1. SpectreRF Overview

- SpectreRF is an optional feature added to Spectre ,and is represented by 6 analyses:
    1. PSS: Periodic Steady State Analysis
    2. PAC: Periodic AC Analysis
    3. PXF: Periodic Transfer Function Analysis
    4. PNOISE: Periodic Noise Analysis

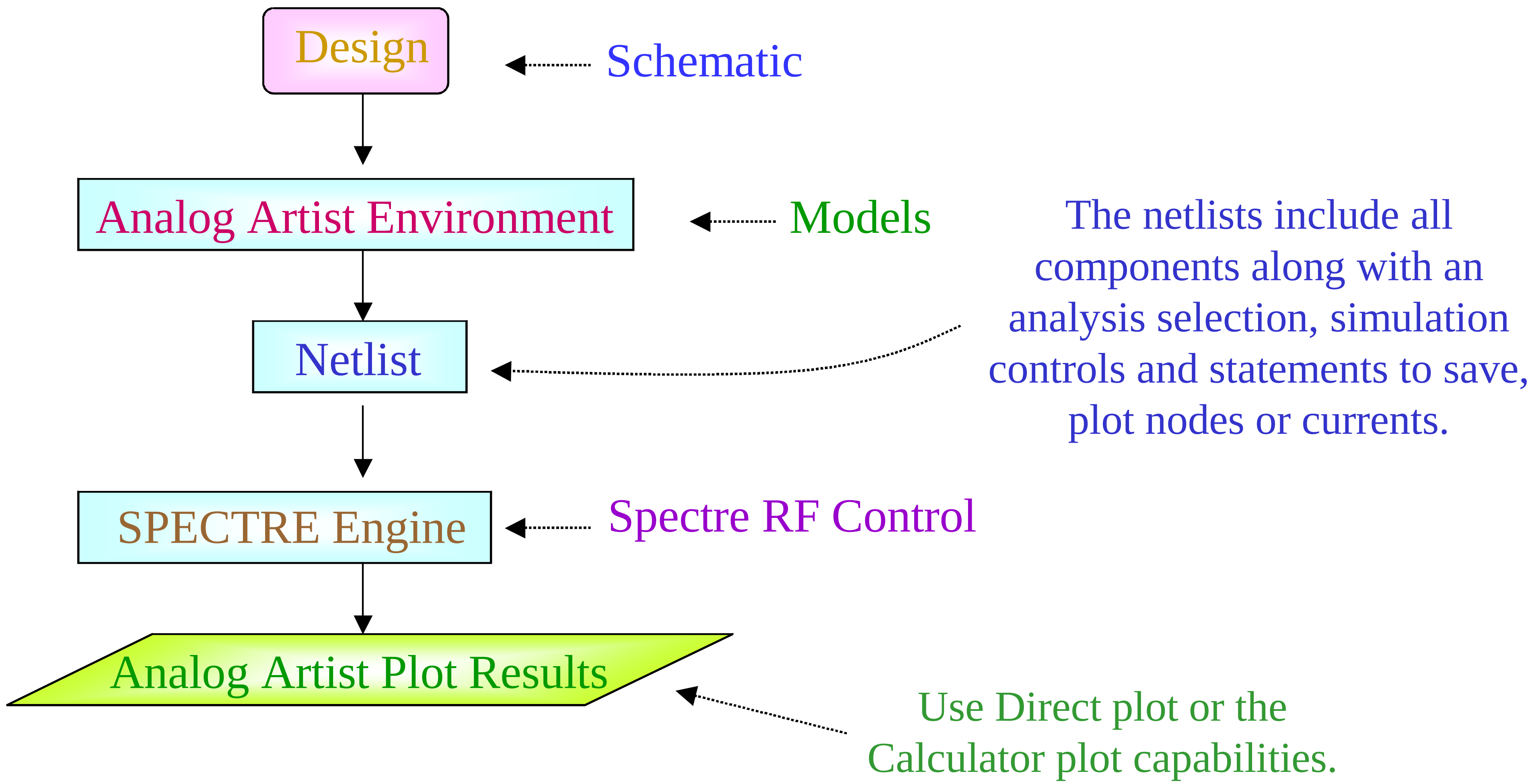
       Tdnoise: Time Domain Noise

       QPNOISE: Quasi-Periodic Noise (not discuss here)
    5. PDISTO: Periodic Distortion Analysis

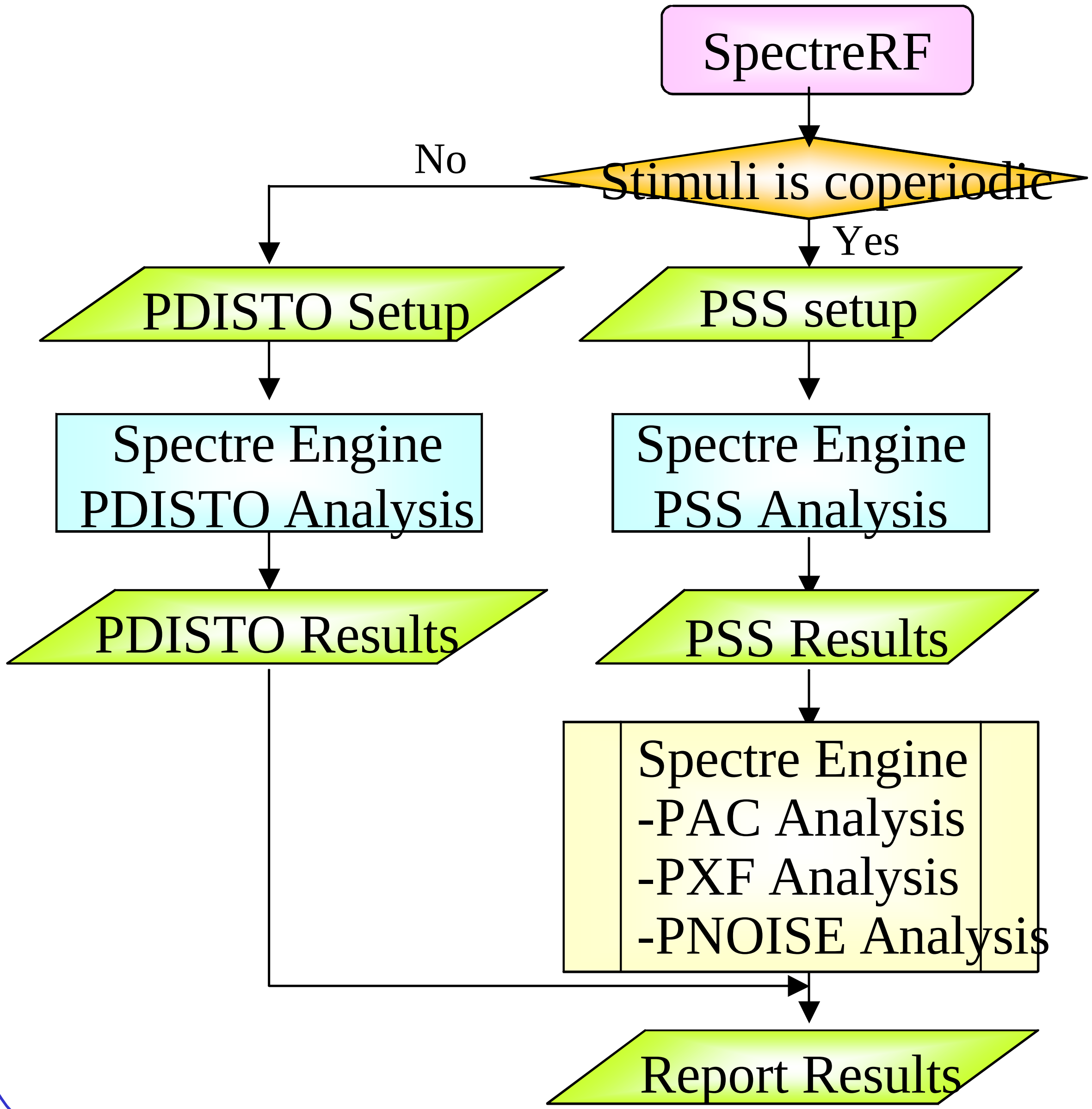
       QPSS: Quasi-Periodic Steady State (not discuss here)
    6. Envelope Analysis (not discuss here)

PAC, PXF, and PNOISE are similar in concept to AC, XF, and Noise. However, they are applied to periodically-driven circuits such as mixers and oscillators.

SpectreRF in a Design Flow



SpectreRF Tool Flow



PSS is a large-signal analysis and determines the period of the small-signal analyses. PSS requires that multiple periodic stimuli be coperiodic.

PDISTO is also a large-signal analysis, and need not to be run after a PSS analysis. PDISTO does not require multiple periodic stimuli to be coperiodic.

SpectreRF Features

- Compute a steady-state solution efficiently and directly
- Handles very large circuits (~ 10,000 transistors)
- Displays results in both time and frequency domains
- Use Discrete Fourier Transform (DFT) for better accuracy
- Displays standard RF measurements, such as s-parameter in Smith chart, NF, IP3, and 1dB compression point in the Analog Artist design environment.
- Performs oscillator analysis.

2. S-Parameter Analysis

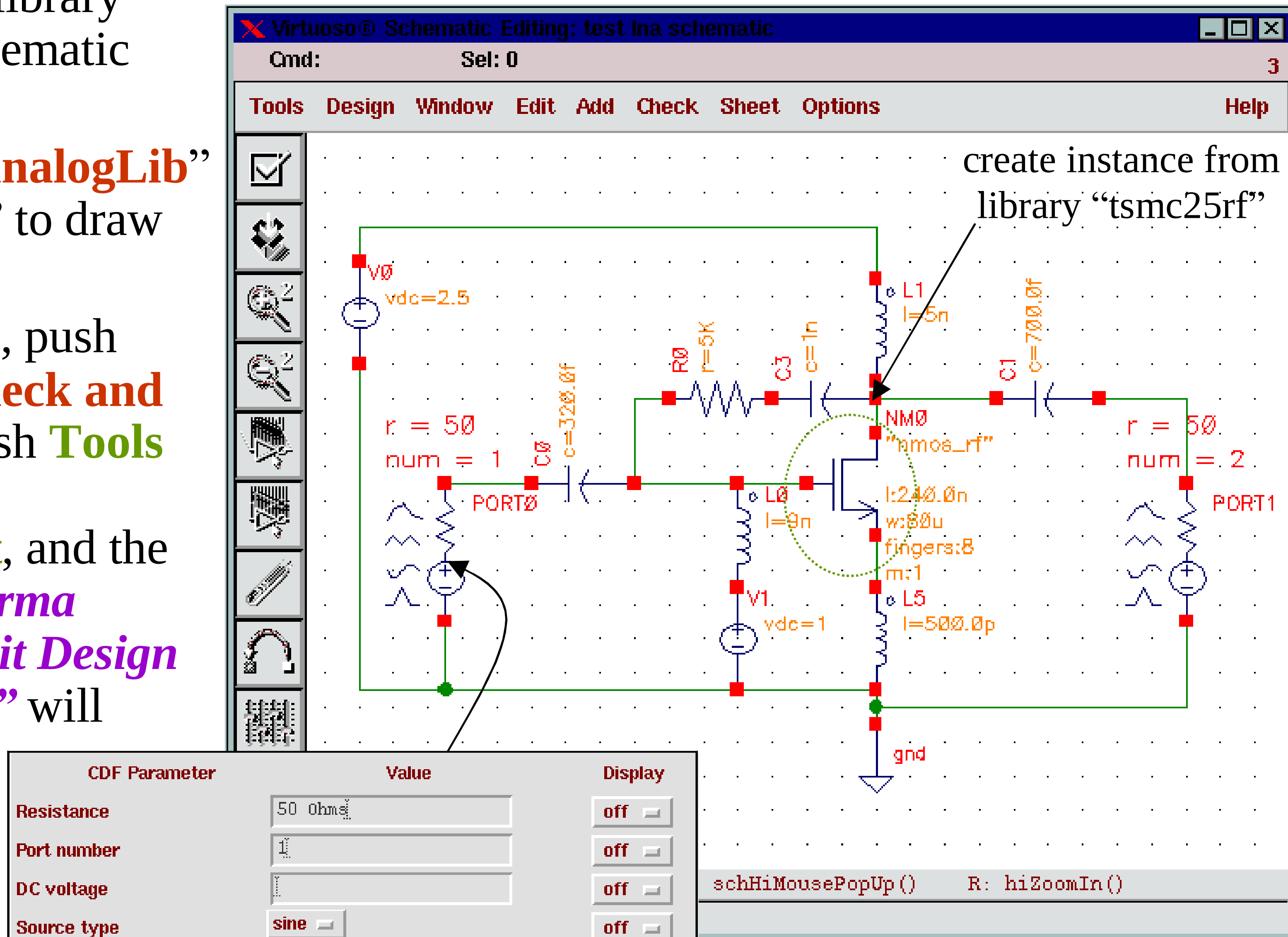
- Linear Simulation:
  - Entirely in the frequency domain
  - A basic RF feature of the Spectre simulator
- Ports:
  - Specify the port number on the ***psin*** (or ***port***); ***psin*** (or ***port***) can act as a source port or a load.
  - Required properties for linear analysis: ***Resistance*** & ***Port number***
- Noise Analysis:
  - Use Nfmin and NF for 2-port circuits ONLY.

Plotting S-Parameter Simulation Results

SP, ZP, YP, HP	s-, z-, y-, and h-parameters
GD	group delay
VSWR	Voltage Standing Wave Ratio
NFmin	minimum noise figure
Gmin	reflection coefficient associated with Nfmin(also known as Γmin, Γopt, or Γon)
Rn	noise sensitivity parameter
rn	normalized equiv. Noise resistance
NF	noise figure
Kf & B1f	stability terms
GT	transducer gain
GA	available gain, assuming conjugate matched output
GP	power gain, assuming conjugate matched input
Gmax	maximum available power gain
Gmsg	maximum stable power gain
Gumx	maximum unilateral power gain
ZM	impedance at port m
NC	noise circles
GAC	available gain circles
GPC	power gain circles
LSB	load stability circles
SSB	source stability circles

Lab1 : S-parameter Analysis

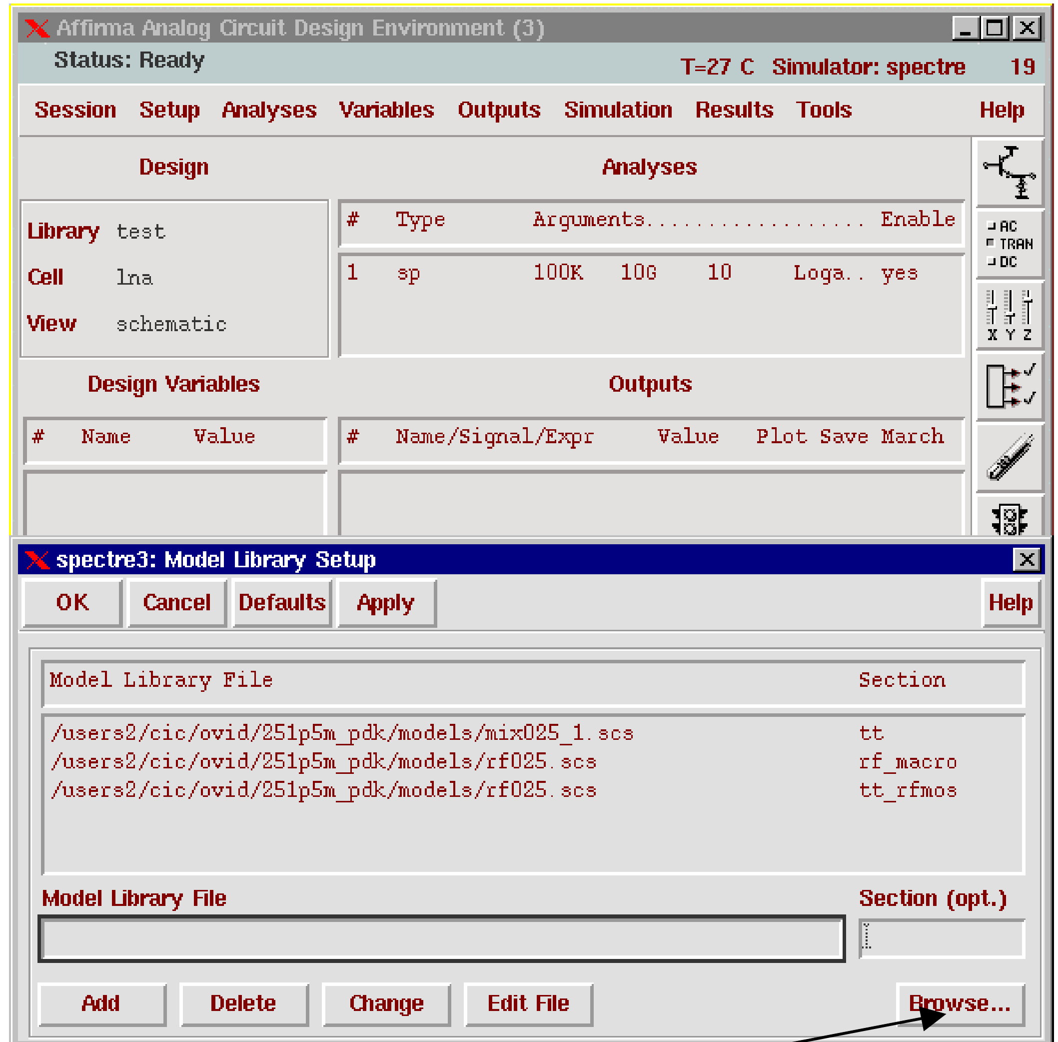
- Create a new library and a new schematic view.
- Use library "**analogLib**" & "**tsmc25rf**" to draw the scheme.
- After drawing, push **Design** → **Check and Save**; then push **Tools** → **Analog Environment**, and the window "***Affirma Analog Circuit Design Environment***" will appear.



Setup Design Environment(1)

- Push **Setup** → **Model Libraries** then the window "***Model Library Setup***" appears. Setup the model library as shown right. Then click OK.

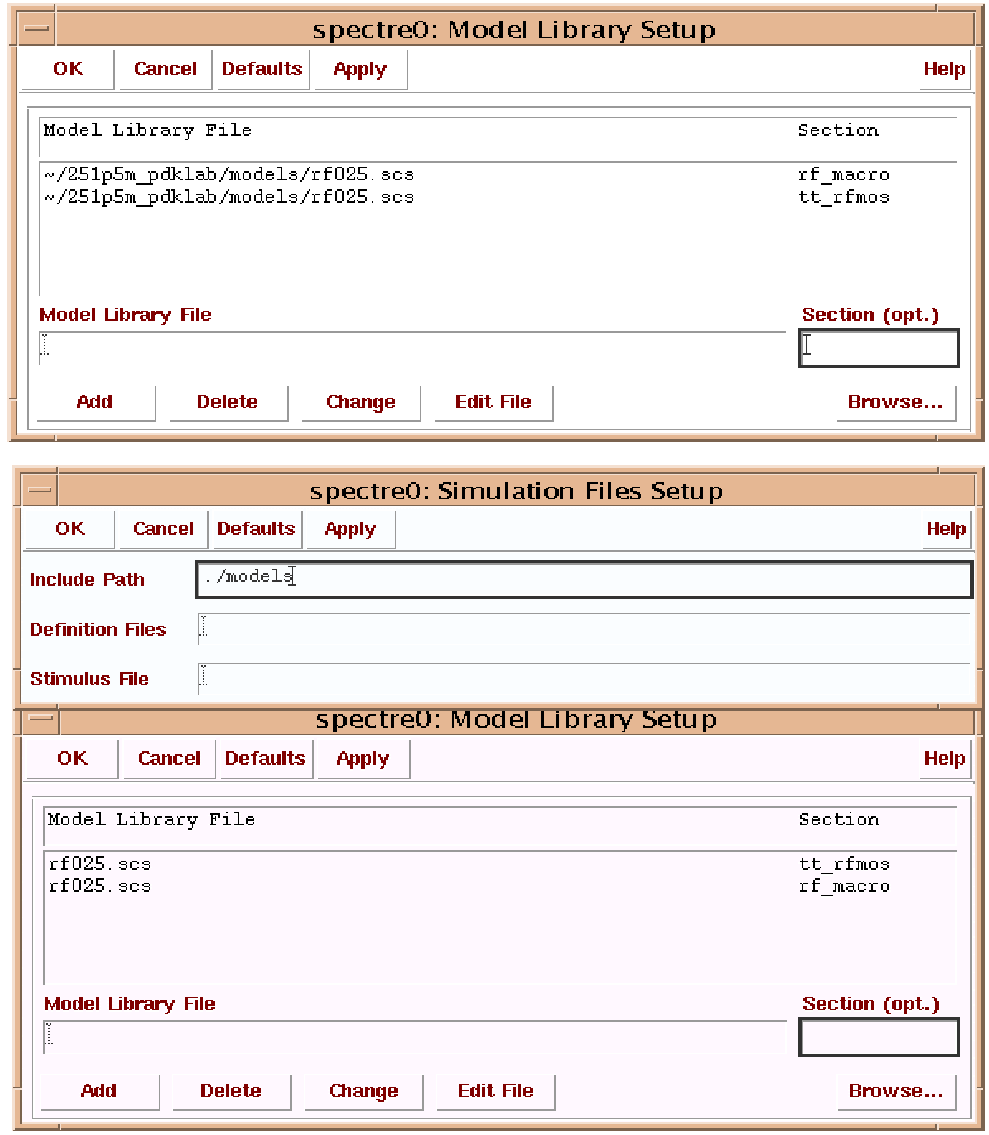
- Push **Setup** → **Simulator/Directory/Host** to designate the project directory. The default project directory is " **~/simulation** " .



Use "**Browse**" to access to the model files

Setup Design Environment(2)

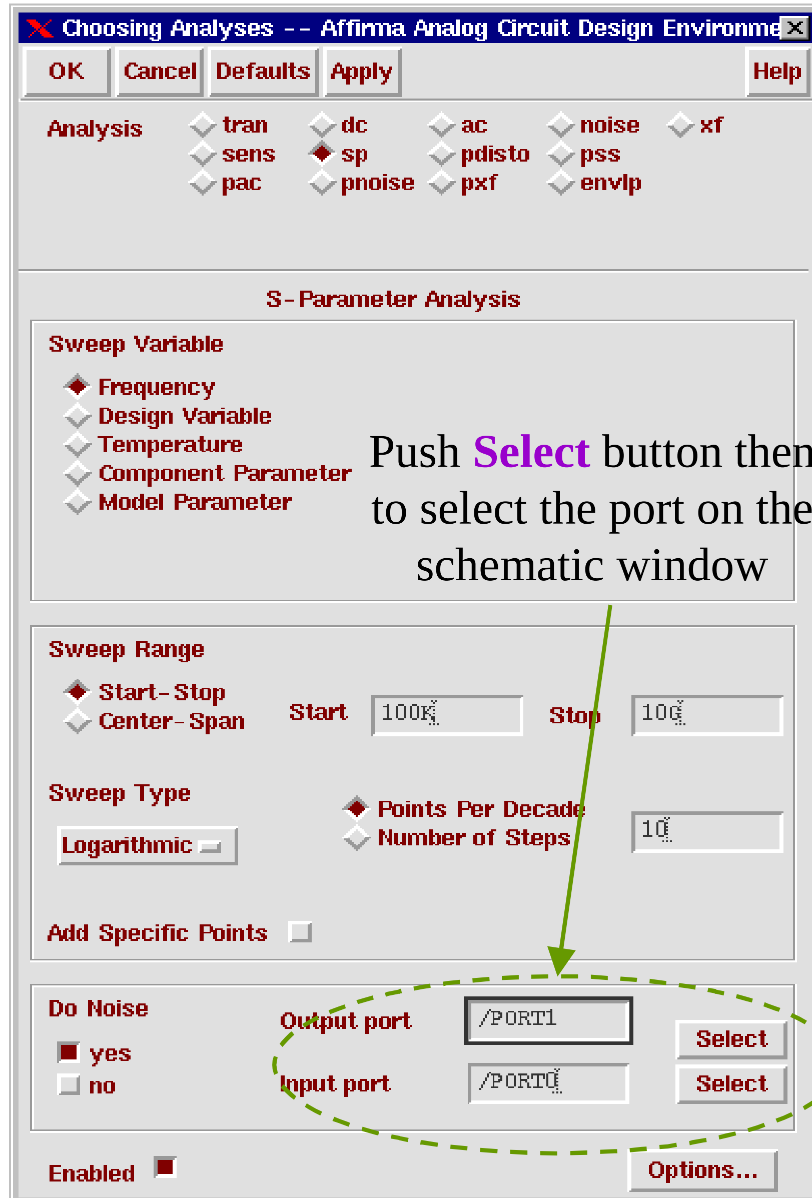
- You can use either an absolute model path or a relative model path
- IF you use the absolute approach, the setup is as shown right-upper.
- To use a relative path ,push "**Setup ® Simulation Files**",than **Setup** → **Model Libraries** .The setup is as shown right.

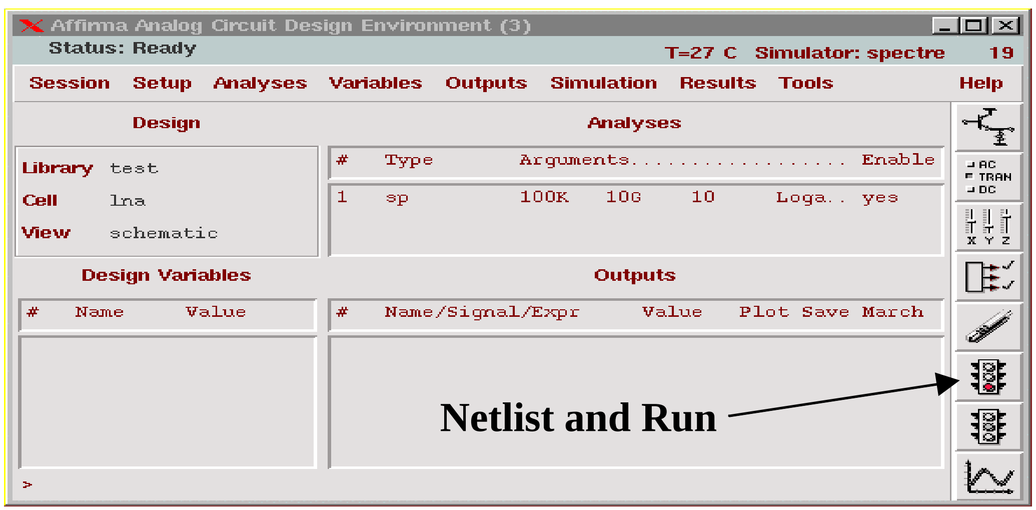


Setup Design Environment(3)

- Push **Analyses** → **Choose** then the window "***Choosing Analyses***" appears. Key in the values as right and push ok, then some information will appear in the "Analyses" domain of the window "***Affirma Analog Circuit Design Environment***".

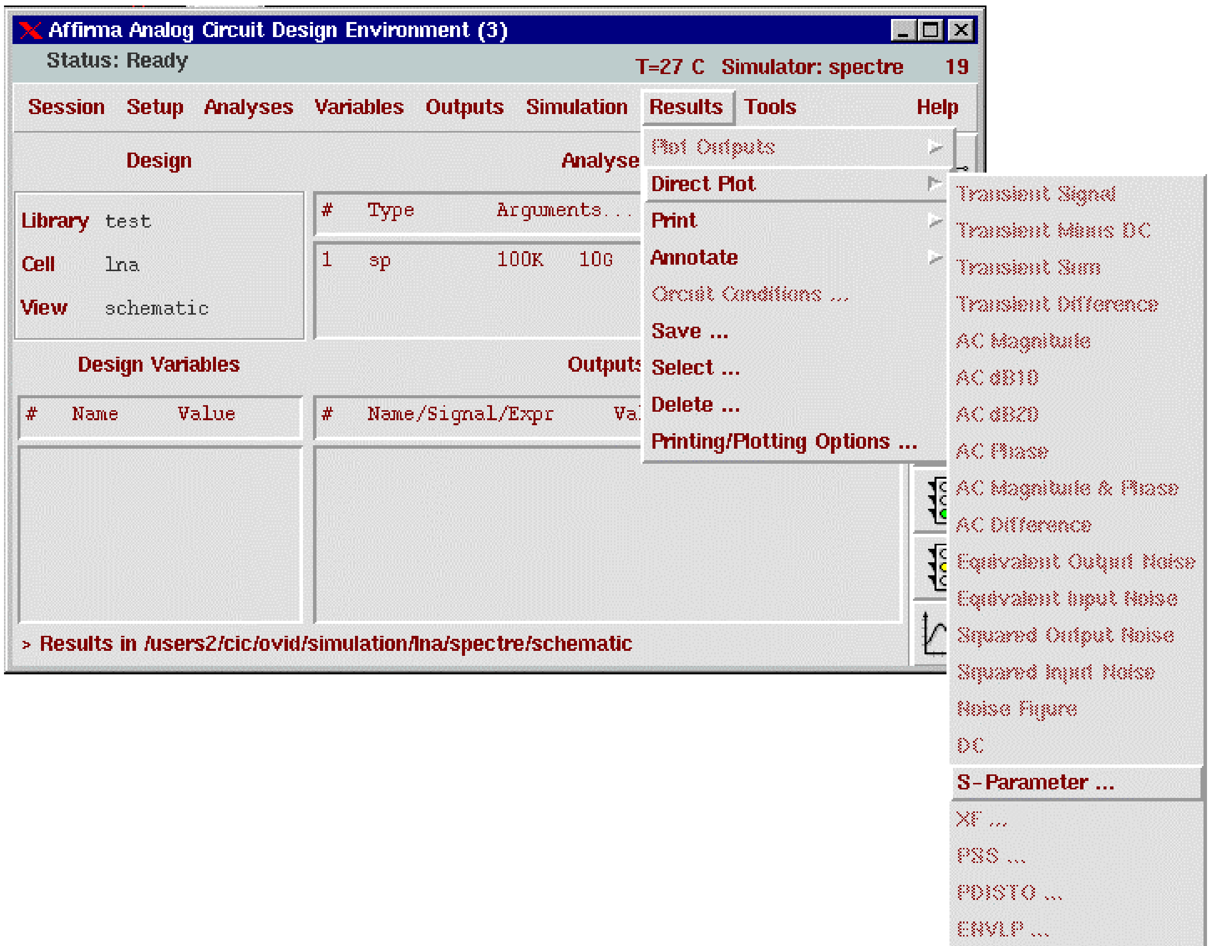
- Push **Simulation** → **Netlist and Run** to run the simulation. The Netlist will be saved under a directory called ***~/simulation.***

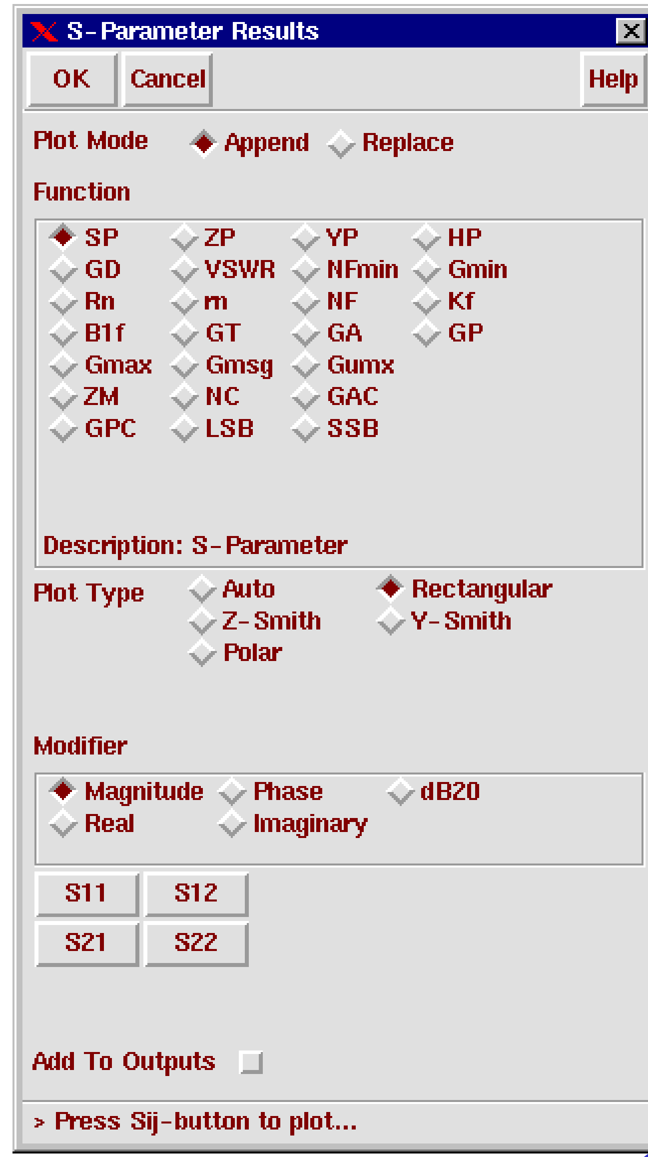




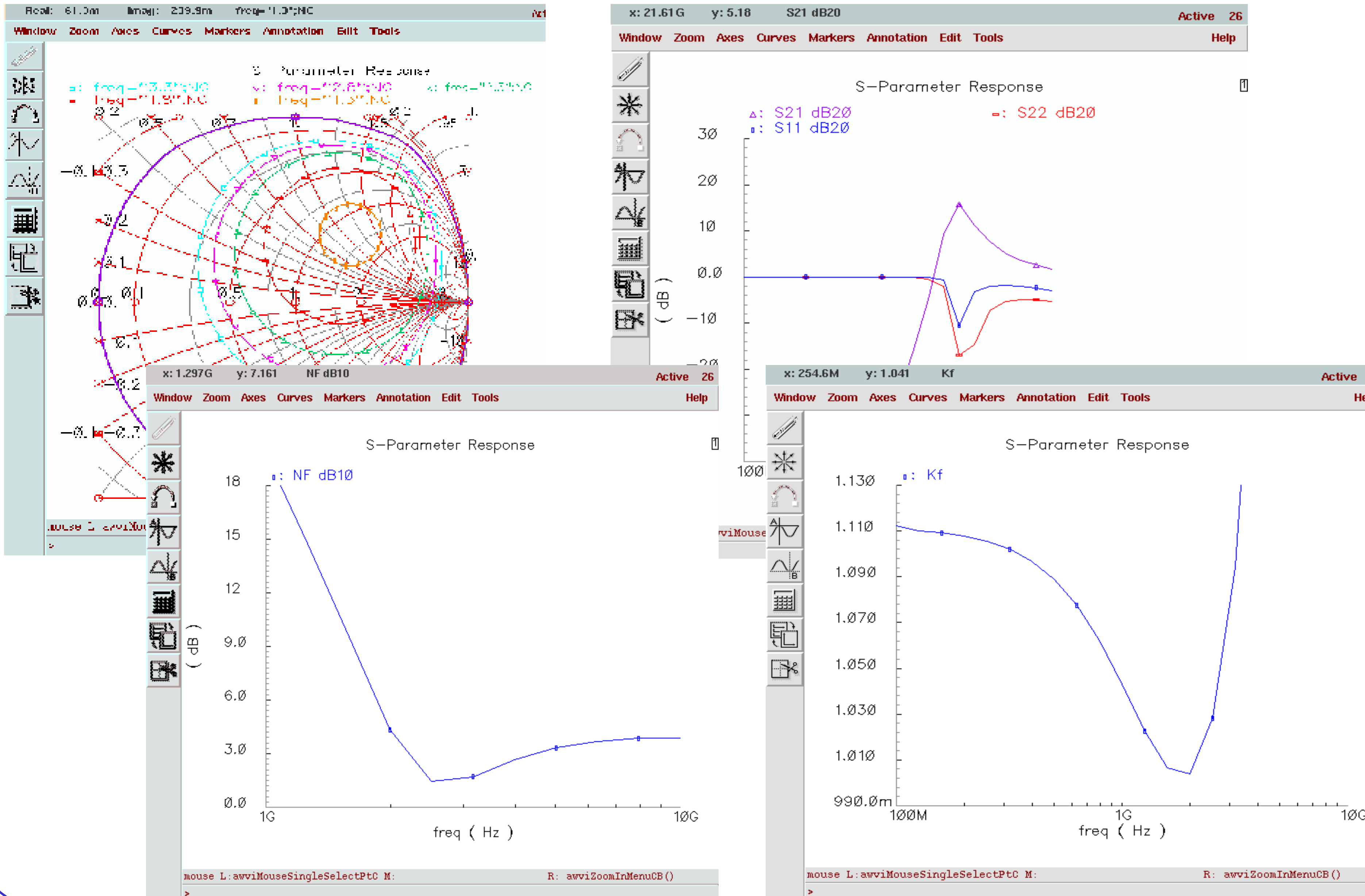
See the Results

- Use the Direct Plot tool to look the results.
- In the "**S-parameter Results**" window choose some parameters to see their results.



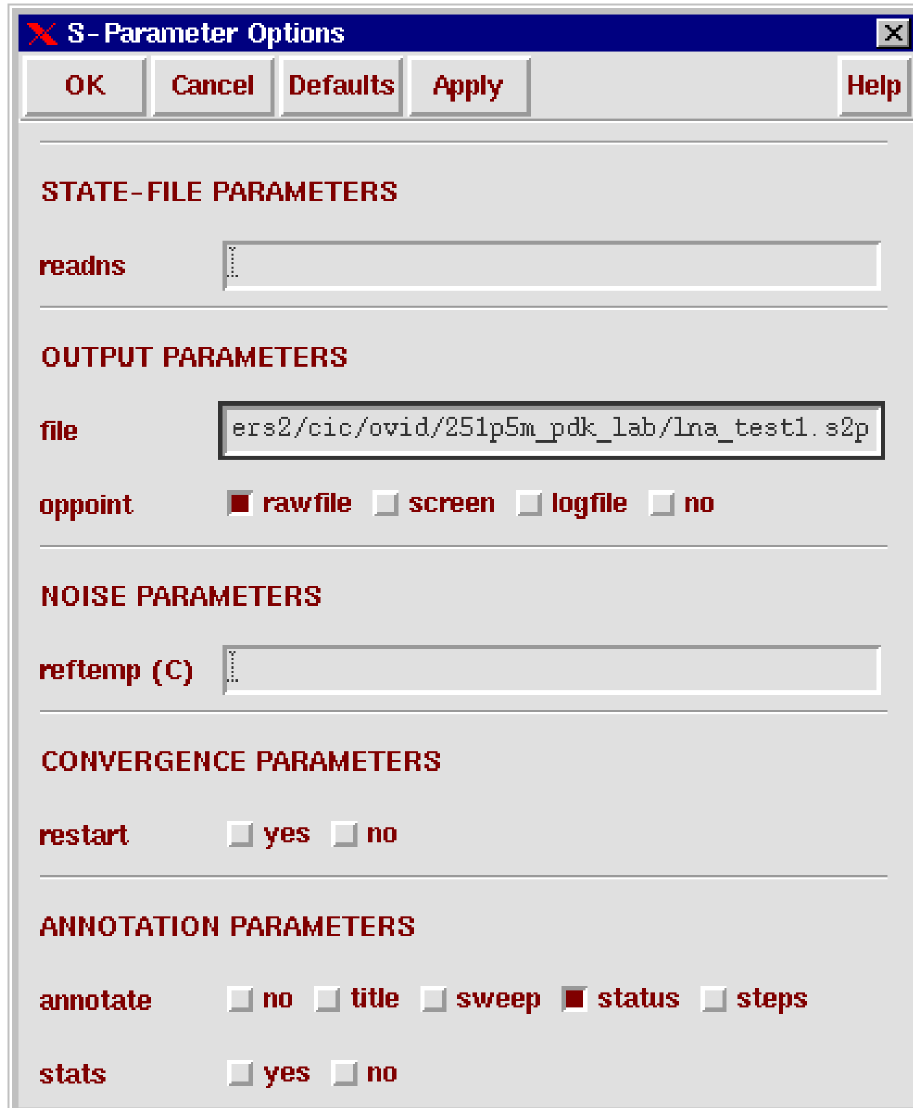


Some Results



Save the results to *.s2p

- Edit the S-Parameter Options, and enter the path to the output S-parameter file in the file field of the OUTPUT PARAMETERS section and OK the S-Parameter Options form.
- And Simulate again. Check if the file is created in the appointed directory.



