



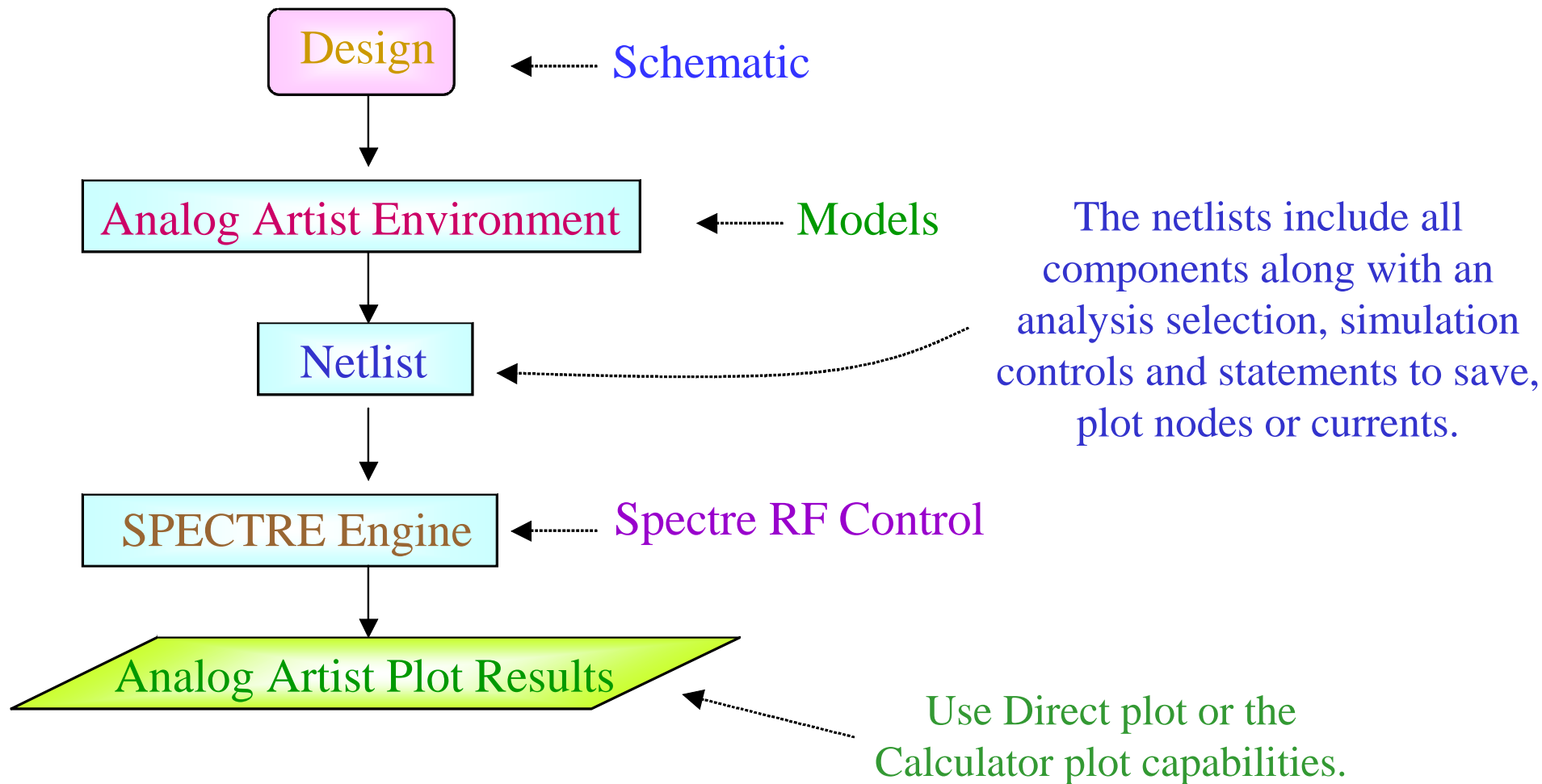
# 1. SpectreRF Overview

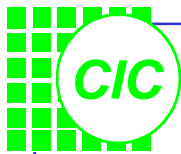
- SpectreRF is an optional feature added to Spectre ,and is represented by 6 analyses:
  1. PSS: Periodic Steady State Analysis
  2. PAC: Periodic AC Analysis
  3. PXF: Periodic Transfer Function Analysis
  4. PNOISE: Periodic Noise Analysis
    - Tdnoise: Time Domain Noise
    - QPNOISE: Quasi-Periodic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

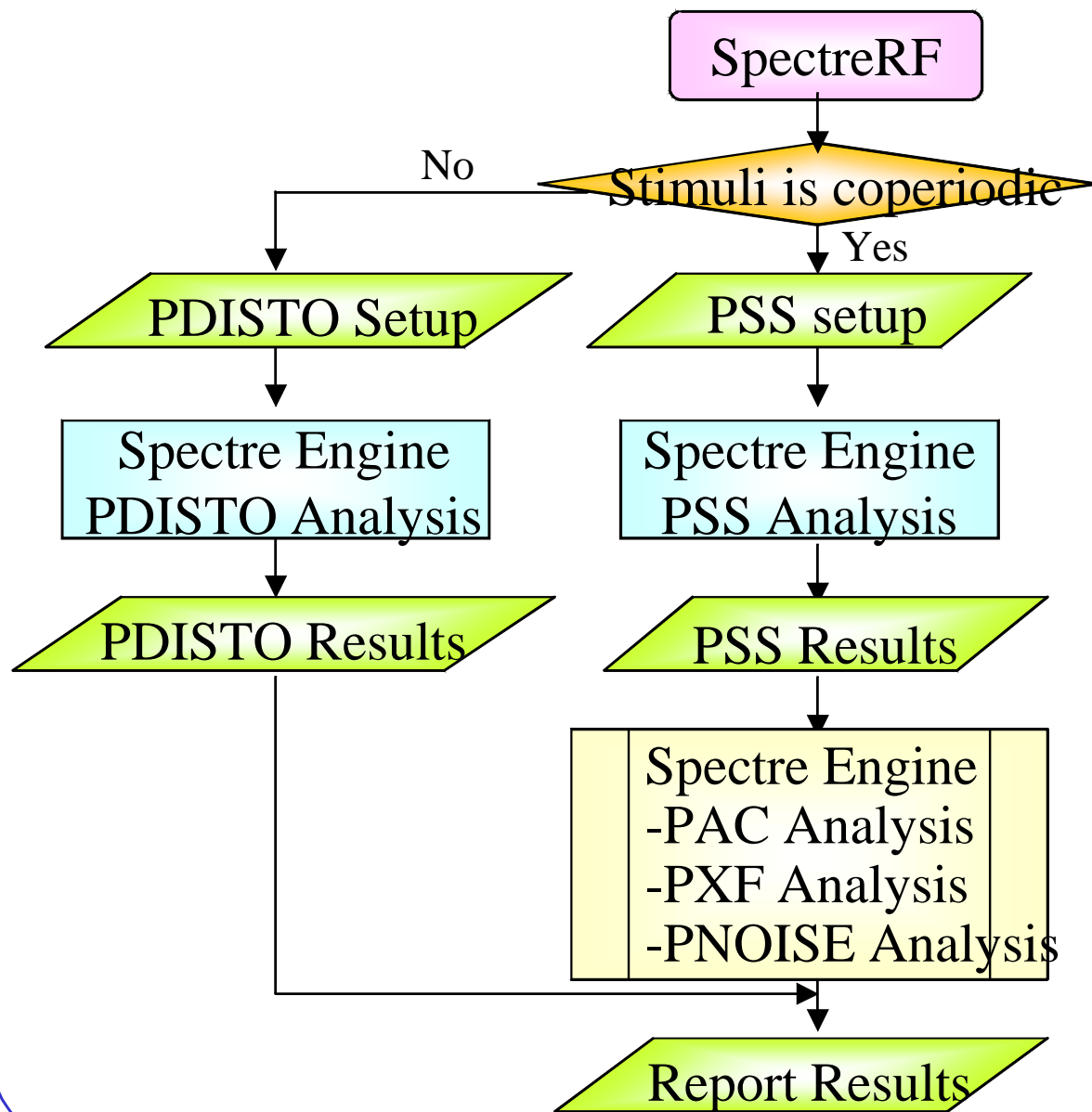


# SpectreRF in a Design Flow



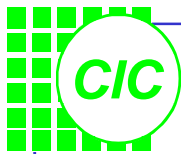


# SpectreRF Tool Flow



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.

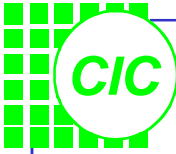


# SpectreRF Features

- 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.

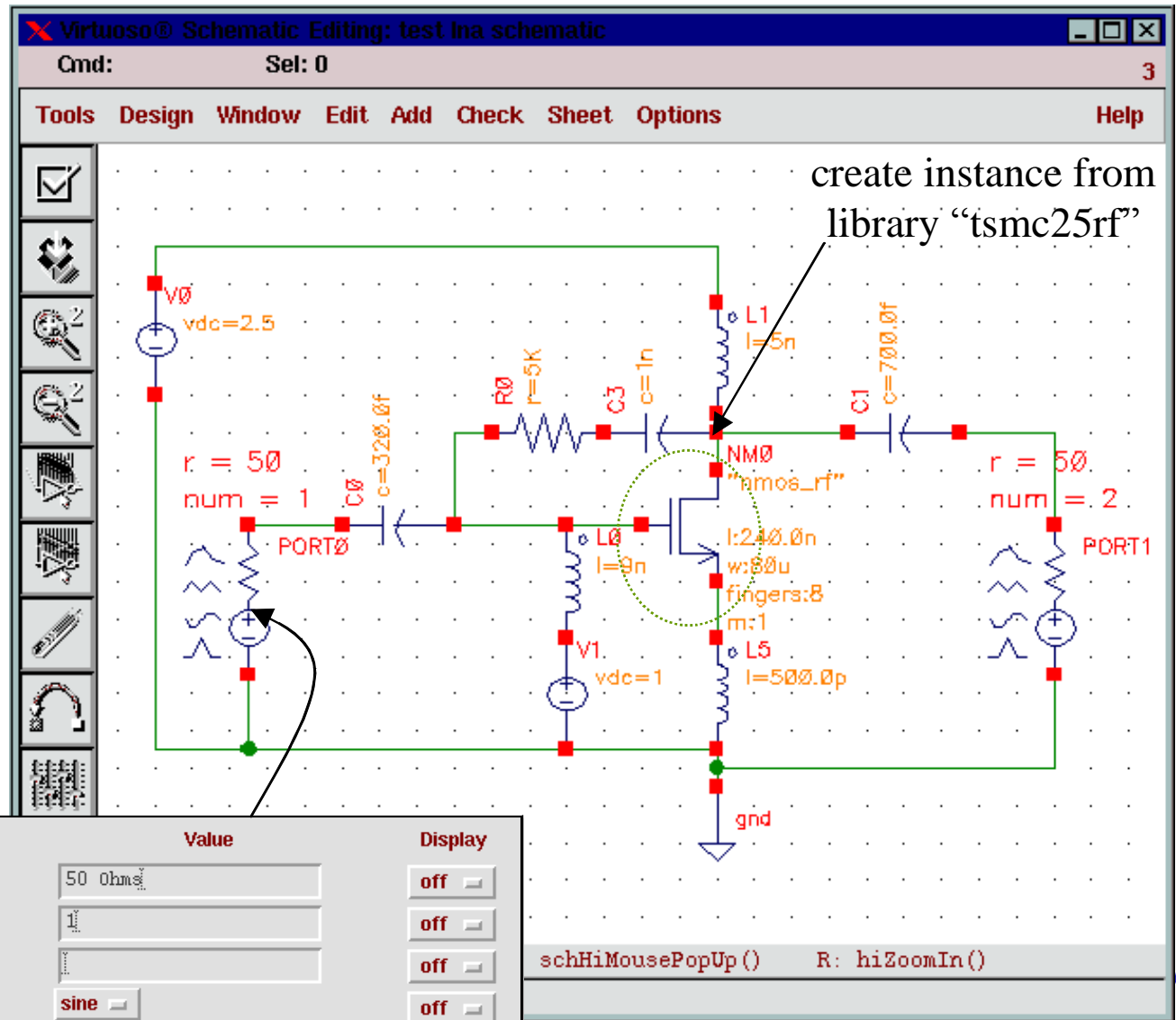


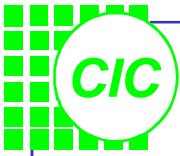
# Plotting S-Parameter Simulation Results

