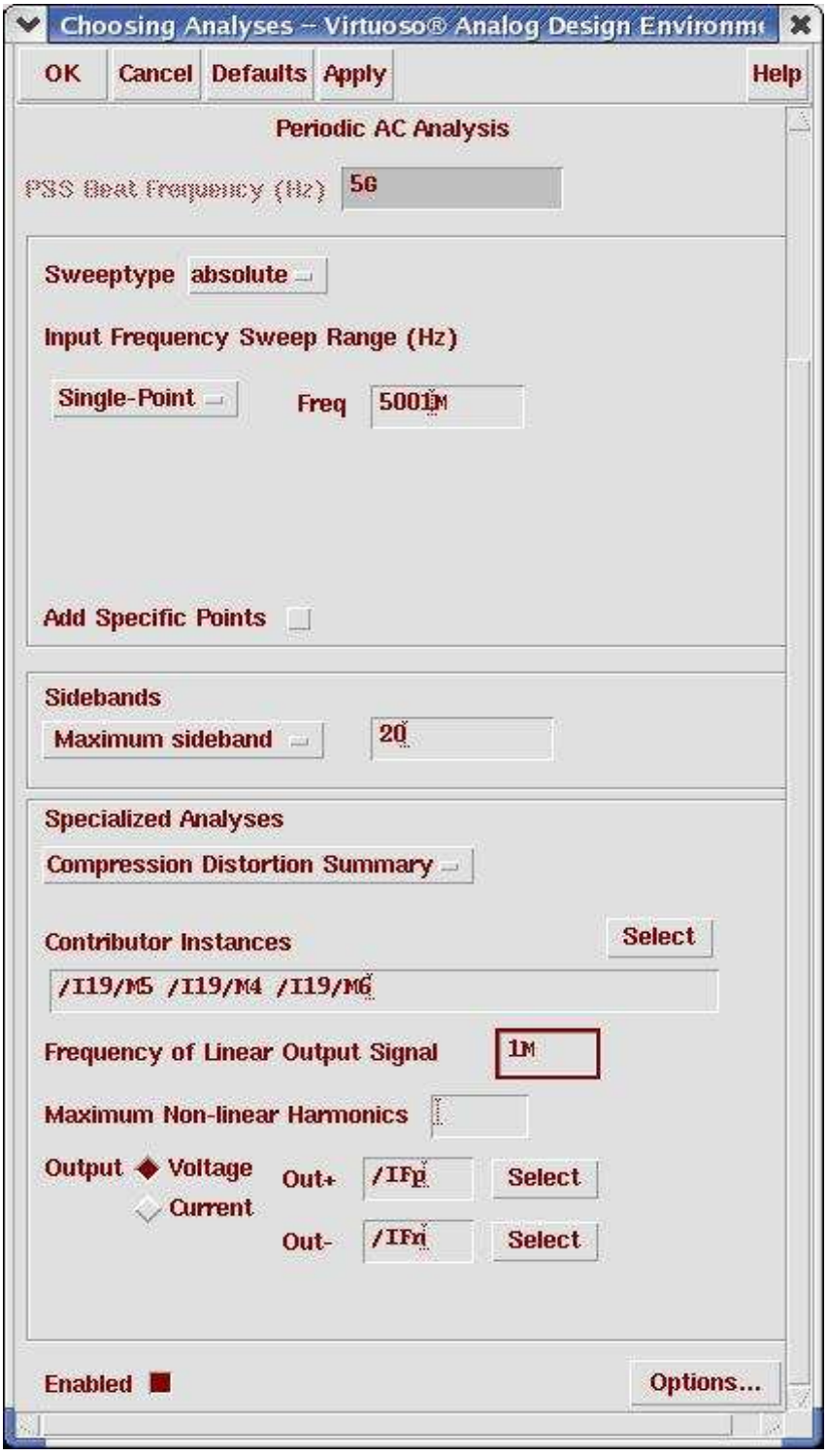
Action12-5: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog Environment** command.

Action12-6: You can choose **Session—Load State**, load state "**Lab12_CompDistorSmry_PAC**" and skip to Action12-11 or …

Action12-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action12-8: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set the form as shown in Action2-6.