S2P File

CIC

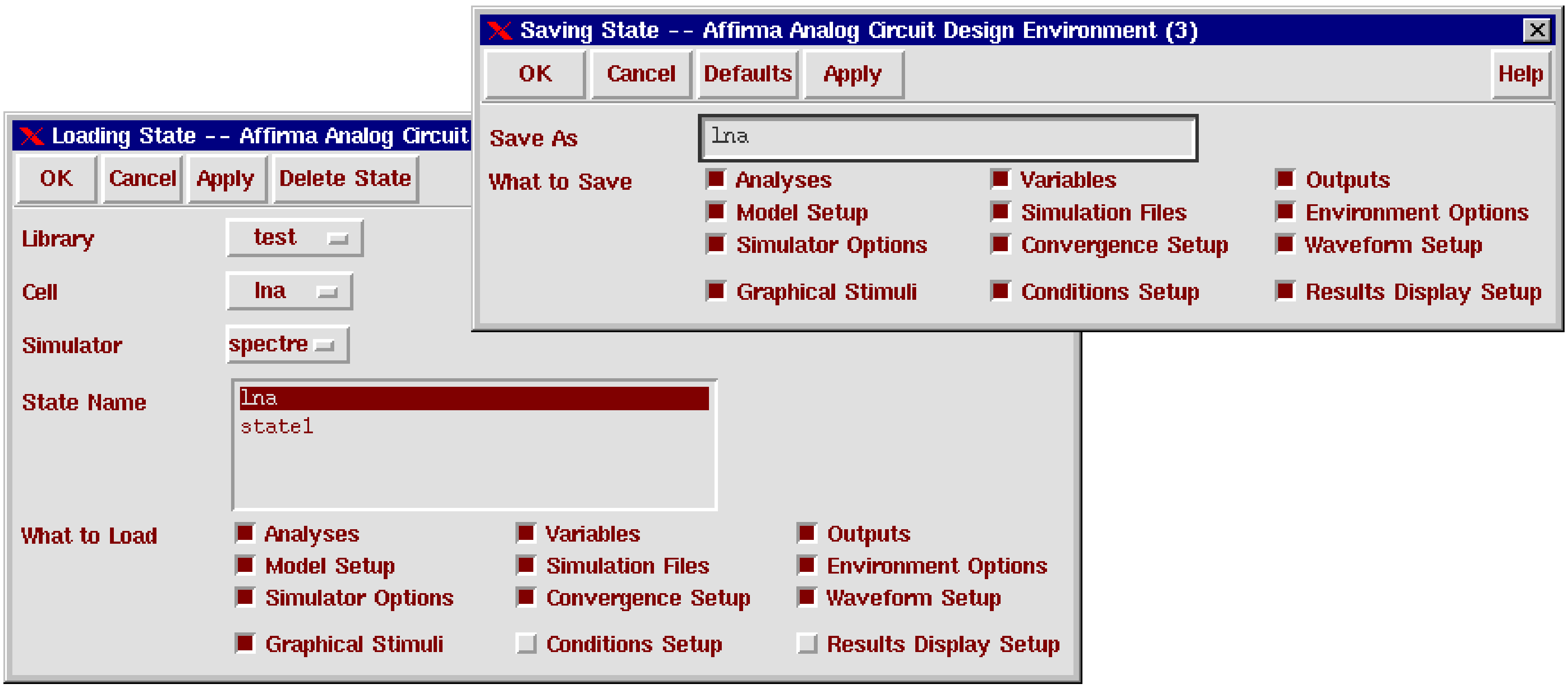
```
; S-parameter data file `/users2/cic/ovid/251p5m_pdk_lab/lna_test1.s2p'.
; Generated by spectre from circuit file `input.scs' during analysis sp.
; 4:23:25 PM, Mon Jun 17, 2002
reference resistance
        port1=50        ; is port PORT0
        port2=50        ; is port PORT1

format  freq:   s11(real,imag)  s21(real,imag)
                s12(real,imag)  s22(real,imag)

1.00000000e+05:           1,-2.01062e-05      -3.50355e-18, 9.24347e-21
                   2.853e-20, 9.08223e-21                 1,-4.39823e-05
1.25892541e+05:           1,-2.53122e-05        -8.798e-18, 1.92698e-20
                 7.41827e-20, 1.87592e-20                 1,-5.53704e-05
1.58489319e+05:           1,-3.18662e-05      -2.20953e-17, 3.98972e-20
                 1.90564e-19, 3.82814e-20                 1,-6.97072e-05
1.99526231e+05:           1,-4.01171e-05      -5.54941e-17, 8.26123e-20
                 4.85624e-19, 7.74999e-20                 1,-8.77562e-05
2.51188643e+05:           1,-5.05045e-05      -1.39383e-16, 1.72265e-19
                 1.23111e-18, 1.56093e-19                 1,-0.000110479
3.16227766e+05:           1,-6.35814e-05      -3.50097e-16, 3.64524e-19
                 3.11055e-18, 3.13372e-19                 1,-0.000139084
```

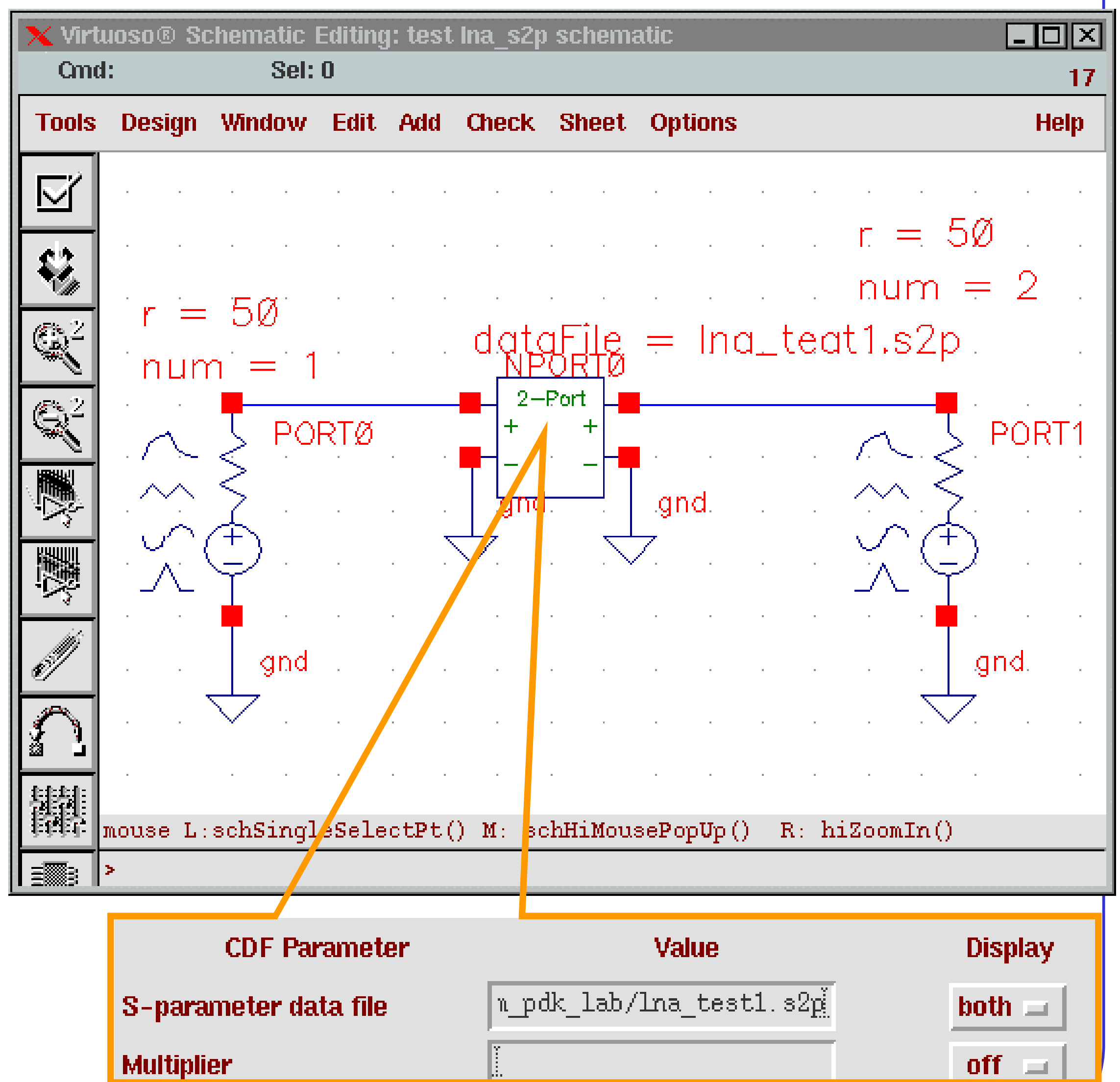
Simulation State

- Push **Session** → **Save State** to save simulation states under a directory called ***~/.artist_states***. Designate a new directory with the **Session** → **Options** command in the simulation window.
- Push **Session** → **Save State** **to load saved states for a design.**



Read the S2P file(1)

- Create a new schematic view.
- Use library "**analogLib**" (***n2port*** cell) to draw the scheme.
- Simulate if the results are the same as before.



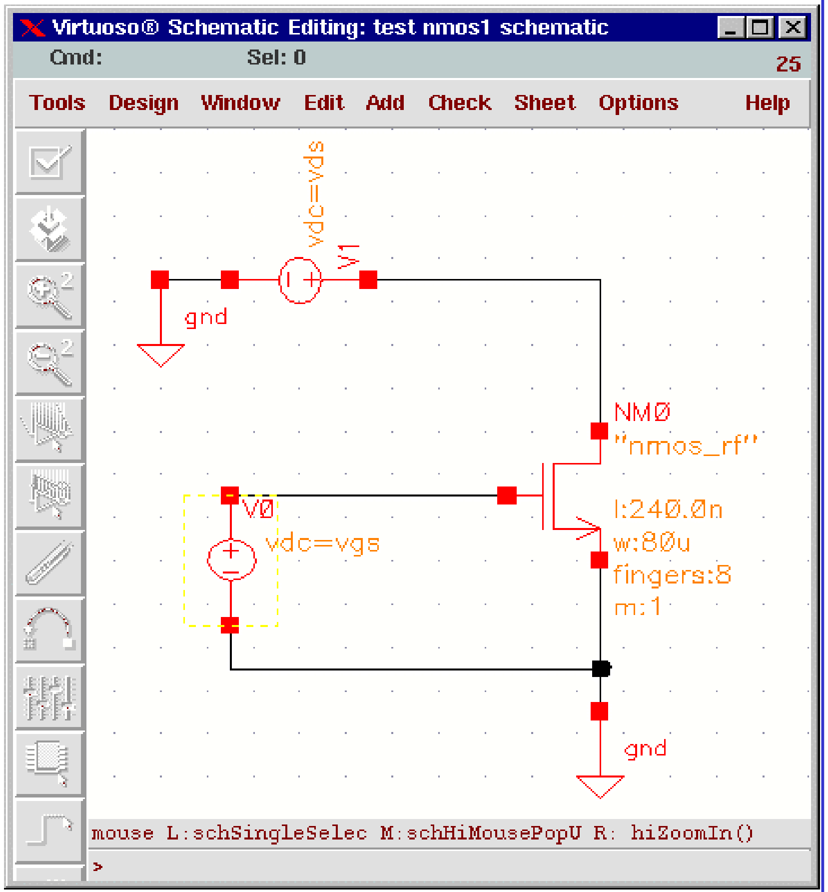
3. Lab2: Swept DC Analysis

- Create a new schematic view and use library "**analogLib**" & "**tsmc25rf**" to draw the scheme.

- . After "**Check and Save**"; then call the window "***Affirma Analog Circuit Design Environment***"**.**

- **Setup** up the **Model Libraries.**

- Push **Variables** → **Copy From Cellview**, and the defined variables appear in the "***Design Variables***" section. Double click on



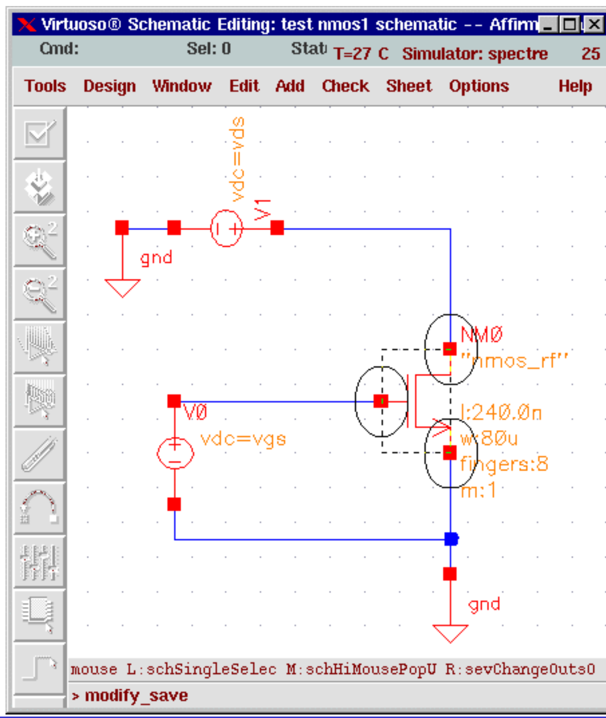
the variable name or push **Variables** → **Edit**, the window "***Editing Design Variables***" appears. Key in the appropriate value for the variables.

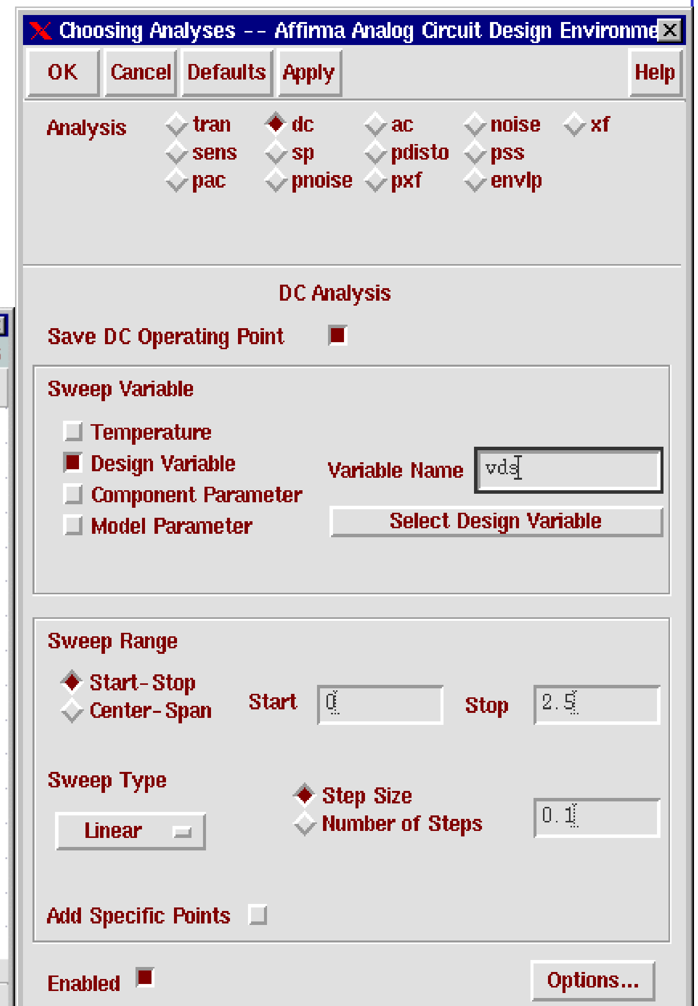
Set up the Design Environment(1)

- Call the window "***Choosing Analyses***" and key in the values as right and push ok.

- To plot power or current at the end of the simulation, you must explicitly save the currents necessary for the calculation before the simulation. The voltages at each node are saved by default.

- Select **Outputs → To Be Saved → Select On Schematic**. In the schematic, select the NMOS. The terminals are circled in the schematic window after you select them. Press **Esc** to end the selections.

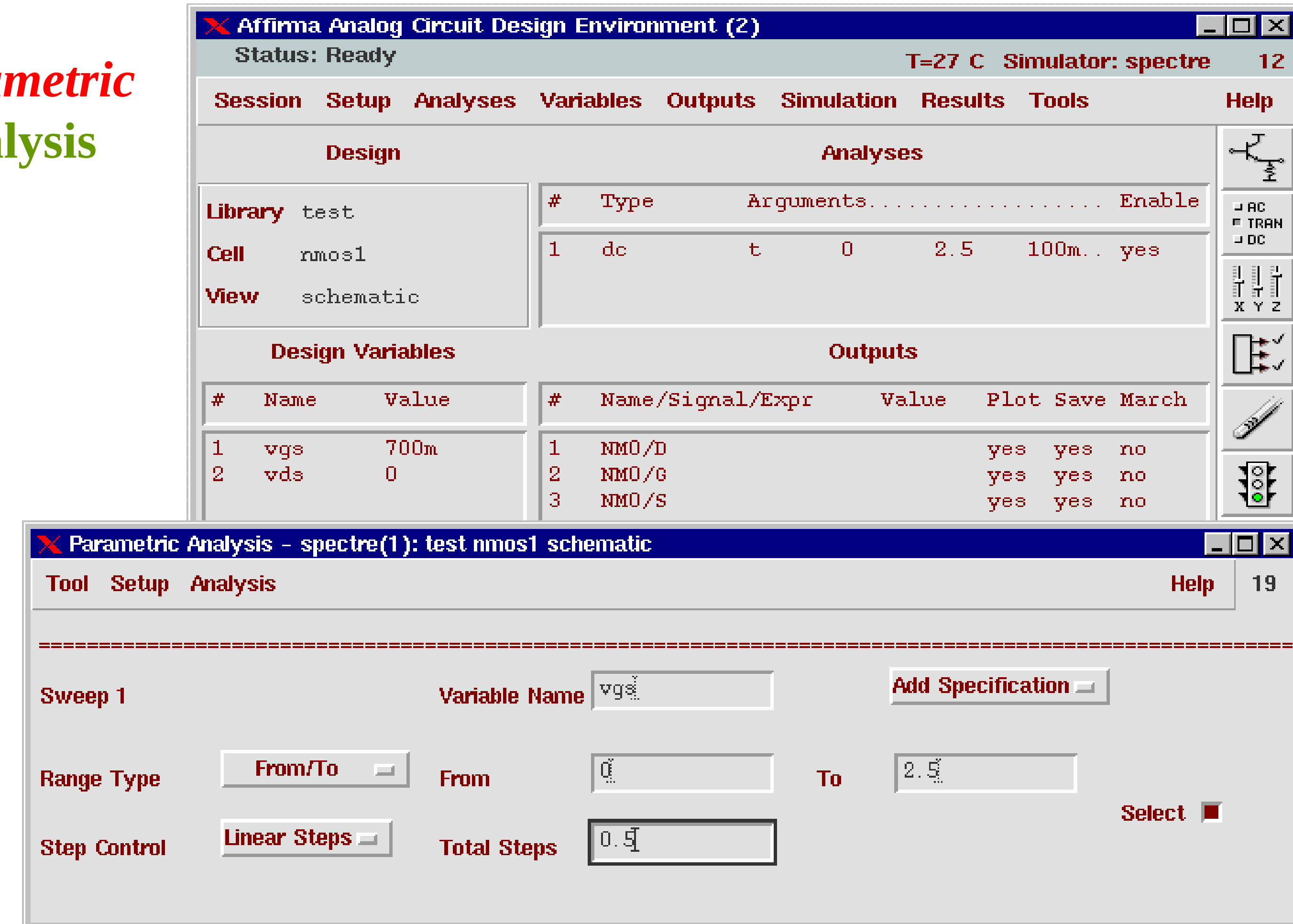




Set up the Design Environment(2)

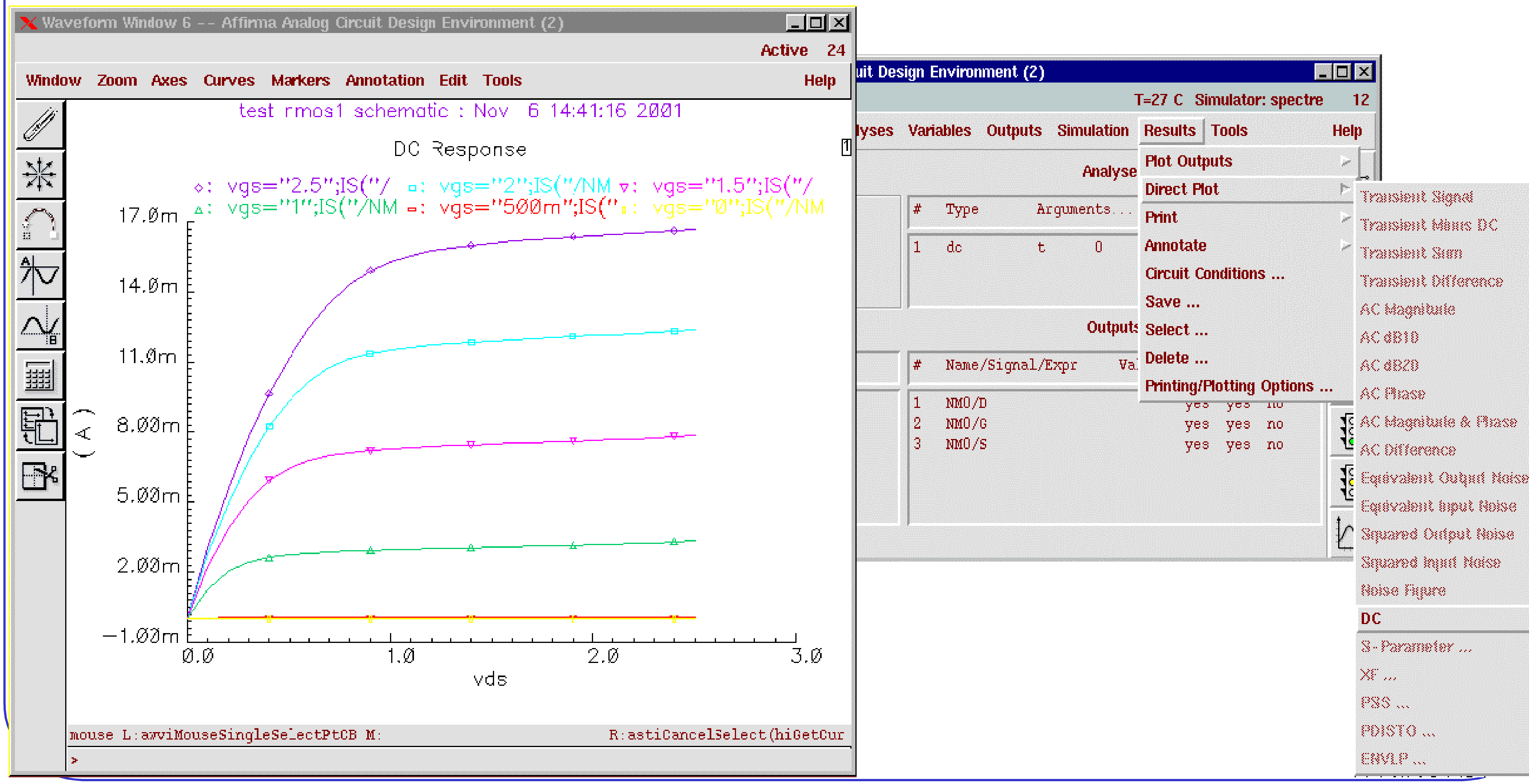
- In the window "***Design Environment***" select **Tools** → **Parametric Analysis…**; the window "***Parametric Analysis***" appears, then key in the values as below .

- In the window "***Parametric Analysis***" select **Analysis** → **Start** to start the simulation.



The Results

- Select **Results** → **Direct Plot** → **DC** and select the terminal "Drain" of the nmos in the schematic window; then push **ESC**, and the results will be showed.



4. Periodic Steady State Analysis

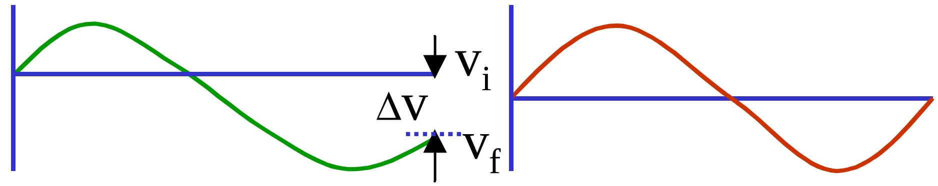
- Directly computes the periodic steady-state response of a circuit in the time domain.
- Iterative Shooting Newton method is employed.
- Calculate frequency translations using the saved matrices at every time point.
- The fundamental frequency of the circuit or system is determined, based on integer multiples of all source frequencies.
- The circuit is evaluated for one period of the common frequency, and the period is adjusted until all node voltages and all branch currents fall within a specified tolerance.

Shooting Newton Method

- PSS operates by efficiently finding an initial condition that results in steady state.

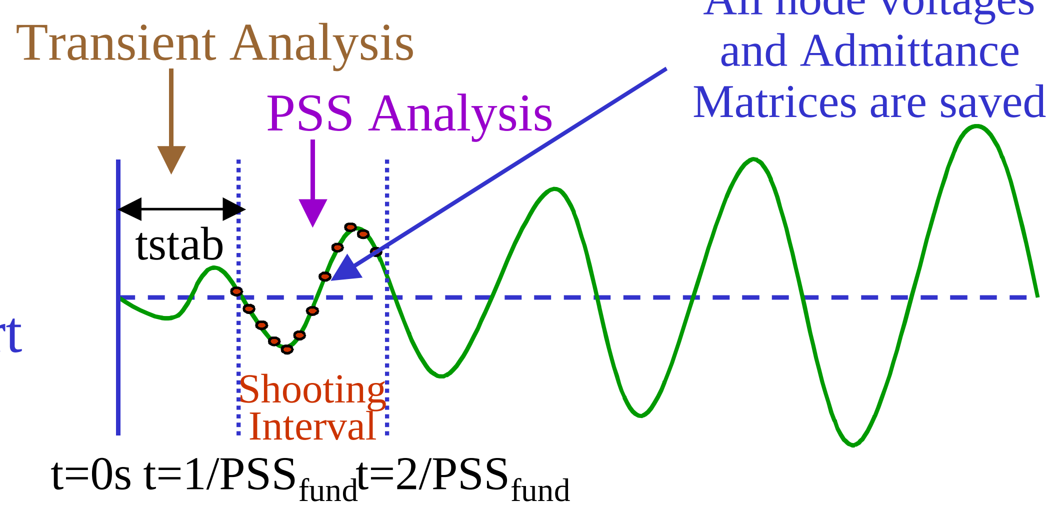
- The first iteration is transient simulation from t=0 to t=1/PSS_{fund} by default. The **tstab** parameter can be adjusted to facilitate convergence.

- The second iteration is PSS analysis between t=tstab to t=(tsatb+1/PSS_{fund}) and compares all voltage and currents at the start and end of the shooting interval. Set the value of **tstab** to keep "start-up behavior" away.

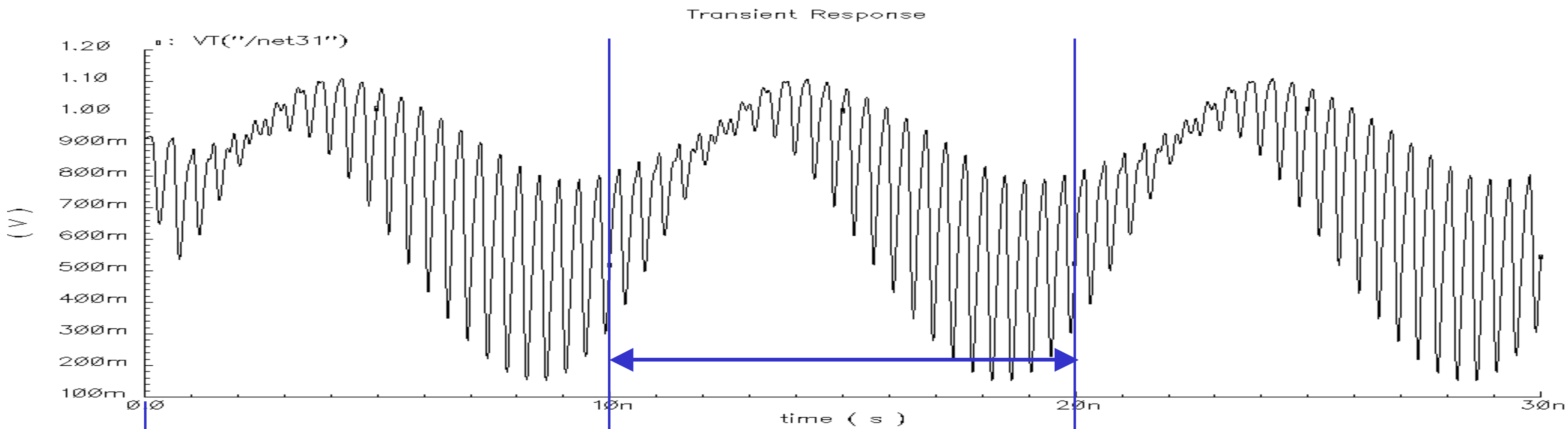


The signal starts at a point v_i doesn't result in periodicity.

The starting point is adjusted by the shooting method to result in periodic steady state.

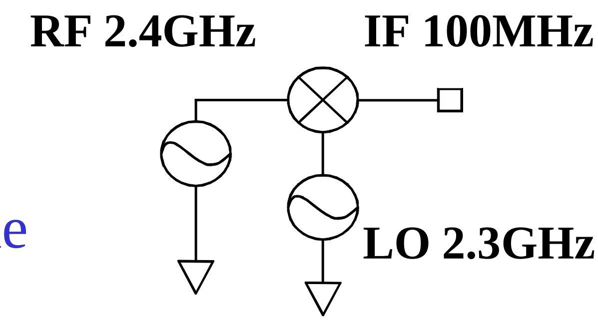


Shooting Newton Method(continued)



$\tau = \dfrac{1}{PSS_{fund}}$ $\tau = \dfrac{1}{PSS_{fund}}$

Transient Analysis PSS Analysis



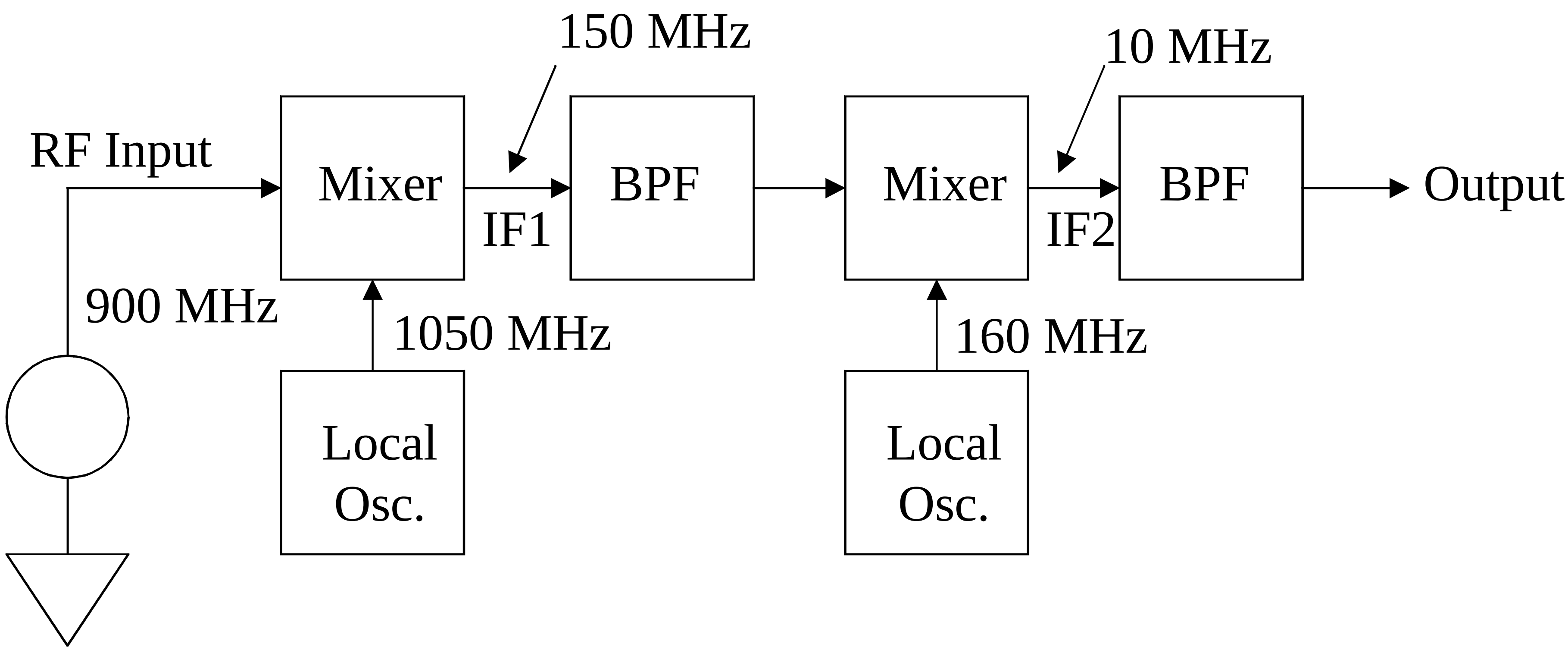
PSS_{fund}=100MHz

- Shooting method takes the last few point data at the end of the shooting interval to adjust the slopes of the waveform at the beginning of the next iteration.
- If 20 iterations do not yield a solution, this might indicate the circuit won't converge to a PSS solution.

PSS Analysis Assumptions

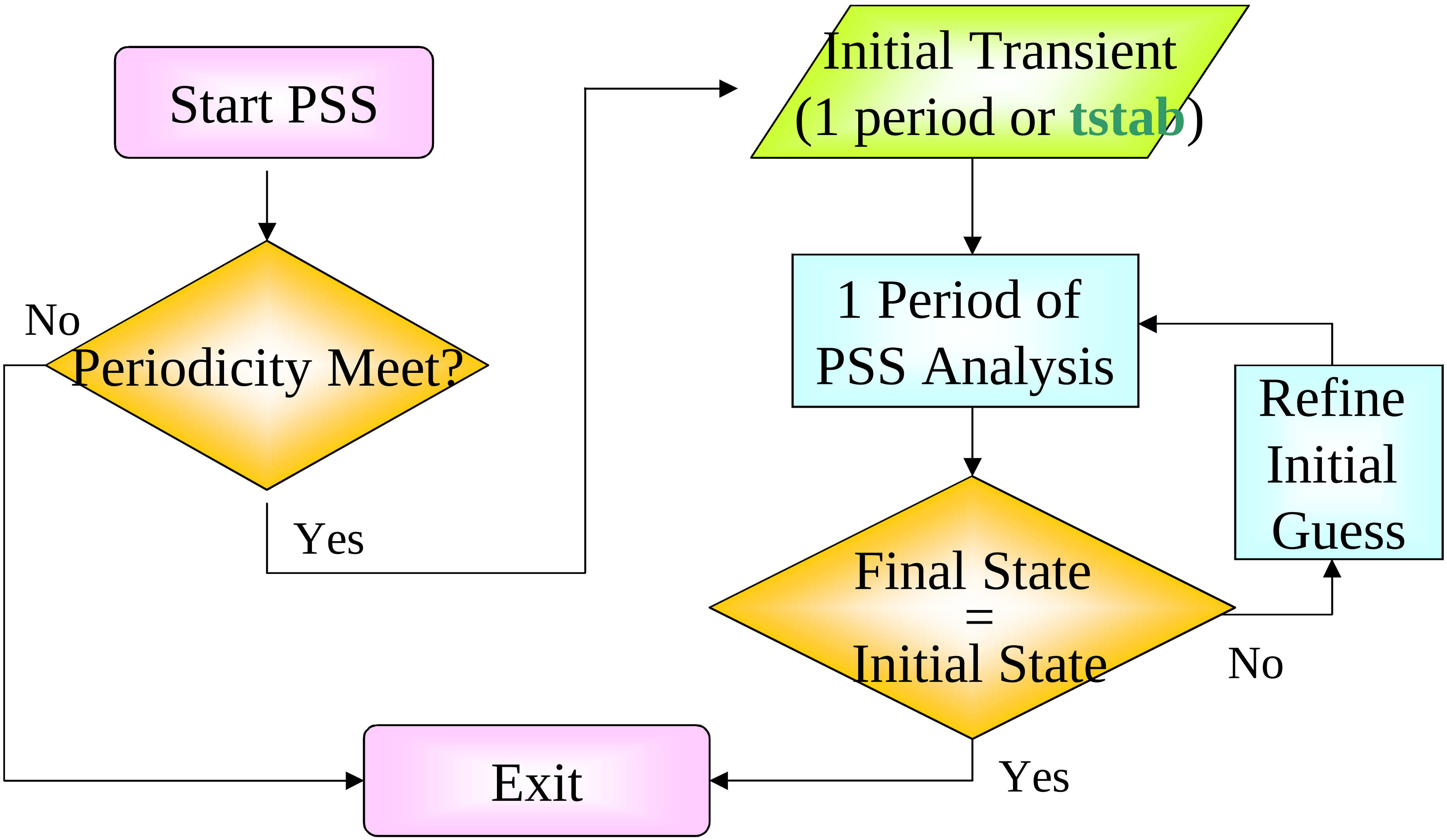
- 1st Assumption : Periodicity
  - All stimuli are periodic and coperiodic with the PSS_{fund} ; All responses are periodic.
  - PSSfund can be set to includes the subharmonics.
  - If periodicity assumptions fail, PSS analysis will not converge.
- 2nd Assumption : Linearity
  - A near-linear relationship need to exist between initial and final points of the shooting interval.

The PSS Fundamental



$$PSS_{fund} = 10 \text{ MHz}$$

PSS Operation



Simulator Accuracy Suggestions

- Do not set "conservative". This will dramatically extend the simulation time.
- ΔV and ΔI $<$ reltol $*$ lteratio $*$ steadyratio
- The suggested settings are recommended for IP3 Analysis, Noise Analysis, or wherever high accuracy is needed.
- Choose the gear2only integration method. The default trap integration method yields underdamping and gearOne yields too much overdamping.

Parameter	Defaults	Suggested Settings
reltol	1e-3	**1e-5**
vabstol	1e-6	**3e-8**
iabstol	1e-12	**1e-13**
Method	trap	**gear2only**

errpreset	reltol	relref	method	maxstep	steady-ratio	lteratio
liberal	x10.0	allglobal	gear2	<0.4/maxacfreq	0.1	3.5
moderate	x1.0	sigglobal	traponly	<0.2/maxacfreq	0.001	3.5
conservative	x0.1	alllocal	gear2only	<0.1/maxacfreq	0.00001	10.0

Normalized Convergence ratio

- When the Conv norm is 1(unity) or less, the simulation meets the matching criterion.

- The PSS messages also display the number of PSS iterations, the number of accepted timesteps, and the total time required for PSS analysis.

$$\text{Conv norm} = \frac{\text{Measured } \Delta V \text{ between start and end of shooting interval}}{\text{reltol*lteratio*steadyratio}}$$

Lab3 : PSS and swept PSS Analysis

Create a new schematic view and use library "**analogLib**" & "**tsmc25rf**" to draw the scheme.