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 $\Gamma_{min}$ , $\Gamma_{opt}$ , or $\Gamma_{on}$ )
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

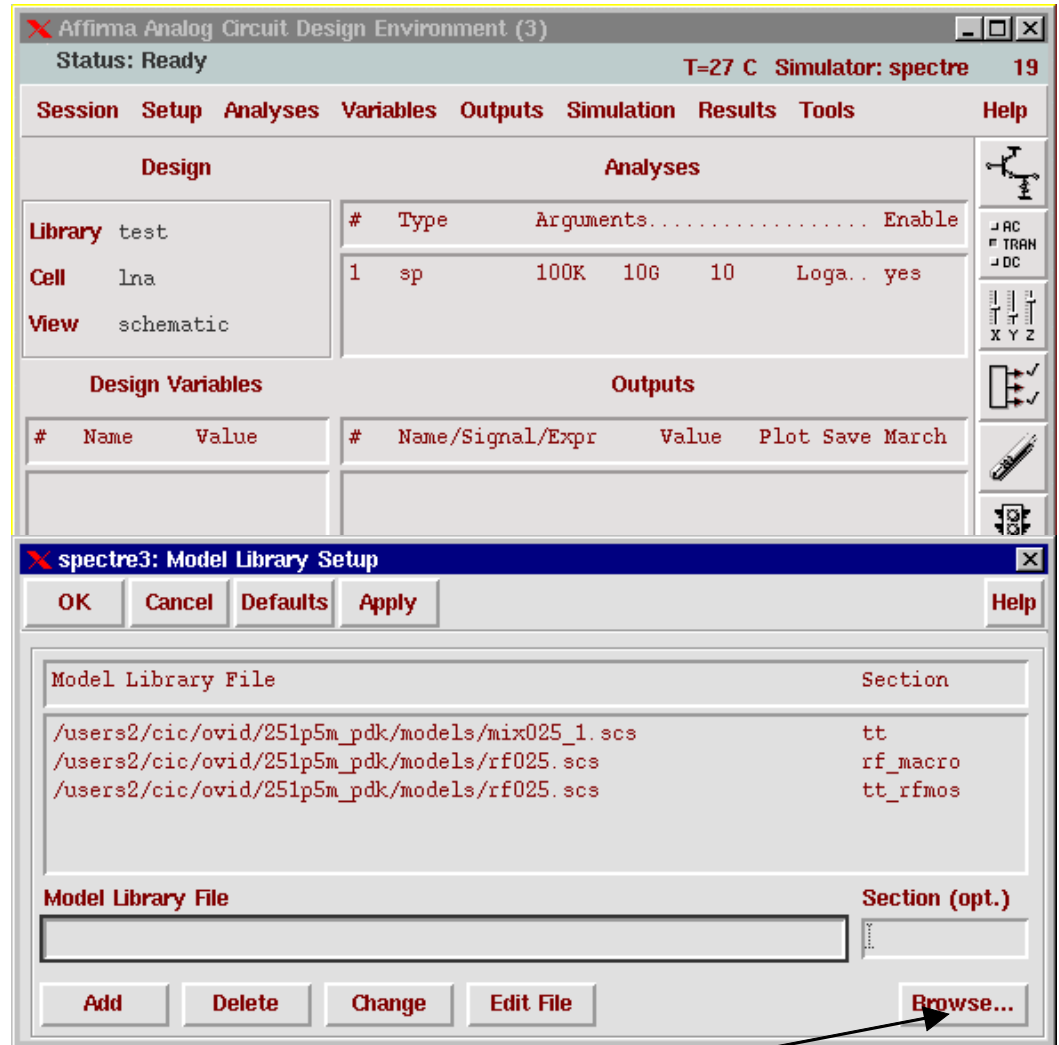
- Create a new library and a new schematic view.
- Use library “**analogLib**” & “**tsmc25rf**” to draw the scheme.
- After drawing, push **Design** → **Check and Save**; then push **Tools** → **Analog Environment**, and the window “*Affirma Analog Circuit Design Environment*” will appear.





# 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** ”.

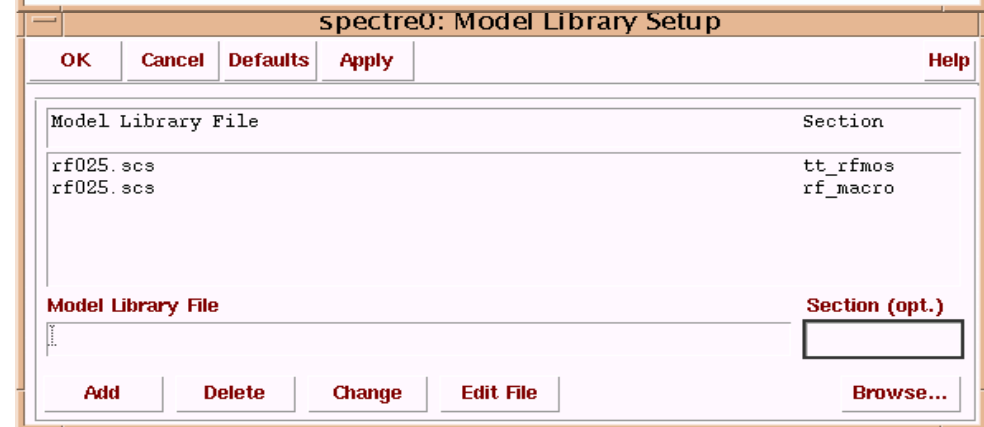
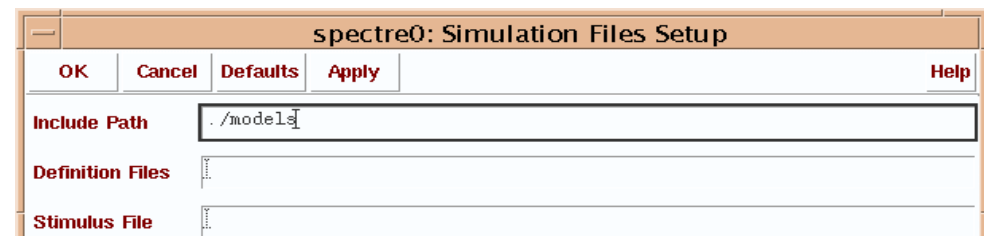
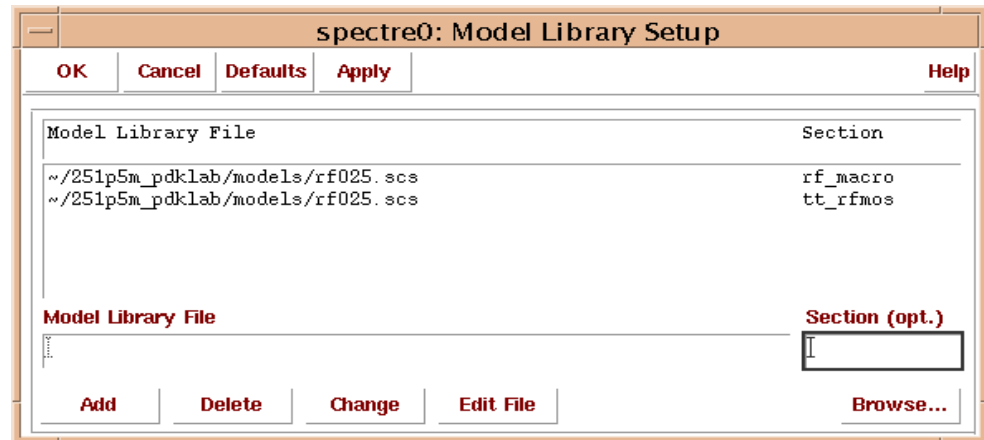


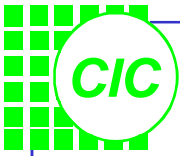
Use “**Browse**” to access to the model files



# Setup Design Environment(2)

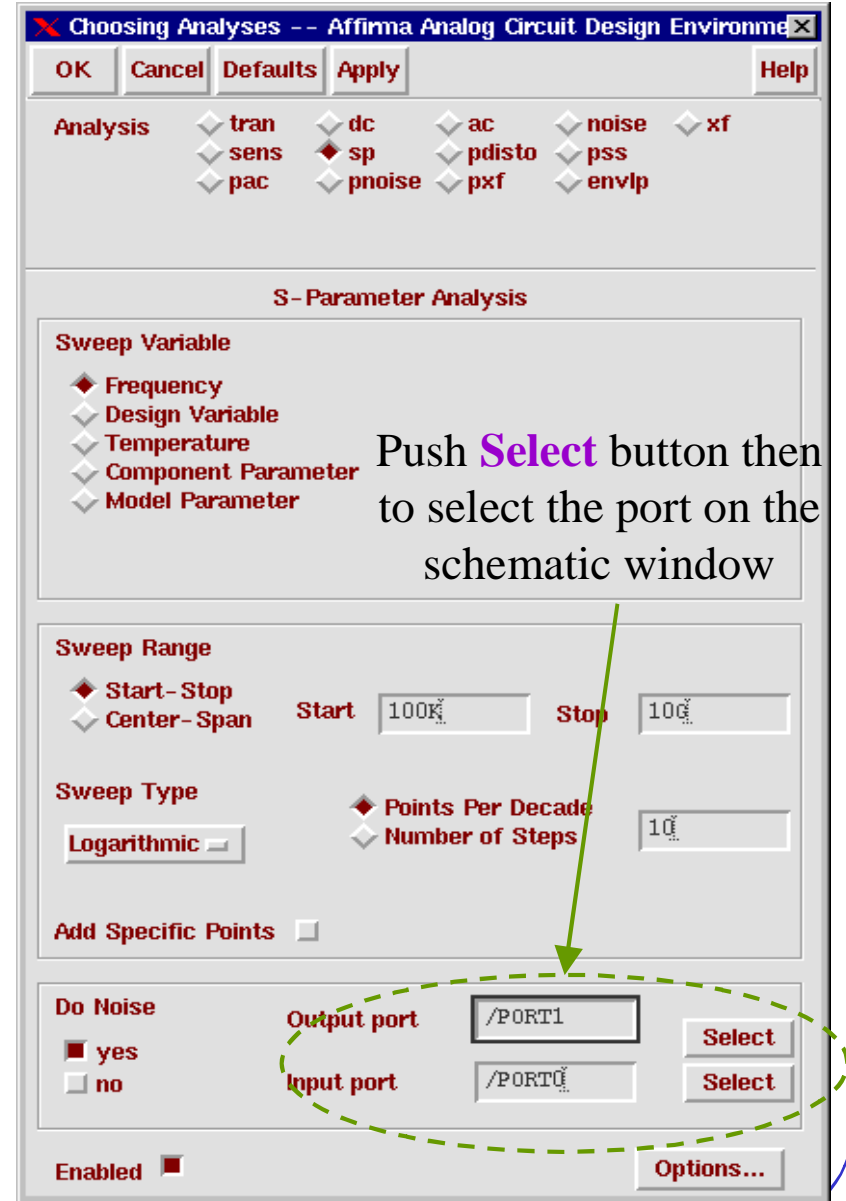
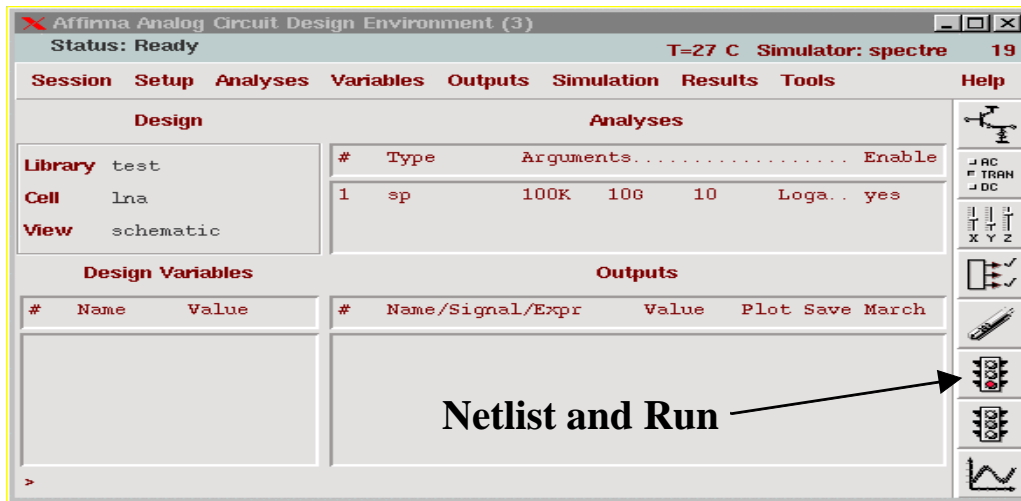
- You can use either an **absolute model path** or a **relative model path**
- IF you use the absolute approach, the setup is as shown right-upper.
- To use a relative path ,push “**Setup ® Simulation Files**”,than **Setup → Model Libraries** .The setup is as shown right.