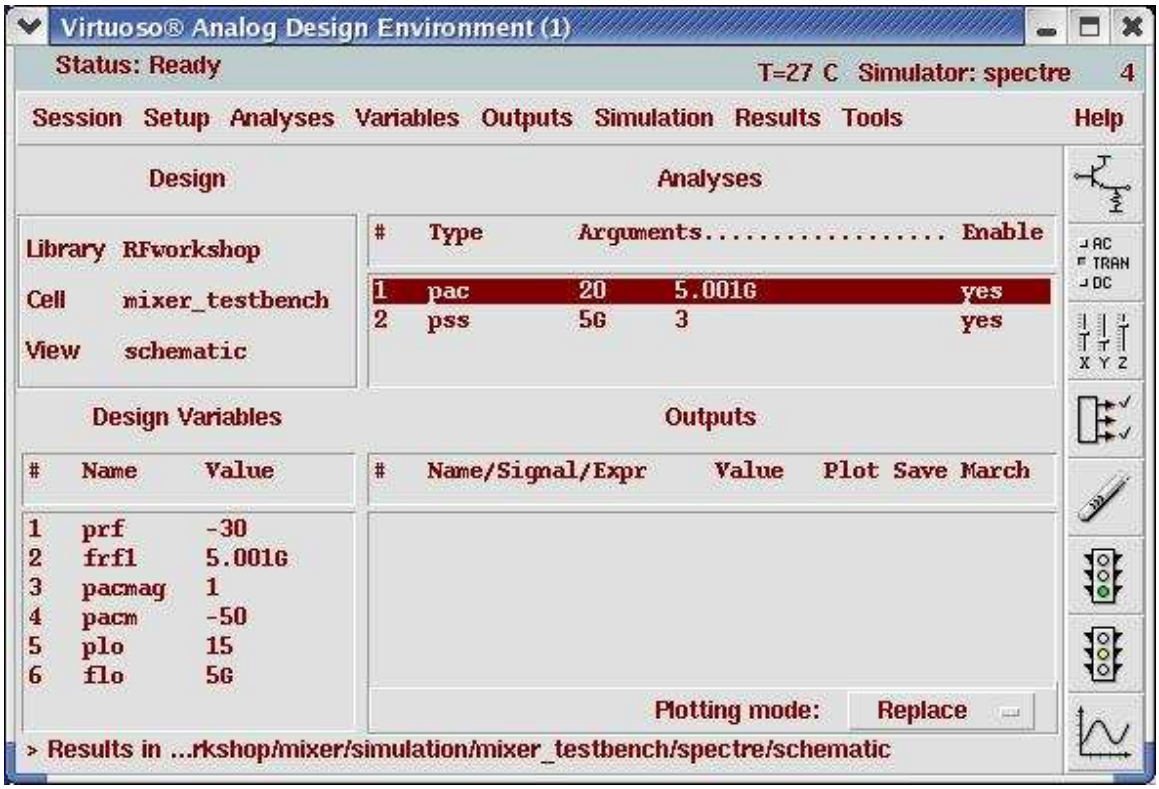
Action12-9: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window and set the form as follows:



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

Action12-10: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:



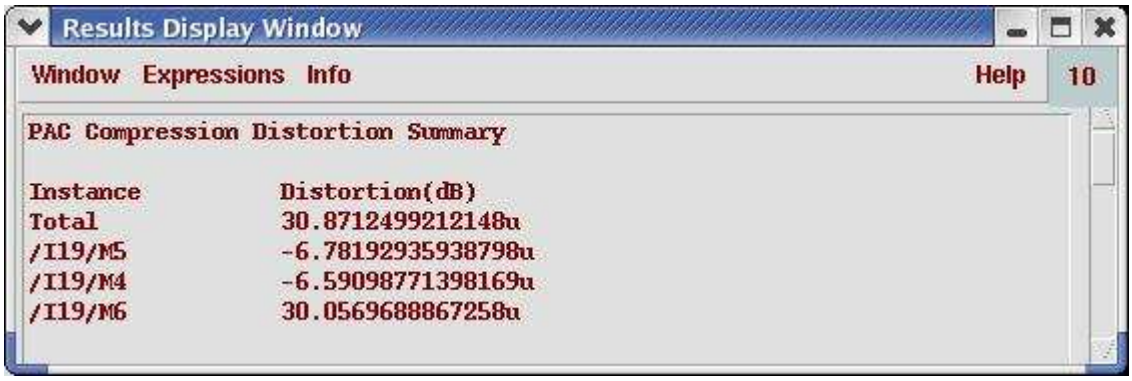
Action12-11: In your Analog Design Environment, choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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

```
************************************
Compression Distortion summary `pac'
************************************
Input freq = 5.001000e+09 Hz
Output freq = 1.000000e+06 Hz

Using the operating-point information generated by PSS analysis `pss'.
f_out = f_in - 1 * fundamental
Calculating non-linearity from all non-linear devices
Calculating non-linearity from I19.M5
Calculating non-linearity from I19.M6
Calculating non-linearity from I19.M4
Total time required for pac analysis `pac' was 810 ms.
```