Port1:

Frequency name: F1

Resistance: 50

Source type: sine

Amplitude(dBm): -40

Frequency: frf

Port2:

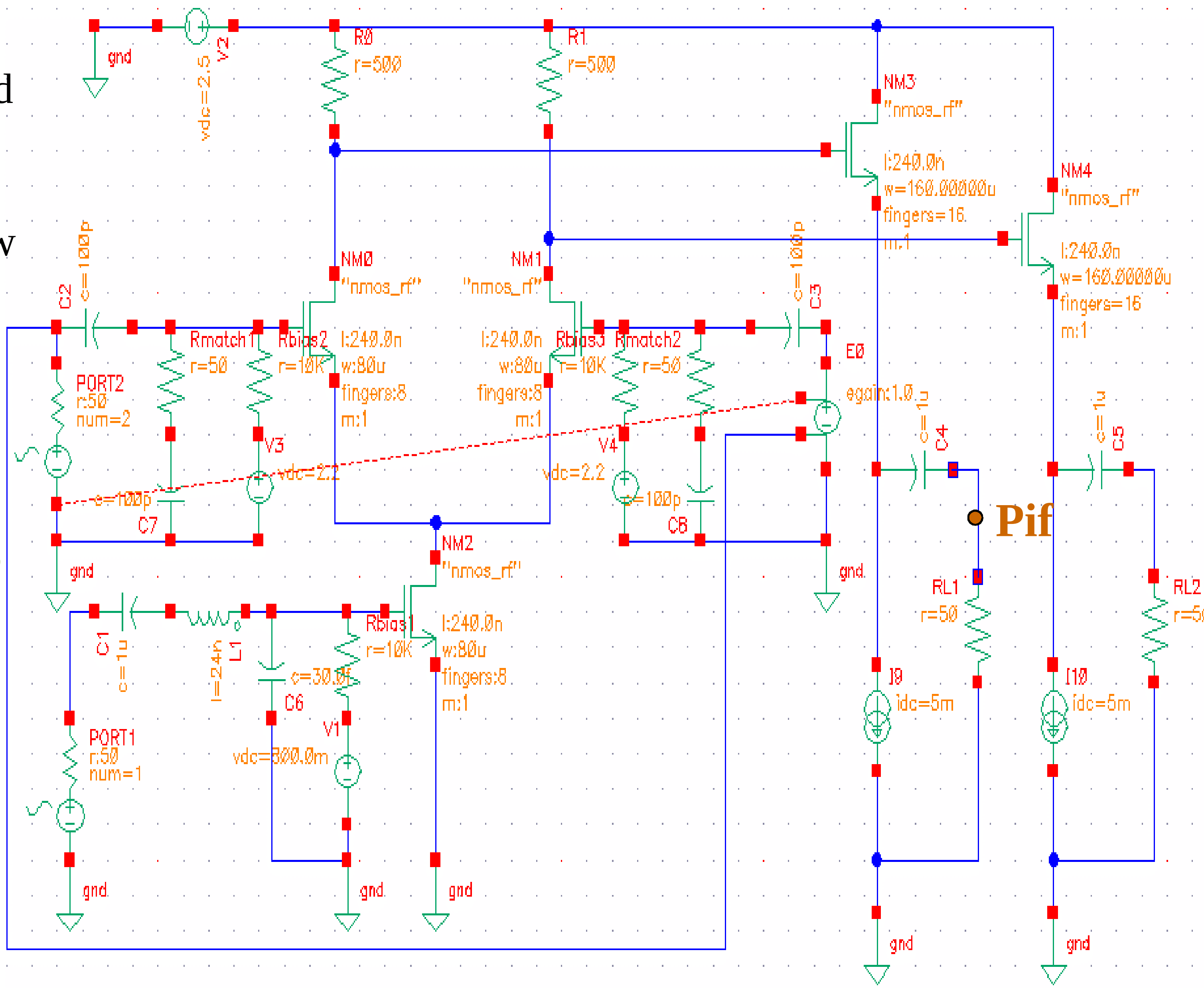
Frequency name: F2

Resistance: 50

Source type: sine

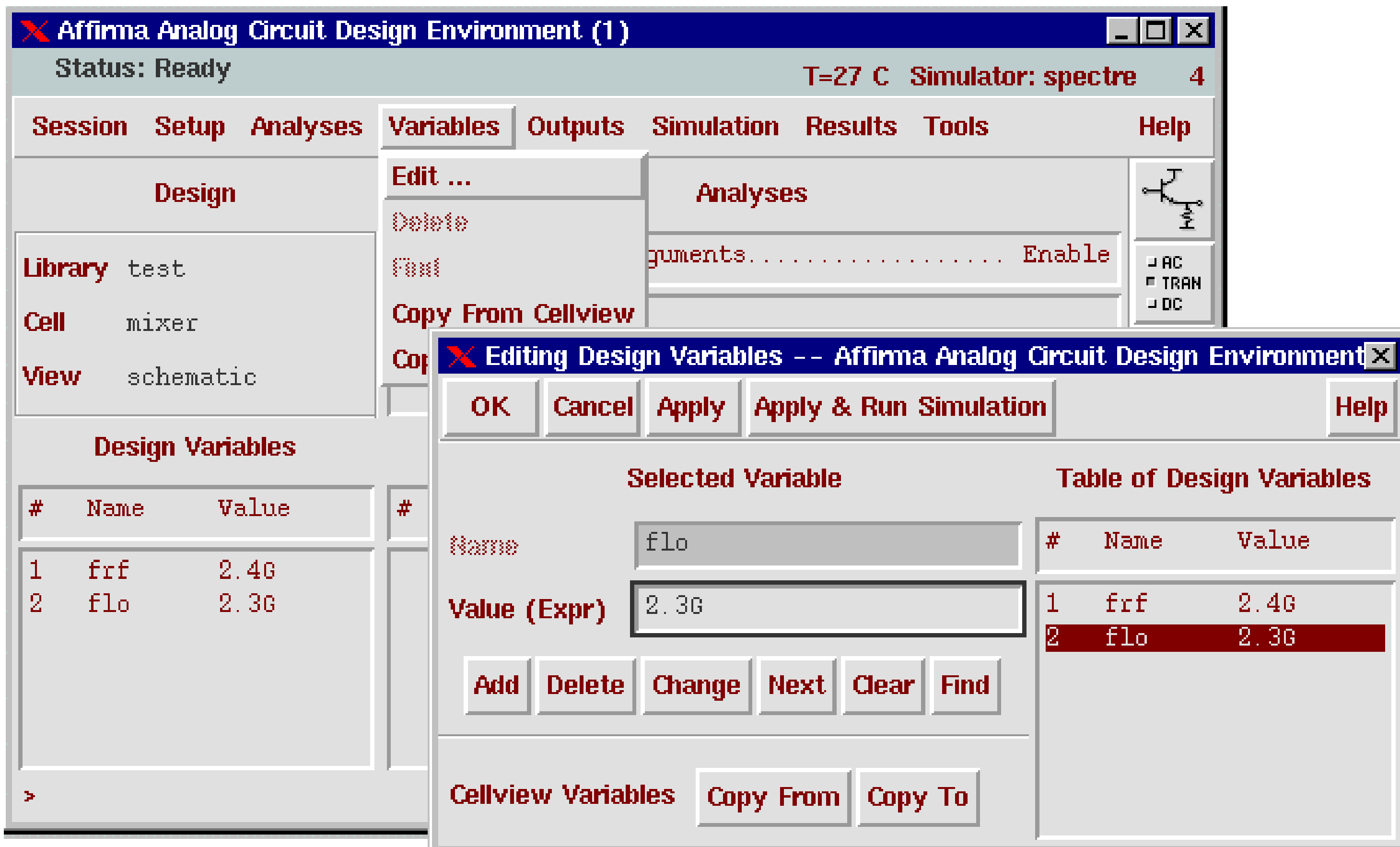
Amplitude(dBm): 8

Frequency: flo



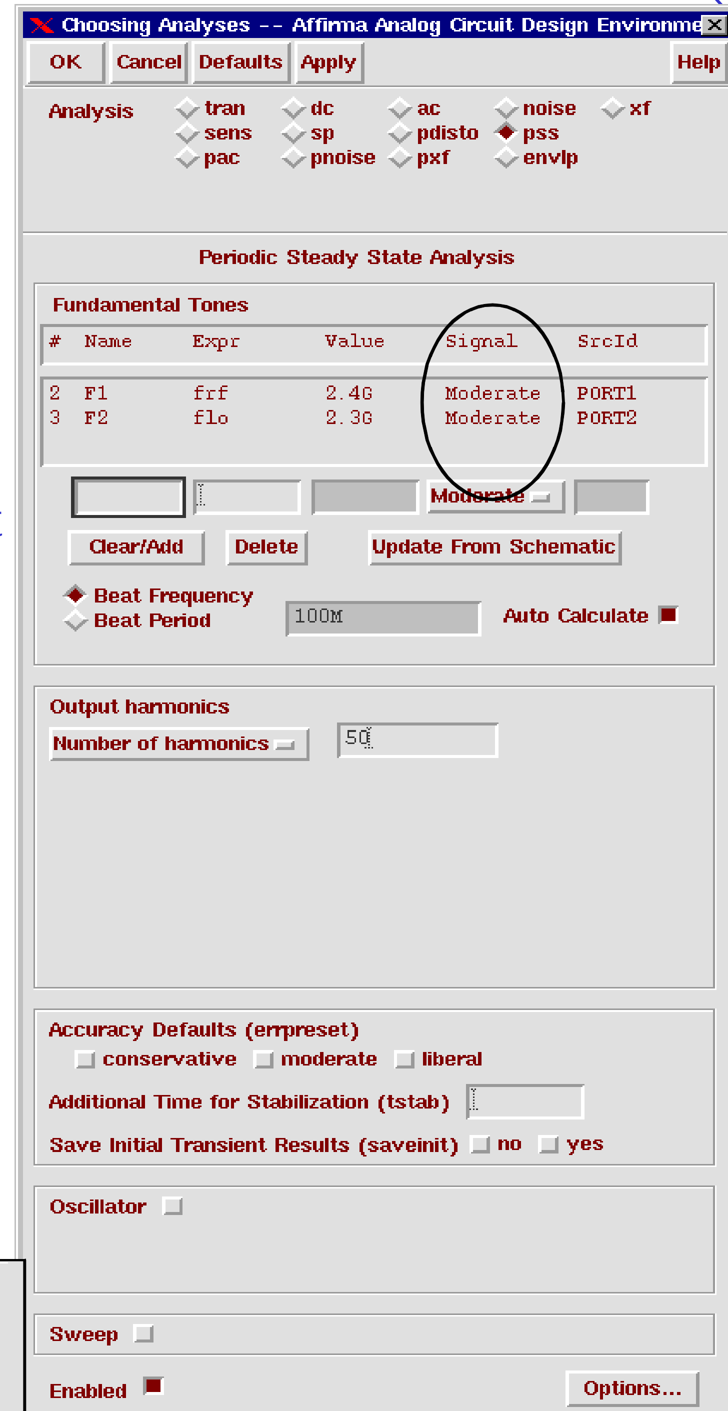
Setup up the PSS Simulation(1)

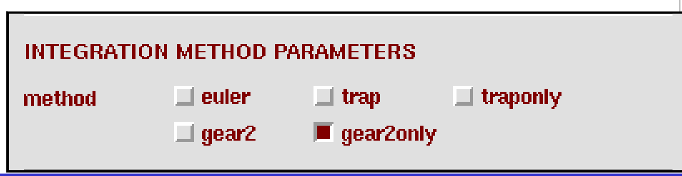
- Model library setup.
- Call the window "***Affirma Analog Circuit Design Environment***"; key in appropriate value for the variables in the "***Design Variables***" section.
- **Analyses** → **Choose**. In the window "***Choosing Analyses***", select pss.



Setup up the PSS Simulation(2)

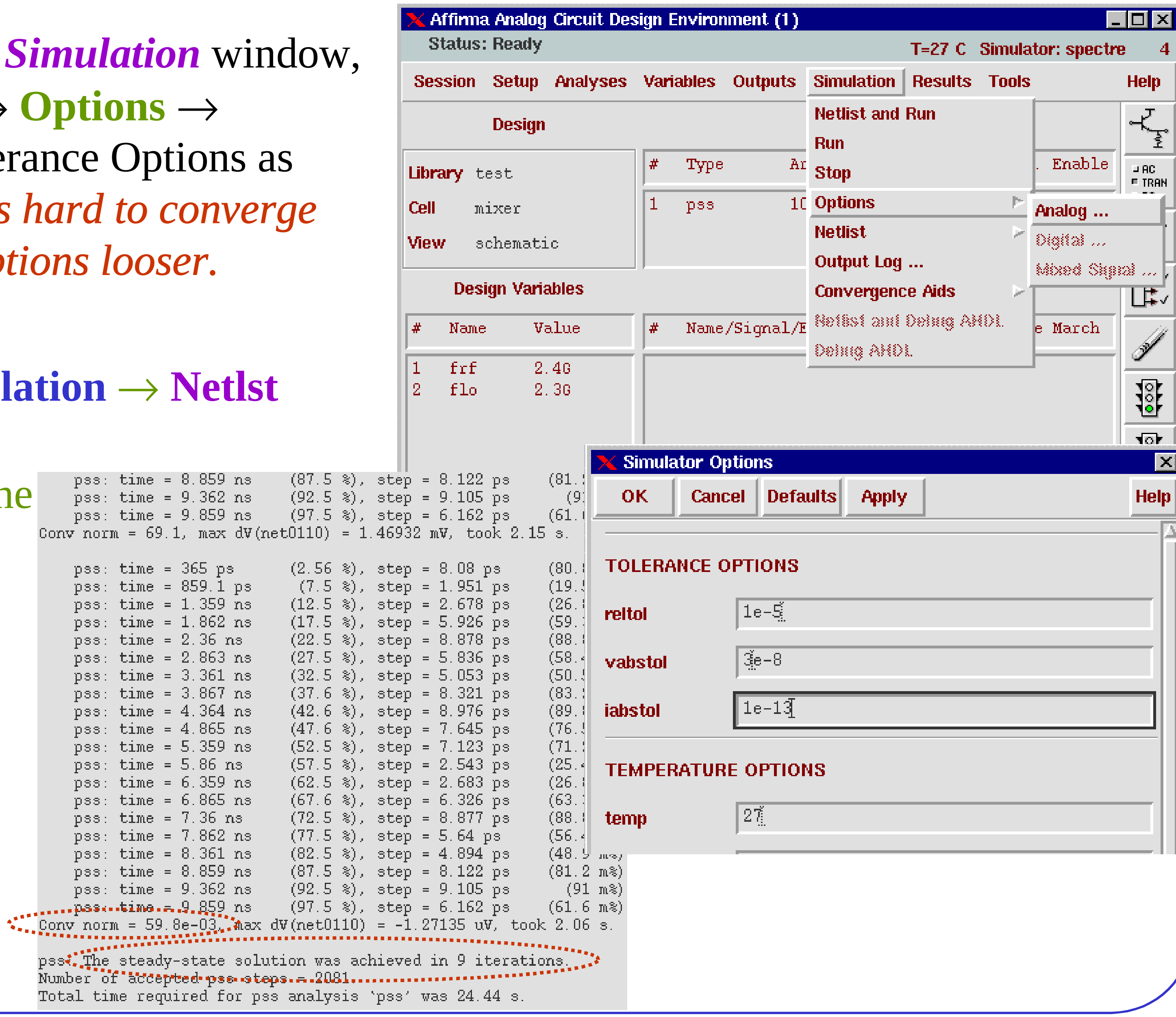
- The Signal field is ONLY applicable to the **pdisto** analysis.
- **Beat Frequency** represents the PSS Fundamental (PSS_{fund}) frequency. This fundamental is the highest frequency that evenly divides into all frequencies in the circuit. You may key in an appropriate value or push **Auto Calculate** button to get an auto-responded value.
- Set the value for **number of harmonics**. The number of harmonics won't affect the simulation accuracy or time.
- Make sure the **Enabled** field is on.
- Click the **Options** button and set the integration method to gear2only.





Setup up the PSS Simulation(3)

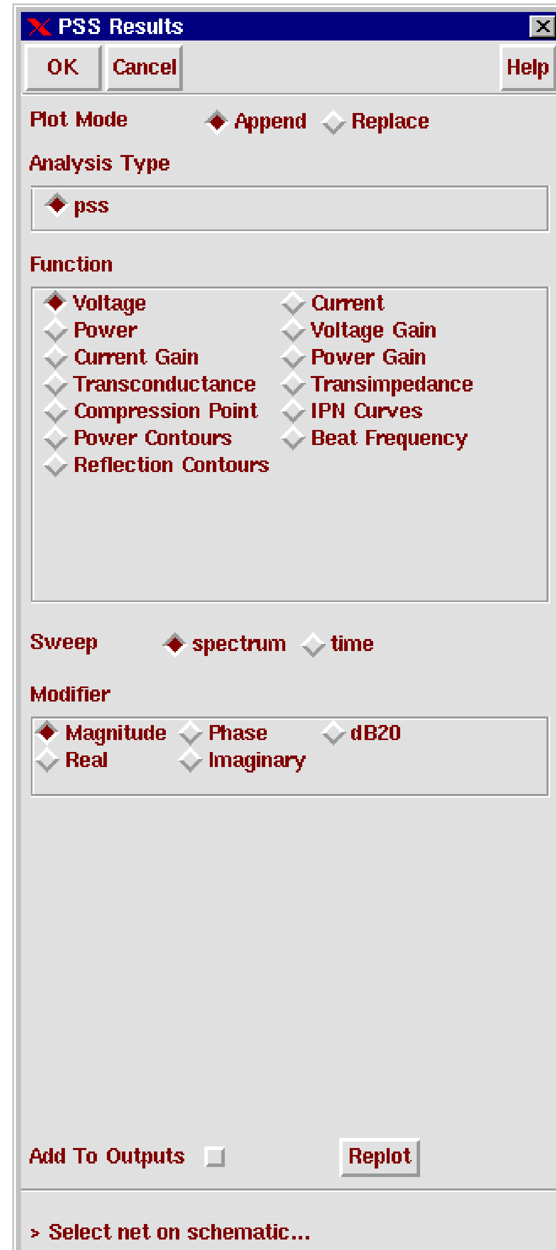
- In the *Analog Artist Simulation* window, select **Simulation** → **Options** → **Analog**. Set the Tolerance Options as recommended. *If it is hard to converge set the Tolerance Options looser.*



- Finally, Select **Simulation** → **Netlst and Run** to start the simulation. Note if the ***Conv norm*** is less than 1 or if the PSS simulation has a convergent result.

Display the Conversion Power Gain-method 1

- In the Analog Artist Simulation Window, select **Results** → **Direct Plot** → **PSS**. Note the prompts on the bottom of the schematic and ***PSS Results*** windows.
- The ***PSS Results*** window **MUST** be on the screen when probing the nodes in the schematic. Don’t push OK.
- In the PSS Results form, use the cursor to select the Pif net and Prf nets on the schematic. Press **Esc** to end this command.
- Click the **Switch Axis Mode** icon on the ***Waveform Window*** or select **Axes** → **To Strip**.



Display the Conversion Power Gain-method 1(continued)

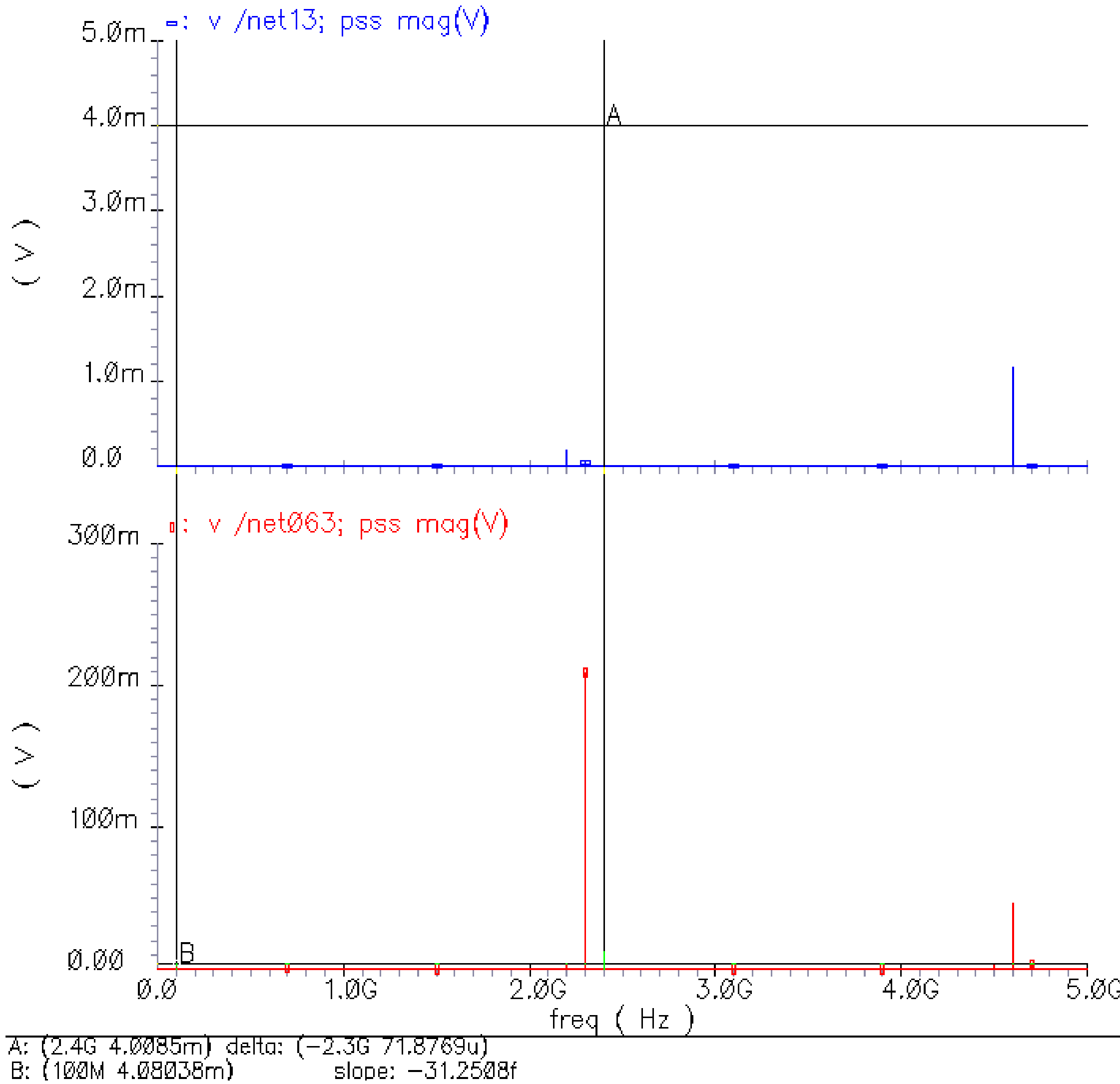
- Click the **Crosshair Marker A** icon and place the marker on the 2.4GHz harmonic of Prf.
- Click the **Crosshair Marker B** icon and place the marker on the 100MHz harmonic of Pif.
- Prf:

  Magnitude: 4.0085m

  Power: ≅ -38 dBm
- Pif:

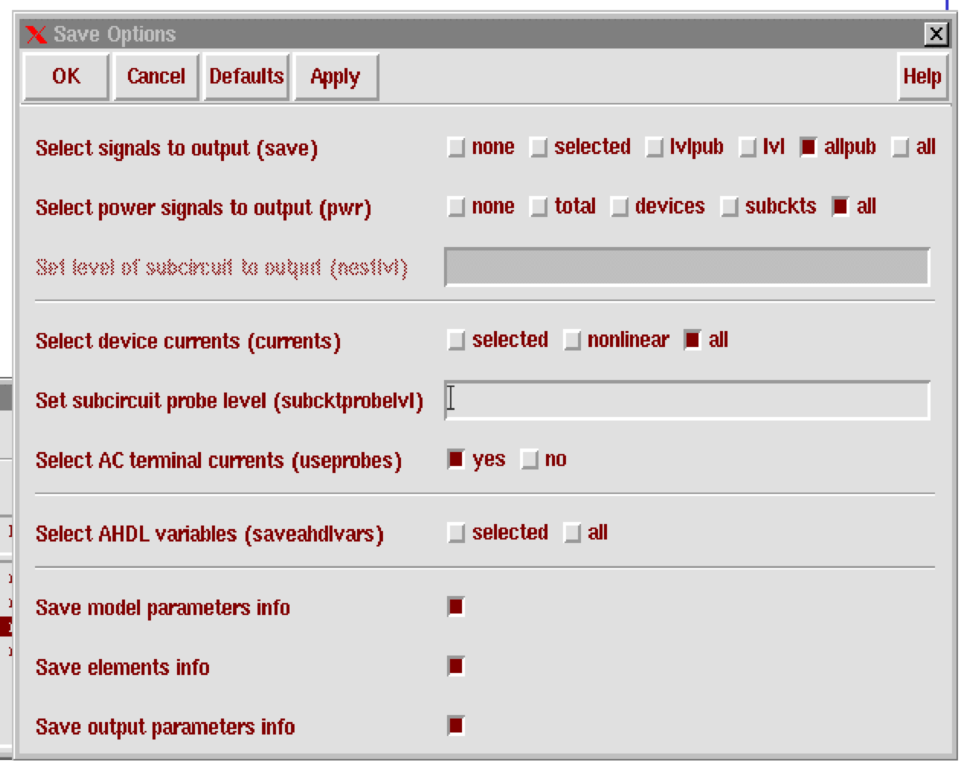
  Magnitude: 4.08038m

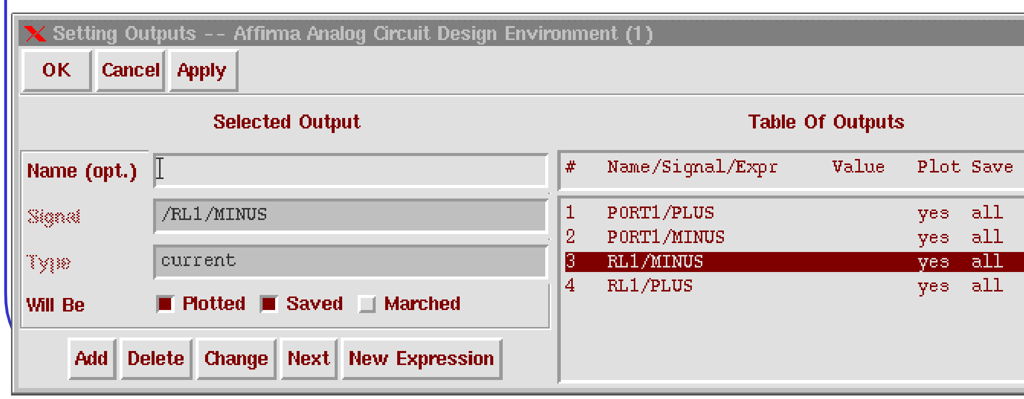
  Power: ≅ -37.8 dBm
- Conversion Power Gain ≅ 0.2dB + 3 dB = 3.2 dB



Display the Conversion Power Gain-method 2

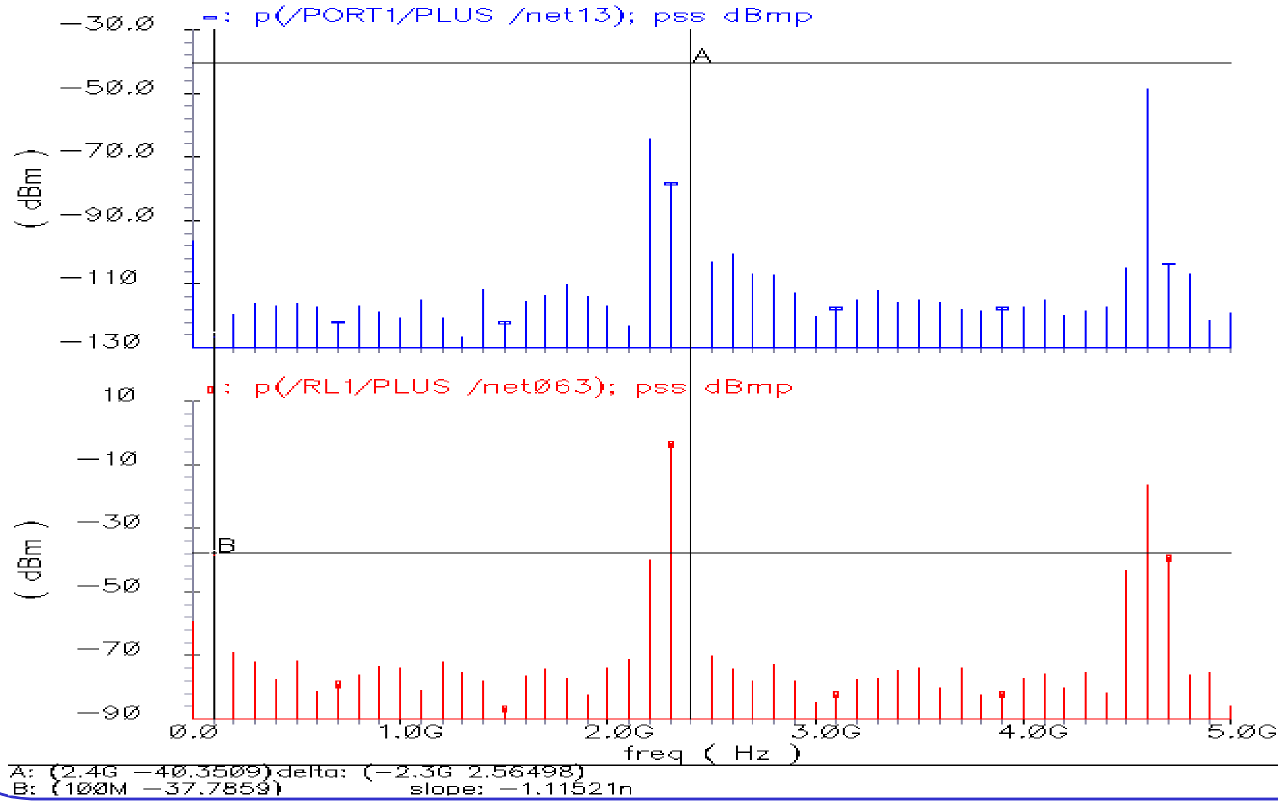
- Select **Output** → **Save All …** and the window "***Save Options***" appears. Set the buttons as below window in order to get the **AC** power!
- Select **Outputs** → **To Be Saved** → **Select On Schematic**. In the schematic, select the **PORT1** and **RL1**. The terminals are circled in the schematic window after you select them. Press **Esc** to end the selections.
- Double click on the name in the Outputs section or select **Outputs** → **Setup**. Set the outputs Will Be Plotted and Saved.

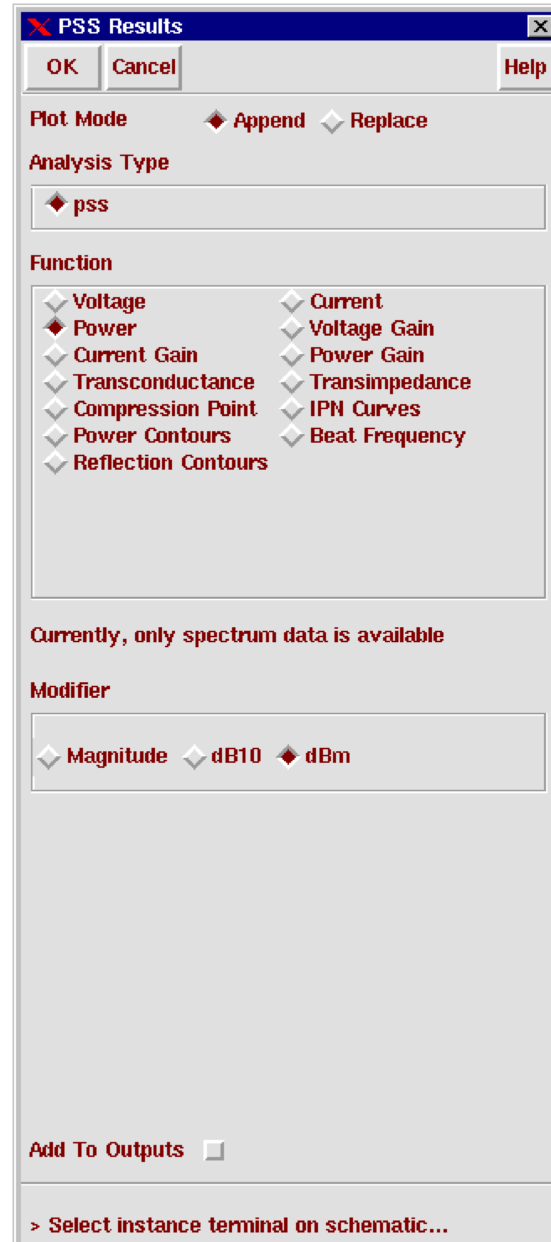




Display the Conversion Power Gain-method 2(Continued)

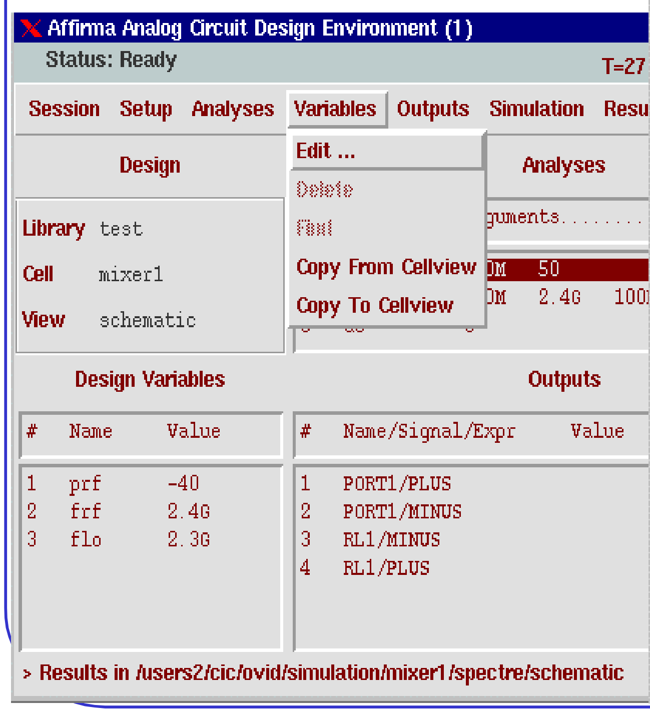
- Push **Netlist and Run** icon to run this simulation.
- Select **Results** → **Direct Plot** → **PSS**. Set the function and modifier as right; Select instance terminal(PORT1 & RL1) in the schematic window. Press Esc to end the selections.
- Compare the results to those of method 1.

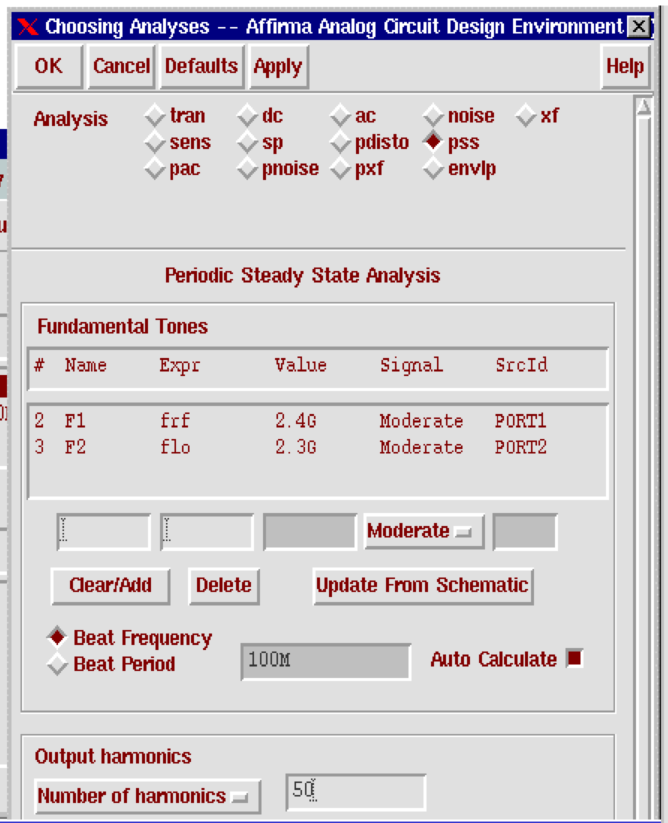


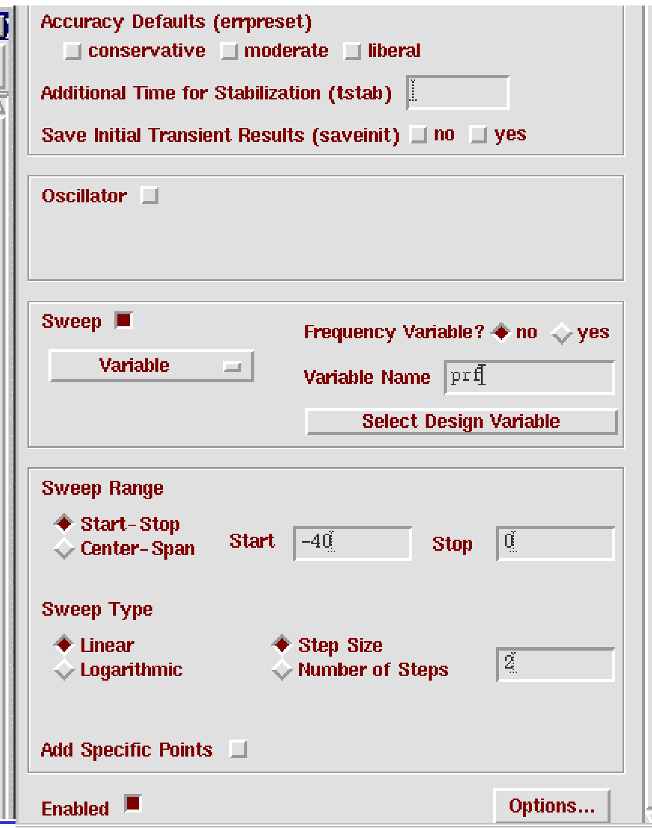


1 dB Compression Point Simulation

- *Change the* **Amplitude(dBm)** *of PORT1 to a variable* ***prf***; *Designate a value to* ***prf*** *in the* ***Design Variables*** *section.*
- In the Choosing Analyses window, turn on the **Sweep** button as shown here. Type in ***prf*** for the **Design Variable Name**, or click the **Select Design Variable** button, and highlight ***prf*** from a list , then click **OK**.
- Remember to check in the **INTEGRATION METHOD PARAMETERS** the method is **gear2only**.
- Select Netlist and Run button.