# 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.

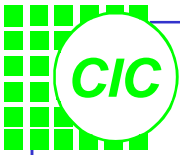
The screenshot shows the Affirma Analog Circuit Design Environment (3) interface. The main window displays the design setup for a circuit simulation. The Results menu is open, showing options like Direct Plot, Print, Annotate, and various plotting options. The S-Parameter Results dialog box is also visible, showing the function selection and plot type options.

Library	test	#	Type	Arguments...
Cell	lna	1	sp	100K 10G
View	schematic			

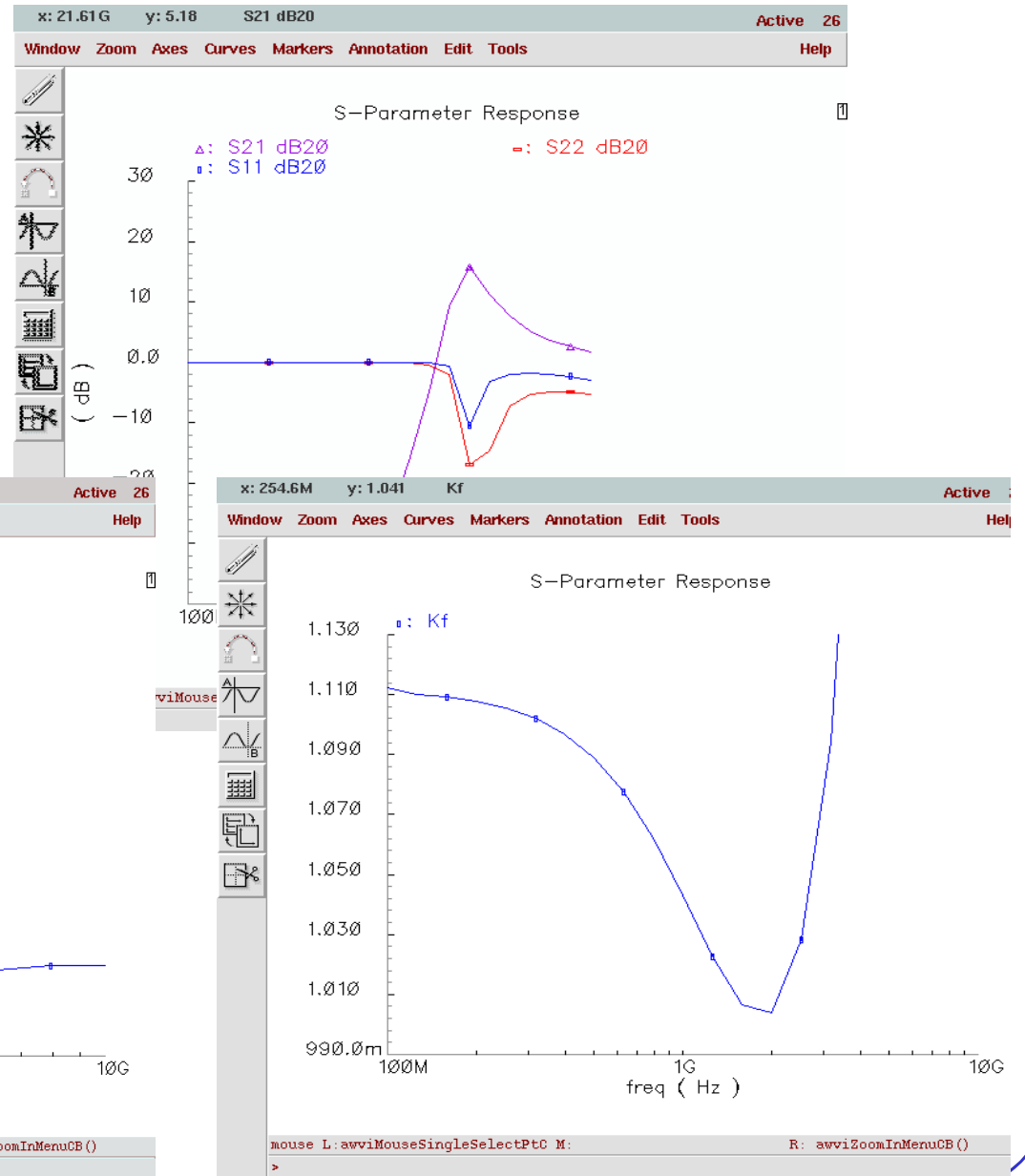
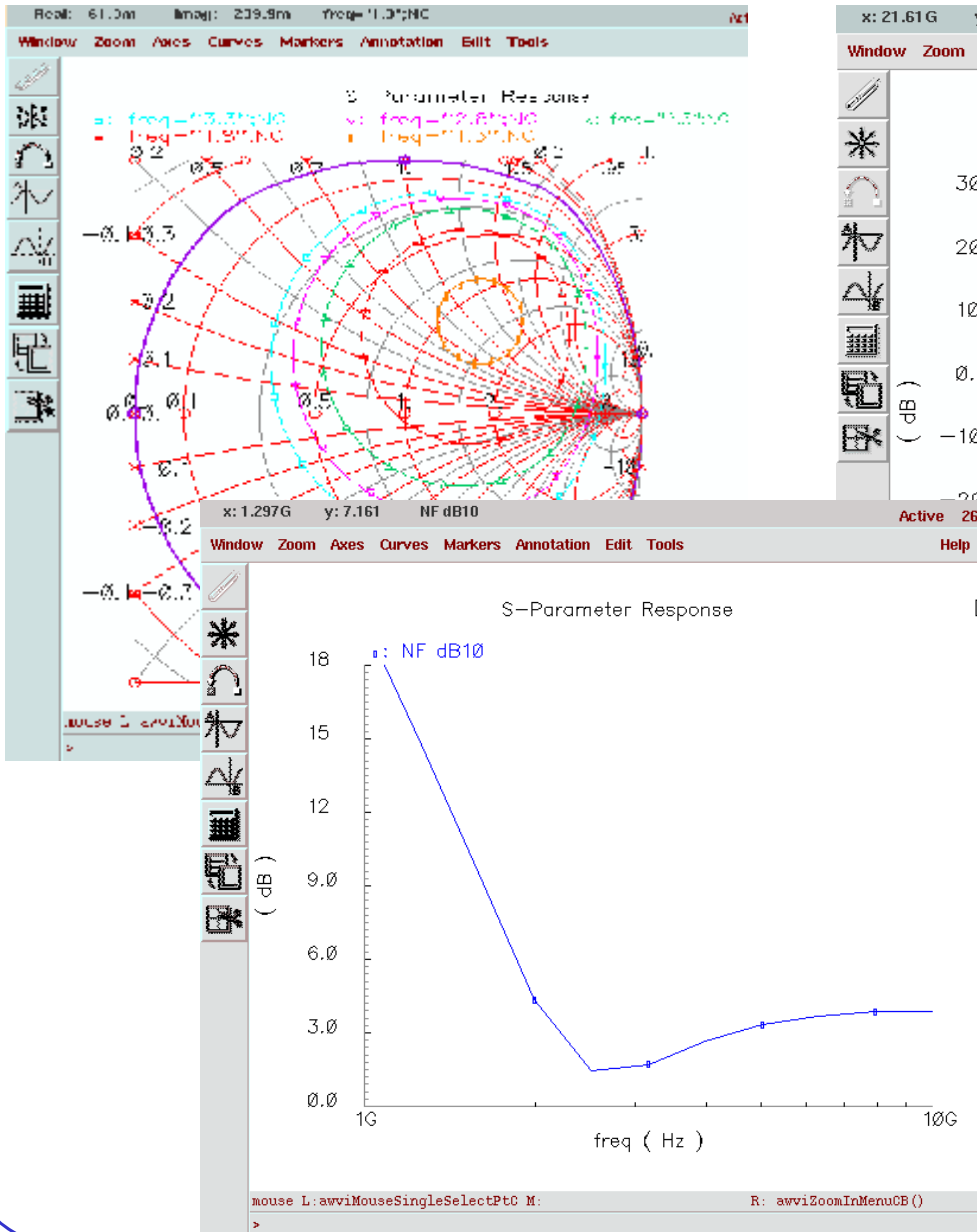
#	Name	Value	#	Name/Signal/Expr	Val

The S-Parameter Results dialog box is shown with the following settings:

- Plot Mode:  Append  Replace
- Function:  SP  ZP  YP  HP  GD  VSWR  NFmin  Gmin  Rn  m  NF  Kf  B1f  GT  GA  GP  Gmax  Gmsg  Gumx  ZM  NC  GAC  GPC  LSB  SSB
- Description: S-Parameter
- Plot Type:  Auto  Rectangular  Z-Smith  Y-Smith  Polar
- Modifier:  Magnitude  Phase  dB20  Real  Imaginary
- Buttons: S11, S12, S21, S22
- Add To Outputs:
- > Press Sij-button to plot...



# 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.

A screenshot of the 'S-Parameter Options' dialog box. The dialog has a title bar with a close button (X) and a standard Windows-style title. Below the title bar are buttons for 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'. The main area is divided into several sections:

- STATE-FILE PARAMETERS**: A text field labeled 'readns' is empty.
- OUTPUT PARAMETERS**: A text field labeled 'file' contains the path 'ers2/cic/ovid/251p5m\_pdk\_lab/lna\_test1.s2p'. Below it are radio buttons for 'oppoint': 'rawfile' (checked), 'screen', 'logfile', and 'no'.
- NOISE PARAMETERS**: A text field labeled 'reftemp (C)' is empty.
- CONVERGENCE PARAMETERS**: A radio button labeled 'restart' with options 'yes' and 'no' (both unchecked).
- ANNOTATION PARAMETERS**: Radio buttons for 'annotate' with options 'no', 'title', 'sweep', 'status' (checked), and 'steps'. Below it are radio buttons for 'stats' with options 'yes' and 'no' (both unchecked).

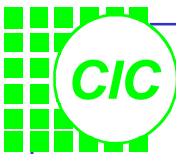


# S2P File

```
CIC
: S-parameter data file `~/users2/cic/ovid/251p5m_pdk_lab/lina_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

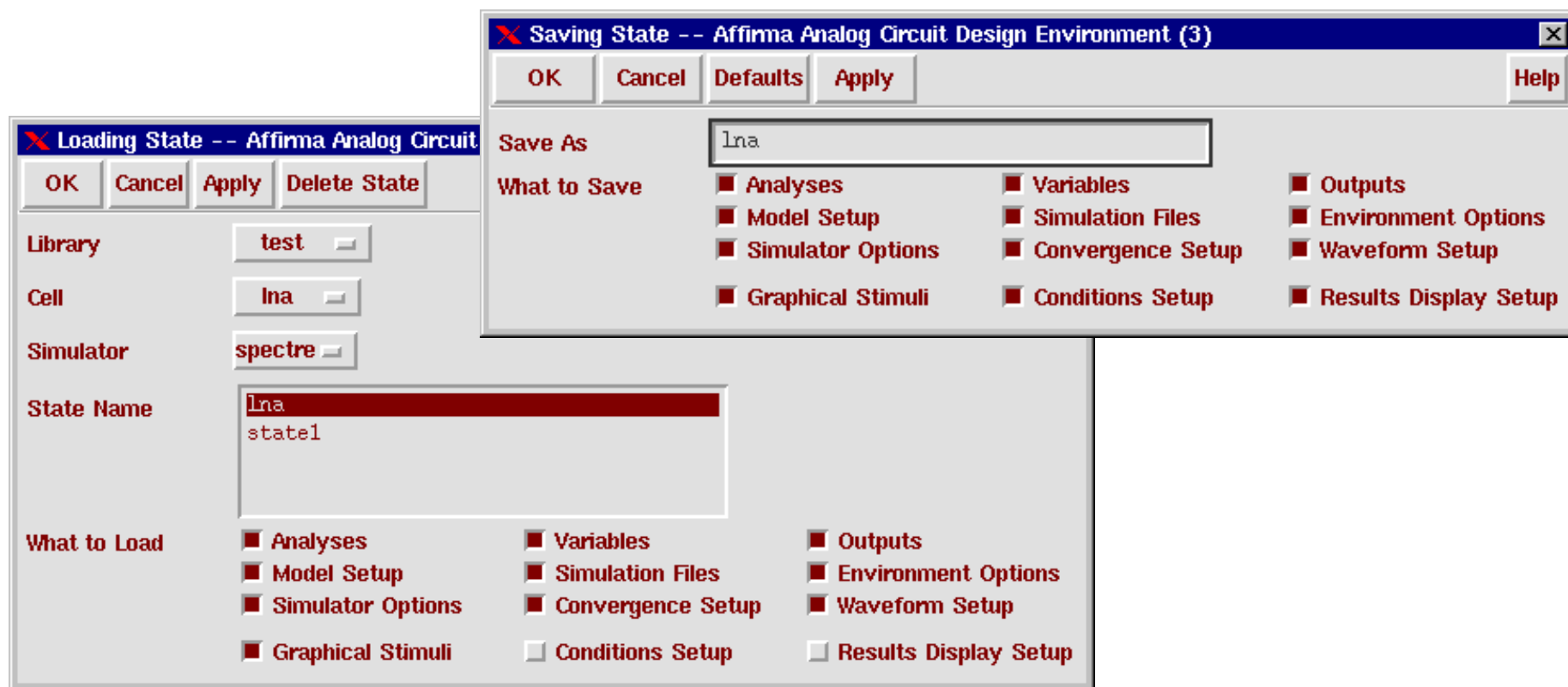
format  freq:    s11(real,imag)  s21(real,imag)
          s12(real,imag)  s22(real,imag)

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
```