Action12-12: After the simulation completes, go to Virtuoso Analog Design Environment, choose **Results—Print—PAC Distortion Summary.**

Results Display Window shows up:



Action12-13: After viewing the results, close it by selecting **Window—Close**.

Lab13: Rapid IP2 (PAC)

Action13-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action13-2: Select the **PORT0** source by placing the mouse cursor over it and clicking the left mouse button. Then in the Virtuoso Schematic Editor select **Edit—Properties—Objects…** The Edit Object Properties window for the port cell should come up. Set the source type to sine.

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

Action13-3: Click **ok** on the Edit Object Properties window to close it.

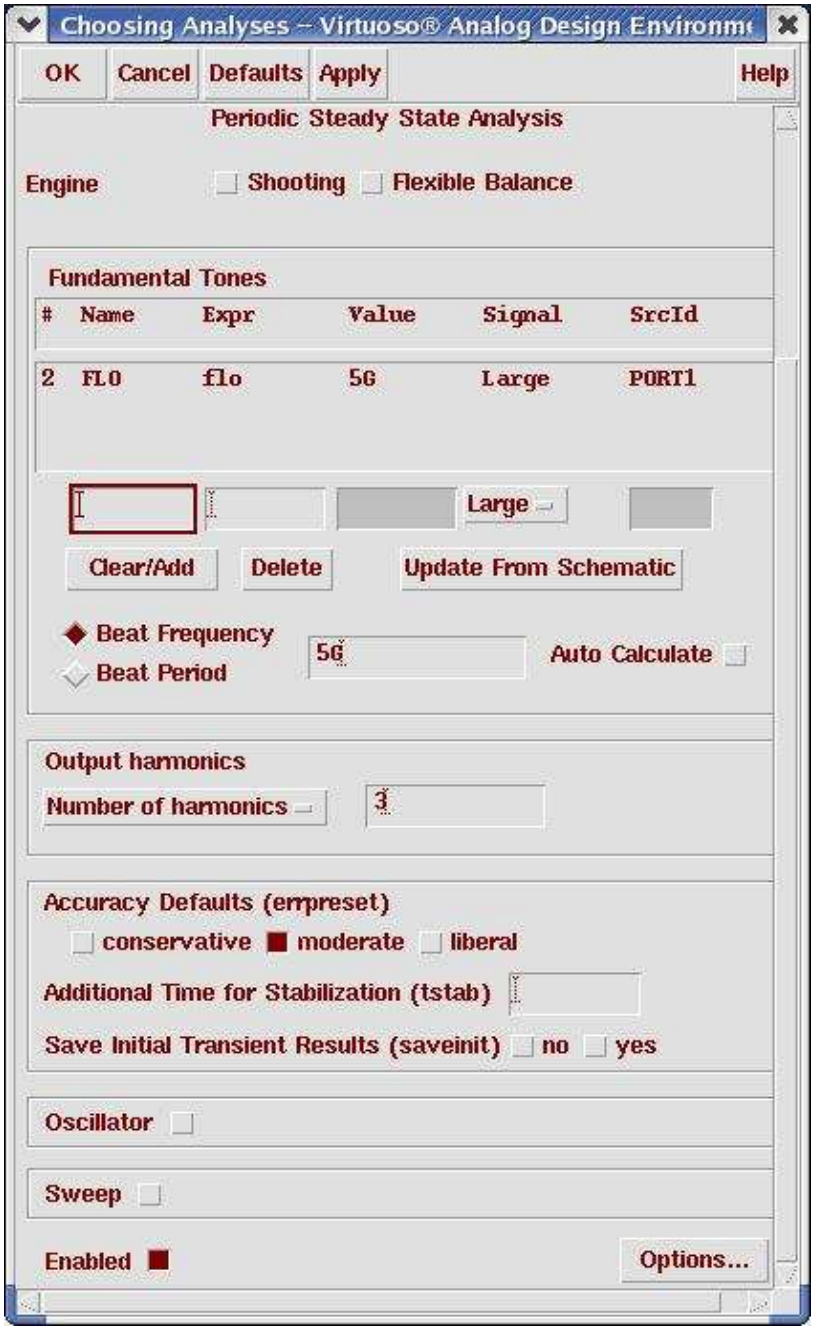
Action13-4: check and save the schematic.