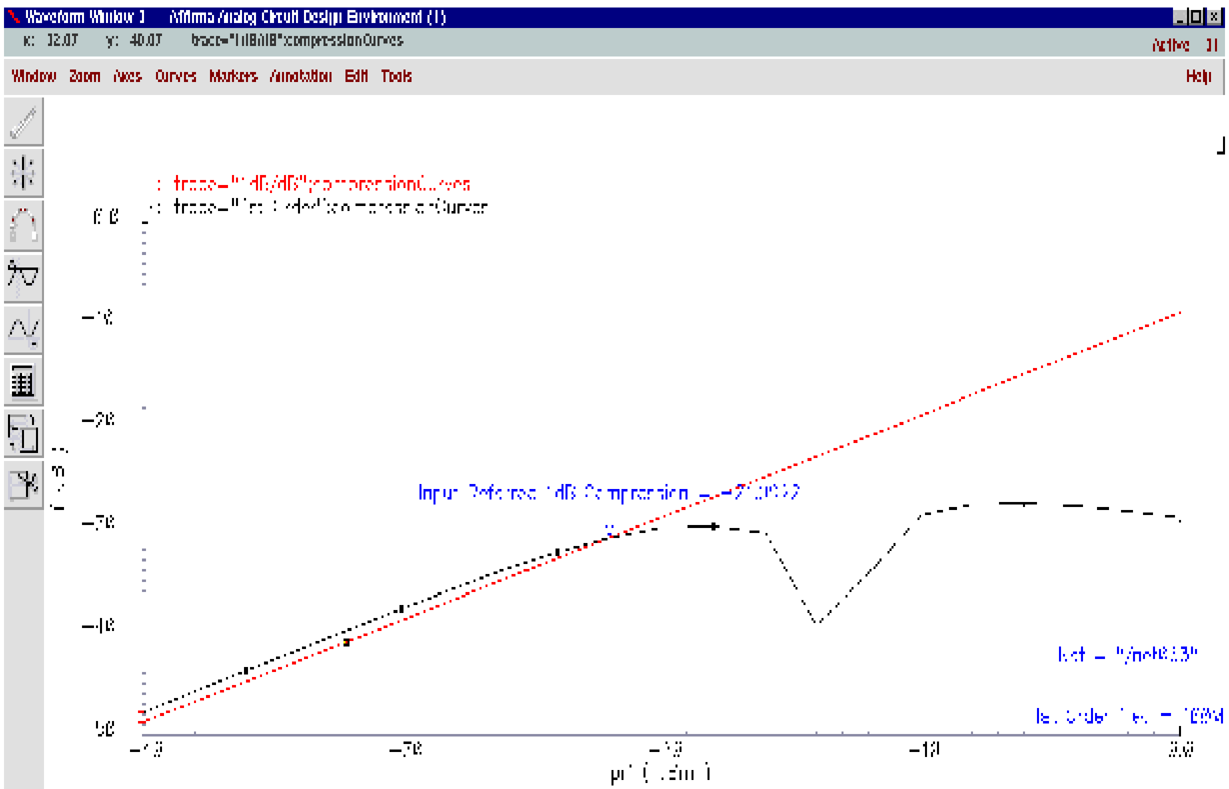


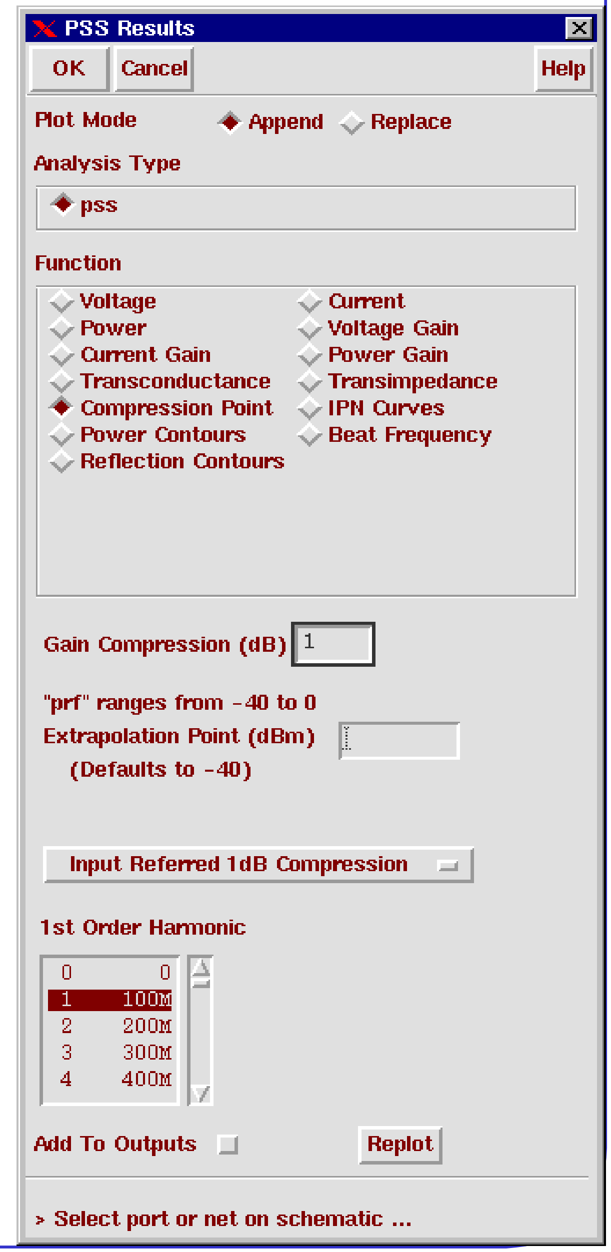




P1 dB Simulation Results

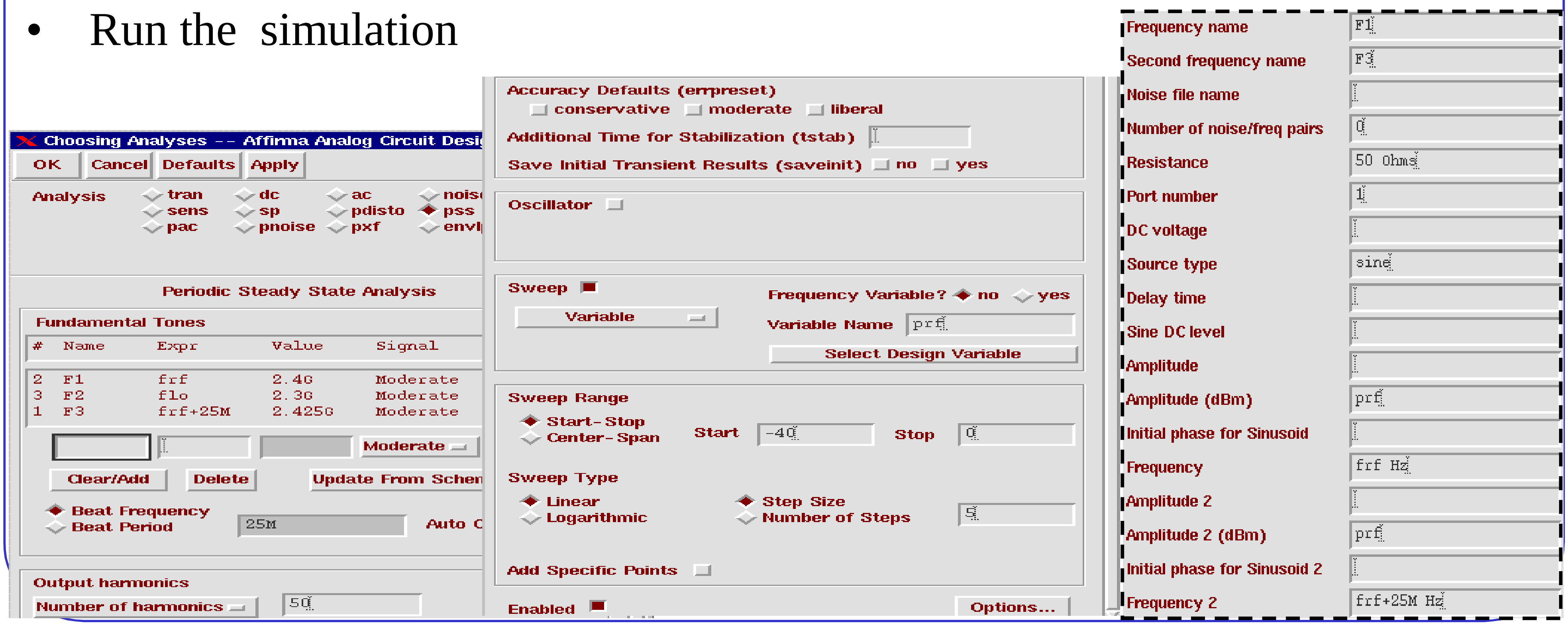
- Use Direct Plot function to see the results. Set up PSS Results form as shown here. Then select the **Pif** net in the schematic. With the cursor still in the schematic window, press **ESC** key to end the **Direct Plot command**.





Simulating IP3

- *PSS by itself is seldom used for IP3 simulation, because the separation between the 2-tone frequency is typically only a few Khz, and leads to a very long simulation time.*
- Edit **PORT1** properties as right. So The **Fundamental (Beat) Frequency** is now 25MHz.
- Set up ***Choosing Analysis*** form appears as shown below and push OK
- Run the simulation

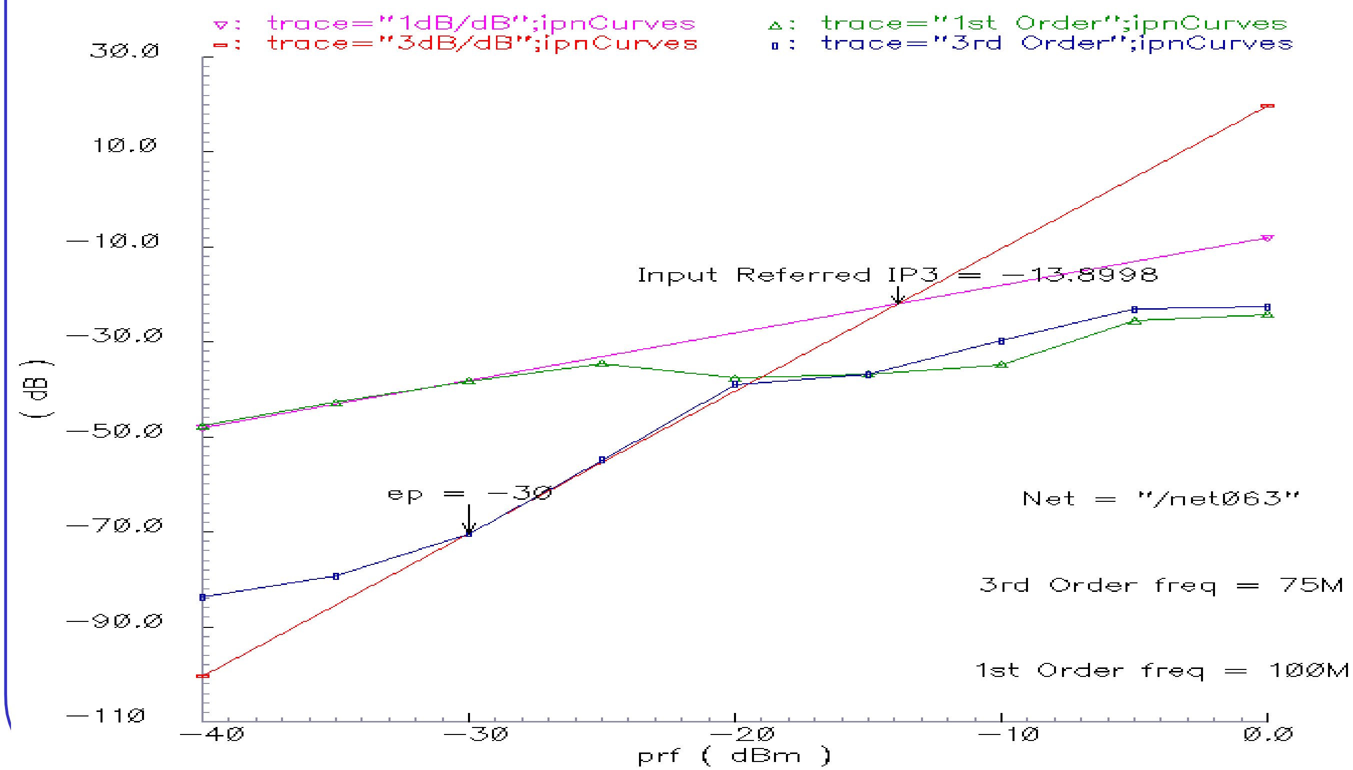


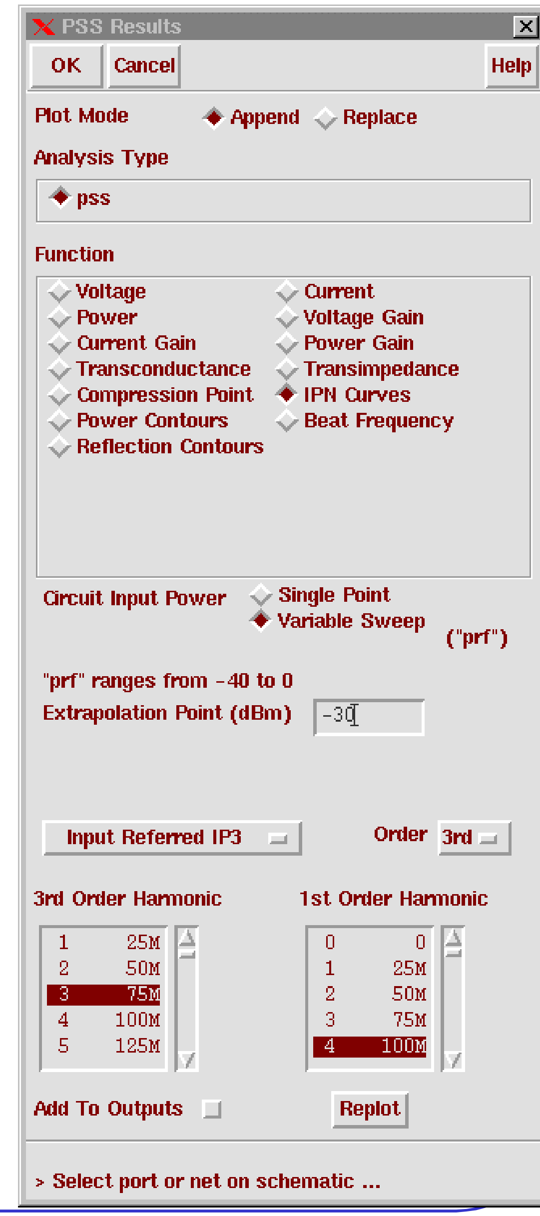
IP3 Results

- Use Direct Plot function to see the results. Set up PSS Results form as shown here. Then select the **Pif** net in the schematic. Press **ESC** key to end the **Direct Plot command**.

3rd order intermodulation product will occur at

$(2 \times 2.4\text{GHz} - 2.425\text{GHz}) - 2.3\text{GHz} =$ **75 MHz**





5. PAC Analysis

- PAC is a small-signal analysis like AC analysis, except the circuit is first linearized around a periodically varying operating point as opposed to a simple DC operating point. Linearizing around a periodically time-varying operating point allows analyzing transfer-functions that include frequency translation.

- When a small sinusoid is applied to a linear circuit that is periodically time-varying, the circuit responds with harmonics.

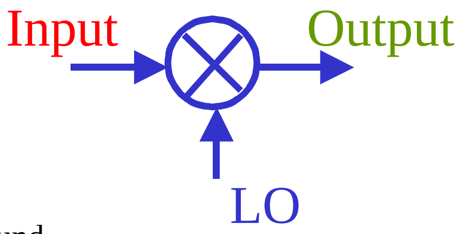
- PAC computes a series of transfer functions, one for each frequency. These transfer functions are unique because the input and output frequencies are offset by the harmonics of the LO.

PAC Analysis Overview

- PAC computes the transfer function from one input to many outputs. PAC is similar in concept to normal small-signal AC analysis, but it also calculates frequency conversion effects.

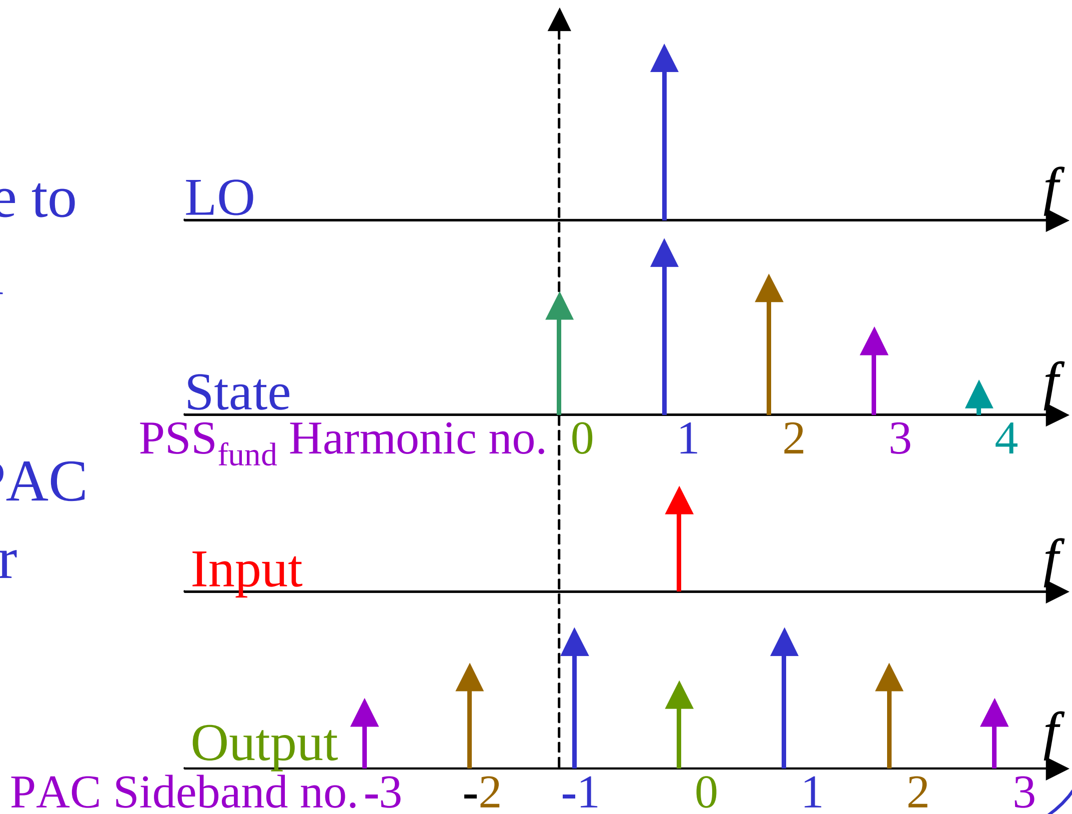
- By setting the ***maxsideband*** value to K_{max}, PAC generates all $2K_{max}+1$ sidebands from $-K_{max}$ to $+K_{max}$.

- The small-signal frequency in a PAC analysis can be arbitrarily close or even equal to the LO frequency.



$$f_{out} = f_{in} + Ki \times PSS_{fund}$$

where $\mathbf{f_{in}}$ represents the input frequency, and **Ki** are the PAC sidebands



Fundamental PAC Assumptions

- The PAC small signal analysis assumes that the circuit responds in a small signal fashion to the sinusoidal stimulus. This is accomplished by keeping the magnitude of the PAC signal at least 10 dB below the 1 dB GCP.
- The harmonics of the small signal PAC tone are not computed, although small signals can be used to measure distortion caused by the large signals present in the PSS analysis.
- For the transfer function to be accurate, a large number of time steps, during the PSS analysis, are needed at the small signal frequency. If the analysis frequency of the small frequency analysis is too high, the accuracy degrades. The ***maxaxfreq*** parameter of the PSS analysis can be used to specify the highest frequency that SpecteRF uses in subsequent small signal analyses.

PAC Analysis Summary

- Specify the following information when running a PAC analysis:

PSS fundamental	The number of harmonics should be no less than the PAC harmonics. *
Input port	Set type to dc and specify PAC magnitude
Input sweep frequency	Sweep, array or single point
Output frequencies of interest	Sidebands or Array of Indices
Results format	Plot results relative to output or absolute value of output frequency. Input is of little value and is not used.

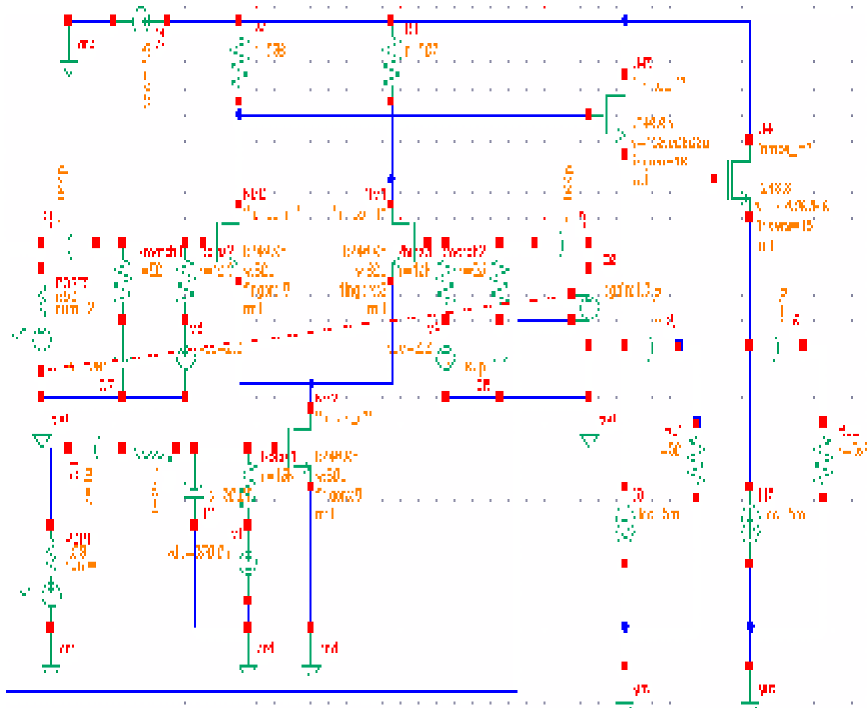
* When setting Output harmonics less than the PAC harmonics, be sure to set the ***maxacfreq*** parameter to assure that the simulator takes sufficient time points to accurately characterize the output waveform in the PSS analysis.

Lab4 : PAC Analysis

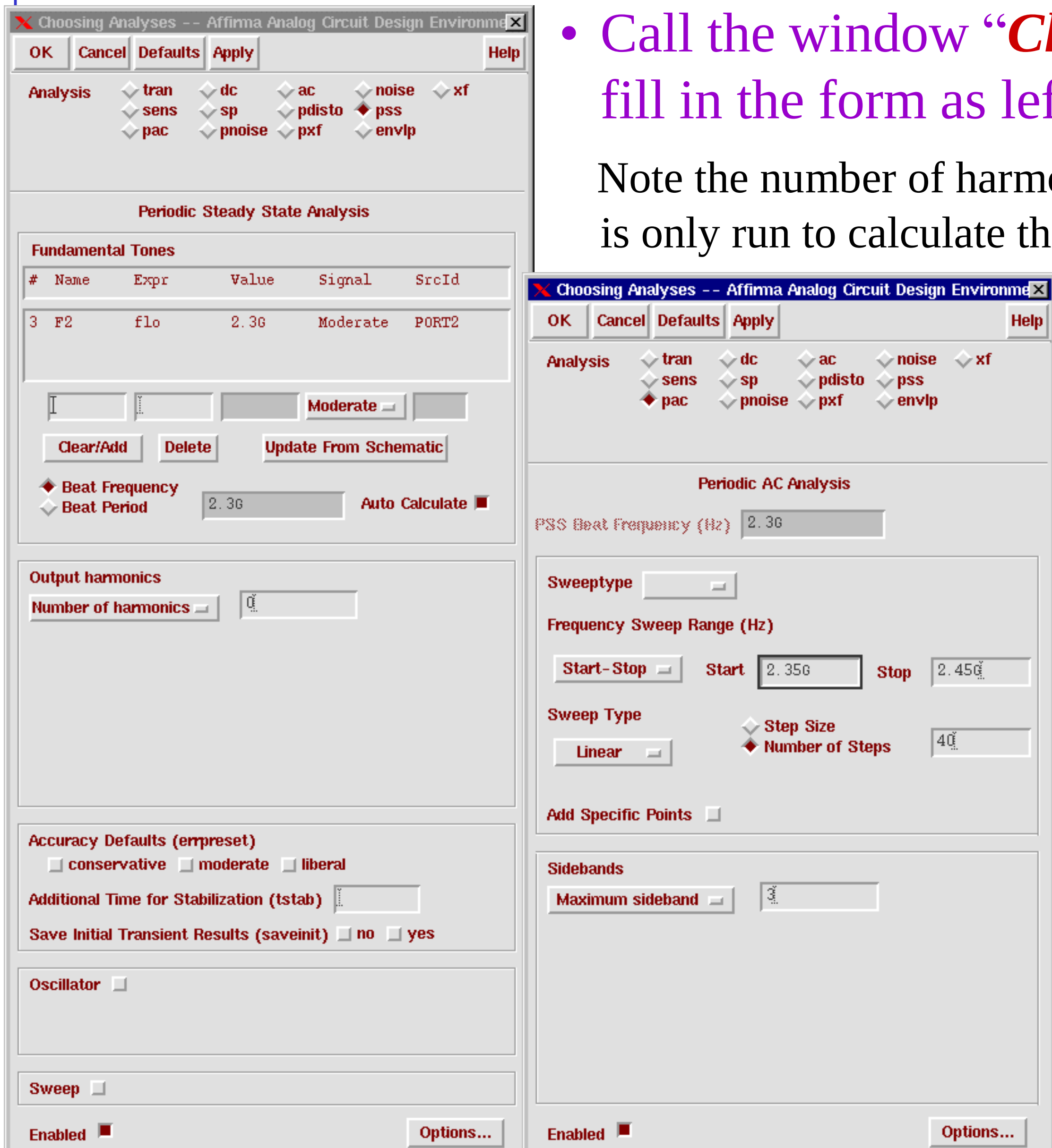
- Use the same schematic as Lab3.
- Modify the parameter values of PORT1 as below table.

Note : When the source type is set to **dc**, this signal will not be checked for coperiodicity with the other signals; this source will be treated as a small signal. When the source is set to **sine**, it will be considered "large signal".

Parameter	Value
Resistance	50
Source type	dc
Frequency	frf
PAC magnitude	1
Amplitude (dBm)	prf
Amplitude2 (dBm)	(blank)
Frequency2	(blank)



Setting Up the PAC Simulation



- Call the window "***Choosing Analyses***"; In the **pss** form, fill in the form as left; then click **Apply**.

  Note the number of harmonics is set to 0, because the PSS simulation is only run to calculate the large-signal, steady state solution.

- Click on **pac** in the ***Choosing Analyses*** form, and setup the form as left; then click **OK** .

  The Frequency Sweep Range sets the sweep range on the psin(port) component at the input port which has a PAC magnitude parameter value specified.

  The value for Maximum sideband is relative to the Fundamental frequency. Since the LO frequency and PSS_{fund} are equal, you get the results of mixing the RF with the 0 through 3rd harmonic of the LO.

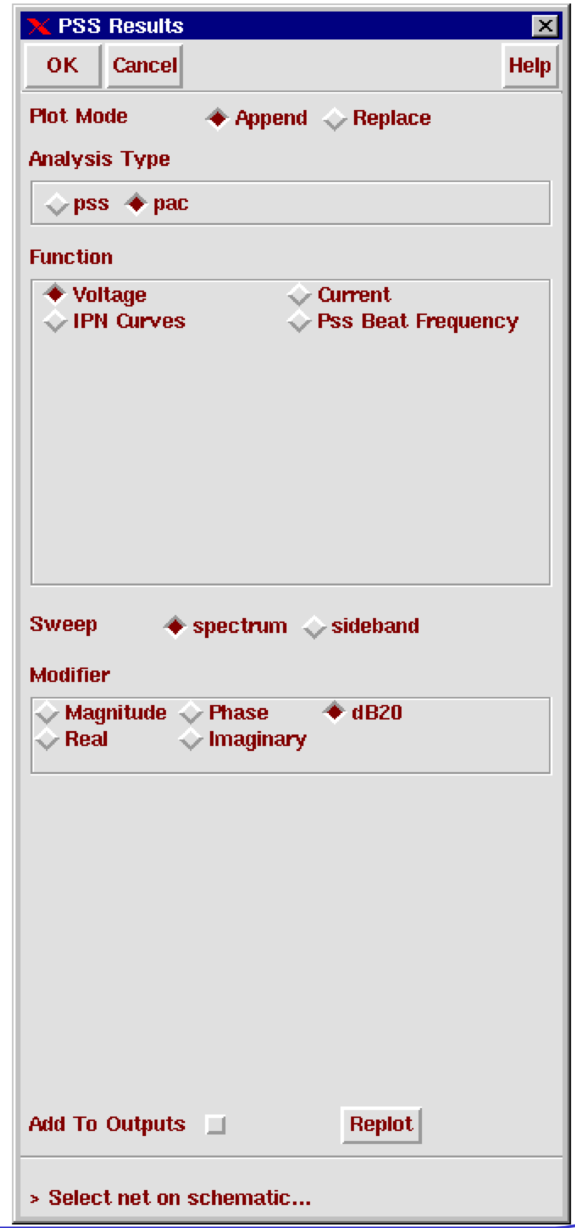
- Select **Netlist and Run**.

Plotting the Conversion Gain

- Note how much faster this simulation runs than the previous method used to calculate CG.

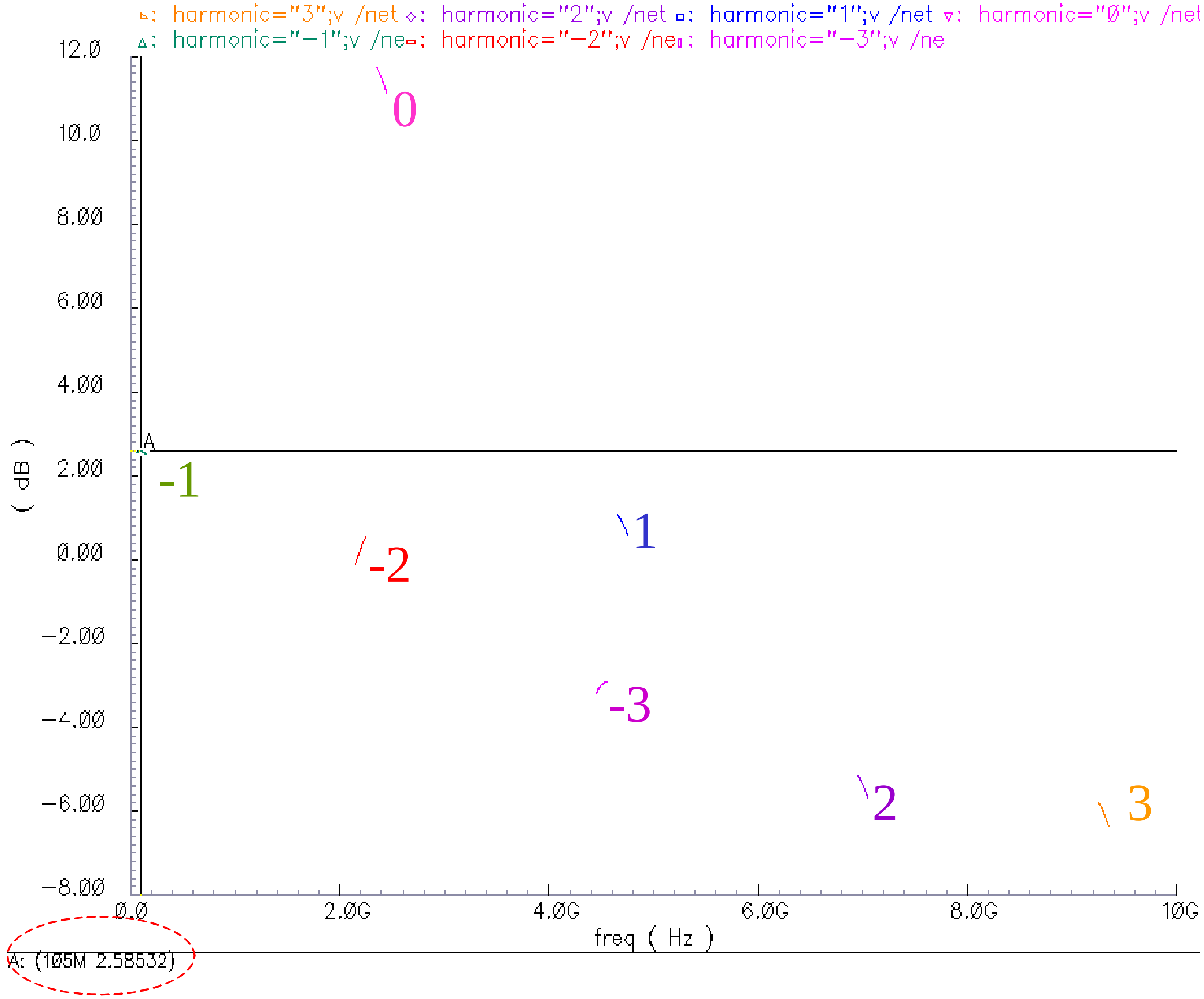
- Use **Direct Plot** function to see the results.

- In the schematic window, select the Pif node, and the result are plotted as next page:



Periodic Steady State Response

- To measure the CG, move the marker to the 100MHz position in the waveform window and read the gain.

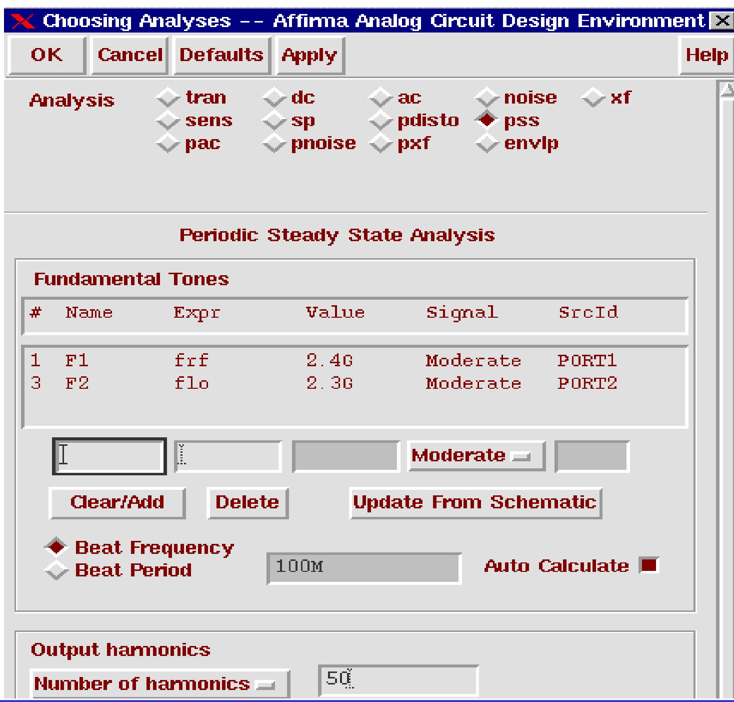


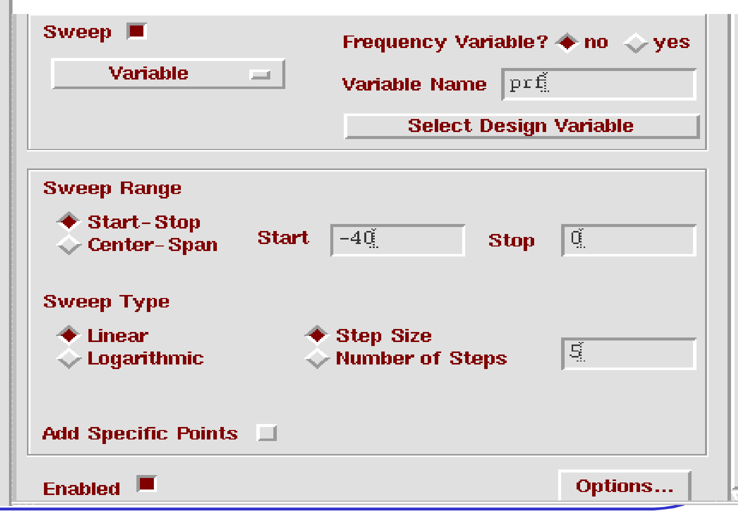
Note if the input and output port are both matched to 50ohm, we get conversion power gain; otherwise we get conversion voltage gain.

Running a Swept Small-Signal IP3 Simulation

- Modify the parameter values of **PORT1** as right table; then **check and save**!
- Select **pss** in the **Choosing Analyses** form, and setup the form as below : Note now the **Fundamental Frequency** is 100 MHz
- Set the **Number of harmonics** to 50 and you have the harmonics available to view; it won't affect the simulation time.
- Click **Apply**! The **Choosing Analyses** form is still active on the screen.

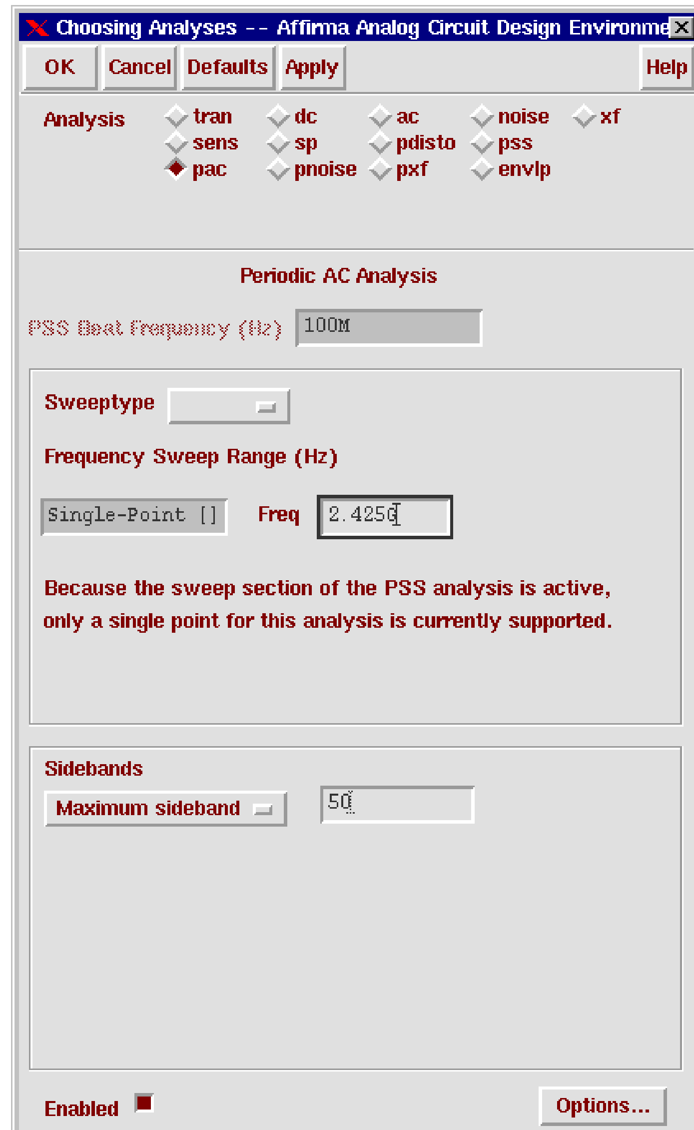
Parameter	Value
Resistance	50
Source type	sine
Frequency	frf
PAC magnitude	(blank)
PAC magnitude (dBm)	prf
Amplitude (dBm)	prf
Amplitude2 (dBm)	(blank)
Frequency2	(blank)





Setup up the PAC Simulation

- In the **Choosing Analyses** form, select **pac**; then set up the form as right:
- This simulation applies a 2.425GHz tone in the PAC analysis to compare the results by the swept PSS. This PAC test tone is typically separated according to "***channel spacing***".
- Click **OK**.
- Select **Netlist and Run**.