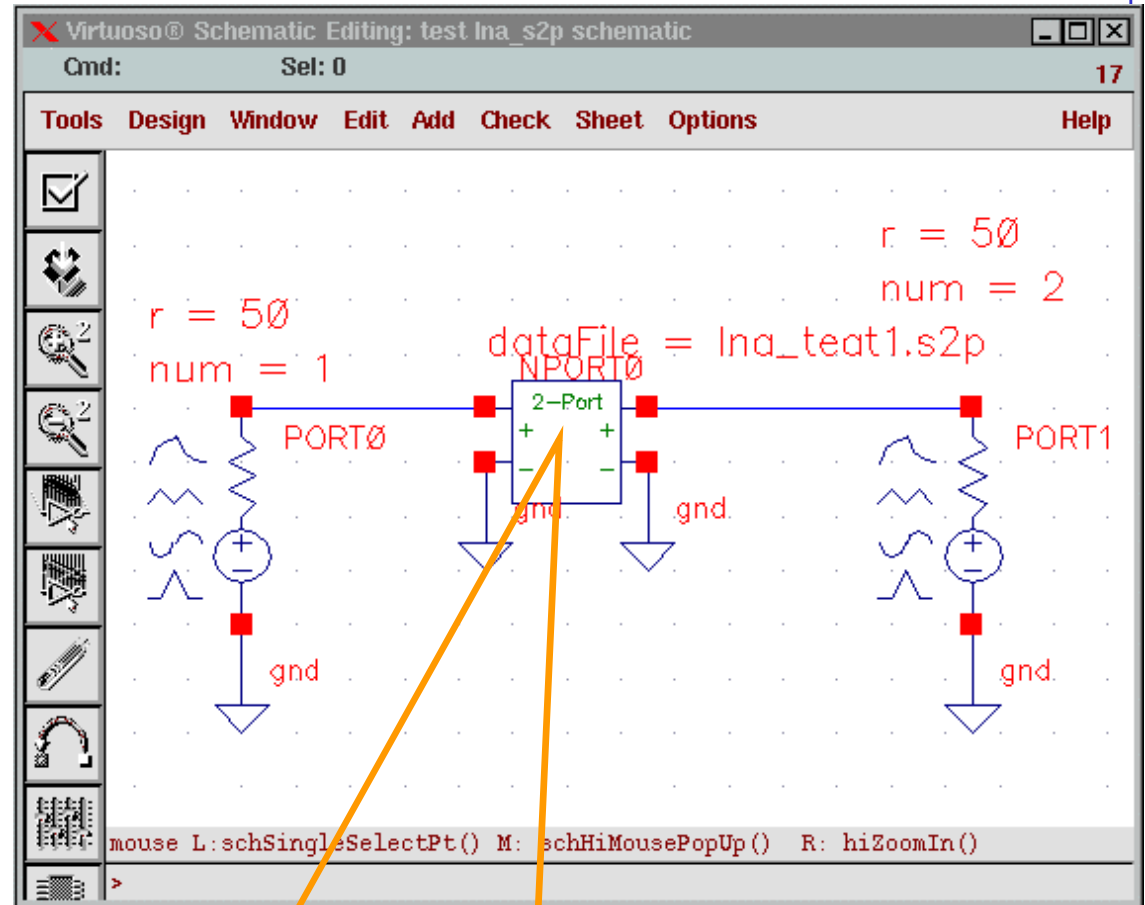
# Simulation State

- 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.



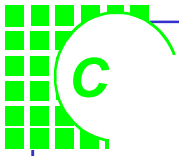
# Read the S2P file(1)

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



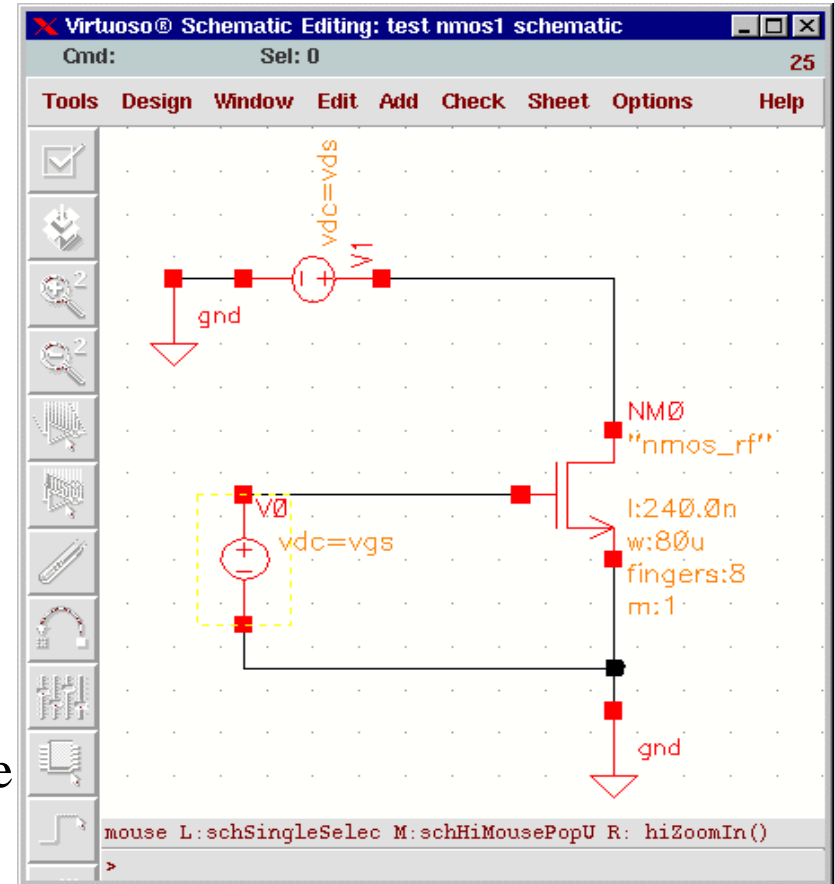
CDF Parameter	Value	Display
S-parameter data file	n_pdk_lab/lna_test1.s2p	both <input type="checkbox"/>
Multiplier		off <input type="checkbox"/>

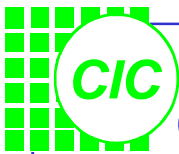




# 3. Lab2: Swept DC Analysis

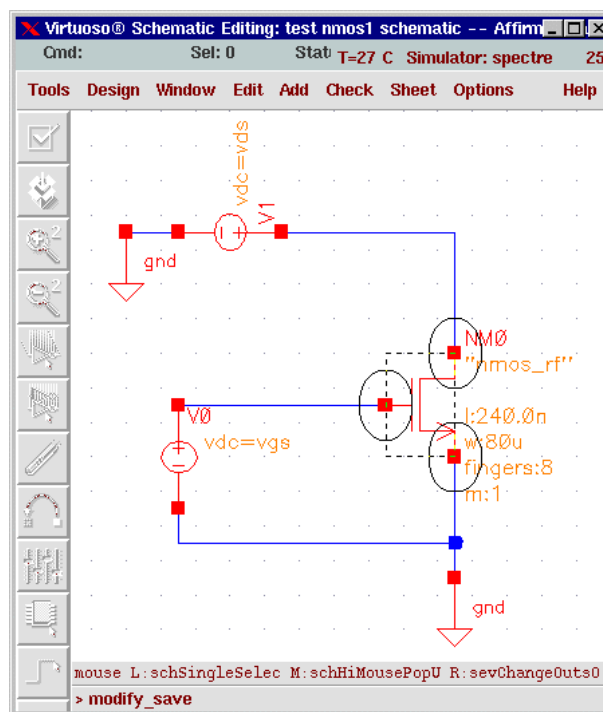
- Create a new schematic view and use library “**analogLib**” & “**tsmc25rf**” to draw the scheme.
- . After “**Check and Save**”; then call the window “*Affirma Analog Circuit Design Environment*”.
- **Setup** up the **Model Libraries**.
- Push **Variables** → **Copy From Cellview**, and the defined 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.



**Choosing Analyses -- Affirma Analog Circuit Design Environment**

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  xf  
 sens  sp  pdisto  pss  
 pac  pnoise  pxf  envlp

**DC Analysis**

Save DC Operating Point

**Sweep Variable**

Temperature  
 Design Variable Variable Name   
 Component Parameter   
 Model Parameter

**Sweep Range**

Start-Stop Start  Stop   
 Center-Span

**Sweep Type**

Step Size   
 Number of Steps

Add Specific Points

Enabled



# Set up the Design Environment(2)

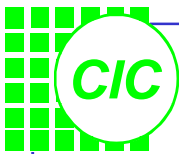
- In the window “*Design Environment*” select **Tools** → **Parametric Analysis...**; the window “*Parametric Analysis*” appears, then key in the values as below .
- In the window “*Parametric Analysis*” select **Analysis** → **Start** to start the simulation.

The screenshot shows the Affirma Analog Circuit Design Environment (2) window. The main window has a menu bar with Session, Setup, Analyses, Variables, Outputs, Simulation, Results, Tools, and Help. The status bar shows T=27 C, Simulator: spectre, and 12. The Design panel shows Library: test, Cell: nmos1, and View: schematic. The Analyses table has one entry: #1, Type: dc, Arguments: t 0 2.5 100m, Enable: yes. The Design Variables table has two entries: #1, Name: vgs, Value: 700m; #2, Name: vds, Value: 0. The Outputs table has three entries: #1, Name/Signal/Expr: NMO/D, Value: yes, Plot: yes, Save: yes, March: no; #2, Name/Signal/Expr: NMO/G, Value: yes, Plot: yes, Save: yes, March: no; #3, Name/Signal/Expr: NMO/S, Value: yes, Plot: yes, Save: yes, March: no. The Parametric Analysis - spectre(1): test nmos1 schematic window is open, showing Sweep 1 with Variable Name: vgs, Range Type: From/To, From: 0, To: 2.5, Step Control: Linear Steps, Total Steps: 0.5, and an Add Specification button.