Action13-5: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action13-6: You can choose **Session—Load State**, load state "**Lab13_Rapid_IP2_PAC**" and skip to Action13-12 or …

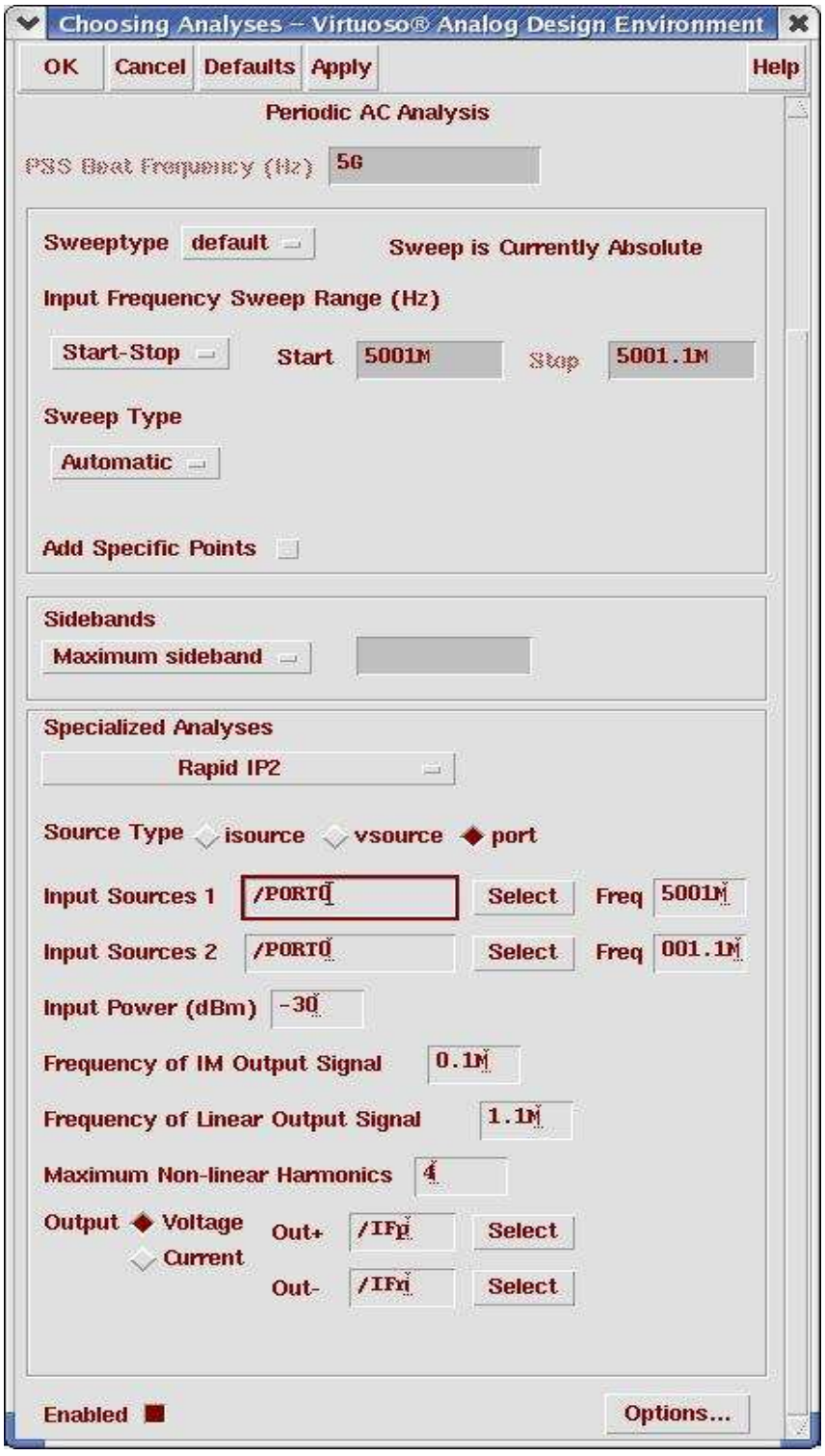
Action13-7: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action13-8: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set the form as follows:



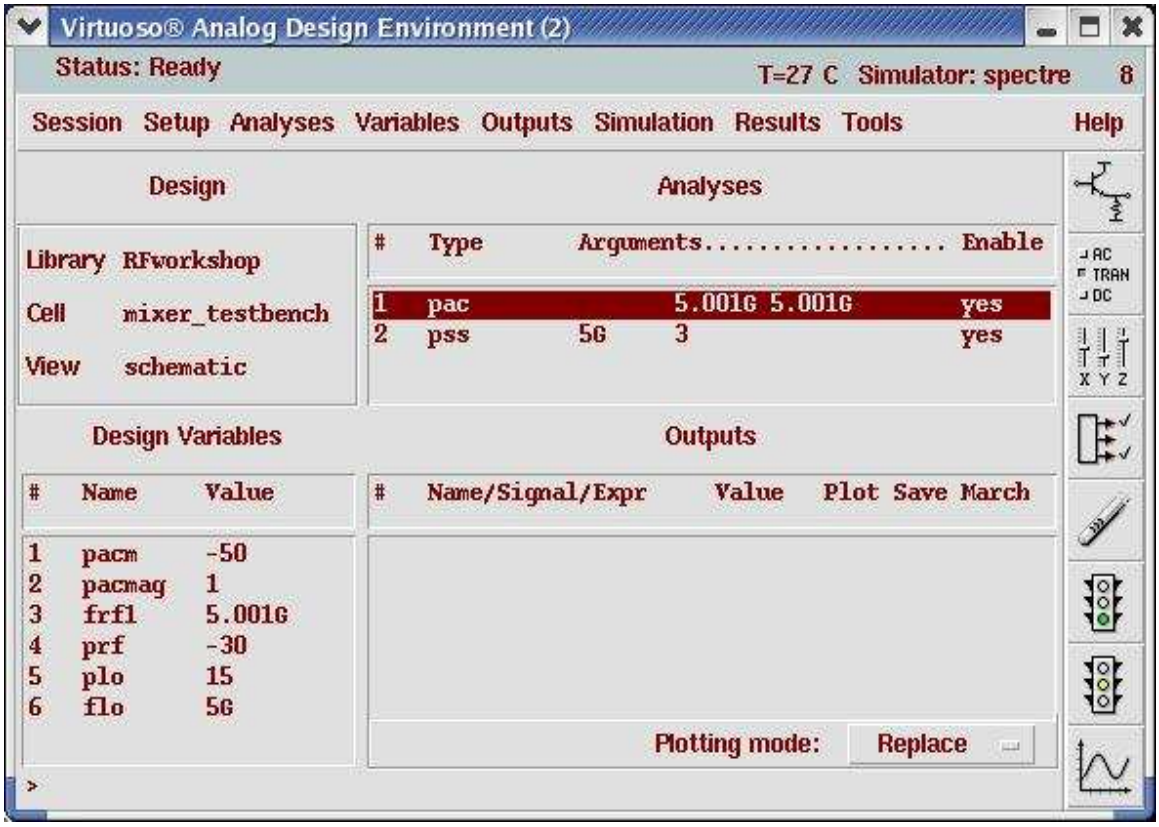
Action13-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **apply** button.