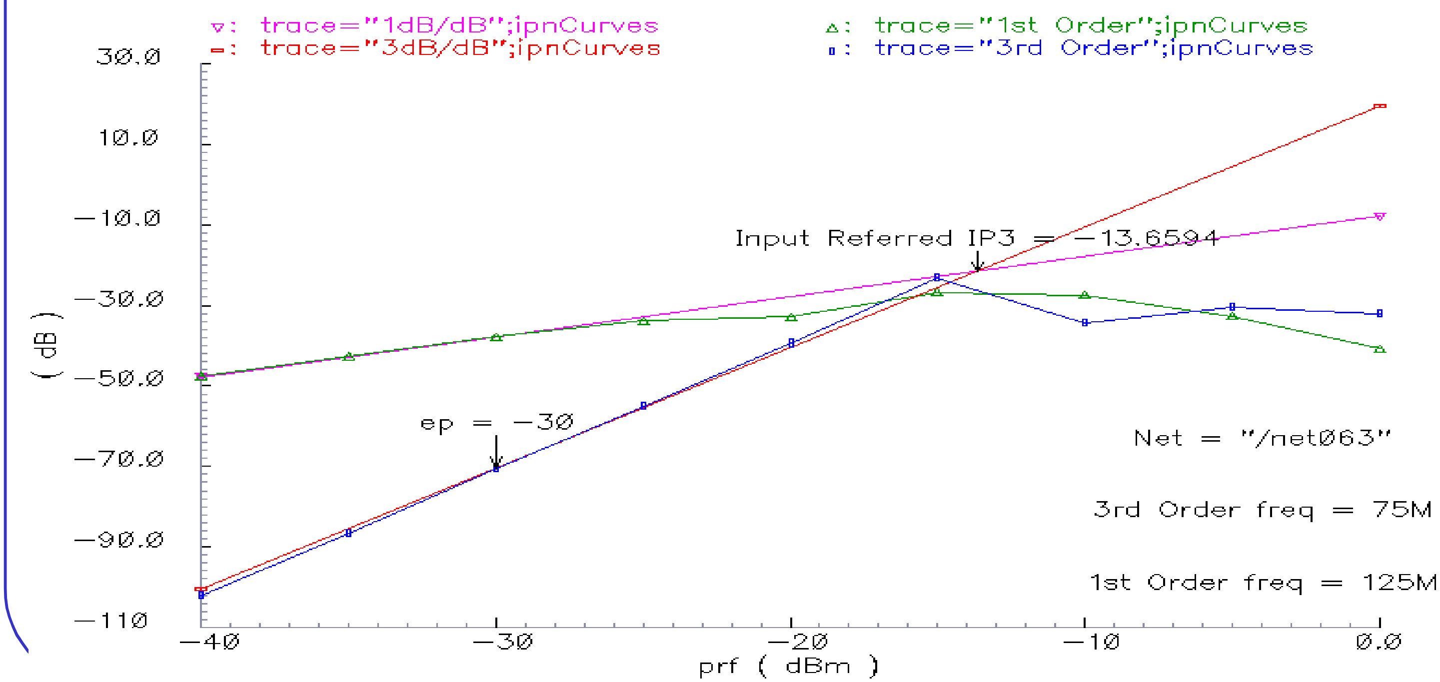


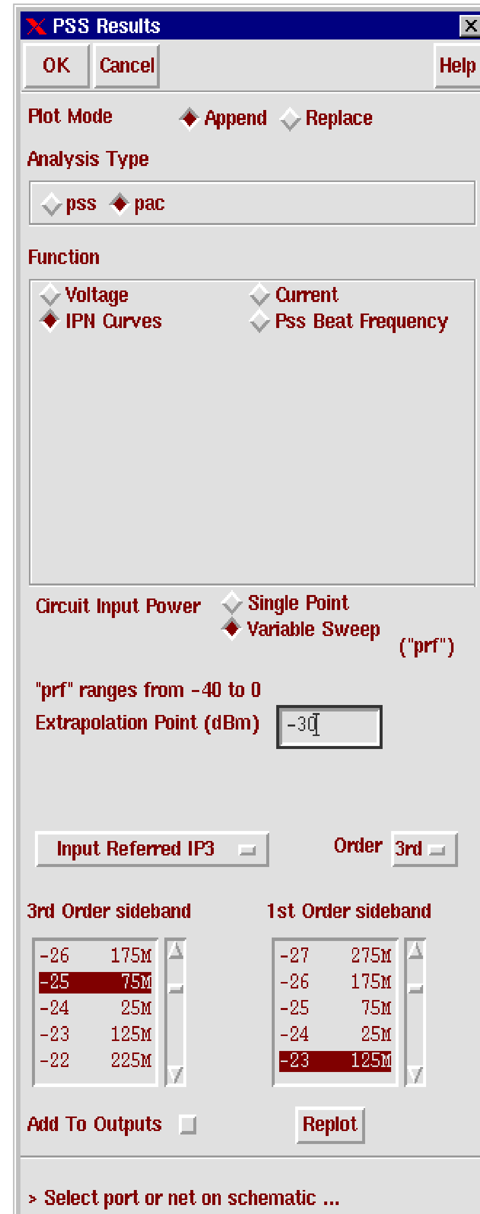
IP3 Results

LO: 2.3 G RF: 2.4 G & 2.425G

1st order harmonics: 100M & 125M 3rd order harmonics: 75M & 150M

- The only 1st and 3rd order pair available from this analysis (due to the 100MHz PSS_{fund}) is ***125M*** and ***75M***.
- Use **Direct Plot** function; select the **Pif** net in the schematic window.
- Compare the IP3 values using 2 different method!





6. PXF Analysis

- The periodic transfer function (**PXF**) analysis directly computes such useful quantities as *conversion efficiency*(the transfer function from input to output at a preferred frequency), *image and sideband rejection*, and *power supply rejection.*

- The primary use of PXF analysis is to measure various conversion gains. This is very valuable when looking at different spurs on the input of a receiver.

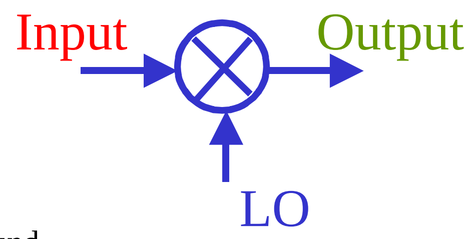
- **PXF** can be a better choice for calculating CG than PAC, because **PXF** will provide information on all of the frequencies on the RF port that are converted to the IF band.

- When simulating oscillators, PXF can determine the ***tstab*** value.

PXF Analysis Overview

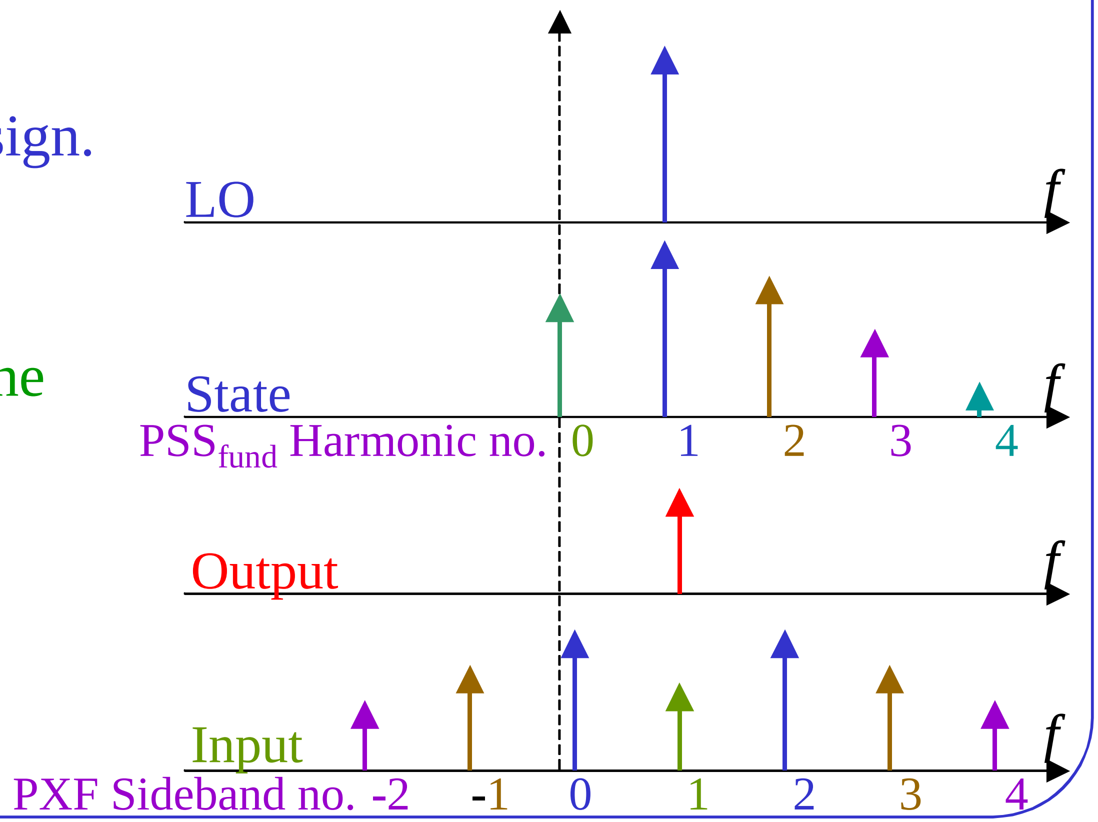
- The **PXF** analysis computes the energy contributions from all source harmonic frequencies to a signal or swept output frequency. In this way, a single output response is the combination of all possible frequency components in the design.

- Set the ***maxsideband***, or the sidebands parameters, to select the periodic small-signal input frequencies of interest, while sweeping the selected output frequency.



$$f_{in}^{k} = f_{out} + k \times PSS_{fund}$$

where $\mathbf{f_{out}}$ represents the output signal frequency; **k** is the PXF sidebands number



Fundamental PXF Assumptions

- The **PXF** small signal analysis assumes that the circuit responds in a small signal fashion to sinusoidal stimulus. **SpectreRF** is not capable of computing the distortion caused by the small signals, although small signals can be used to measure distortion caused by the large signals present in the PSS analysis.

- To increase accuracy, choose a large number of time steps during PSS analysis. If the analysis frequency of the small signal analysis is too high, the accuracy of the results degrade. The ***maxacfreq*** parameter of the PSS analysis specifies the highest frequency uses in subsequent small signal analyses.

PXF Analysis Summary

- Specify the information in this table when running a PXF analysis.

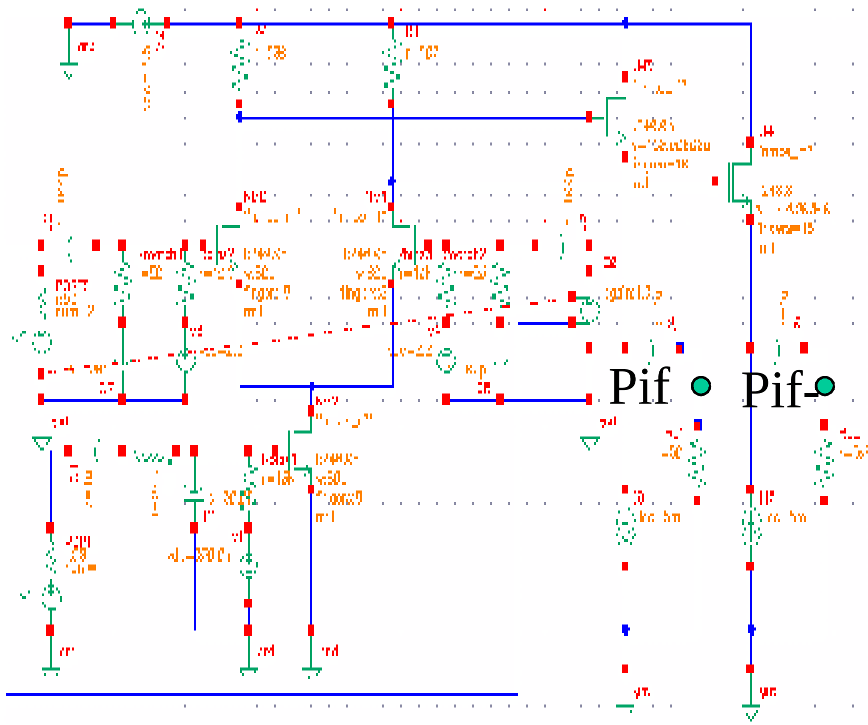
PSS fundamental	The number of harmonics should be no less than the PXF harmonics. *
Output net (v) or	Specify in form
Voltage source (i)	(To measure current, put a 0v battery in series with the branch.)
Output sweep frequency	Sweep, array or single point
Input frequencies of interest	Sidebands
Results format	Plot results relative to input or absolute input value of input frequency. Output is of little value and is usually not used.

* When setting Output harmonics to 0, be sure to set the ***maxacfreq*** parameter to assure that the simulator takes sufficient time points to accurately characterize the output waveform in the PSS analysis.

Lab5 : PXF Analysis

- Because **PXF** is a small signal analysis, only one large signal tone, the LO, is required. Set the **PORT1** as follows:

Parameter	Value
Resistance	50
Source type	dc
Frequency	frf
PAC magnitude (dBm)	(blank)
Amplitude (dBm)	prf
Amplitude2 (dBm)	(blank)
Frequency2	(blank)

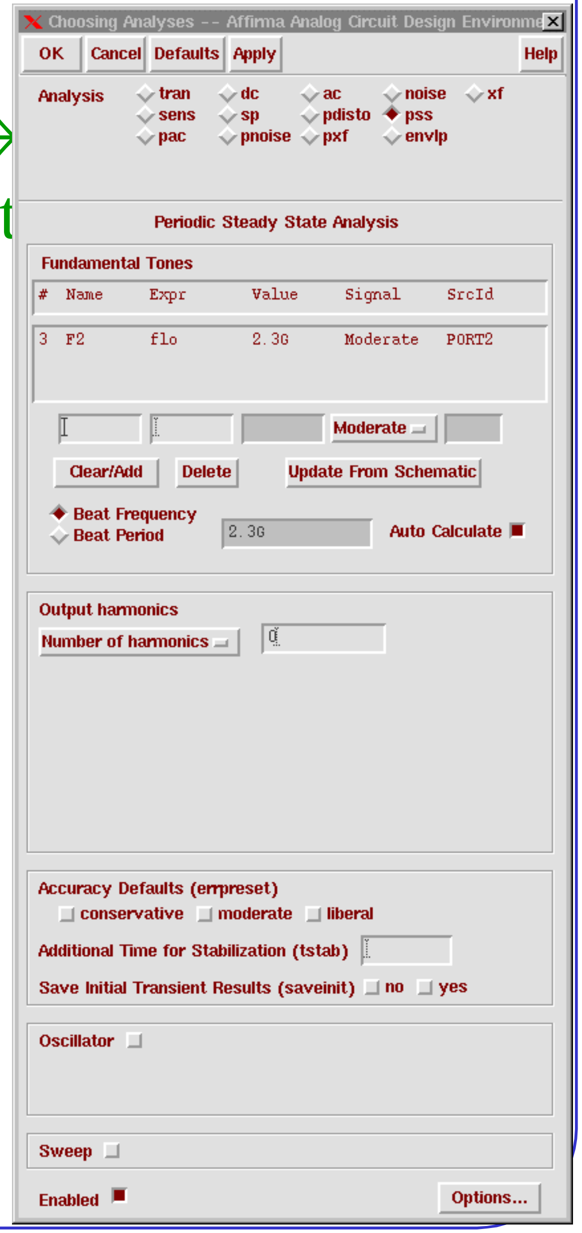


Setting Up the PXF Simulation(1)

- In the Simulation window, select **Analyses** → **Choose**; turn off the **pac** analysis. Then select the **pss** analysis, and set up the form as right:

- Note the number of harmonics is set to **0**, because the **PSS** simulation is only run to calculate the large-signal, steady state solution. Therefore set a value for ***maxacfreq*** in the **PSS** Options form. Set ***maxacfreq*** to 4 GHz.

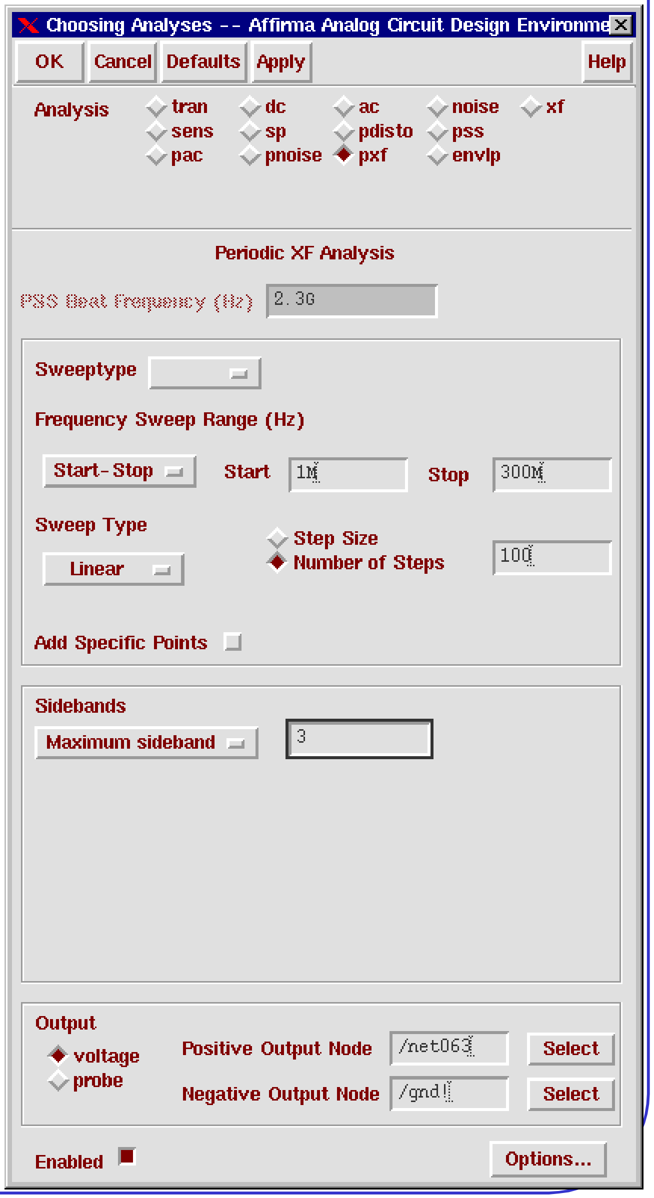
- Click **Apply** in the ***Choosing Analyses*** form.



Setting Up the PXF Simulation(2)

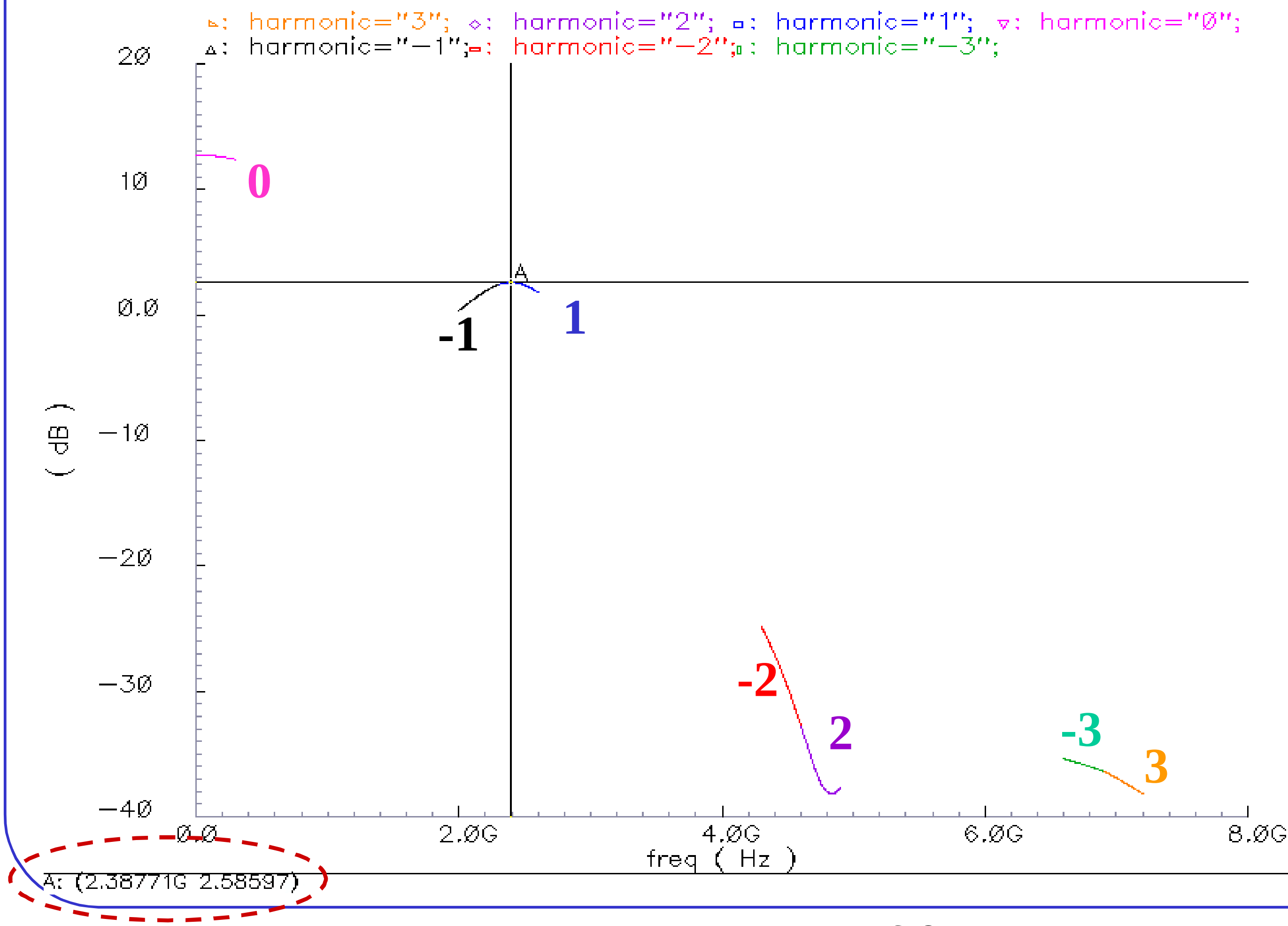
- Click on **pxf** in the ***Choosing Analyses*** form, and setup the form as left; then click **OK** .
- The ***Frequency Sweep Range*** is specified from 1MHz to 300 MHz. The **PXF** analysis will calculate all inputs that produce this range of frequencies at the **Pif** port.
- To set the ***Positive Output Node***, click the Select button, and select the **Pif** node in the schematic.

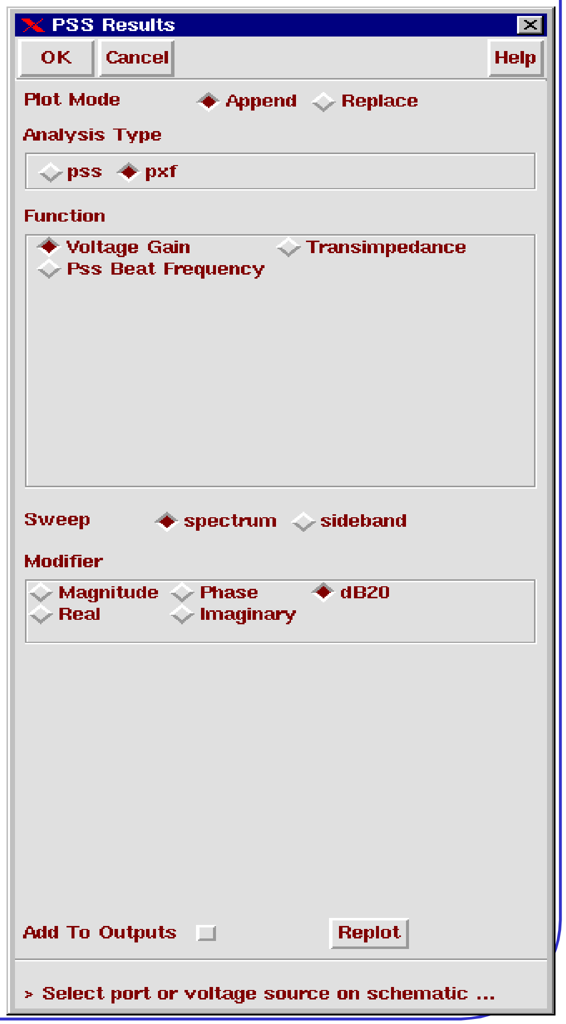
- Click the **Netlist and Run.**



Plotting the RF to IF Conversion Gain

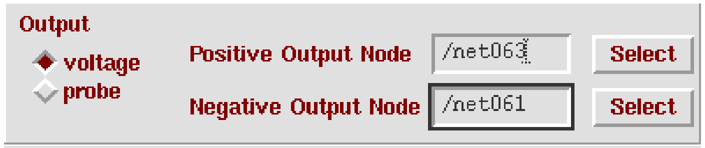
- Use **Direct Plot** function to see the results. In the PSS Results form, select **pxf** button. Follow the prompts at the bottom of the form, and select the port component (PORT1) in the schematic

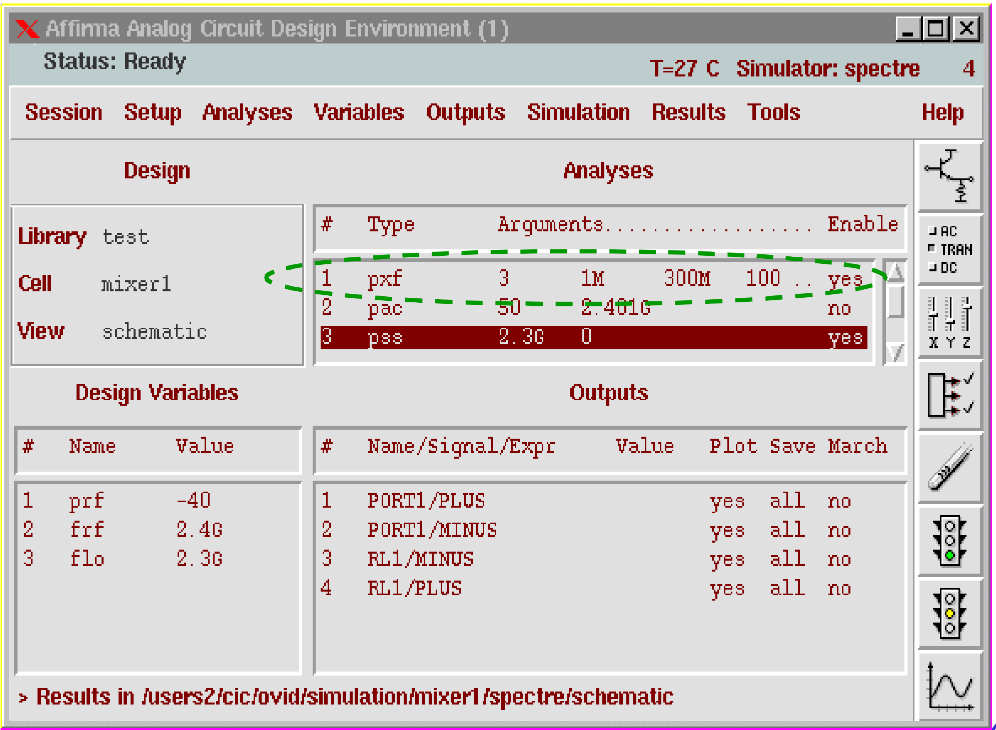




Power Supply Rejection

- Double click on the **pxf** analysis in the window "***Design Environment***", and the ***Choosing Analyses*** form appears. Change the **Negative Output Node** to **Pif-** (/net016) in the pxf form, then click ok.

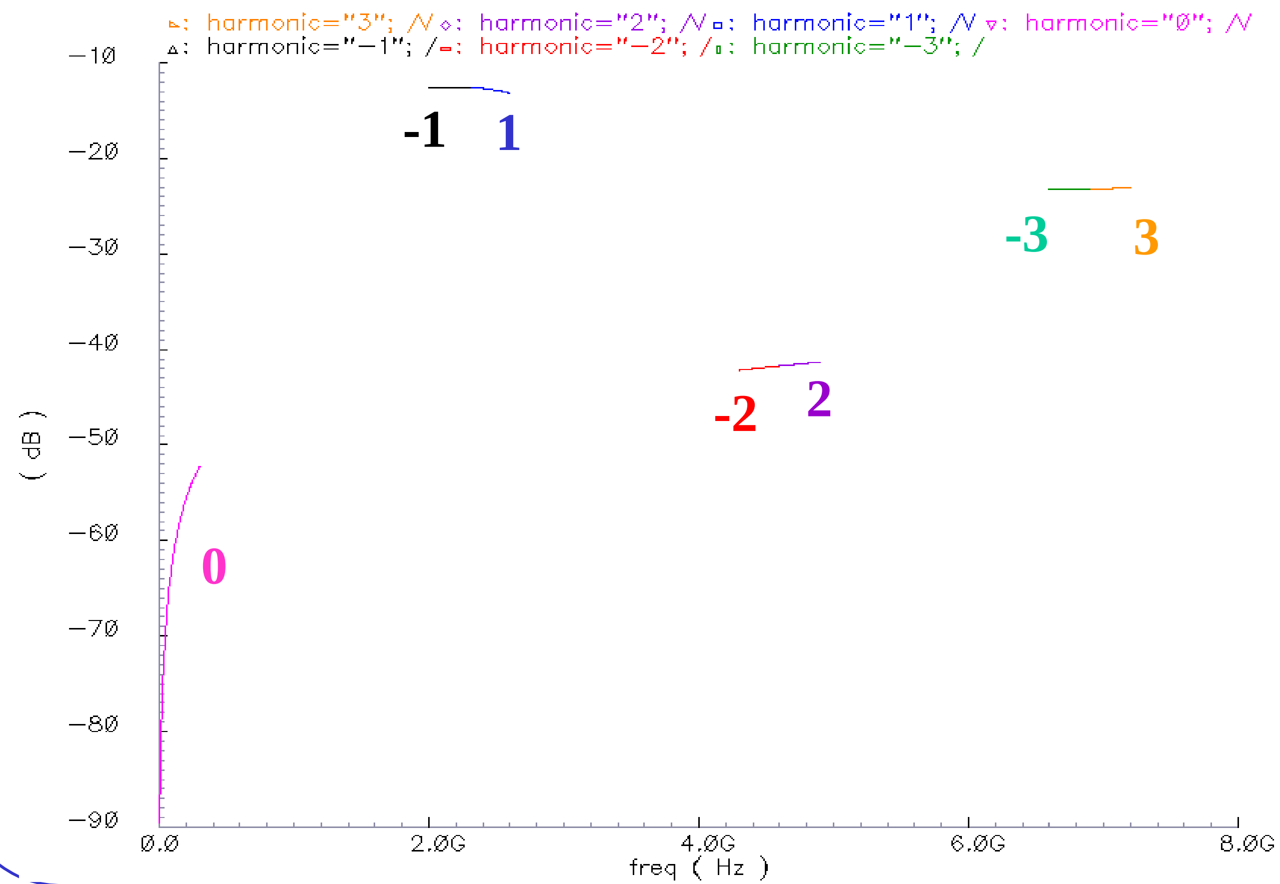


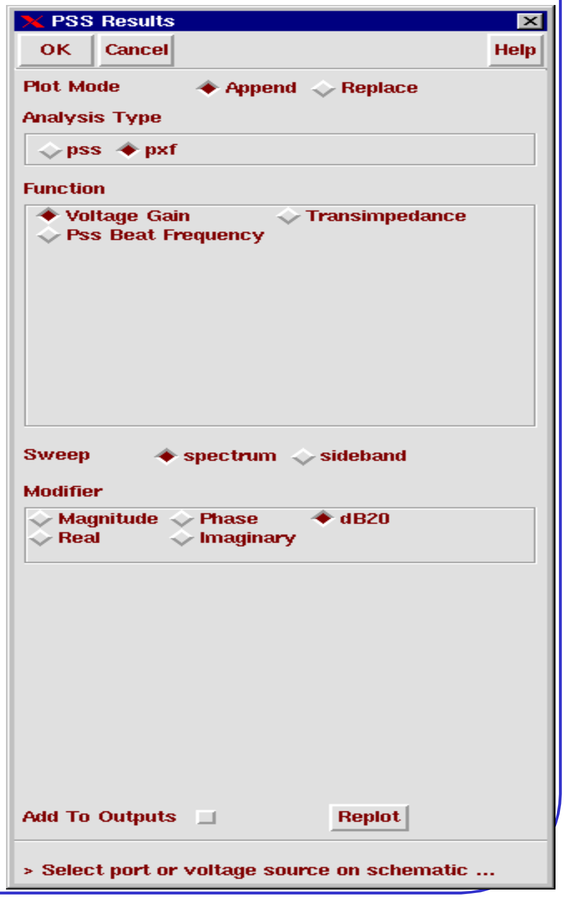


- Run the simulation.

Plotting the Power Supply Rejection

- Use **Direct Plot** function to see the results. In the PSS Results form, select pxf button. Follow the prompts at the bottom of the form, and select the DC supply (vdc=2.5v) in the schematic

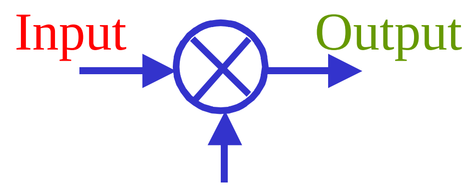




7. PNOISE Analysis

- PNOISE analysis, unlike conventional noise analysis, computes frequency convention effects, noise folding, aliasing.
- For noise sources that are bias dependent, such as shot noise sources, the time-varying operating point acts to modulate the noise sources. The transfer function from the noise source to the output is also periodically time-varying, and so acts to modulate the contribution of the noise source to the output. The effect of a periodically time-varying bias point on the noise generated by the various components in the circuit is also included.
- Include the effects of thermal noise, shot noise, and flicker noise.

PNOISE Analysis Overview(1)

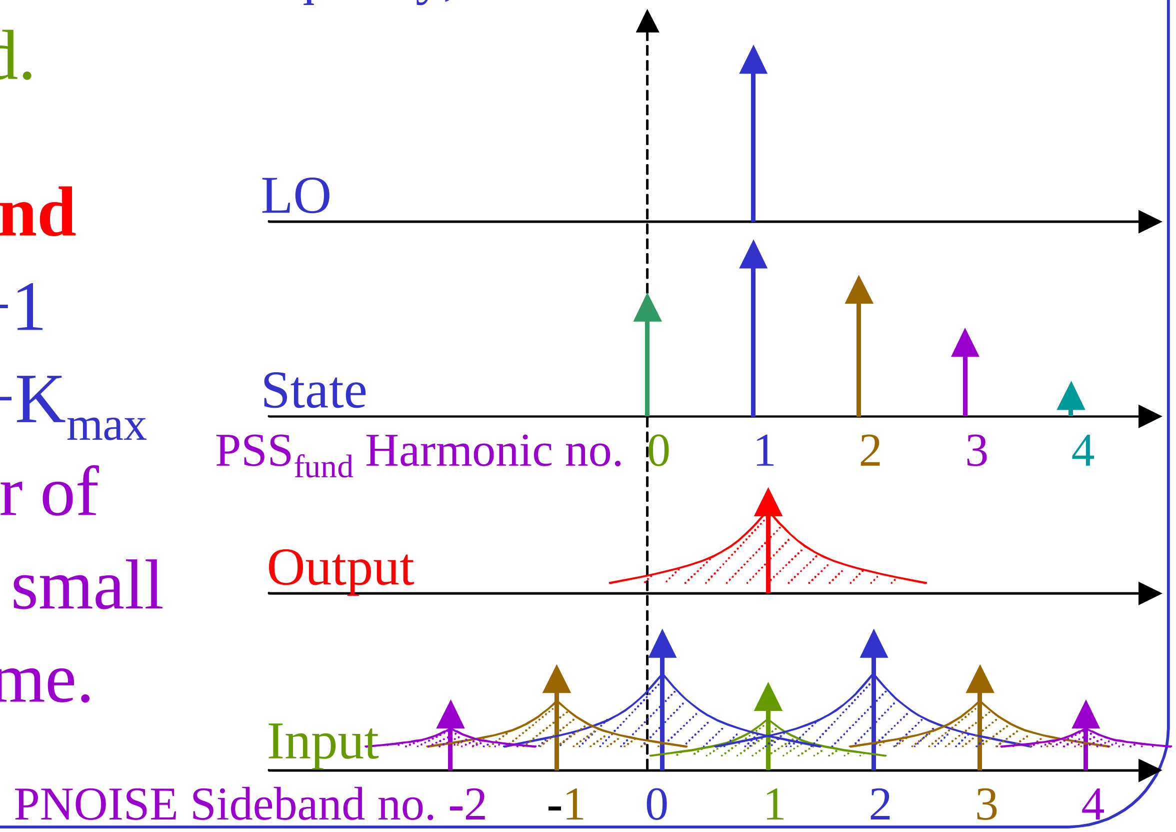


- The final result of the analysis is the sum of the noise contributions from both the up-converted and down-converted output frequency specified.

$$f_{noise\_source} = f_{out} + K_i \times PSS_{fund}$$

where f_{out} represents the output signal frequency; **Ki** is the PNOISE sidebands no.

- By setting the **maxsideband** value to K_{max}, all $2 \times K_{max}+1$ sidebands from $-K_{max}$ to $+K_{max}$ are generated. The number of requested sidebands has a small effect on the simulation time.



PNOISE Analysis Overview(2)

- When the reference sideband has any value other **0**, Single Sideband (**SSB**) NF is calculated. To determine the reference sideband, run a **PXF** analysis.

- The ***Noise Summary Table*** displays the following data:
  - Noise contribution (value and %) for each component in the circuit
  - Total output noise
  - Total input referred noise

Fundamental PNOISE Assumptions

- The small signal analyses compute transfer function by using time-domain techniques. The time steps used in these time-domain computations are the same as those in PSS analysis. For accuracy, the PSS analysis needs to have many data points at the highest frequency that you want to analyze in the noise analysis.

- More sidebands yield greater accuracy, but they take longer to simulate and use more disk space. If the analysis frequency of the small signal analysis is too high, the **Spectre simulator** warns. Use the **maxacfreq** parameter of the **PSS** analysis to specify the highest frequency for **SpectreRF** to use in subsequent small signal analyses.

PNOISE Analysis Summary

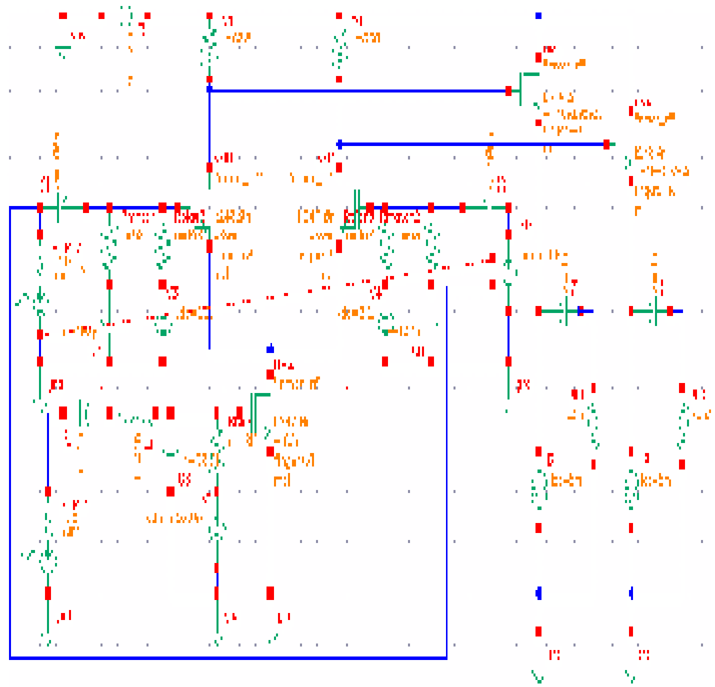
- Specify the information in this table when running a PNOISE analysis.

PSS fundamental	The number of harmonics will likely be no less than the PNOISE harmonics.
Output net (v) or Voltage source (i)	Specify in form
Output sweep frequency	Sweep, array or single point
Input frequency contributors	Sidebands
Input Sources	Port, voltage or current sources
Reference Sideband	Noise figure and Input referred noise

Lab6 : Noise Figure

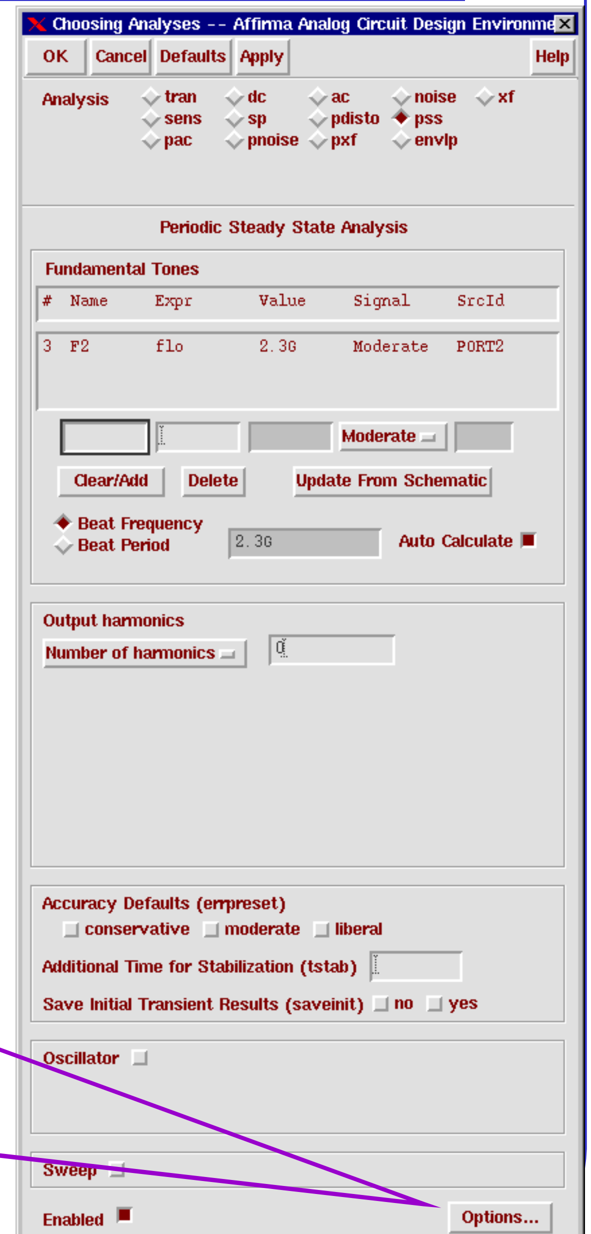
- Modify the parameter values of **PORT1** as follows:
- In the Simulation window, select **Analyses** → **Choose**; turn off the **pxf** analysis.