#	Type	Arguments	Enable
1	dc	t 0 2.5 100m	yes

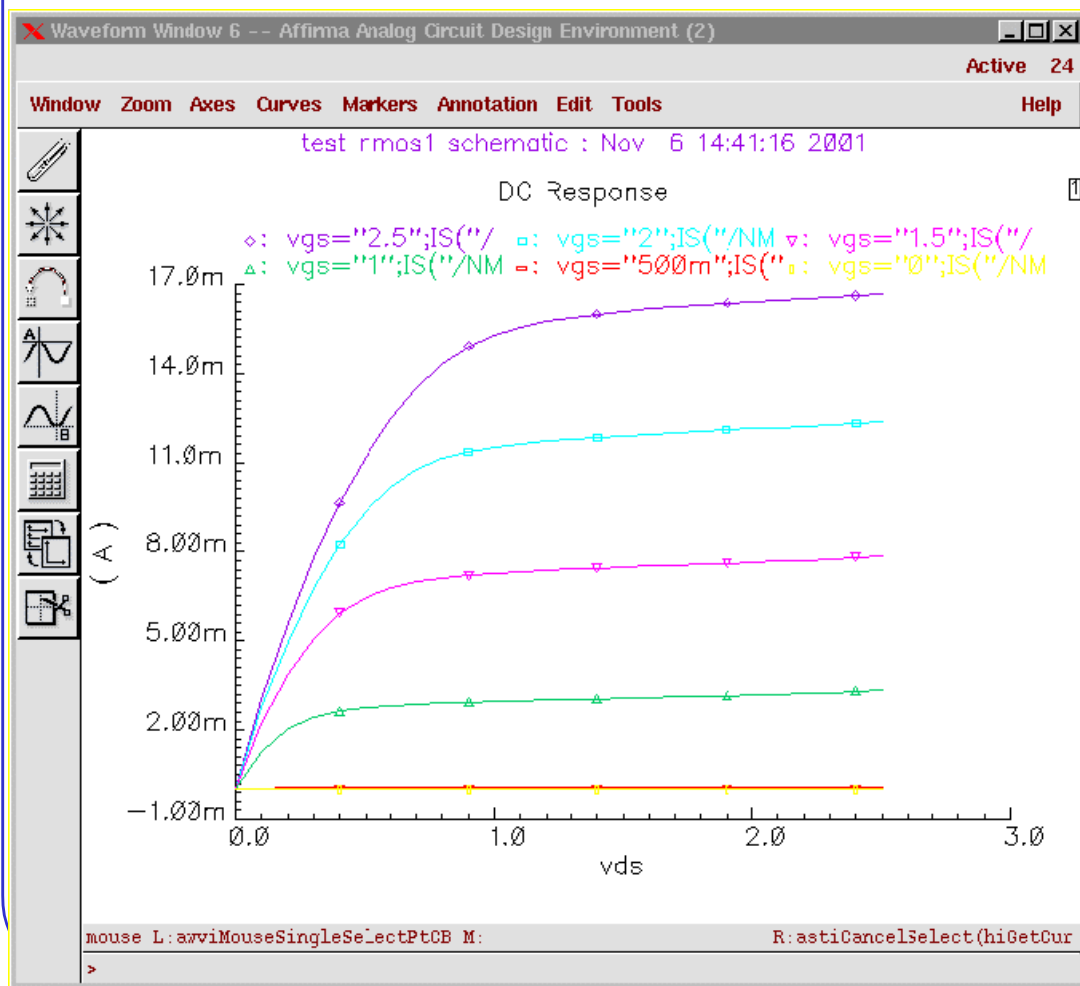
#	Name	Value
1	vgs	700m
2	vds	0

#	Name/Signal/Expr	Value	Plot	Save	March
1	NMO/D	yes	yes	yes	no
2	NMO/G	yes	yes	yes	no
3	NMO/S	yes	yes	yes	no



# 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.



Unit Design Environment (2) T=27 C Simulator: spectre 12

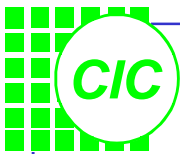
Analyses Variables Outputs Simulation Results Tools Help

Analyse Plot Outputs

- Direct Plot
  - Transient Signal
  - Transient Minus DC
  - Transient Sien
  - Transient Difference
  - AC Magnitude
  - AC dB10
  - AC dB20
  - AC Phase
  - AC Magnitude & Phase
  - AC Difference
  - Equivalent Output Noise
  - Equivalent Input Noise
  - Squared Output Noise
  - Squared Input Noise
  - Noise Figure
  - DC**
  - S-Parameter ...
  - XF ...
  - PSS ...
  - PDISTO ...
  - ENVLP ...
- Print
- Annotate
- Circuit Conditions ...
- Save ...
- Outputs Select ...
- Delete ...
- Printing/Plotting Options ...

#	Type	Arguments...	Value
1	dc	t 0	

#	Name/Signal/Expr	Value
1	NM0/D	yes yes no
2	NM0/G	yes yes no
3	NM0/S	yes yes no

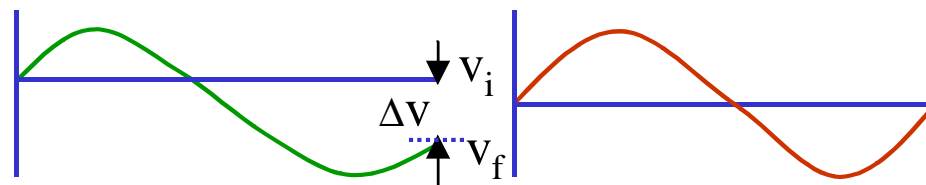


## 4. Periodic Steady State Analysis

- 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.

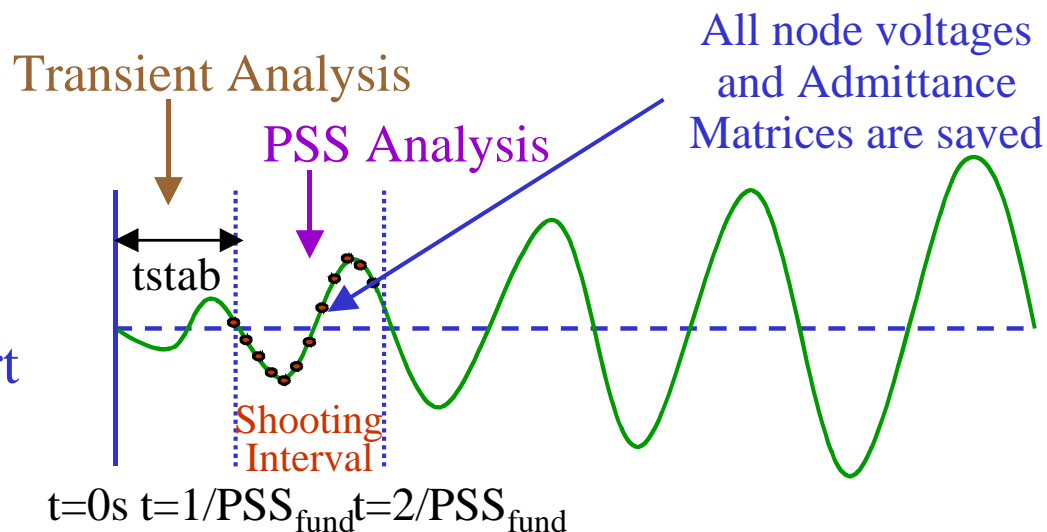
# Shooting Newton Method

- 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  $t=(tstab+1/PSS_{fund})$  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.

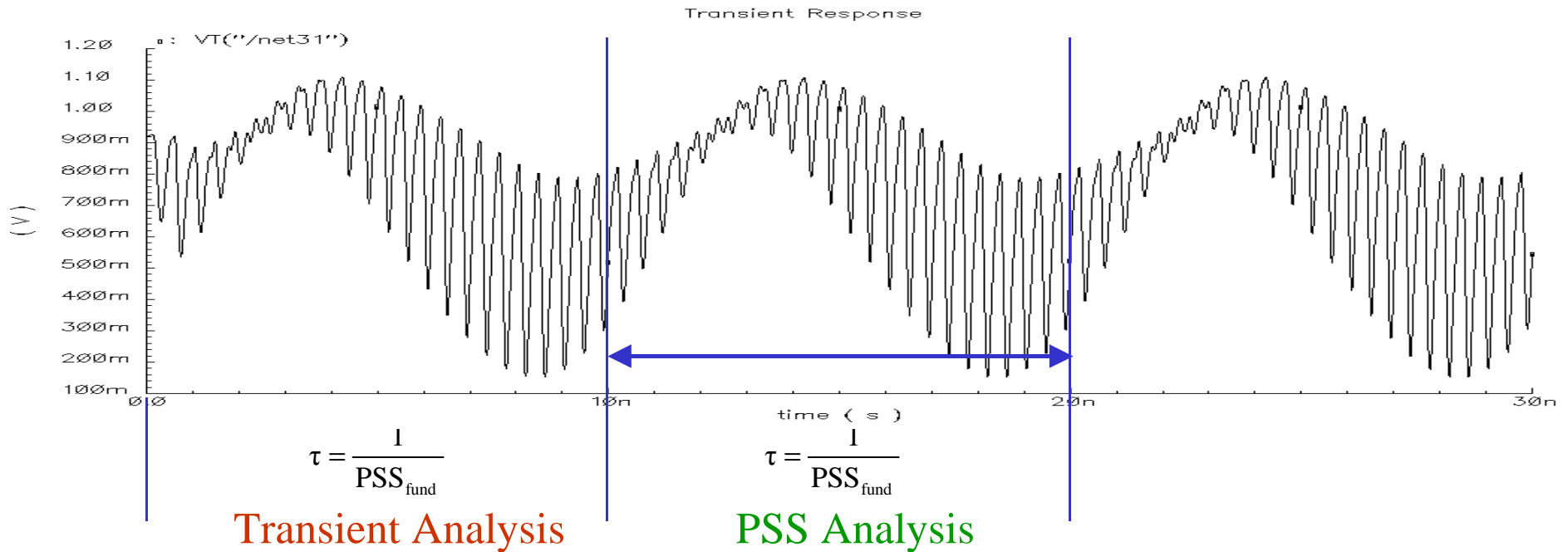


The signal starts at a point  $v_i$  doesn't result in periodicity.

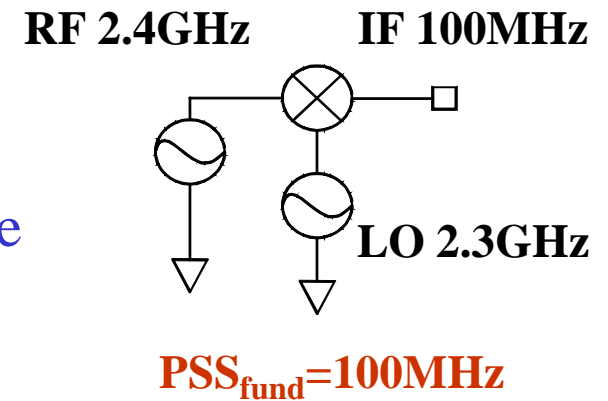
The starting point is adjusted by the shooting method to result in periodic steady state.

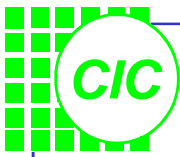


# Shooting Newton Method(continued)



- 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.