Action13-10: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window. In the field of Specialized Analyses, choose rapid IP2. Set the Input Source 1 to /PORT0 by select PORT0 on the schematic. Push the ESC key on your keyboard 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 should look like:



Action13-11: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:



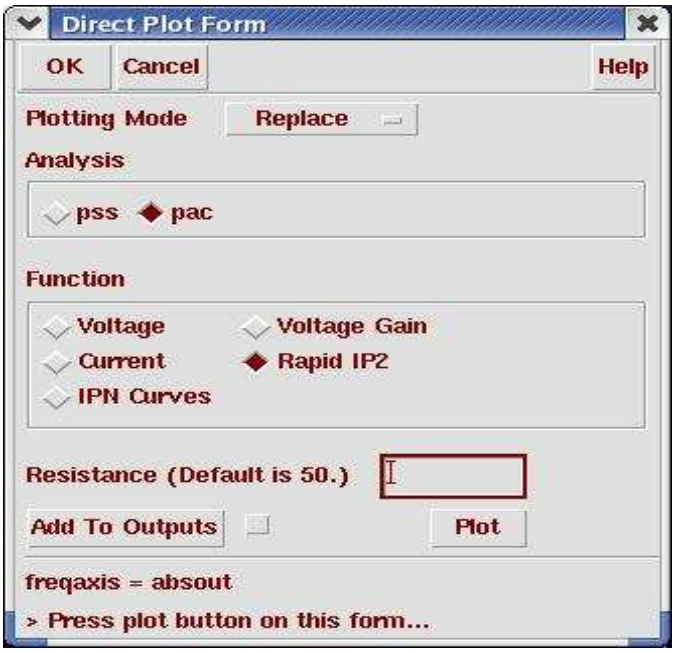
Action13-12: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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

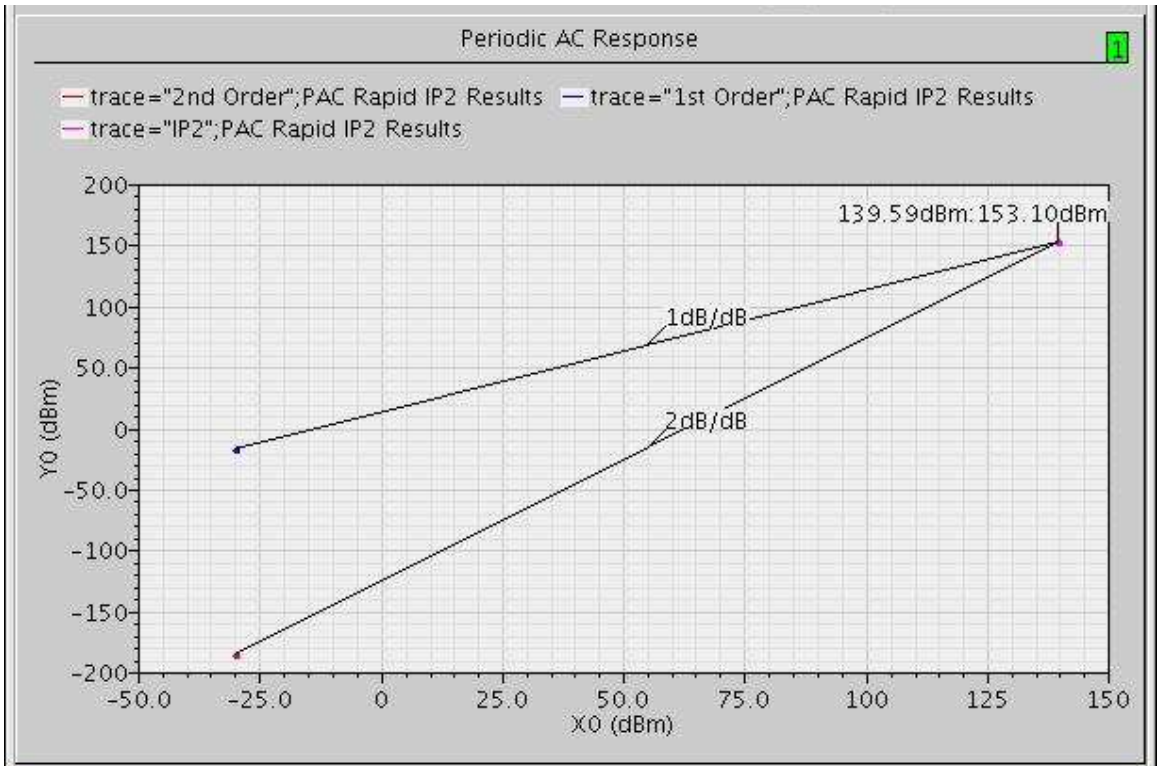
```
*********************
IP2 measurement `pac'
*********************
Input RF1 freq = 5.001000e+09 Hz
Input RF2 freq = 5.001100e+09 Hz
Output IM1 freq = 1.100000e+06 Hz
Output IM2 freq = 1.000000e+05 Hz

Using the operating-point information generated by PSS analysis `pss'.
Linear output:
f_out = f_in_2 - 1 * fundamental

IM2 output:
f_IM2 = f_in_2 - f_in_1 - 0 * fundamental
Total time required for pac analysis `pac' was 1.09 s.
```

Action13-13: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.