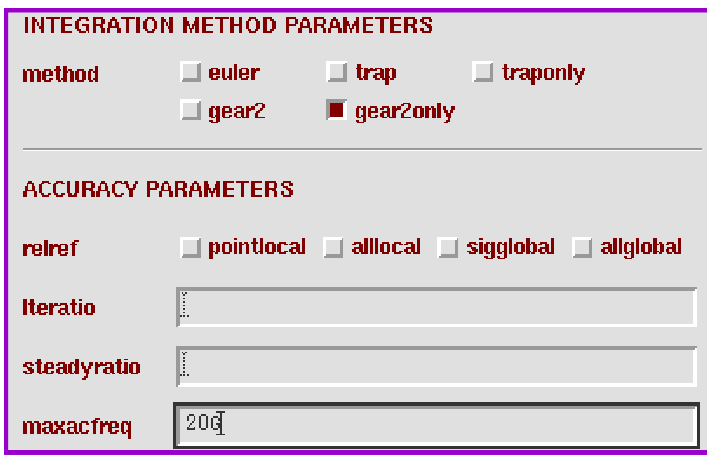
Parameter	Value
Resistance	50
Source type	dc
Frequency	frf
PAC magnitude (dBm)	(blank)
Amplitude (dBm)	(blank)
Amplitude2 (dBm)	(blank)
Frequency2	(blank)



Setting Up the PNOISE Simulation(1)

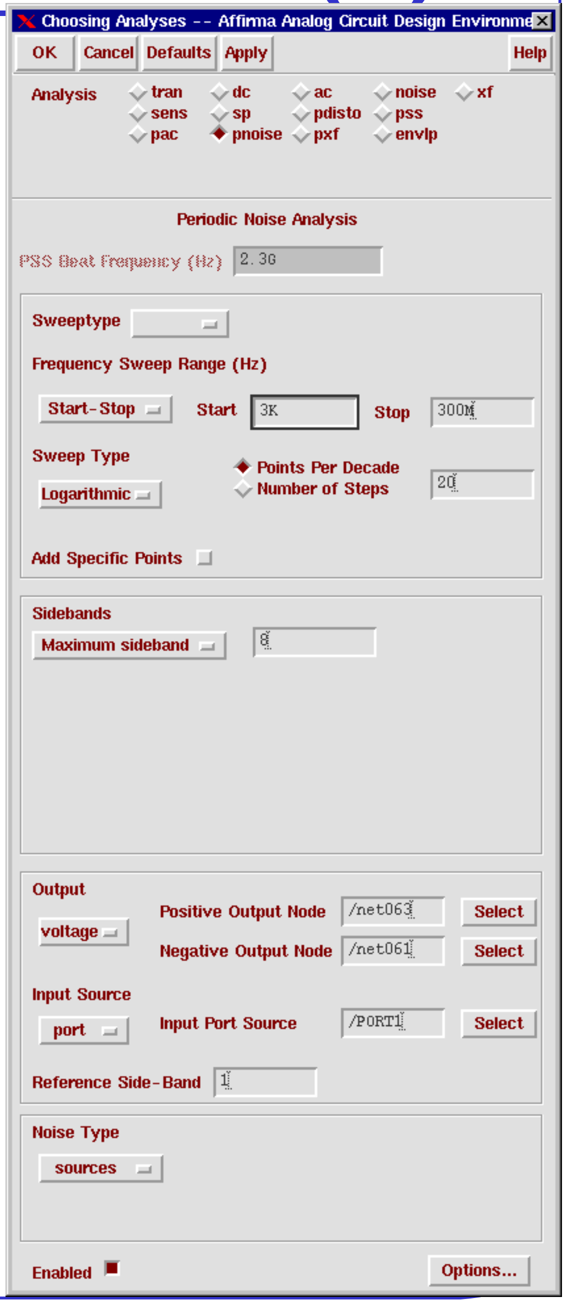
- Then select the **pss** analysis, and set up the form as right:
- Set a value for **maxacfreq** in the **PSS** Options form. Set **maxacfreq** to 20GHz. Remember to set the integration method to **gear2only**.
- Click **Apply** in the ***Choosing Analyses form***.





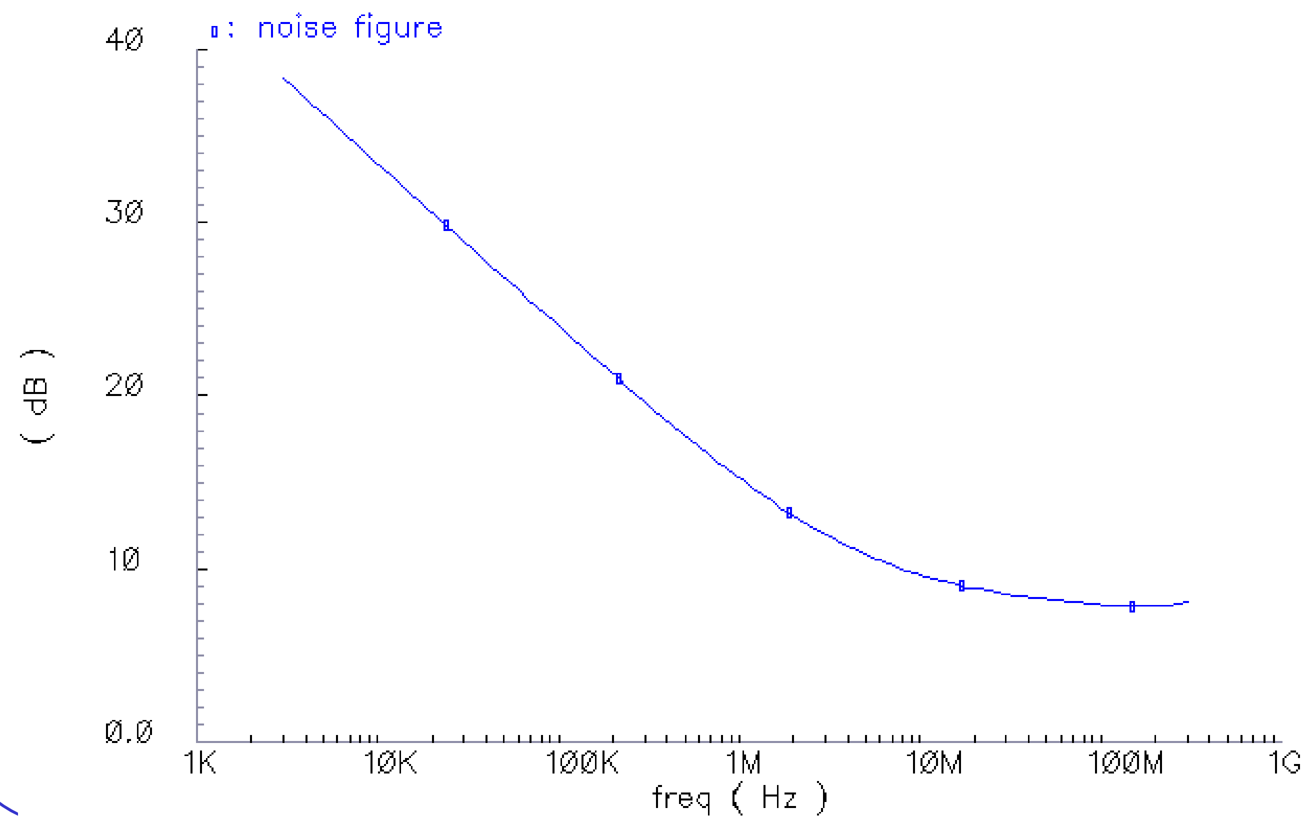
Setting Up the PNOISE Simulation(2)

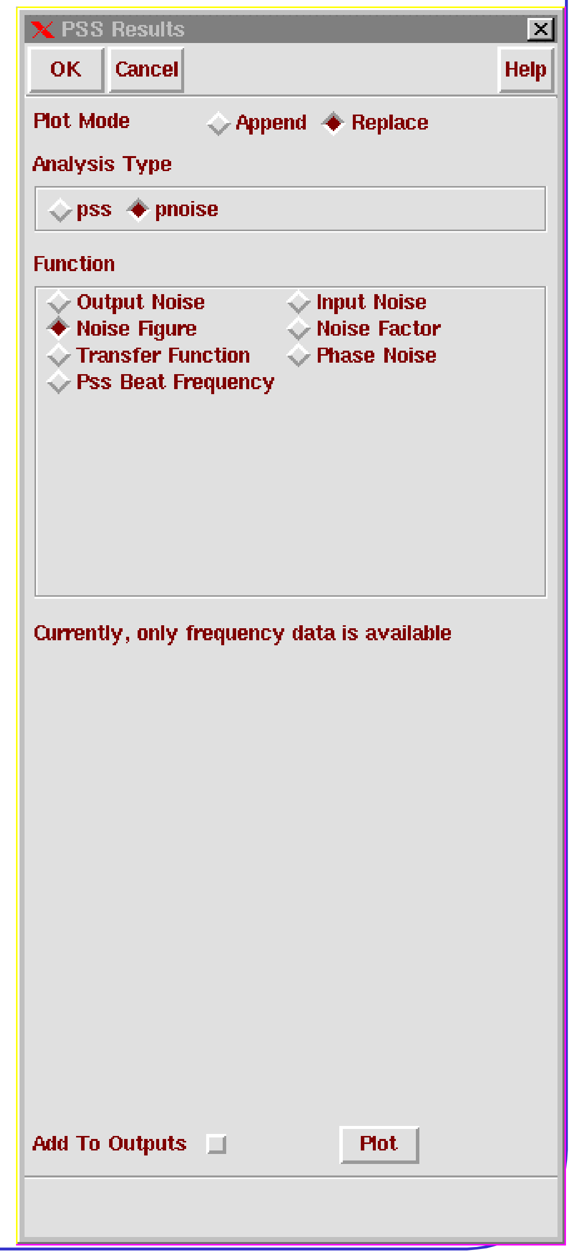
- Click on **pnoise** in the ***Choosing analyses form***, and set up the form as right:
- A **Maximum sideband** of 8 implies PNOISE will calculate the noise out to 8 harmonics of the PSS_{fund}, or 18.4 GHz.
- To set the **Positive/Negative Output Node**, click the **Select** button, and select the **Pif/Pif-** node in the schematic window.
- Click the **Select** button and select PORT1 component in the schematic to set the **Input Port Source**.
- To obtain the **Reference Side-Band**, run **PXF** analysis.
- Finally, push **OK**; then **Netlist and Run**.



Plotting the NF

- Use **Direct Plot** function to see the results. In the ***PSS Results form***, select **pnoise** button. Click **Plot** button, and the waveform window displays the results.

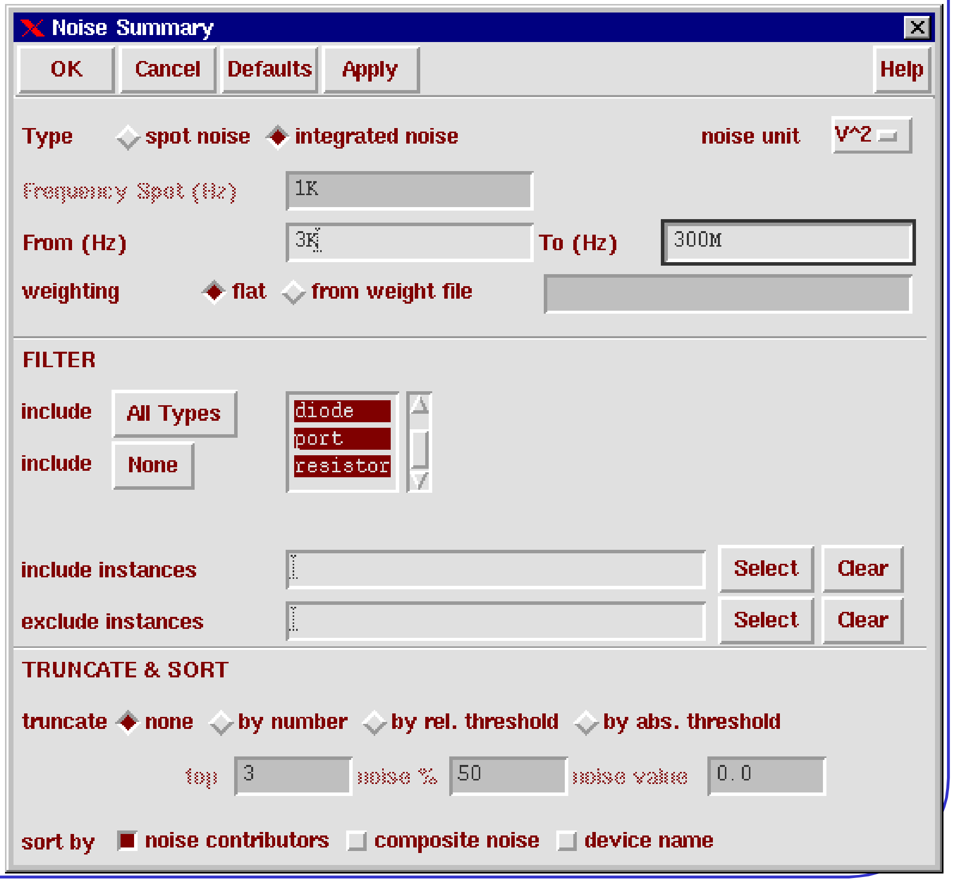




Printing the Noise summary Report

- It is valuable to know the main contributions of noise in a system. This information is readily available from a **PNOISE** simulation.

- In the ***Analog Artist Simulation*** window, select **Results → Print → PSS Noise Summary**. The ***Noise Summary*** form appears. Fill the form as shown here.



The Noise Summary Table

- Click **OK** in the ***Noise Summary form***, and the ***Noise Summary Table*** displays.
- Note what are the main contributions of noise.

Results Display Window

Window Expressions Info Help 52

Device	Param	Noise Contribution	% Of Total
/PORT1	rn	7.90045e-10	25.76
NM2.rg	rn	2.16638e-10	7.06
NM0.mcore	id	2.14053e-10	6.98
NM1.mcore	id	2.13385e-10	6.96
/R1	rn	2.10986e-10	6.88
/R0	rn	2.07627e-10	6.77
/Rbias1	rn	1.98573e-10	6.47
NM4.mcore	fn	1.30932e-10	4.27
NM3.mcore	fn	1.30236e-10	4.25
NM0.mcore	fn	9.85642e-11	3.21
NM1.mcore	fn	9.64867e-11	3.15
NM2.mcore	id	5.95926e-11	1.94
NM4.mcore	id	5.04592e-11	1.65
NM3.mcore	id	5.04257e-11	1.64
/RL1	rn	4.37251e-11	1.43
/RL2	rn	4.34147e-11	1.42
/PORT2	rn	3.98763e-11	1.30
/Rmatch1	rn	3.77525e-11	1.23
⋮			
NM0.rs	fn	0	0.00
NM0.rg	fn	0	0.00
NM0.rds	fn	0	0.00
NM0.rd	fn	0	0.00
NM0.mcore	rd	0	0.00
NM0.mcore	rs	0	0.00
/Rmatch2	fn	0	0.00
/Rmatch1	fn	0	0.00
/Rbias3	fn	0	0.00
/Rbias2	fn	0	0.00
/Rbias1	fn	0	0.00
/RL2	fn	0	0.00
/RL1	fn	0	0.00
/R1	fn	0	0.00
/R0	fn	0	0.00
/PORT2	ext_file_noise	0	0.00
/PORT1	ext_file_noise	0	0.00

Integrated Noise Summary (in V^2) Sorted By Noise Contributors
Total Output Noise = 3.06722e-09
Total Input Referred Noise = 4.38152e-10

Time Domain Noise

- The noise in RF circuits is generated by sources that can typically be modeled as periodically time-varying. Noise that is periodically time-varying is also called ***cyclostationary*** noise.

- Might or might mot be independent (correlated).

- Becomes intricate with nonlinear elements, with memory, or driven by time-varying signals.

Time Domain Noise Overview

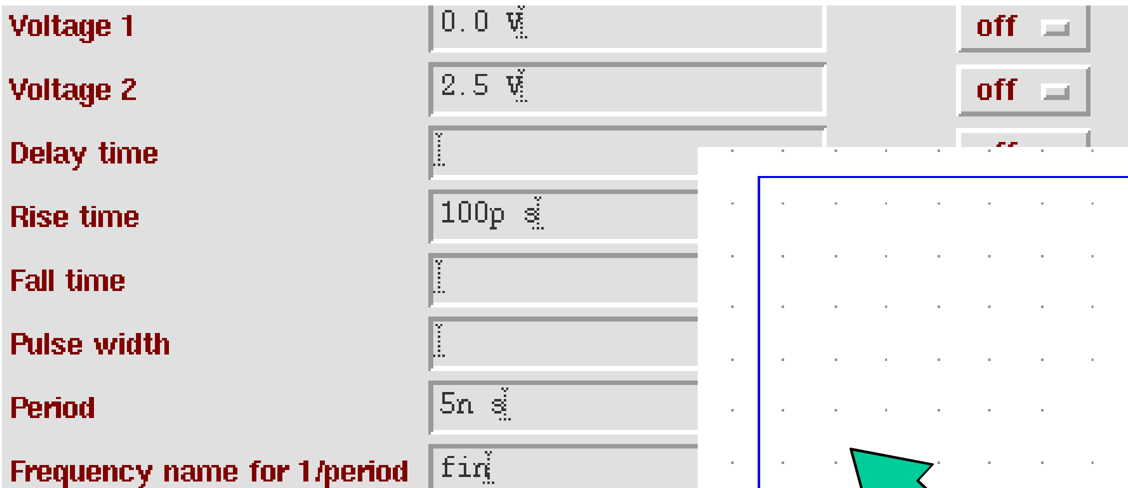
There have been 3 new noise type parameters added to PNOISE analysis:

1. sources:
   - Compute time-averaged total noise power at a signal output, in the frequency domain.

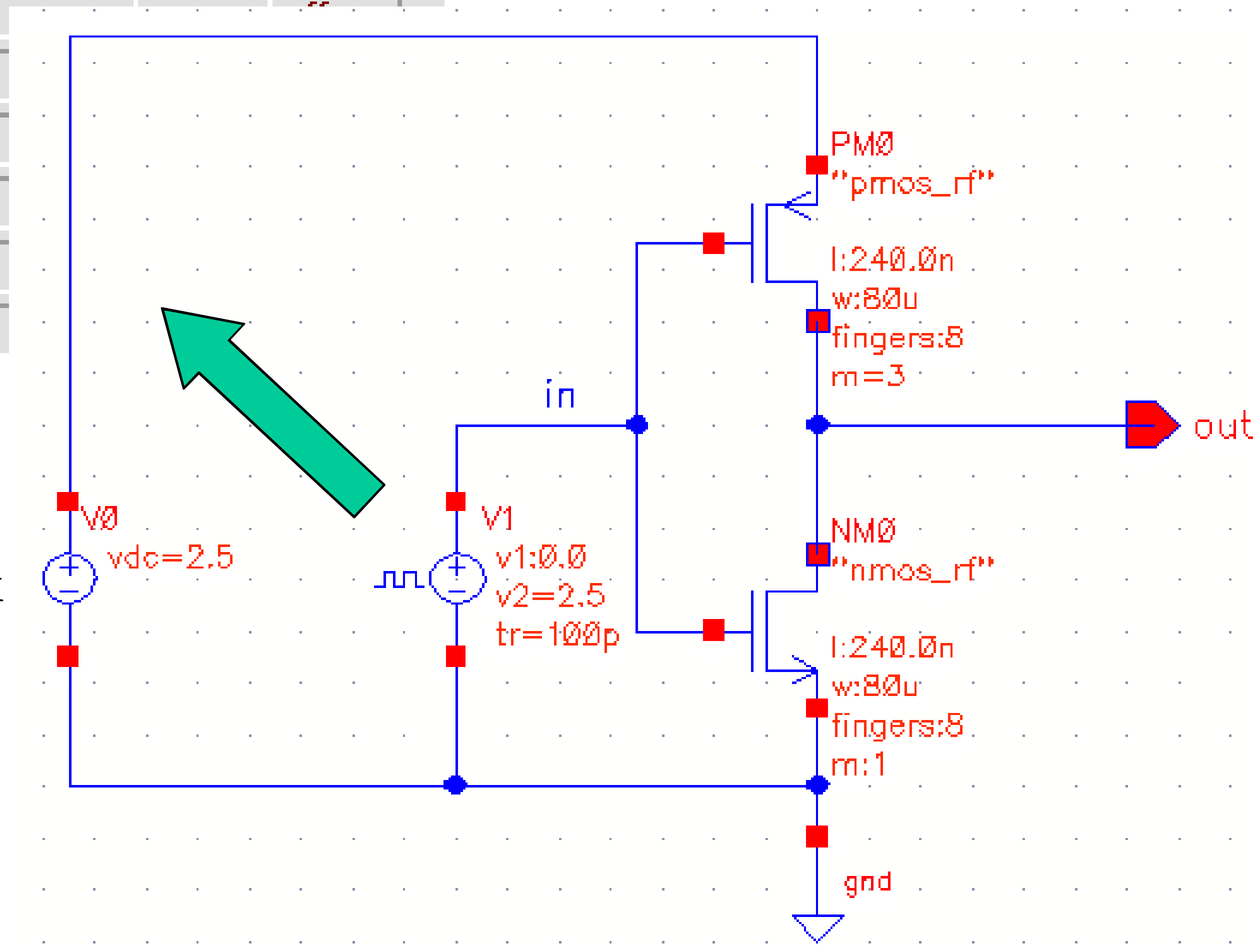
2. timedomain:
   - Calculates the time-varying instantaneous noise power in a circuit with periodically driven components
   - Setting the ***NOISE Skip Count***=N parameter will only compute the noise at every Nth timepoint in the PSS waveform.

3. correlations:
   - Calculate correlations in noise at different ports of a multiport circuit

Lab7: Calculating Time-Varing Instantaneous Noise Power



Voltage 1	0.0 V	off
Voltage 2	2.5 V	off
Delay time		
Rise time	100p s	
Fall time		
Pulse width		
Period	5n s	
Frequency name for 1/period	fin	



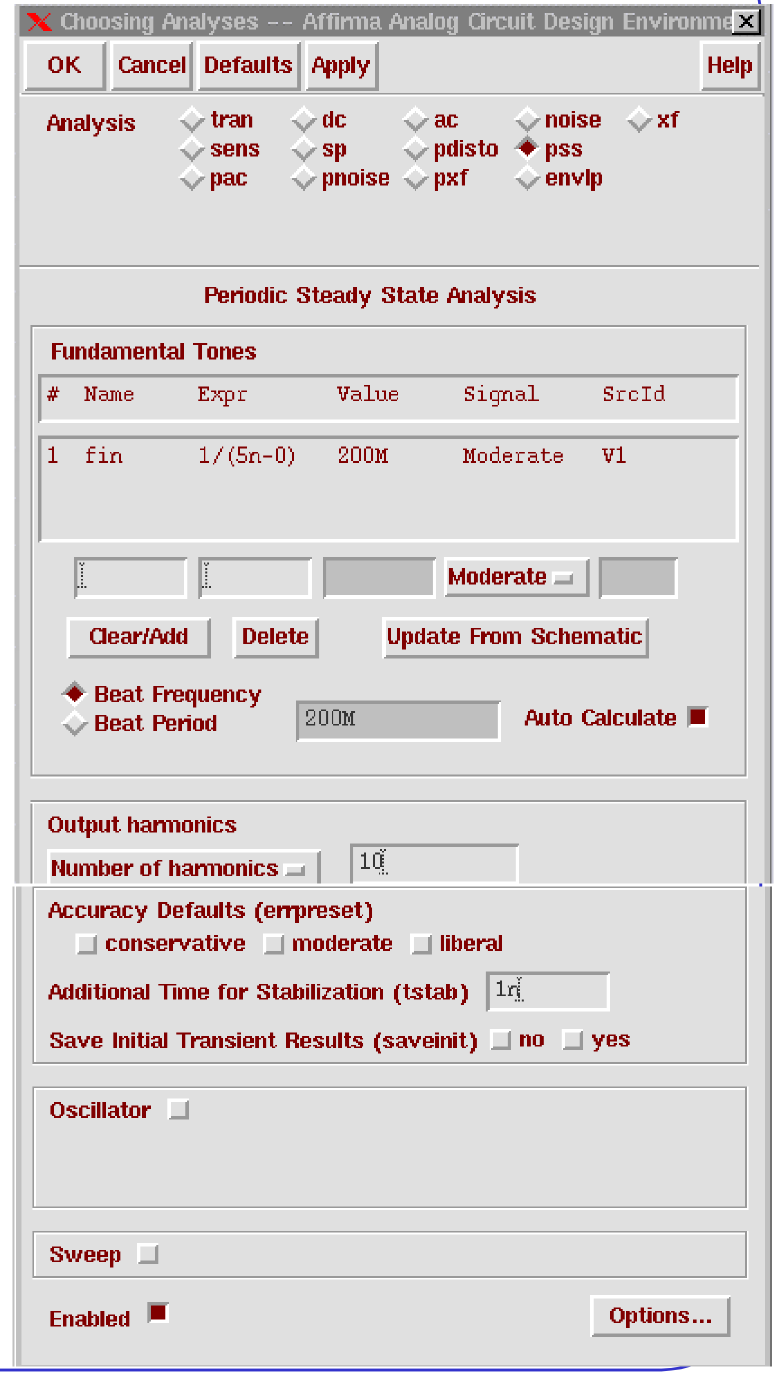
- Create a new schematic view.
- Use library "**analogLib**" & "**tsmc25rf**" to draw the scheme.
- After drawing, **Check and Save!**

Setting Up the PNOISE Simulation(1)

- Open the Design Environment window and set up a PSS analysis as shown right:

- Click the Options button and set the method to gear2only.

- Click Apply.

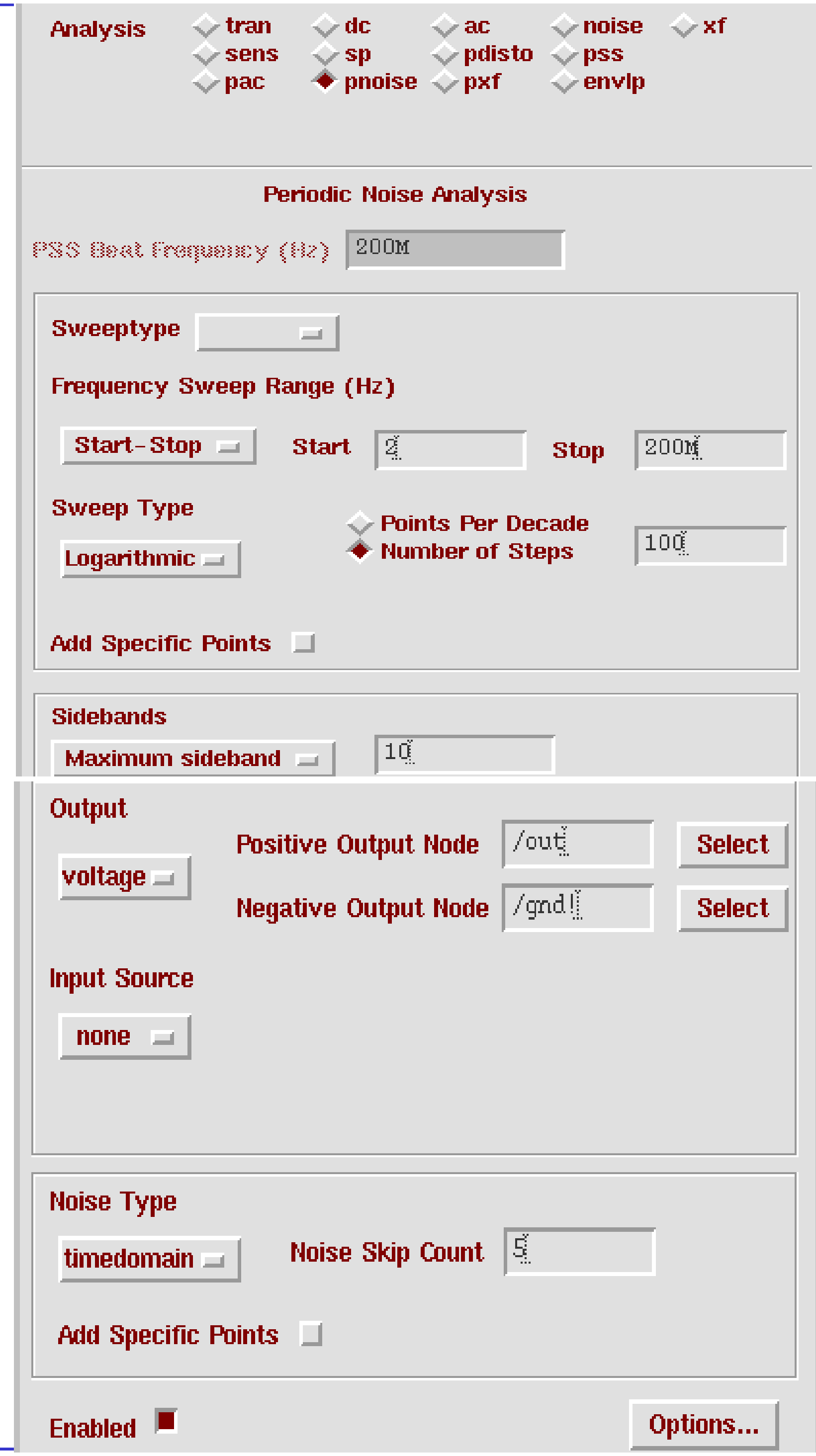


Setting Up the PNOISE Simulation(2)

- On the PNOISE analysis form, select **timedomain** in the NOISE Type field.

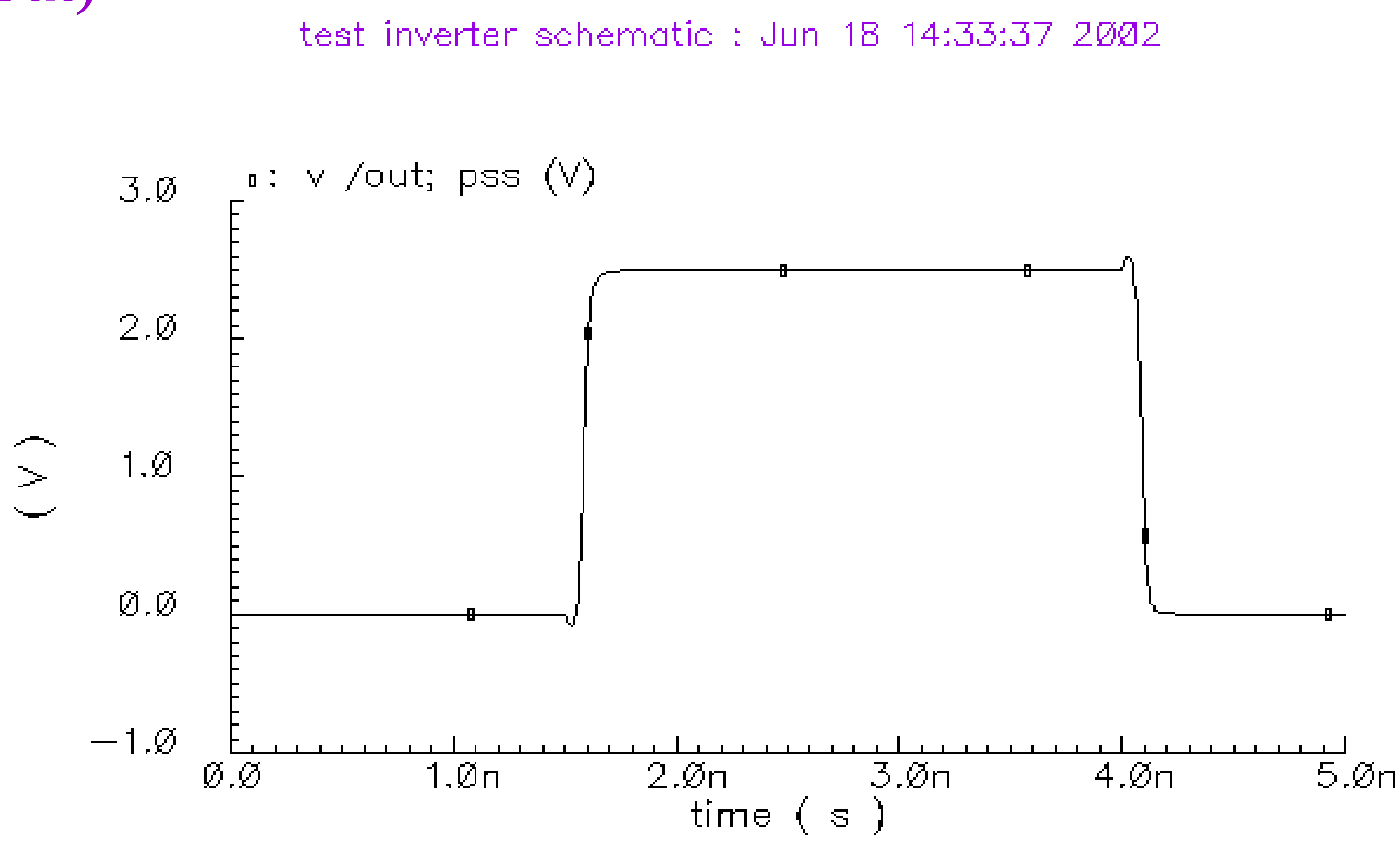
- Set up a PNOISE analysis as shown right:

Note: If the Noise Skip Count is set to an integer p, then noise will be calculated at every p+1 points. When the Noise Skip Count is 0 (default), it calculates the noise at every timepoint in the final PSS solution.



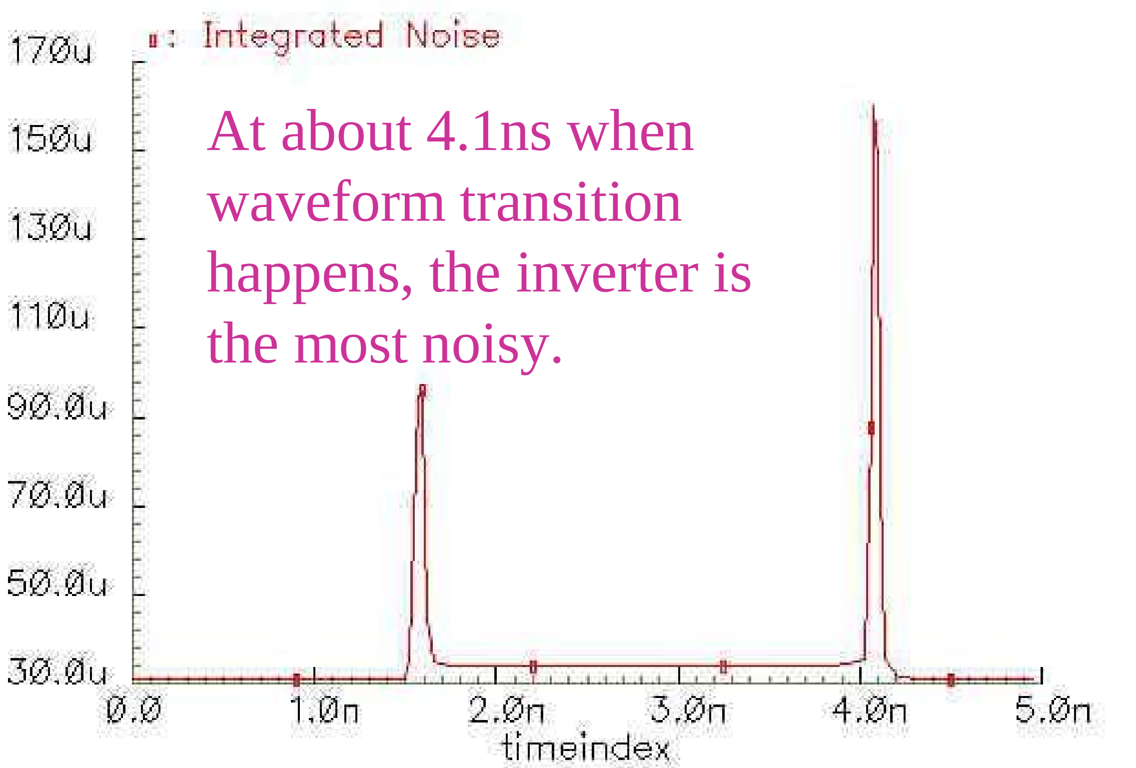
Plotting Time Domain Results

- Click the **Netlist and Run** icon to start the simulation.
- Use Direct Plot function to view the time domain plot of v(out)

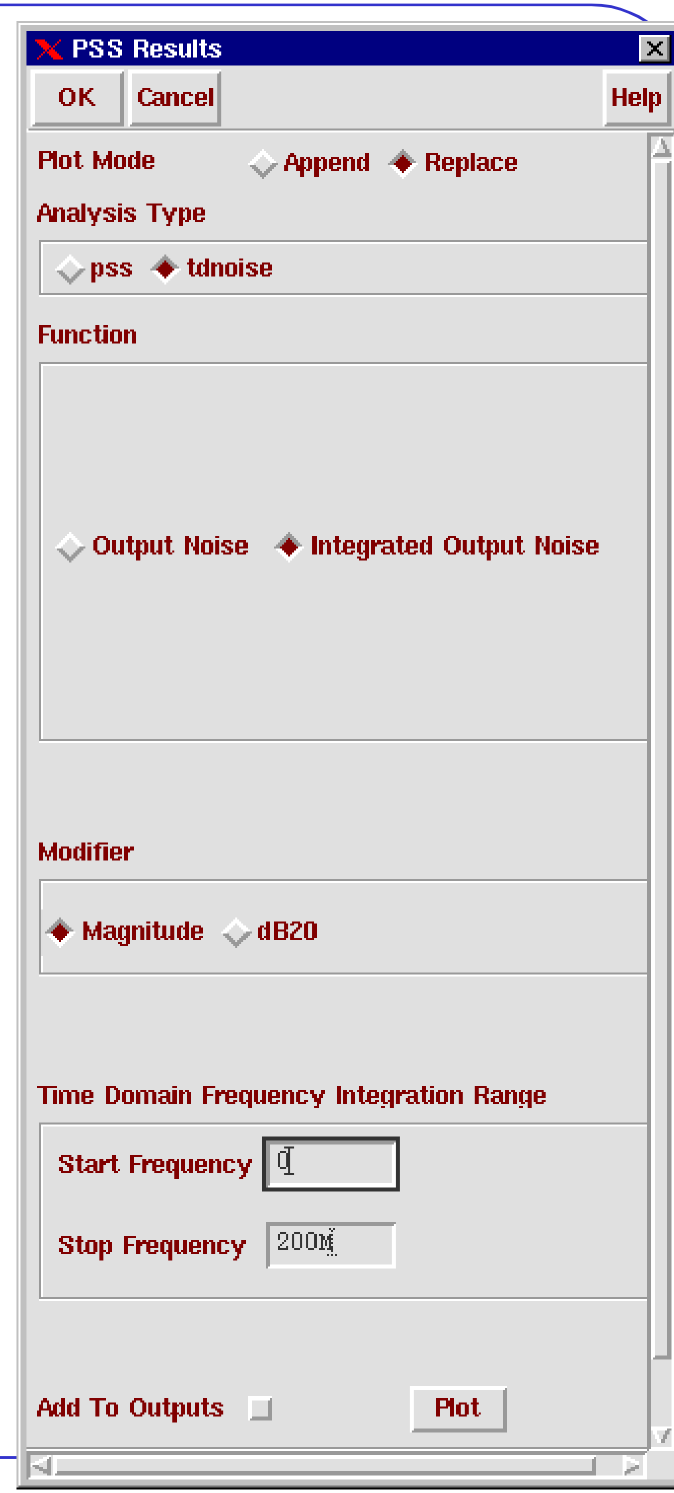


Plotting Time Domain Noise Results

- In the PSS Results form, click **tdnoise** and set up the form as shown right:
- Click **Plot** button.