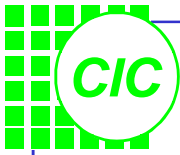




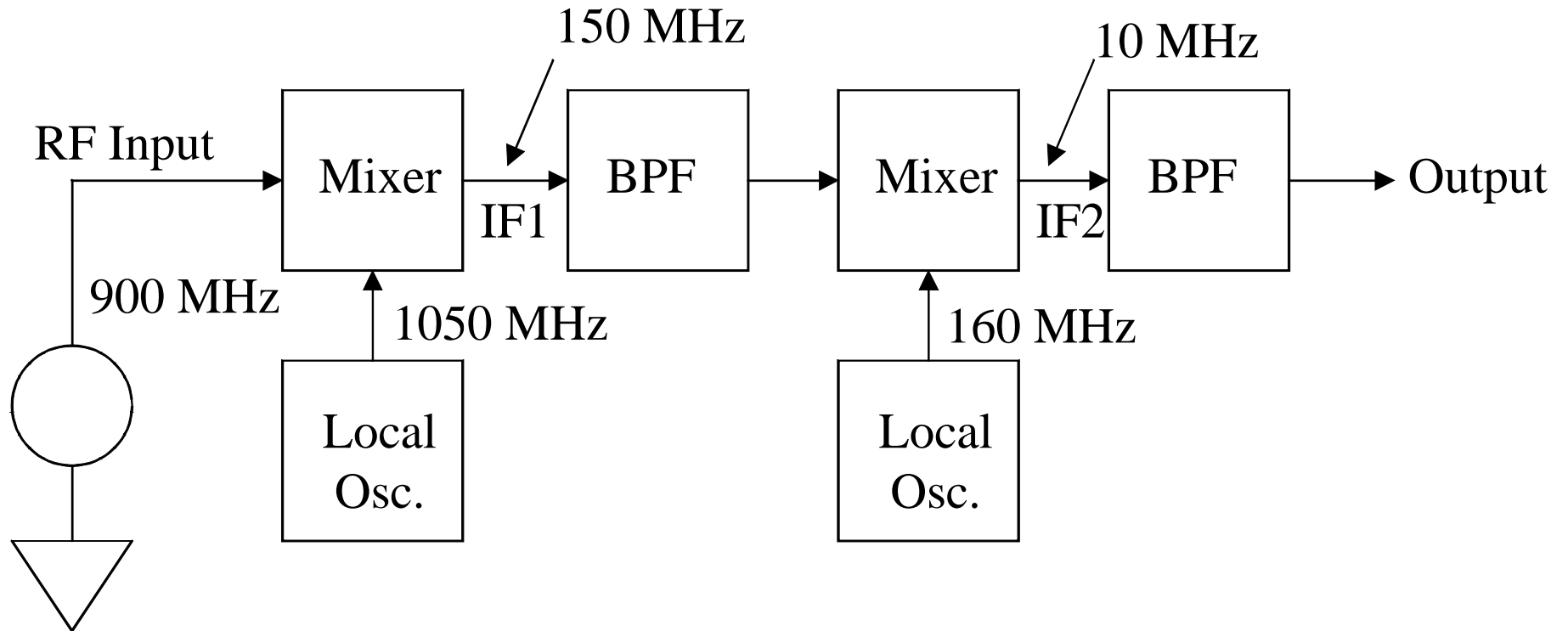
# PSS Analysis Assumptions

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

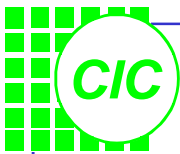




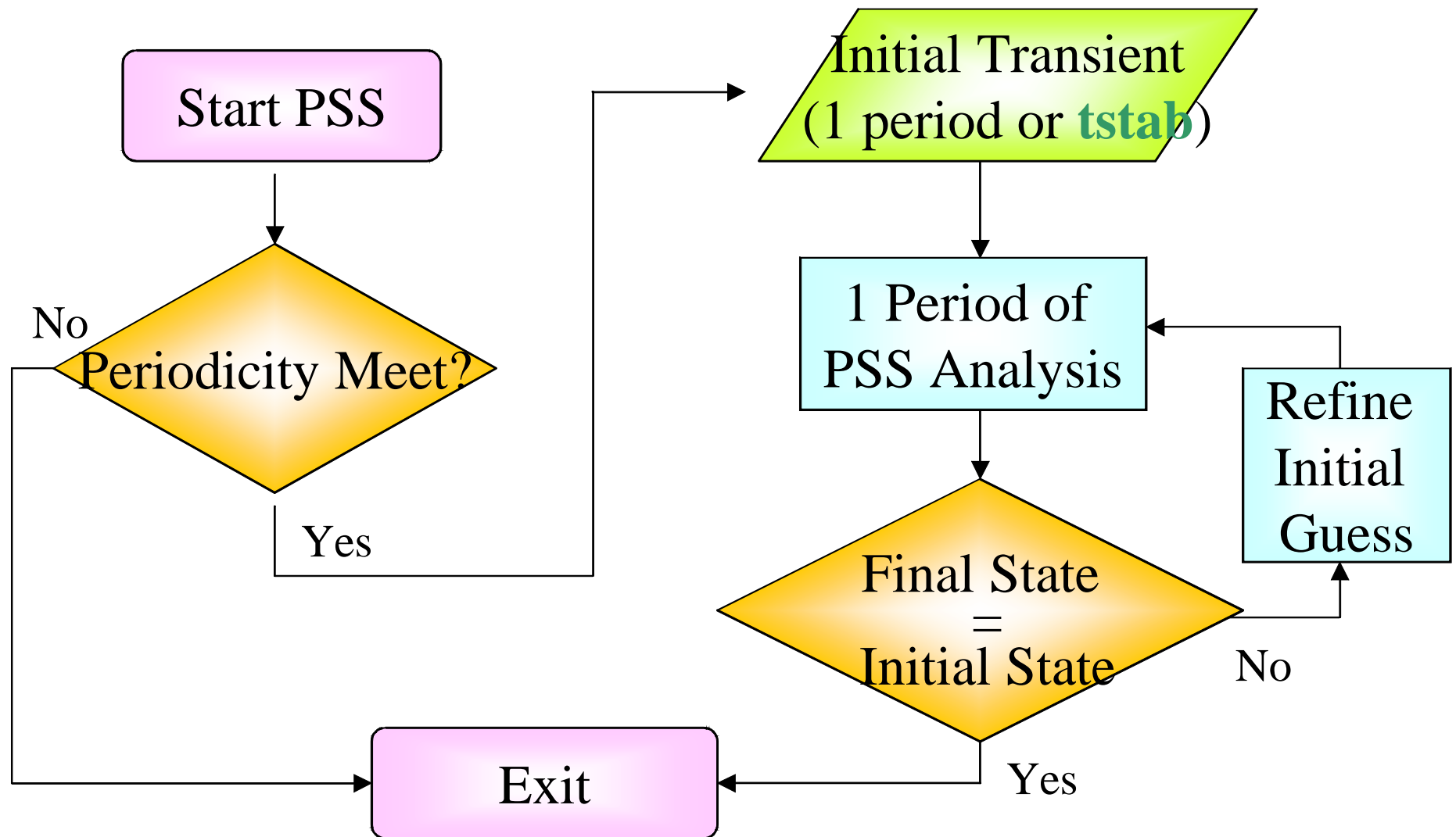
# The PSS Fundamental

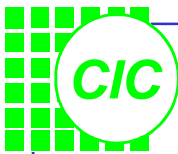


$$PSS_{\text{fund}} = 10 \text{ MHz}$$



# PSS Operation





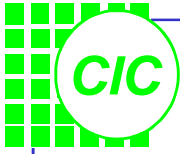
# Simulator Accuracy Suggestions

- Do not set “conservative”. This will dramatically extend the simulation time.
- $\Delta V$  and  $\Delta I < \text{reitol} * \text{lteratio} * \text{steadyratio}$
- 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
reitol	1e-3	<b>1e-5</b>
vabstol	1e-6	<b>3e-8</b>
iabstol	1e-12	<b>1e-13</b>
Method	trap	<b>gear2only</b>

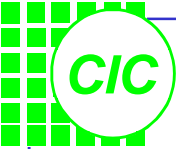
errpreset	reitol	relref	method	maxstep	steady-ratio	Iteratio
liberal	x10.0	allglobal	gear2	<0.4/maxacfreq	0.1	3.5
moderate	x1.0	sigglobal	traponly	<0.2/maxacfreq	0.001	3.5
conservative	x0.1	alllocal	gear2only	<0.1/maxacfreq	0.00001	10.0



# 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.

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

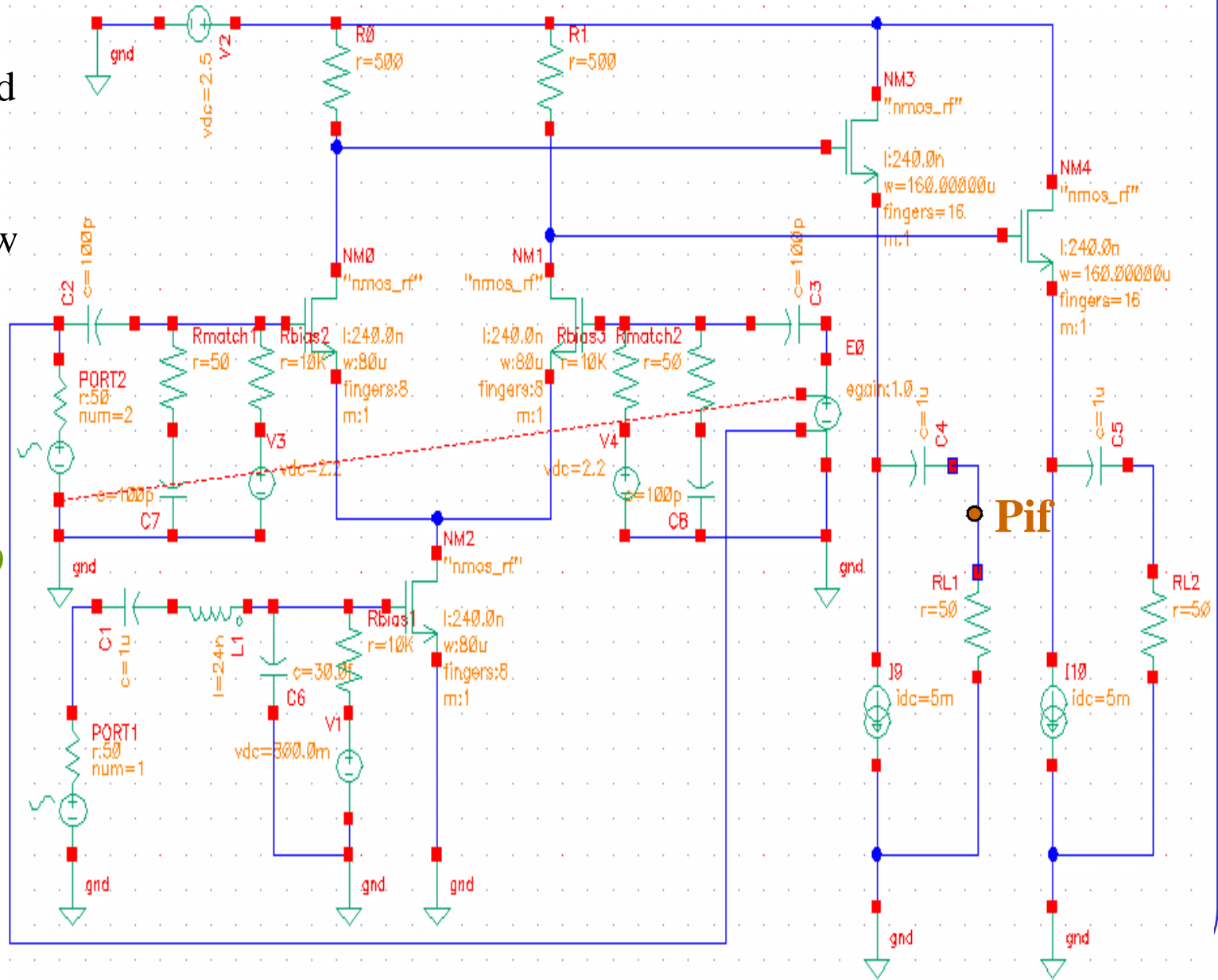


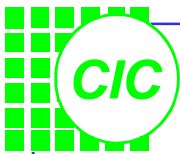
# Lab3 : PSS and swept PSS Analysis

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.

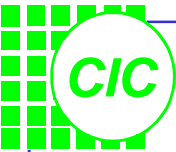
The screenshot shows the Affirma Analog Circuit Design Environment interface. The main window has a menu bar with options: Session, Setup, Analyses, Variables, Outputs, Simulation, Results, Tools, and Help. The status bar indicates 'Status: Ready', 'T=27 C', 'Simulator: spectre', and '4'. The 'Design' section shows 'Library: test', 'Cell: mixer', and 'View: schematic'. The 'Design Variables' section contains a table with two rows:

#	Name	Value	#
1	frf	2.4G	
2	flo	2.3G	

An 'Editing Design Variables' dialog box is open, showing the 'Selected Variable' section with 'Name: flo' and 'Value (Expr): 2.3G'. The 'Table of Design Variables' section contains a table with two rows, where the second row is highlighted:

#	Name	Value
1	frf	2.4G
2	flo	2.3G

The dialog box also includes buttons for 'OK', 'Cancel', 'Apply', 'Apply & Run Simulation', and 'Help'. At the bottom, there are 'Cellview Variables' and 'Copy From' / 'Copy To' buttons.



# 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**.

INTEGRATION METHOD PARAMETERS

method	<input type="checkbox"/> euler	<input type="checkbox"/> trap	<input type="checkbox"/> traonly
	<input type="checkbox"/> gear2	<input checked="" type="checkbox"/> gear2only	

Choosing Analyses -- Affirma Analog Circuit Design Environment

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  xf  
 sens  sp  pdisto  pss  
 pac  pnoise  pxf  envlp

Periodic Steady State Analysis

Fundamental Tones

#	Name	Expr	Value	Signal	SrcId
2	F1	frf	2.4G	Moderate	PORT1
3	F2	flo	2.3G	Moderate	PORT2

Moderate

Clear/Add Delete Update From Schematic

Beat Frequency  
 Beat Period 100M Auto Calculate

Output harmonics  
Number of harmonics 50

Accuracy Defaults (emreset)  
 conservative  moderate  liberal

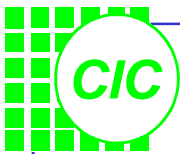
Additional Time for Stabilization (tstab) 1

Save Initial Transient Results (saveinit)  no  yes

Oscillator

Sweep

Enabled  Options...



# 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** → **Netlist and Run** and **Run** to start the simulation. Note if the *Conv norm* is less than 1 or if the PSS simulation has a convergent result.