Action13-14: In the Direct Plot Form, select the **pac** button in analysis field and choose **Rapid IP2** in the Function field.



Action13-15: Click **Plot** button.



Action13-16: Close the waveforms window, click **Cancel** in the Direct Plot form.

Lab14: IM2 Distortion Summary (PAC)

Action14-1: If not already open, open the *schematic* view of the *mixer_testbench* design in the library *RFWorkshop*

Action14-2: Make sure the source type of **PORT0** is sine.

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

Action14-3: Check and save the schematic.

Action14-4: From the Mixer_testbench schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.

Action14-5: You can choose **Session—Load State**, load state "**Lab14_IM2DistorSmary_PAC**" and skip to Action14-10 or …

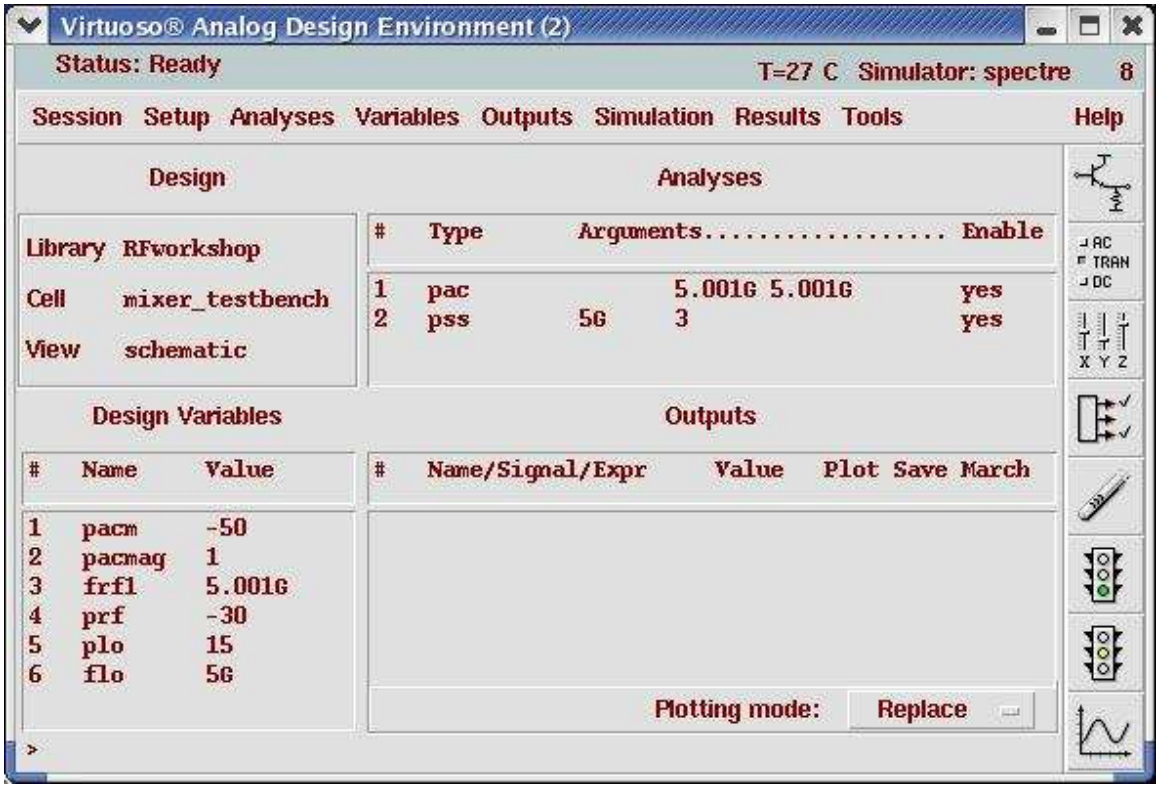
Action14-6: In Vituoso Analog Design Environment, choose **Analyses—Choose…**

Action14-7: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set the form as you did in Rapid IP2 simulation.