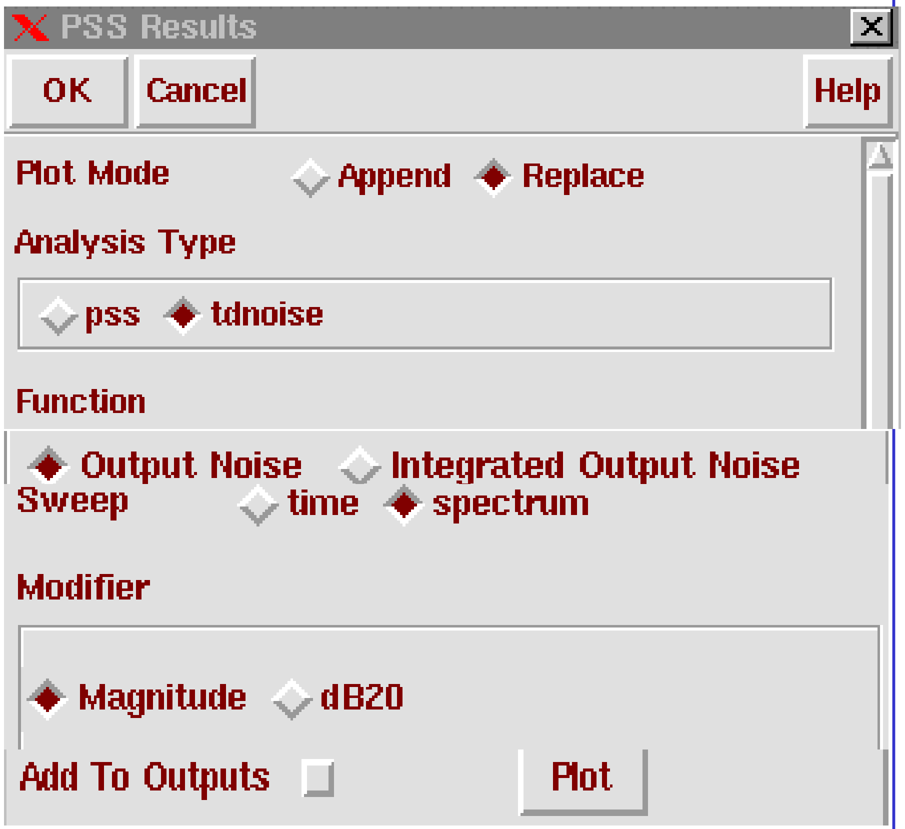
At about 4.1ns when waveform transition happens, the inverter is the most noisy.



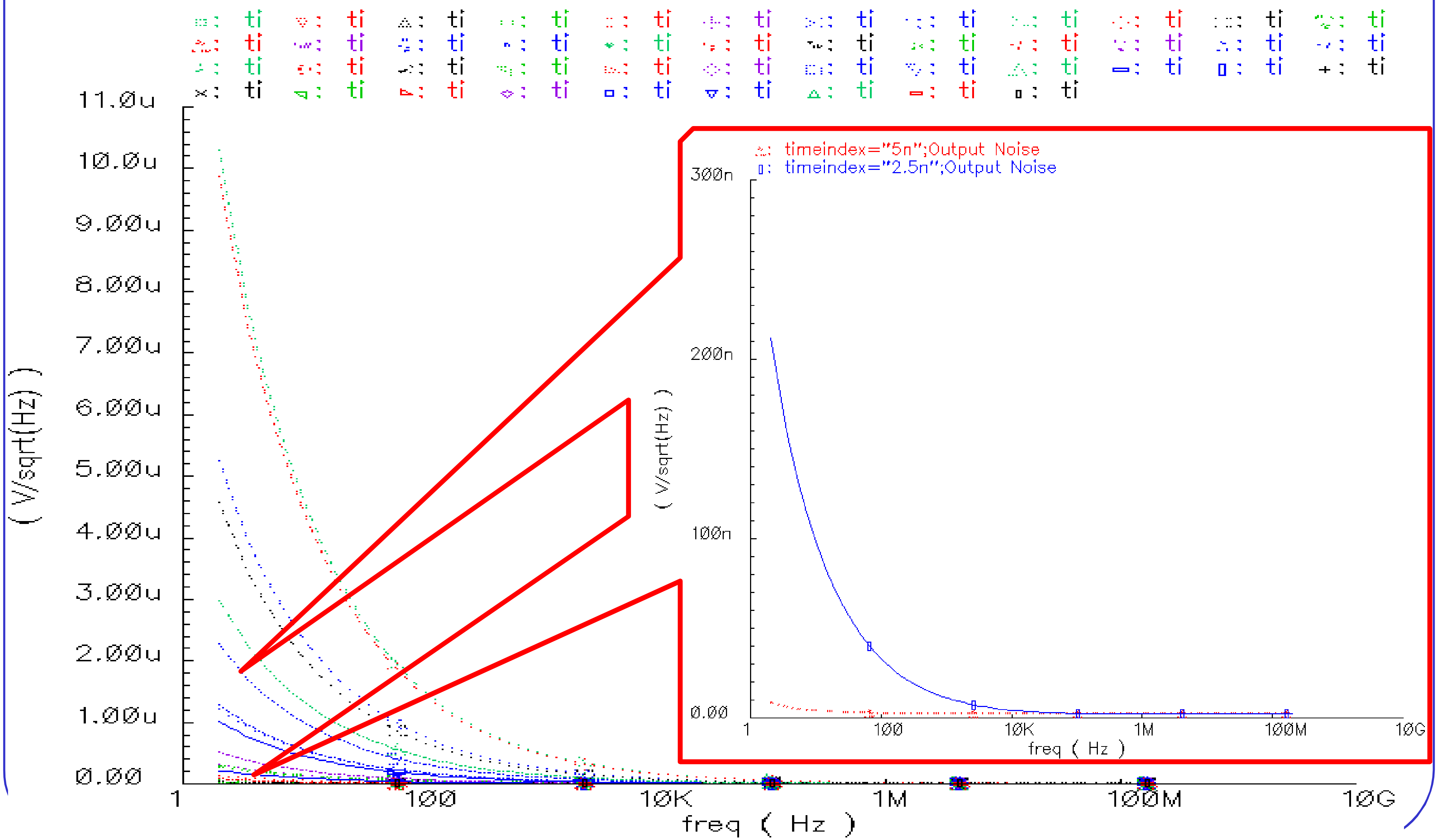
Plotting Time Domain Noise Results on Spectrum(1)

- To display the spectrum of the noise results, set up the PSS Results form as show right:

- Click Plot. See the result in the next page.



Plotting Time Domain Noise Results on Spectrum(2)



8. Periodic Distortion Analysis

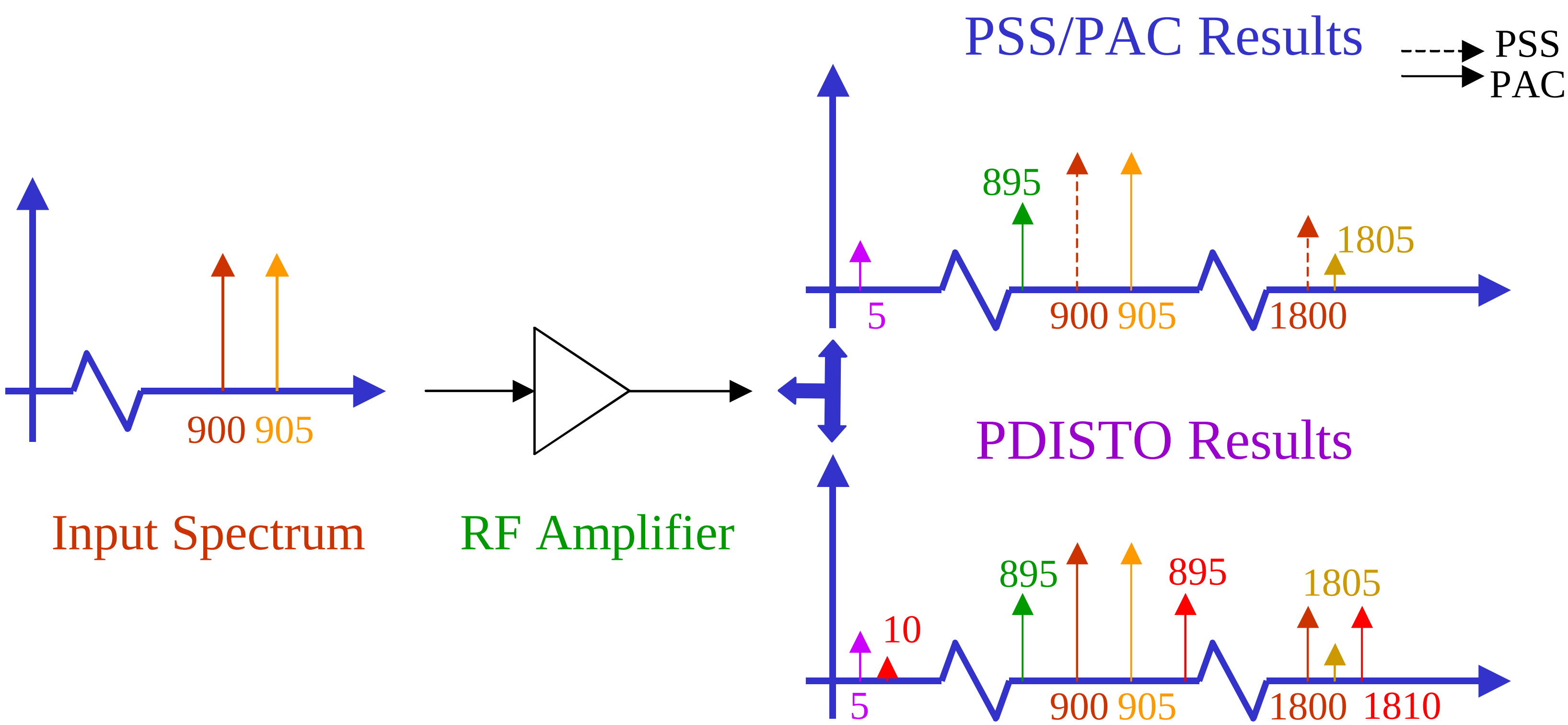
- **PDISTO** is an analysis that invokes a series of **PSS** like analyses over all input frequencies, their harmonics, and the intermodulations of the frequencies and harmonics.
- Similar to **PAC**, the **PDISTO** analysis calculates the responses of circuits that exhibits frequency translations. However, instead of simulating small signal behavior, **PDISTO** models the response from moderately large input signals.
- Use **PDISTO** to calculate intermodulation distortion from two or more large input signals. **PDISTO** treats one particular input signal as the **large** signal, and the others as **moderate** signals.
- **PDISTO** allows arbitrary signal signal inputs, including sums of sinusoids that might not be periodic, it as a **quasi-periodic** extension of **PSS**. **PDISTO** can be thought of as an extension of **PAC** that allows signal signal inputs, capable of producing third-order products, to be used.

PNOISE Analysis Overview

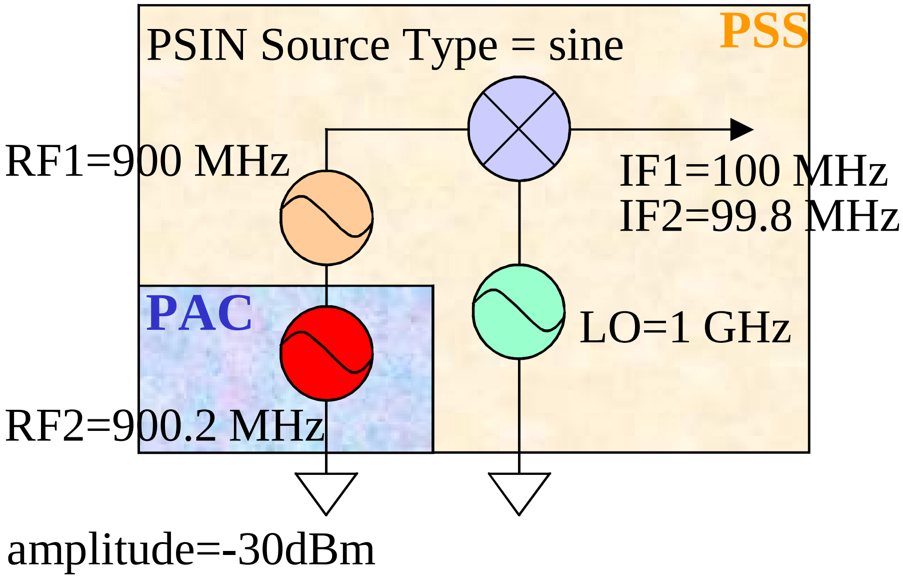
- Internal to the simulator, one input is treated as the **large** signal, which causes the most nonlinearity or the largest response in the circuit.
- Other signals are treated as **moderate** and do not need to be harmonically related to the large signal or integer multiples of each other.
- The **moderate** signals can be large enough to create distortion (near $\mathbf{P_{-1dB}}$ point)
- The ability to **sweep PDISTO** provides a way to perform intermodulation distortion calculations with multiple input signals, considered as large signals.

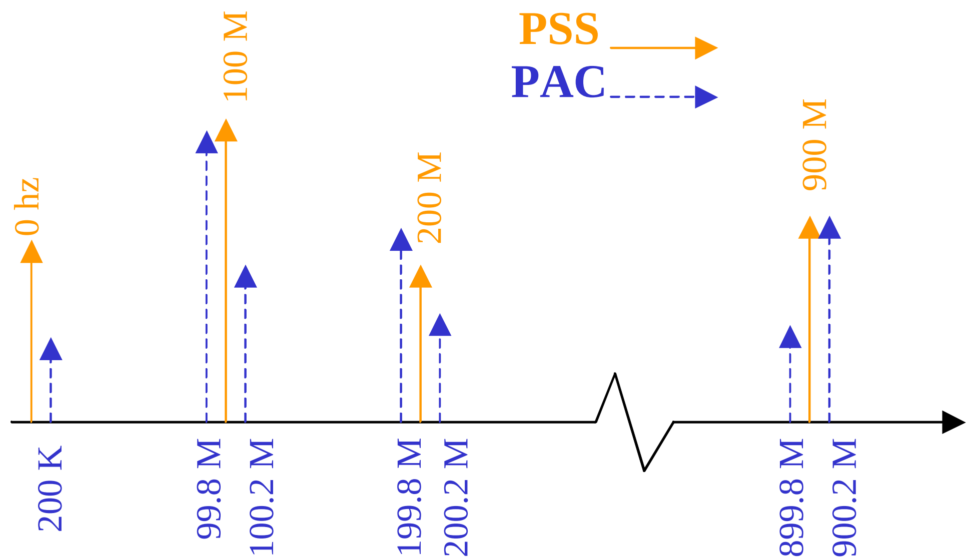
PDISTO v.s. PAC

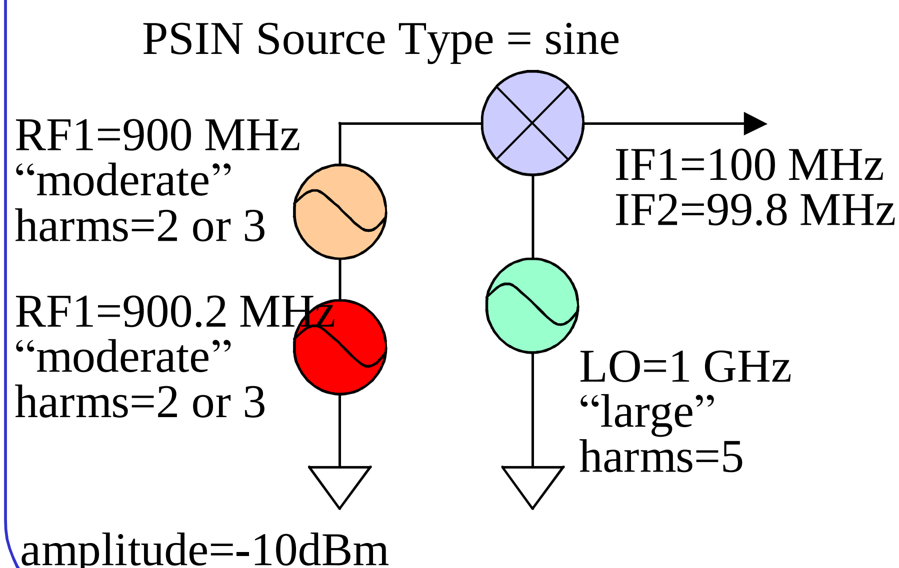
- **PDISTO** analysis yields more information than **PSS** followed by a **PAC** analysis, when modeling intermodulation distortion.

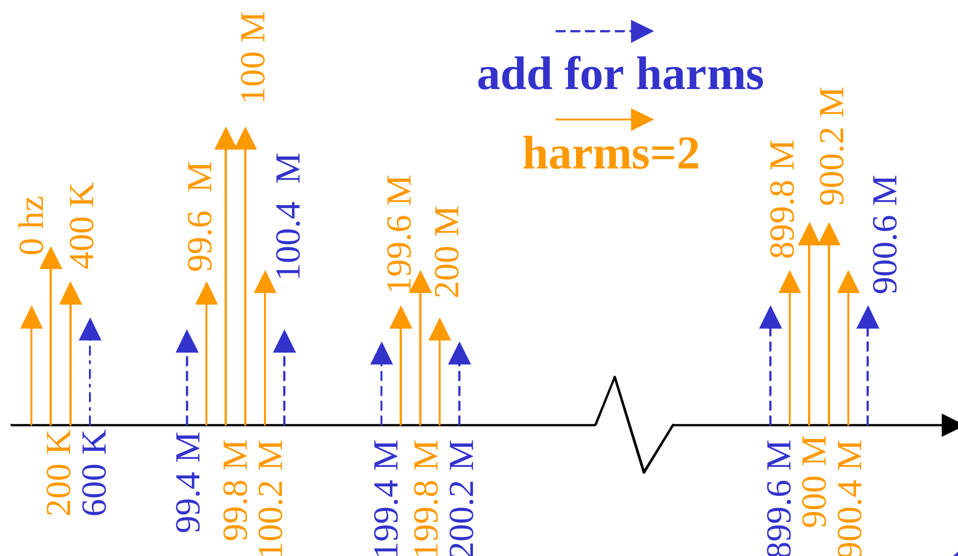


Comparing PDISTO and PAC(1)









Comparing PDISTO and PAC(2)

- The number of harmonics of the **large** signal does not affect the simulation time, where the number of harmonics on the **moderate** signals greatly affects simulation time.

- Always specify at least 1 harmonic on each signal in a **PDISTO** analysis.

- **PDISTO** analysis does not take as long as a **PSS** analysis with a small PSS Fundamental, but it is longer than a **PSS/PAC** analysis.

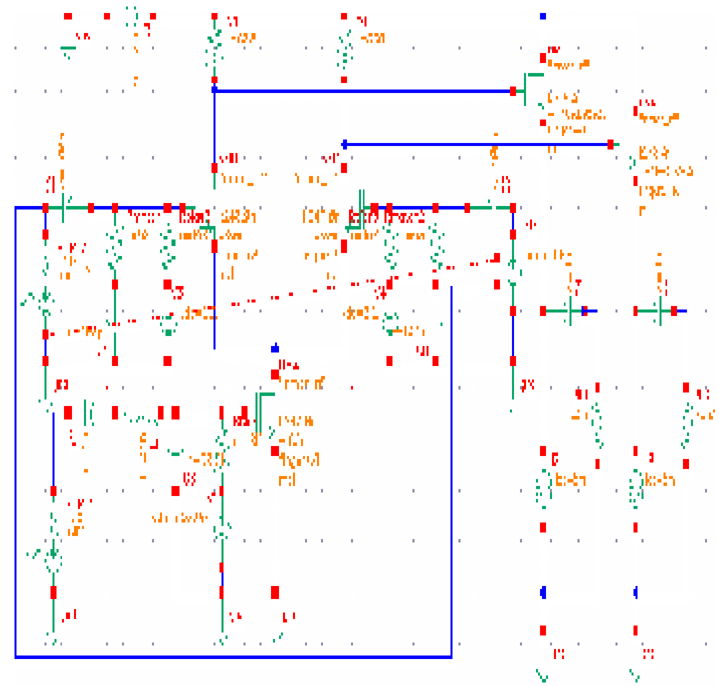
PDISTO Assumptions

- Unlike **PSS**, **PDISTO** does not required multiple inputs be commensurate or coperiodic. However, they still must be periodic.
- For coperiodic, well separated signals, use **PSS.**
- For signals that are closely spaced or not coperiodic, use **PDISTO**.
- For circuits driven by 2 or more moderate signals or at unrelated frequencies, use **PDISTO**.
- If only one periodic signal is large enough to create distortion, choose **PSS** followed by **PAC** or **PXF**.

Lab8 : Simulation with PDISTO

- Modify the parameter values of **PORT1** as follows:
- In the Simulation window, select **Analyses** → **Choose**; turn off the **pss** and **pnoise** analysis

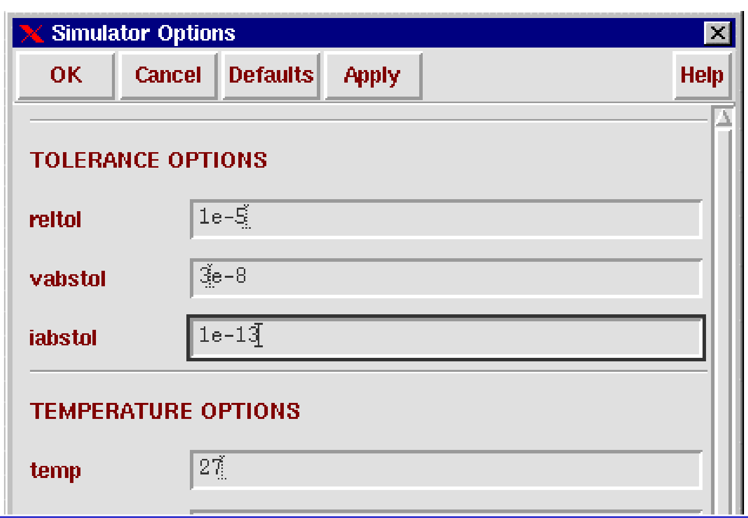
Parameter	Value
Resistance	50
Source type	sine
Frequency	frf
Amplitude (dBm)	prf
Amplitude2 (dBm)	prf
Frequency2	frf +1M

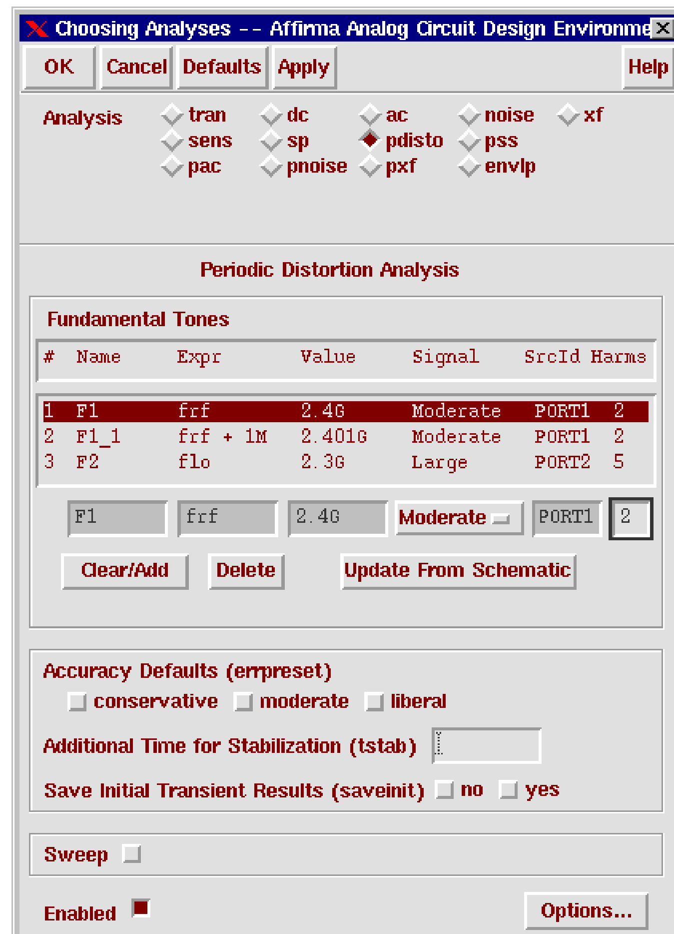


Setting Up the PDISTO Simulation(1)

- In the *Choosing Analyses* form, select **pdisto** for the analysis. Use the **Clear/Add** button to change the values in the Fundamental tones list box as shown right.
- Remember to select **gear2only** button in the Options form.
- Select **Simulation-Options-Analog**, and set the Tolerance Options as recommended.

  If the signals are truly large, relax reltol to 1e-4.

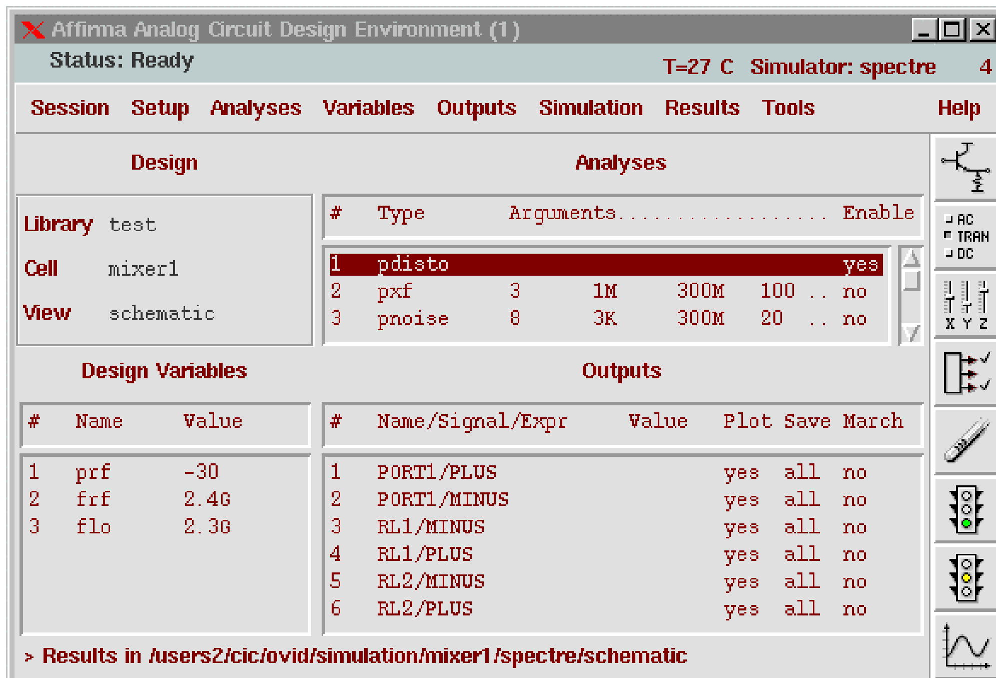




Setting Up the PDISTO Simulation(2)

- Remember to select the output terminals ***to be saved and plotted*** before the simulation.

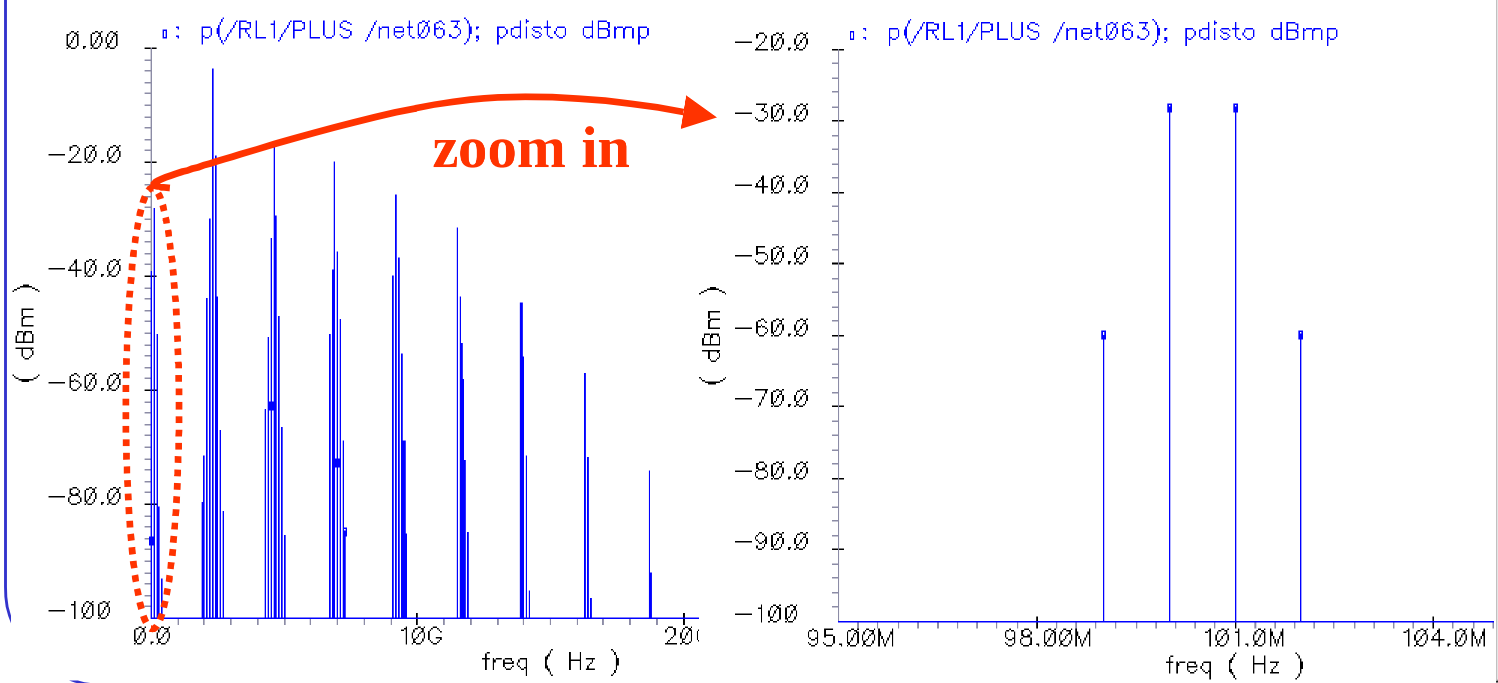
- Increase the power of the input RF signals from –40 dBm to –30 dBm.(P-1dB for this circuit is –22 dBm) In the **PSS/PAC** analysis, you used a **PAC** tone that was at least 10 dB below the 1 dB compression point to prevent violating the small signal assumptions associated with the **PAC** analysis. This restriction does not apply to **PDISTO**.

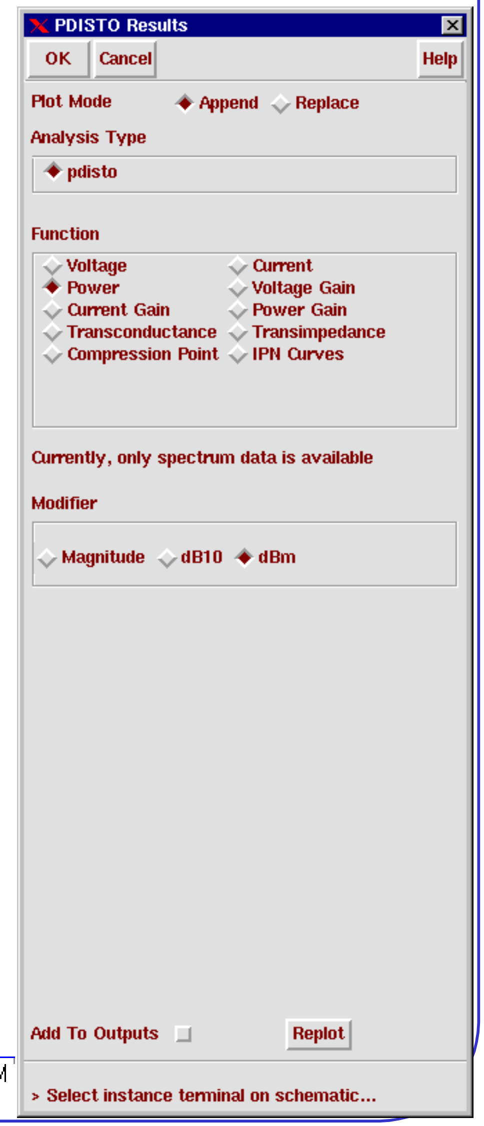


- Select Netlist and Run button

Plotting Simulation Results

- Use **Direct Plot** function to see the results. Follow the prompts at the bottom of the form, and select instance terminal (RL1) in the schematic

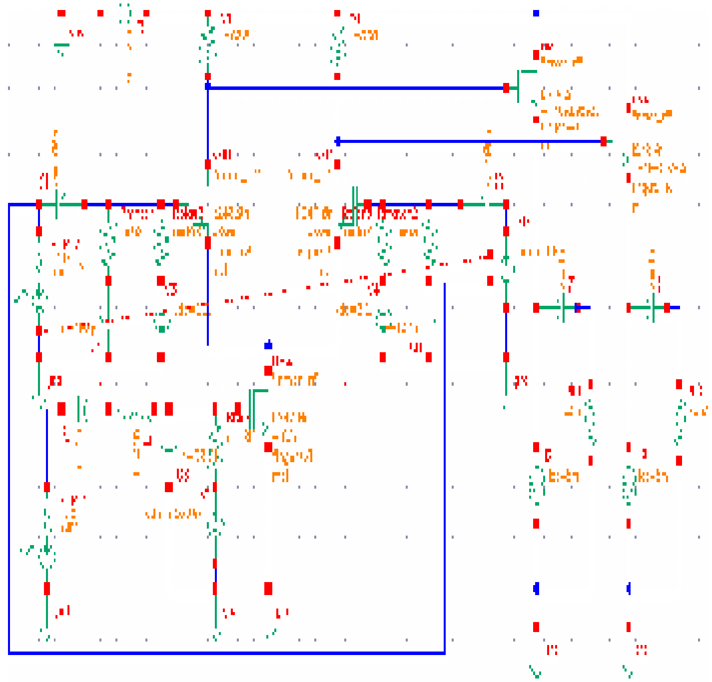




Simulation IP3 with PDISTO(1)

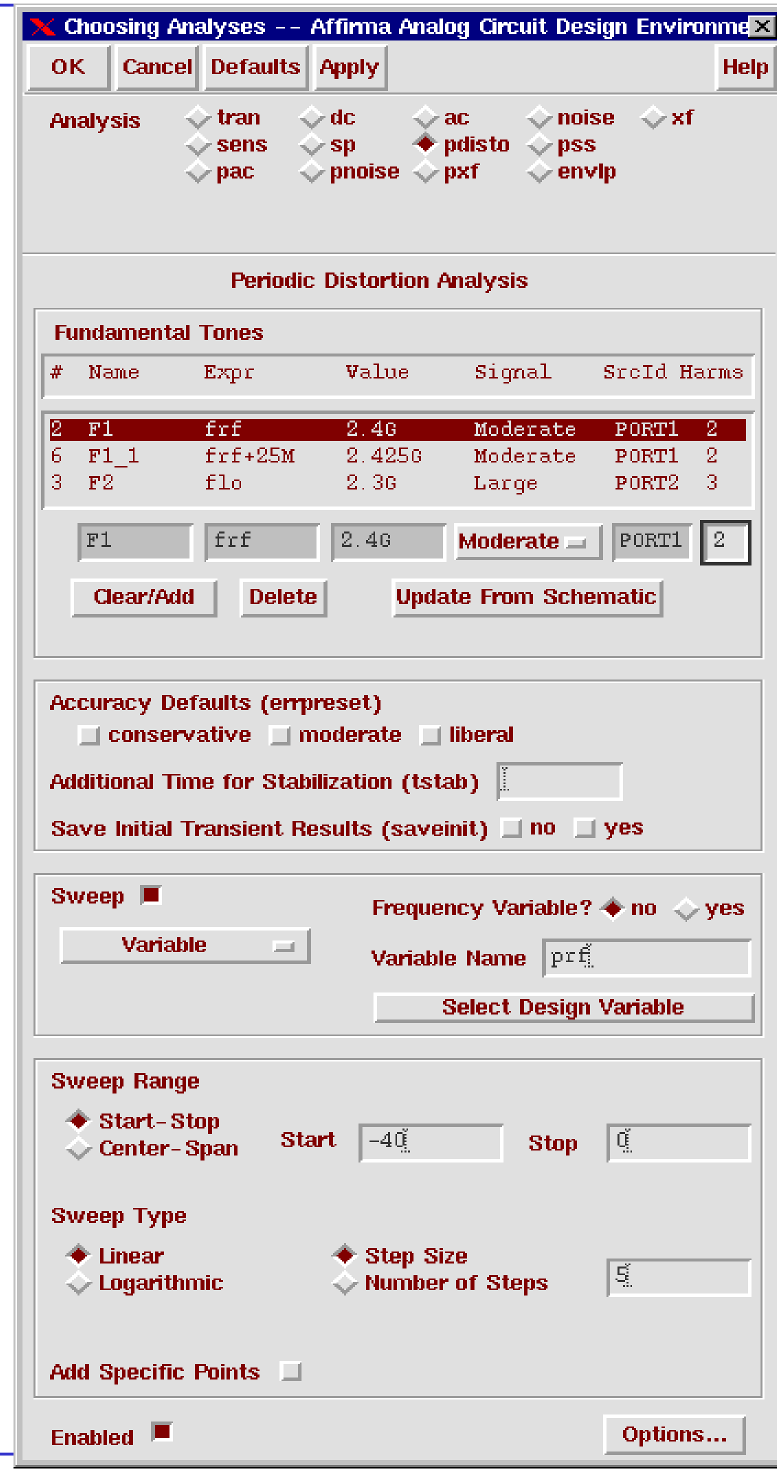
- The setup for this measurement is very similar to the one used for the swept PSS simulation, except you will be using PDISTO with two moderate tones and one large reference signal.
- Modify the parameter values of **PORT1** as follows:
- Check and save.

Parameter	Value
Resistance	50
Source type	sine
Frequency	frf
Amplitude (dBm)	prf
Amplitude2 (dBm)	prf
Frequency2	frf +25M



Simulation IP3 with PDISTO(2)

- In the ***PDISTO Analyses*** form, use the **Clear/Add** button to change the values in the Fundamental tones list box. Set up the Sweep Range as shown right.
- Remember to choose the **gear2only** method and set the Tolerance Options as recommended or relax ***reltol*** to appropriate value. Click OK.
- Run the simulation.