The screenshot shows the Affirma Analog Circuit Design Environment (1) window. The main window displays the Design tab with a table of Design Variables:

#	Name	Value	#	Name/Signal/E
1	frf	2.4G		
2	flo	2.3G		

The Simulation menu is open, showing the Options sub-menu. The Simulator Options dialog box is also open, showing the following settings:

**TOLERANCE OPTIONS**

- reltol: 1e-5
- vabstol: 3e-8
- iabstol: 1e-13

**TEMPERATURE OPTIONS**

- temp: 27

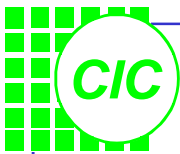
The simulation results window shows the following output:

```
pss: time = 8.859 ns (87.5 %), step = 8.122 ps (81.2 m%)
pss: time = 9.362 ns (92.5 %), step = 9.105 ps (91 m%)
pss: time = 9.859 ns (97.5 %), step = 6.162 ps (61.6 m%)
Conv norm = 69.1, max dV(net0110) = 1.46932 mV, took 2.15 s.

pss: time = 365 ps (2.56 %), step = 8.08 ps (80.8 m%)
pss: time = 859.1 ps (7.5 %), step = 1.951 ps (19.5 m%)
pss: time = 1.359 ns (12.5 %), step = 2.678 ps (26.78 m%)
pss: time = 1.862 ns (17.5 %), step = 5.926 ps (59.26 m%)
pss: time = 2.36 ns (22.5 %), step = 8.878 ps (88.78 m%)
pss: time = 2.863 ns (27.5 %), step = 5.836 ps (58.36 m%)
pss: time = 3.361 ns (32.5 %), step = 5.053 ps (50.53 m%)
pss: time = 3.867 ns (37.6 %), step = 8.321 ps (83.21 m%)
pss: time = 4.364 ns (42.6 %), step = 8.976 ps (89.76 m%)
pss: time = 4.865 ns (47.6 %), step = 7.645 ps (76.45 m%)
pss: time = 5.359 ns (52.5 %), step = 7.123 ps (71.23 m%)
pss: time = 5.86 ns (57.5 %), step = 2.543 ps (25.43 m%)
pss: time = 6.359 ns (62.5 %), step = 2.683 ps (26.83 m%)
pss: time = 6.865 ns (67.6 %), step = 6.326 ps (63.26 m%)
pss: time = 7.36 ns (72.5 %), step = 8.877 ps (88.77 m%)
pss: time = 7.862 ns (77.5 %), step = 5.64 ps (56.4 m%)
pss: time = 8.361 ns (82.5 %), step = 4.894 ps (48.94 m%)
pss: time = 8.859 ns (87.5 %), step = 8.122 ps (81.2 m%)
pss: time = 9.362 ns (92.5 %), step = 9.105 ps (91 m%)
pss: time = 9.859 ns (97.5 %), step = 6.162 ps (61.6 m%)
Conv norm = 59.8e-03, max dV(net0110) = -1.27135 uV, took 2.06 s.

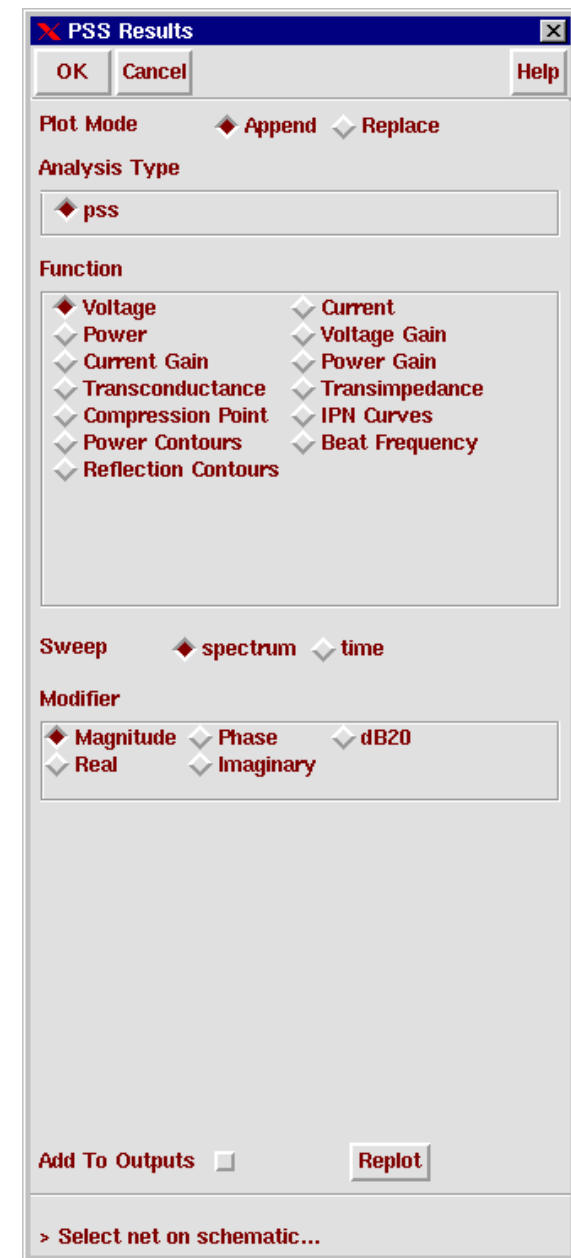
pss: The steady-state solution was achieved in 9 iterations.
Number of accepted pss steps = 2081.
Total time required for pss analysis 'pss' was 24.44 s.
```

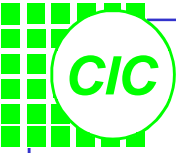




# Display the Conversion Power Gain- method 1

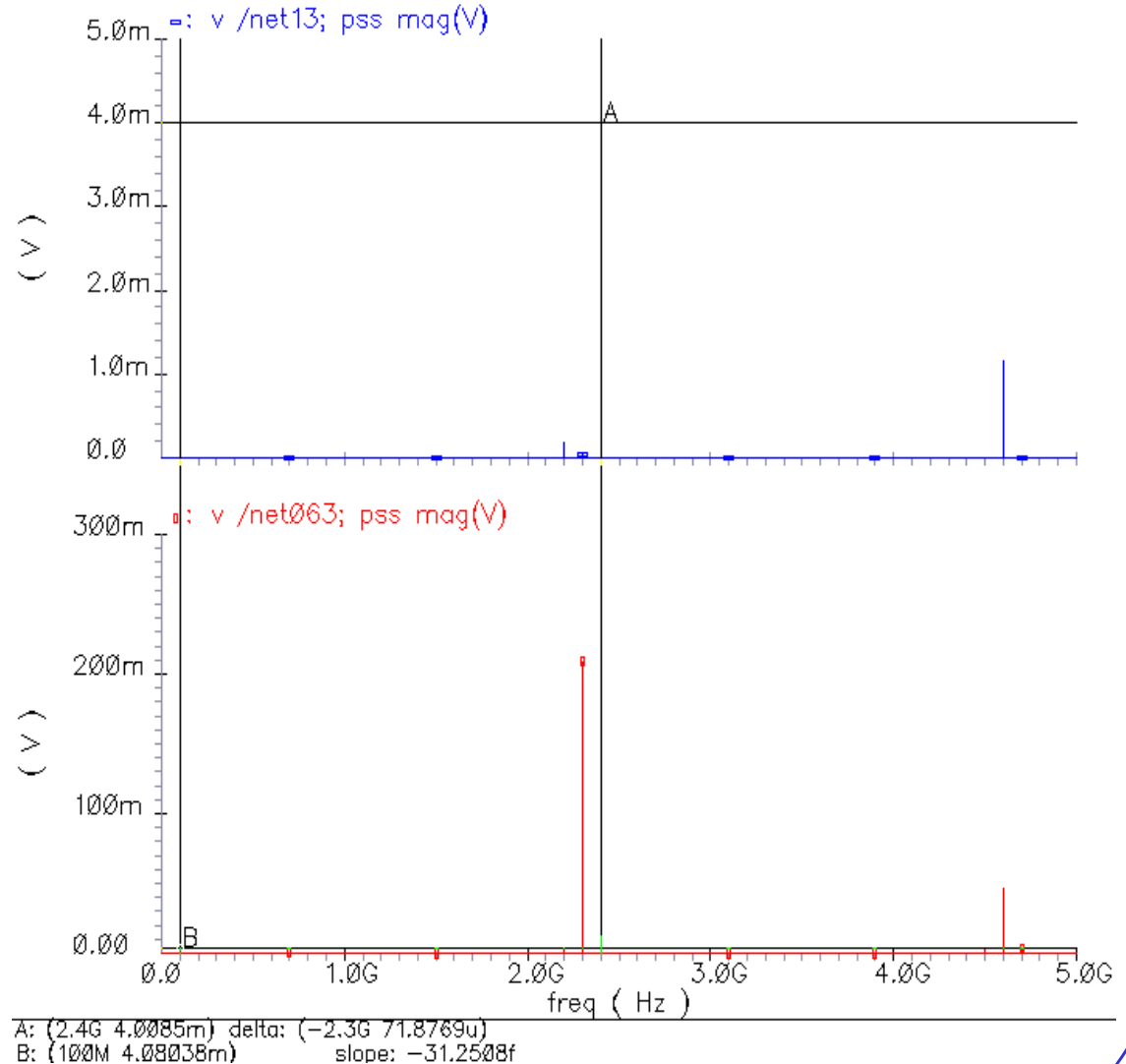
- In the Analog Artist Simulation Window, select **Results** → **Direct Plot** → **PSS**. Note the prompts on the bottom of the schematic and *PSS Results* windows.
- The *PSS Results* window **MUST** be on the screen when probing the nodes in the schematic. Don't push OK.
- In the PSS Results form, 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**.





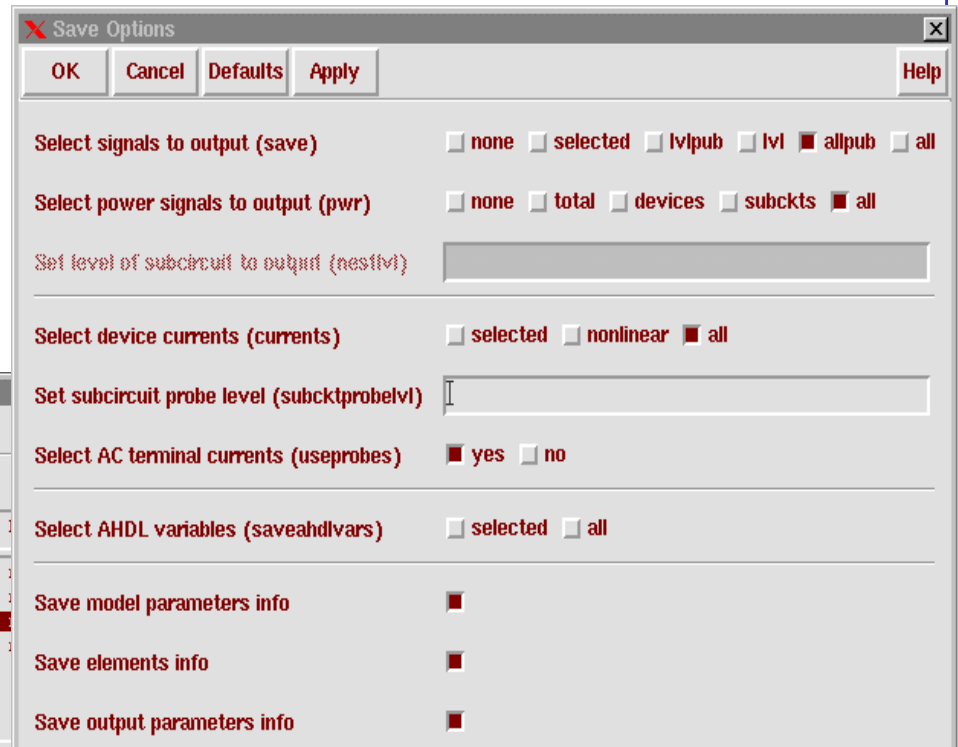
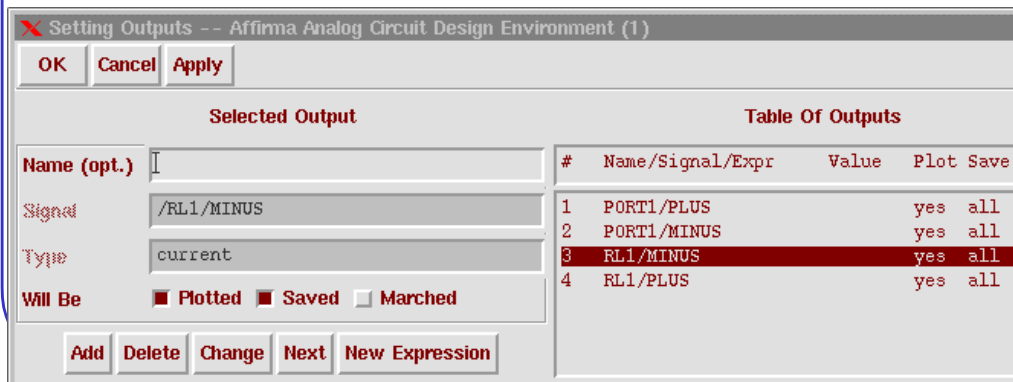
# Display the Conversion Power Gain- method 1(continued)