Action14-8: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window. In the filed of Specialized Analyses, choose IM2 Distortion Summary. Set the Input Source 1 to /PORT0 by select PORT0 on the schematic. Push the ESC key on your keyboard 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 should look like:

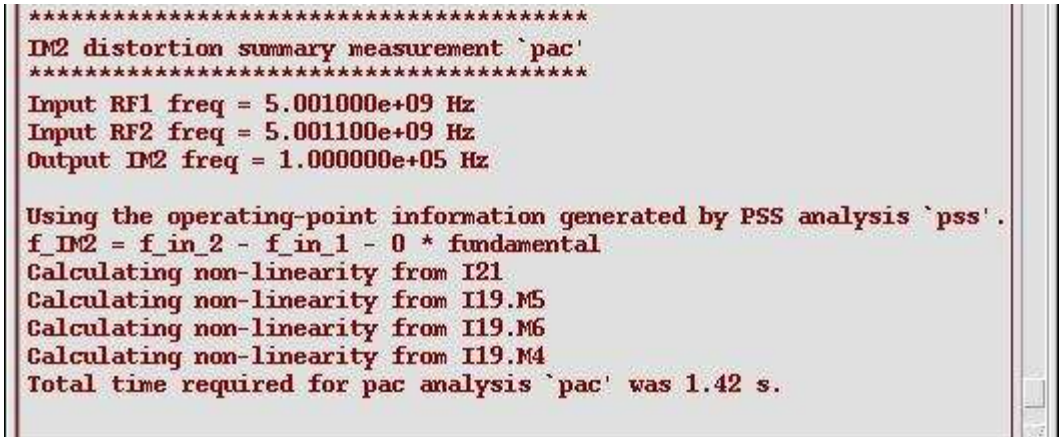
Action14-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:



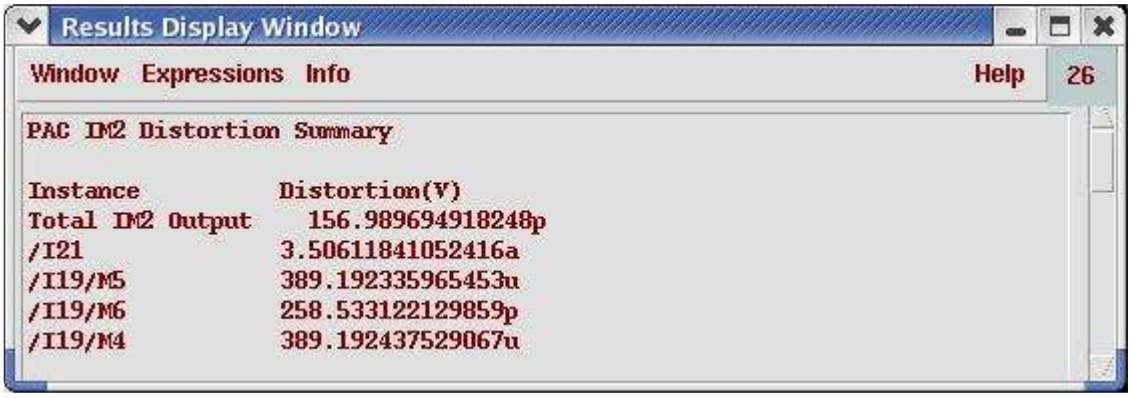
Action14-10: In your Analog Design Environment, Choose **Simulation—Netlist and Run** or click the **Netlist and Run** icon to start the simulation.

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



Action14-11: In the Virtuoso Analog Design Environment, Choose **Results—Print—PAC Distortion Summary.**

The Results Display Window shows the PAC IM2 Distortion Summary.



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

Action14-12: After viewing the distortion summary report, close it by selecting **Window—Close**.

Conclusion

This workshop highlights how to use SpectreRF to simulate a Mixer efficiently and extract design parameters such as IP3, 1dB compression point or port-to-port isolation. Various techniques using PSS, Pnoise, PAC and QPSS analyses are being presented. SpectreRF Flexible Balance and Time domain algorithms accuracies are also demonstrated and compared.

.

Reference

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

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

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

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