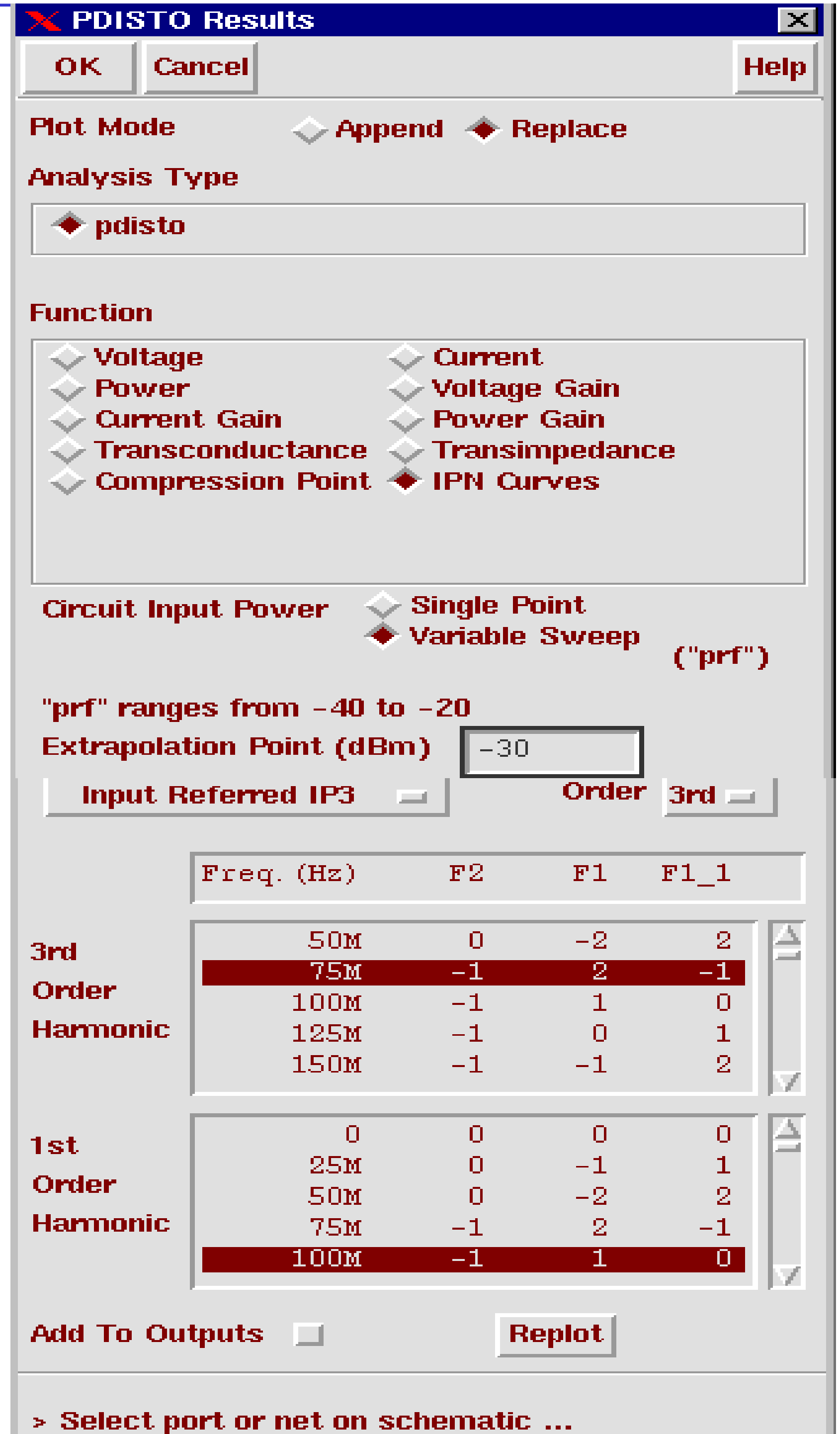
Displaying the IP3 Plot(1)

- Use **Direct Plot** function to see the results. Set up the **PDISTO Results** form as shown right. Follow the prompts at the bottom of the form, and select instance terminal (RL1) in the schematic

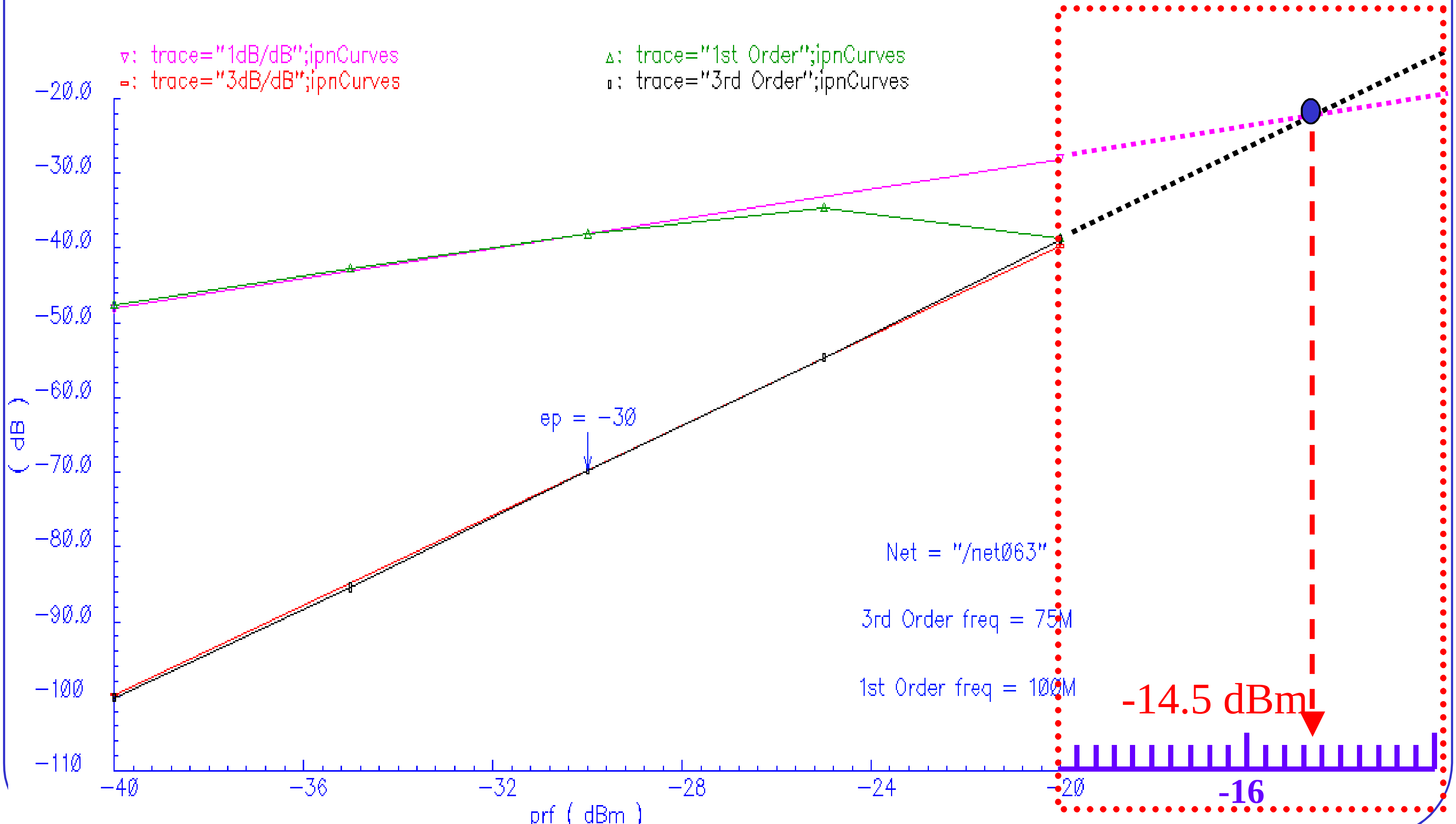
LO: 2.3 G   RF: 2.4 G & 2.425G

1^{st} order harmonics: 100M & 125M

3^{rd} order harmonics: 75M & 150M



Displaying the IP3 Plot(2)



9. Oscillator and Phase Noise Analysis

- SpectreRF-PSS analysis can be performed on autonomous or nondriven circuits, such as oscillators.
- Oscillator analysis includes two phases:
  - **The initial transient phase**:

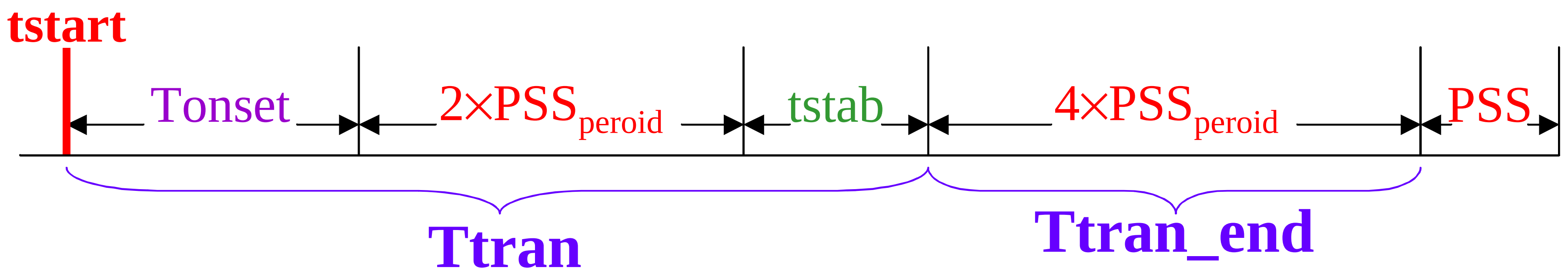
    The PSS monitors the potential difference between the two nodes specified and the waveforms in the circuits, and this analysis develops a better estimate of the oscillation period of the circuit.
  - **The shooting phase**:

    The circuit is simulated repeatedly while the length of the period and the initial conditions are adjusted to achieve a periodic steady state solution.

Troubleshooting Oscillators

- Does not converge – increase ***tstab***
- Improve the estimate of the period. Be especially carefully that the period specified is not too short.
- Change the value of the method parameter from ***gear2only*** to ***trap*** or ***traponly***.
- Does not converge – increase ***maxperiods***
- If the shooting iteration approaches convergence and fails, increase the value of the ***steadyratio*** parameter, but never set ***steadratio*** larger than 0.1.
- Change the value of the tolerance parameter.

Oscillator PSS Algorithm



- tstart - Start time for transient analysis.(default is 0)
- Tonset – Time when the last stimulus waveform becomes periodic.
- PSS_{period} – the guess period entered by the user.
- tstab – additional stabilization time entered by the user.
- maxstep = (Ttran / 50)(default).
- The algorithm then adds a further 4 periods of our guess frequency of transient analysis in order to measure the oscillator frequency.

Oscillator Algorithm and maxstep

- Default ***maxstep*** > period if Ttran > 50 oscillator periods. The oscillator might not start correctly or a metastable state might be found by the simulator.

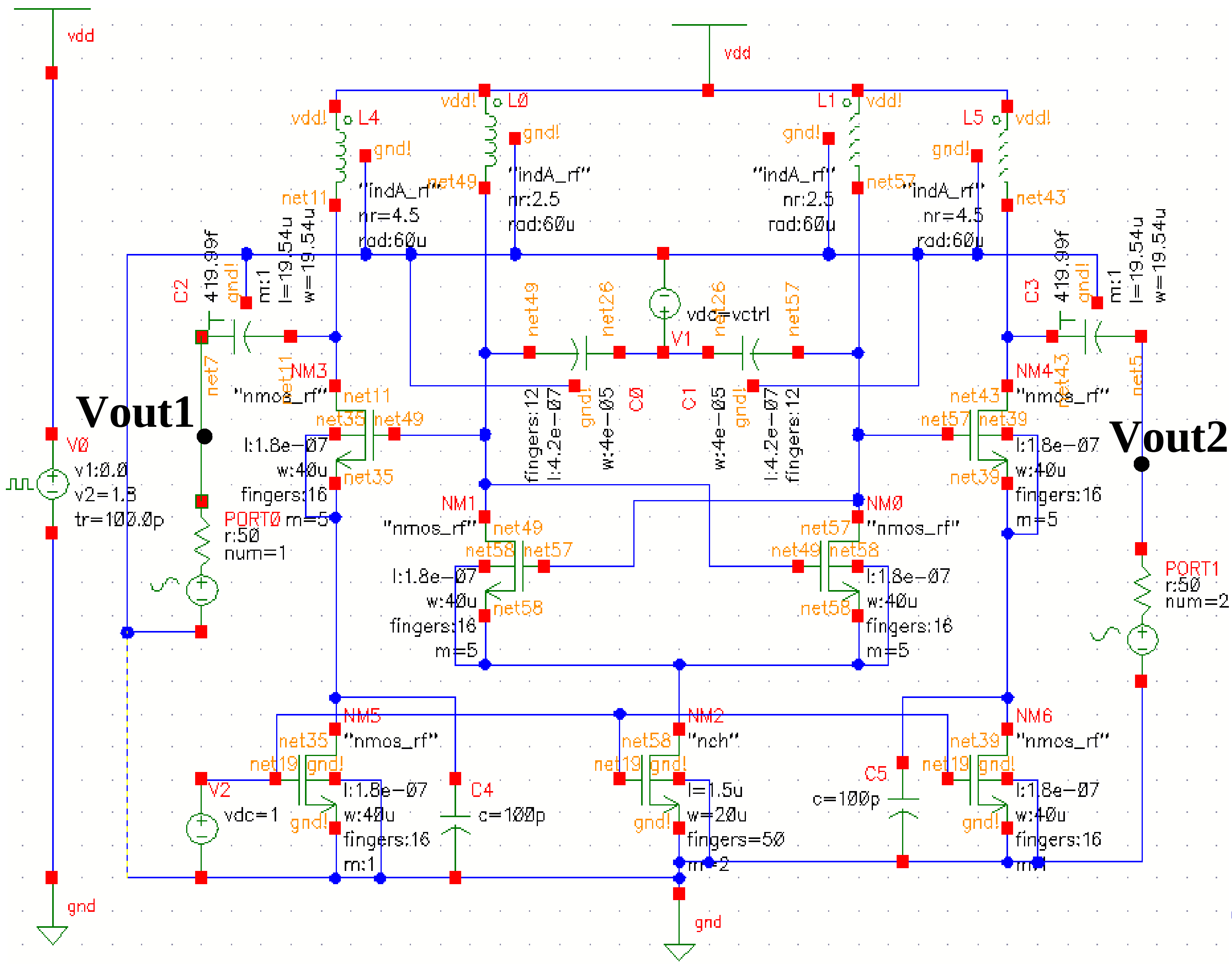
- Use tighter convergence criteria or set ***maxstep*** < 1/(200×FreqOsc)

- In PSS shooting iterations stage, maxstep is the smallest of:
  - ***maxstep*** manual entry
  - PSSperiod/(maxharm×40)
  - 1/(***maxacfreq***×5)
  - PSS Beat Frequency/200

- Setting a high harmonic in the PSS analysis or setting ***maxacfreq*** will only effect the maxstep of the PSS shooting iterations but **NOT** the maxstep of the initial transient section.

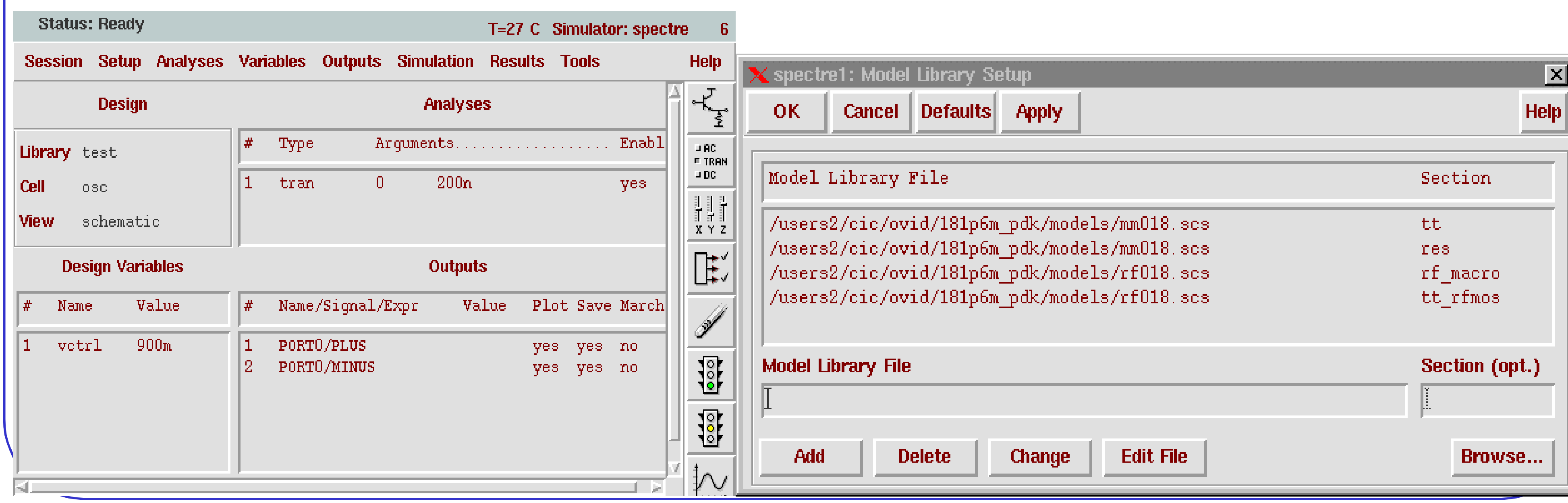
Lab9 : Tunable Oscillator – Transient Analysis

- Create a new schematic view.
- Use library “**analogLib**” & “**tsmc18rf**” to draw the scheme.
- Use a **vpulse** source to ***kick-start*** the oscillator.



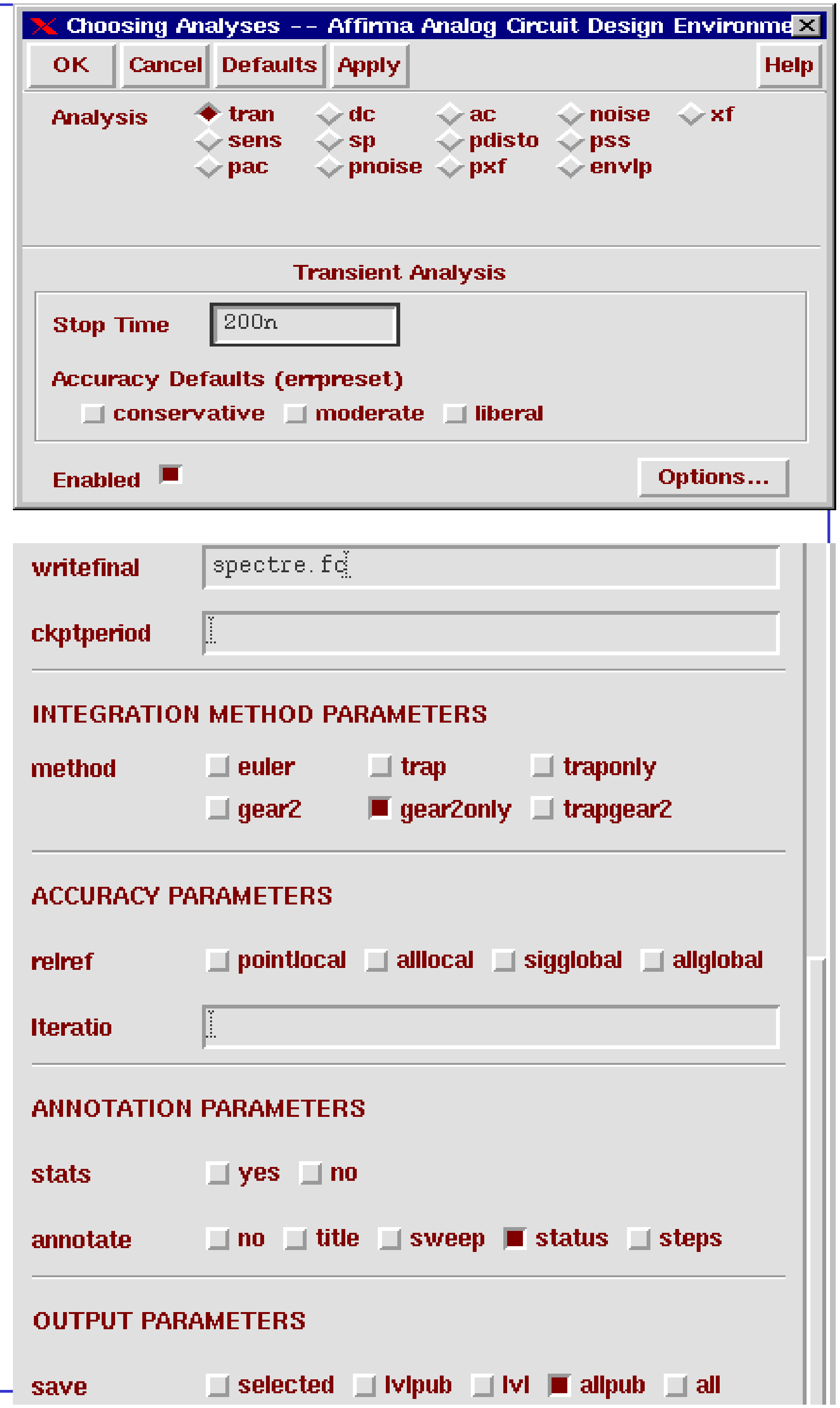
Set Up the Design Environment

- In the ***Design Environment*** form select **Setup → Model Libraries** to set up the model library as show below.
- Select **Variables → Copy From Cellview** to set the variable **vctrl** to be some value.



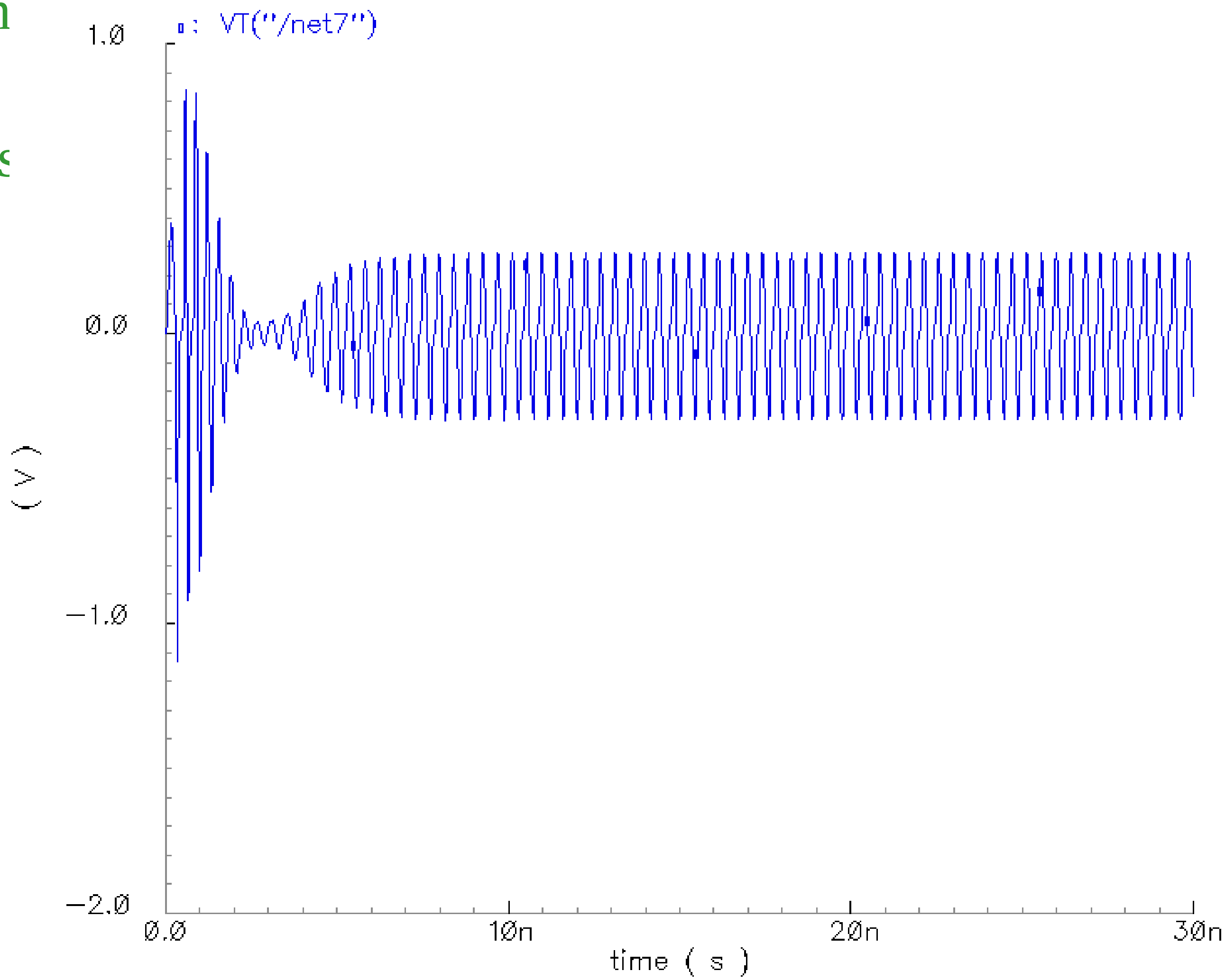
Transient Simulation set up

- Select **Analyses** → **Choose** to set up the transient simulation as right window.
- Set up the form and option form as shown right:
- Push **Netlist and Run** button.



Display the Transient Results

- In the ***Analog Artist Simulation*** window, select **Results** → **Direct Plot** → **Transient Signal**; then select **Vout1** node in the schematic and press **ESC** key to end the selection. The Vout1 transient node voltage appears in the Waveform window.



Oscillator Notes

- When applying initial conditions to start an oscillator, first run a transient analysis to get the voltages for a few nodes in the circuit. To set the initial conditions for the next run, select **Simulation** – **Convergence Aids** – **Initial Condition**.

- In the Transient Options form, set a value such as ***spectre.fc*** for the **writefinal** parameter in the ***STATE FILE PARAMETERS*** section. The ***spectre.fc*** file will have all of the final conditions on the nodes in the circuit.

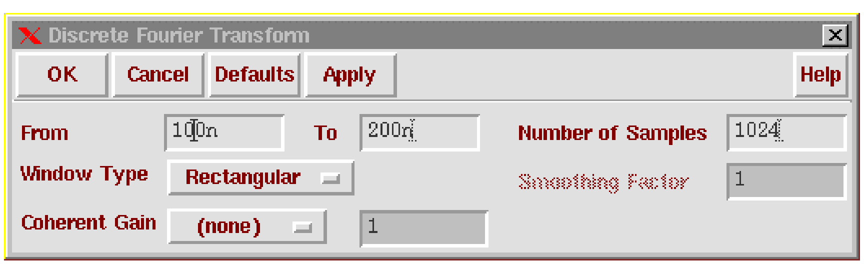
- Before running another transient or PSS analysis, set **readns** to ***spectre.fc*** in the ***CONVERGENCE PARAMETERS*** section of the Options form.

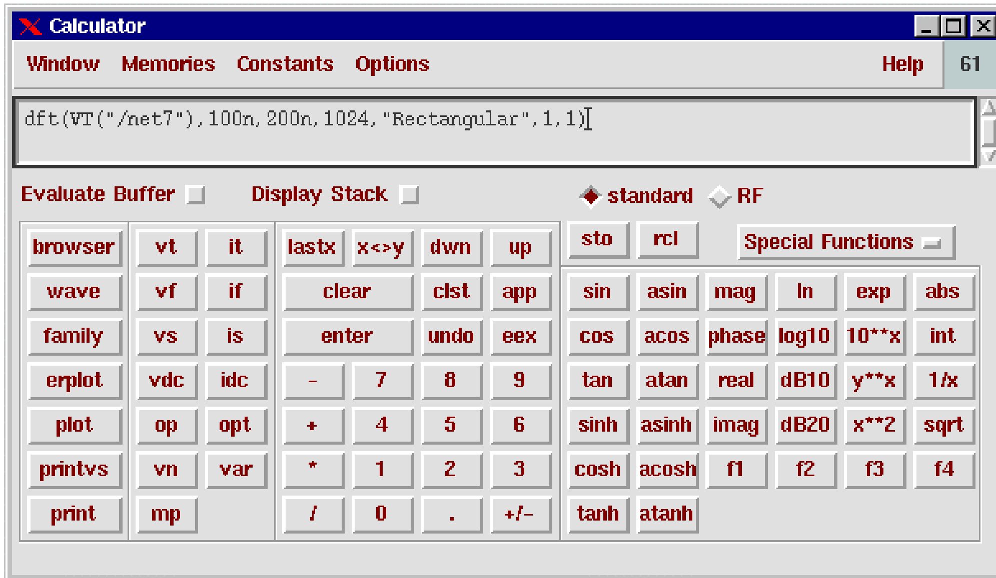
spectre.fc file

```
# CHECKPOINT_VERSION 1
# Generated by spectre from circuit file `input.scs' during analysis tran.
# 4:41:44 PM, Thur Jun 20, 2002
# Number of equations = 77
NM2:int_d       1.38143129288017
NM2:int_s       4.27510716664238e-05
net5     -0.150439862589224
net7     0.202993505669876
net11    2.25021017248973
net19    1
net26    0.9
net35    0.779556456246658
net39    0.776441932715426
net43    1.06676286864526
net49    1.6759459972117
net57    1.92889948704024
net58    1.38147572492745
PORT0:p 0.00405987011339751
V0:p     -0.0732141329919096
V1:p     0.000358606009066479
V2:p     1.1956494590698e-05
vdd!     1.8
C0.diode:int_a   0.899821441348785
C0.n2    -0.105920570737303
C1.diode:int_a   0.900467622111242
C1.n2    0.115359200575381
C2.ls:1 -0.00405987011339751
C2.n1    0.211926869155686
C2.n2    0.203959736445305
```

Use the DFT Function

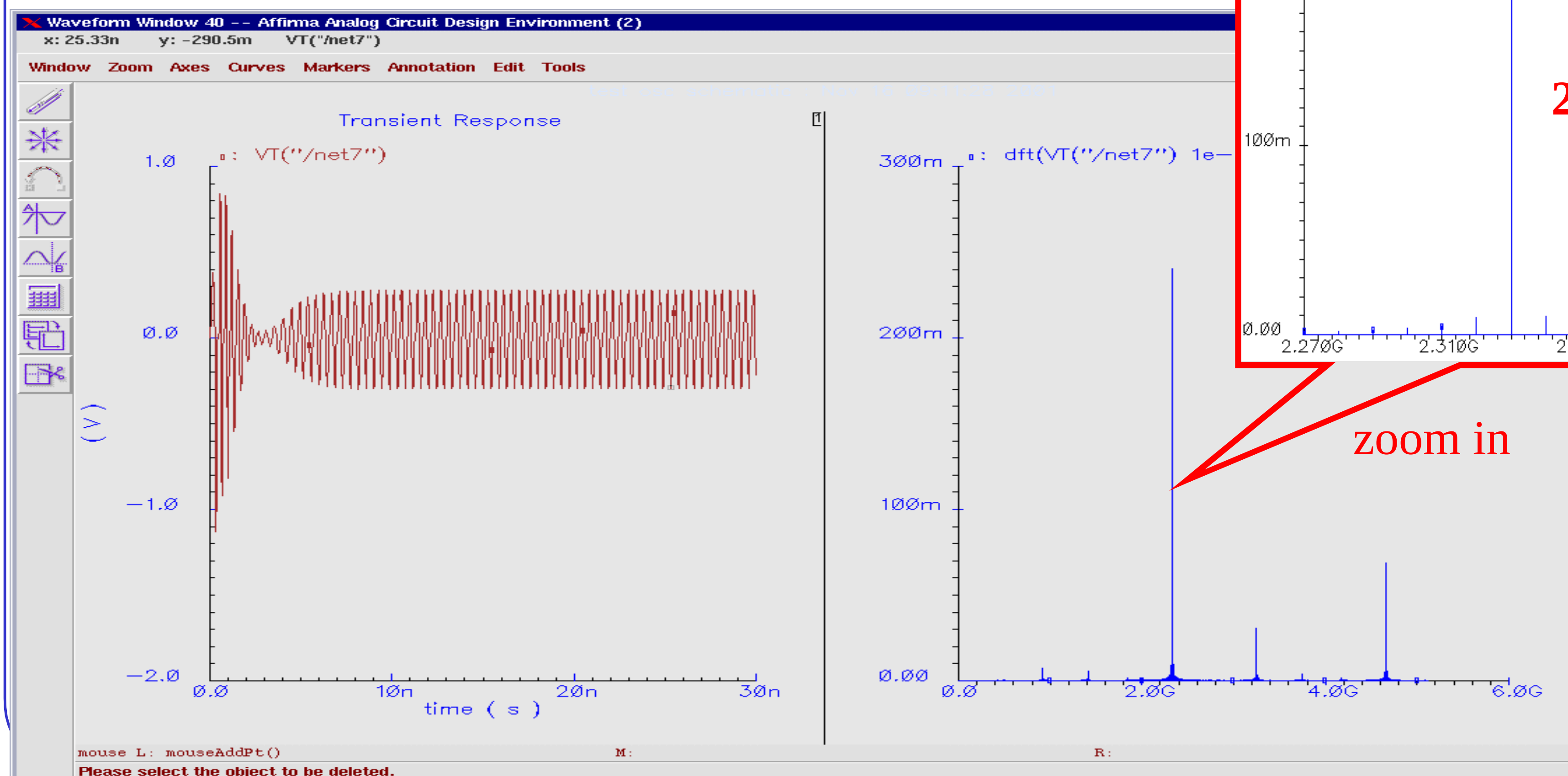
- In the Waveform window, click the **Add Subwindow** icon, then a subwindow with a label of ***2*** in the upper right corner is added.
- Click the **Calculator**, then the calculator appears.
- Click the **vt** button in the Calaulator and follow the prompt at the bottom of the schematic window. Then select the **Vout1** node in the schematic and press **Esc**; click and hold **Special Functions** and select **dft** form from the ***Special Function list.***
- Fill in the form as follows: And click OK.

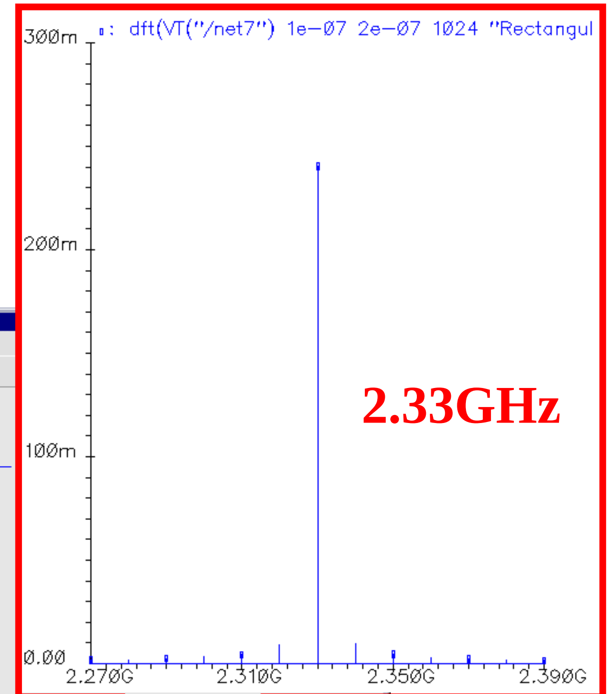




The Frequency Domain Results

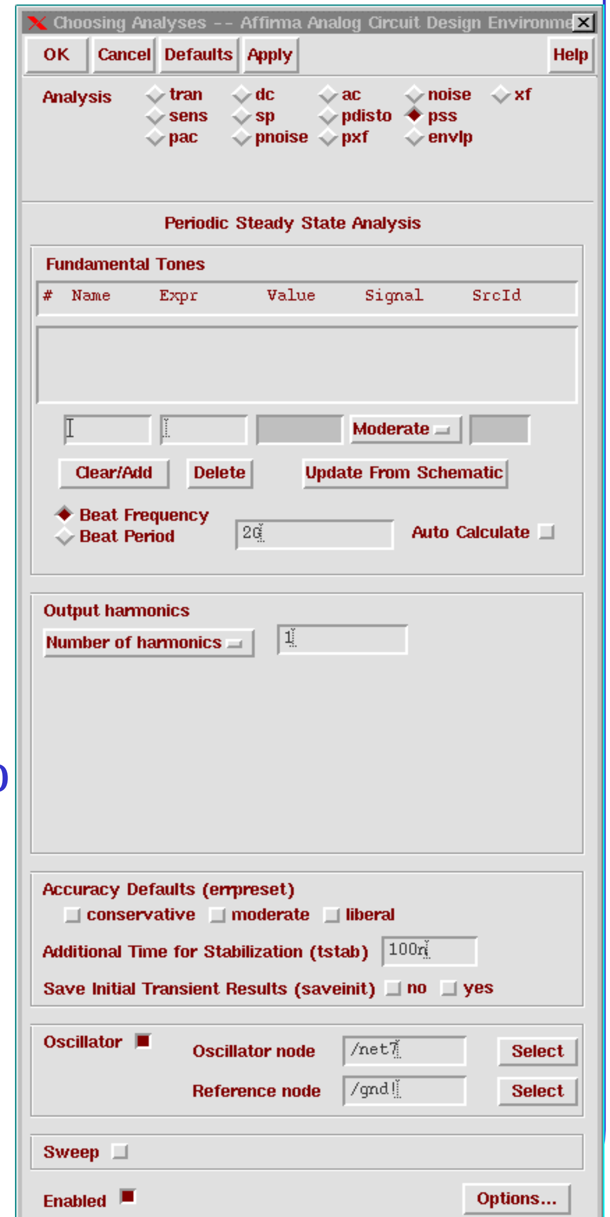
- Finally, in the Calculator, click erplot
- Note the initial estimate of the oscillation frequency is developed.





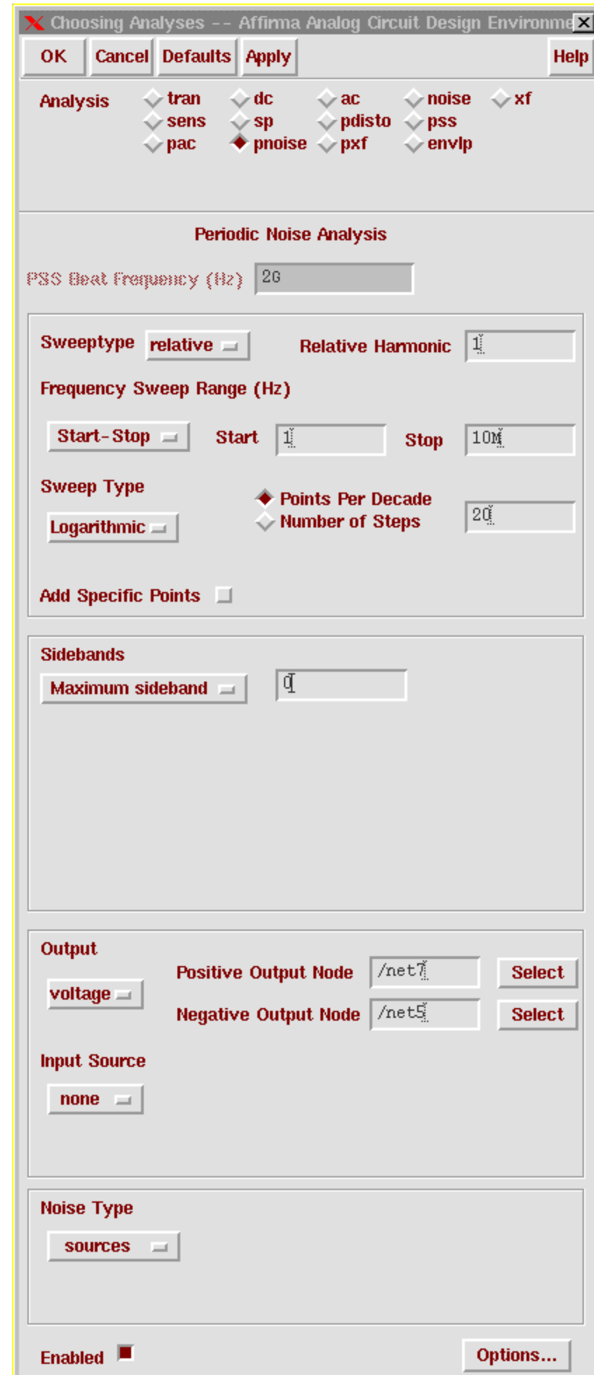
PSS/PNOISE Analysis(1)

- In the ***Choosing Analysis*** window, turn off the transient analysis; select the **pss** analysis and set up the form as right:
- An estimate of 2GHz was selected for Beat Frequency. It's recommended to estimate a lower frequency than expected to help in the convergence.
- The value of **tstab** is set to 100n to inform the simulator that the oscillation needs 100ns to stabilize to a steady-state waveform.
- Remember to choose the **gear2only** method in the options form.
- Click **Apply**.



PSS/PNOISE Analysis(2)

- Next, click the **pnoise** button, and set up the PNOISE analysis as right:
- The phase noise from 1 Hz to 10 MHz, ***relative*** to the derived oscillation frequency, will be calculated.
- The Sidebands field is set to a Maximum sideband of 0. In this case, you are interested in the upconverted 1/f device noise to the oscillation frequency. To account for higher harmonics of the oscillator that also contribute noise, change this value.
- No Input Source is specified.
- Click **OK**.



Run PSS & PNOISE Simulation

- Click the Run Simulation icon and use Direct function to see the results.
- Compare the oscillation frequency with the previous transient results.
- Click Plot icon, and the waveform window appears.