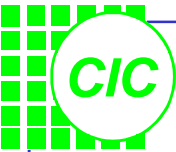
- Click the **Crosshair Marker A** icon and place the marker on the 2.4GHz harmonic of Prf.
- Click the **Crosshair Marker B** icon and place the marker on the 100MHz harmonic of Pif.
- Prf:
  - Magnitude: 4.0085m
  - Power:  $\cong -38$  dBm
- Pif:
  - Magnitude: 4.08038m
  - Power:  $\cong -37.8$  dBm
- Conversion Power Gain  $\cong 0.2\text{dB} + 3 \text{ dB} = 3.2 \text{ dB}$



# Display the Conversion Power Gain-method 2

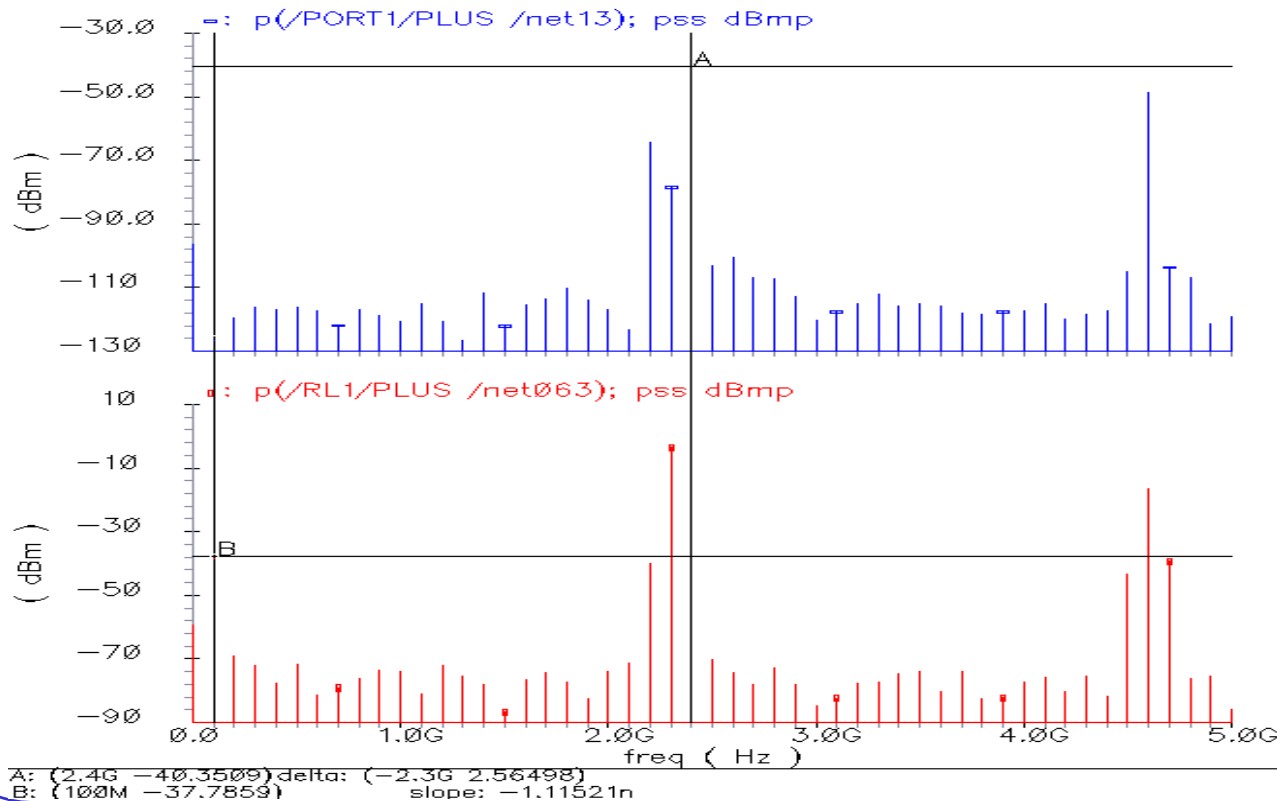
- Select **Output** → **Save All ...** and the window “*Save Options*” appears. Set the buttons as below window in order to get the **AC** power!
- Select **Outputs** → **To Be Saved** → **Select On Schematic**. In the schematic, select the **PORT1** and **RL1**. The terminals are circled in the schematic 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.



**PSS Results**

OK Cancel Help

Plot Mode  Append  Replace

Analysis Type

pss

Function

<input type="checkbox"/> Voltage	<input type="checkbox"/> Current
<input checked="" type="checkbox"/> Power	<input type="checkbox"/> Voltage Gain
<input type="checkbox"/> Current Gain	<input type="checkbox"/> Power Gain
<input type="checkbox"/> Transconductance	<input type="checkbox"/> Transimpedance
<input type="checkbox"/> Compression Point	<input type="checkbox"/> IPN Curves
<input type="checkbox"/> Power Contours	<input type="checkbox"/> Beat Frequency
<input type="checkbox"/> Reflection Contours	

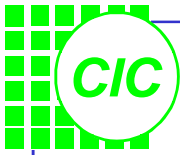
Currently, only spectrum data is available

Modifier

Magnitude  dB10  dBm

Add To Outputs

> Select instance terminal on schematic...



# 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.

Affirma Analog Circuit Design Environment (1)  
Status: Ready T=27

Session Setup Analyses Variables Outputs Simulation Results

Design Edit ... Delete Fund Copy From Cellview Copy To Cellview

Library test  
Cell mixer1  
View schematic

Design Variables			Outputs		
#	Name	Value	#	Name/Signal/Expr	Value
1	prf	-40	1	PORT1/PLUS	
2	frf	2.4G	2	PORT1/MINUS	
3	flo	2.3G	3	RL1/MINUS	
			4	RL1/PLUS	

> Results in /users2/cic/ovid/simulation/mixer1/spectre/schematic

Choosing Analyses -- Affirma Analog Circuit Design Environment

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  xf  
 sens  sp  pdisto  pss  
 pac  pnoise  pxf  envlp

Periodic Steady State Analysis

Fundamental Tones					
#	Name	Expr	Value	Signal	SrcId
2	F1	frf	2.4G	Moderate	PORT1
3	F2	flo	2.3G	Moderate	PORT2

Moderate

Clear/Add Delete Update From Schematic

Beat Frequency   
Beat Period 100M Auto Calculate

Output harmonics  
Number of harmonics 50

Accuracy Defaults (errpreset)  
 conservative  moderate  liberal

Additional Time for Stabilization (tstab)

Save Initial Transient Results (saveinit)  no  yes

Oscillator

Sweep  Frequency Variable?  no  yes  
Variable Variable Name prf  
Select Design Variable

Sweep Range  
 Start-Stop Start -40 Stop 0  
 Center-Span

Sweep Type  
 Linear Step Size  
 Logarithmic Number of Steps

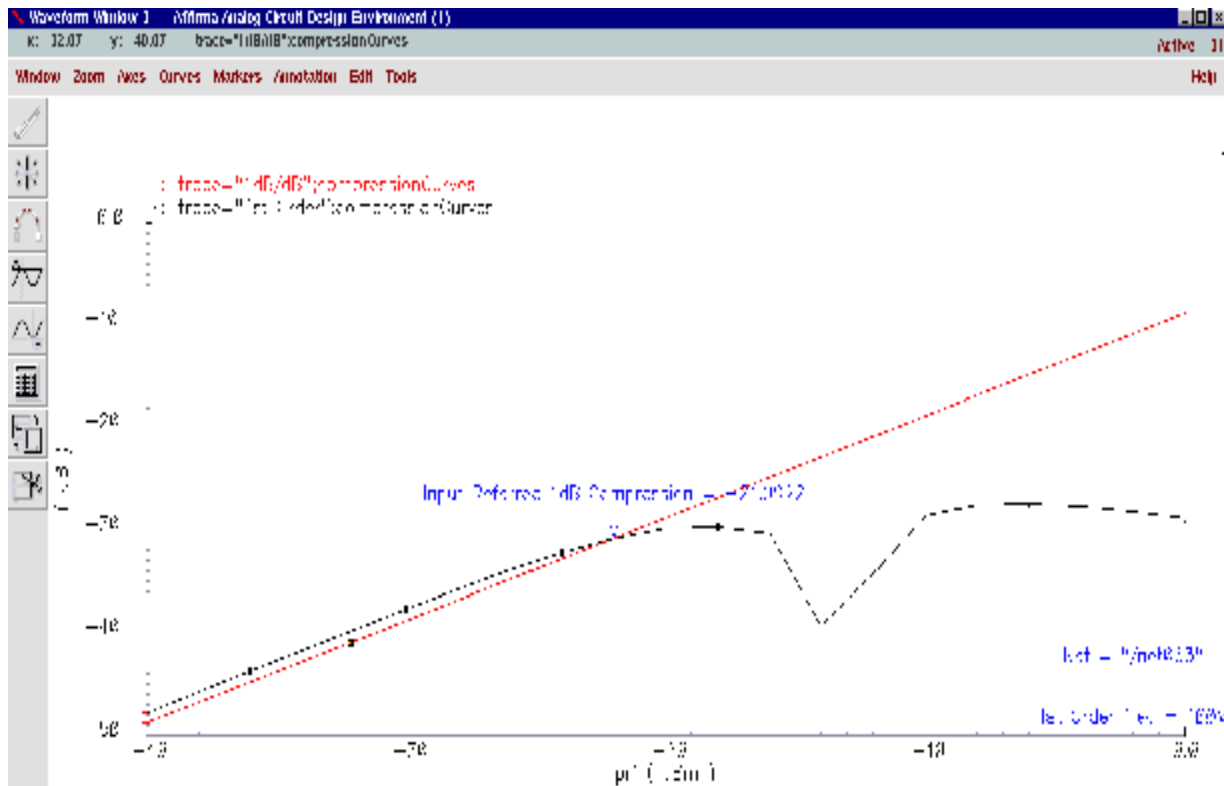
Add Specific Points

Enabled  Options...



# 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.



**PSS Results** [X] [OK] [Cancel] [Help]

Plot Mode  Append  Replace

Analysis Type

pss

Function

- Voltage
- Power
- Current Gain
- Transconductance
- Compression Point
- Power Contours
- Reflection Contours
- Current
- Voltage Gain
- Power Gain
- Transimpedance
- IPN Curves
- Beat Frequency

Gain Compression (dB)

"prf" ranges from -40 to 0

Extrapolation Point (dBm)

(Defaults to -40)

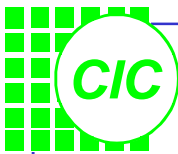
Input Referred 1dB Compression

1st Order Harmonic

0	0
1	100M
2	200M
3	300M
4	400M

Add To Outputs  [Replot]

> Select port or net on schematic ...



# 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

**Choosing Analyses -- Affirma Analog Circuit Design**

OK Cancel Defaults Apply

Analysis:  tran  dc  ac  noise  pss  envl  
 sens  sp  pdisto  
 pac  pnoise  pxf

**Periodic Steady State Analysis**

**Fundamental Tones**

#	Name	Expr	Value	Signal
2	F1	frf	2.4G	Moderate
3	F2	flo	2.3G	Moderate
1	F3	frf+25M	2.425G	Moderate

Clear/Add Delete Update From Schem

Beat Frequency:  Beat Period: 25M Auto C

Output harmonics: Number of harmonics: 50

Accuracy Defaults (empreset):  conservative  moderate  liberal

Additional Time for Stabilization (tstab):

Save Initial Transient Results (saveinit):  no  yes

Oscillator:

Sweep:  Variable Frequency Variable?:  no  yes Variable Name: prf1 Select Design Variable

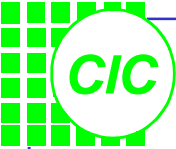
Sweep Range:  Start-Stop Center-Span Start: -40 Stop: 0

Sweep Type:  Linear  Logarithmic Step Size: 10 Number of Steps:

Add Specific Points:

Enabled:  Options...

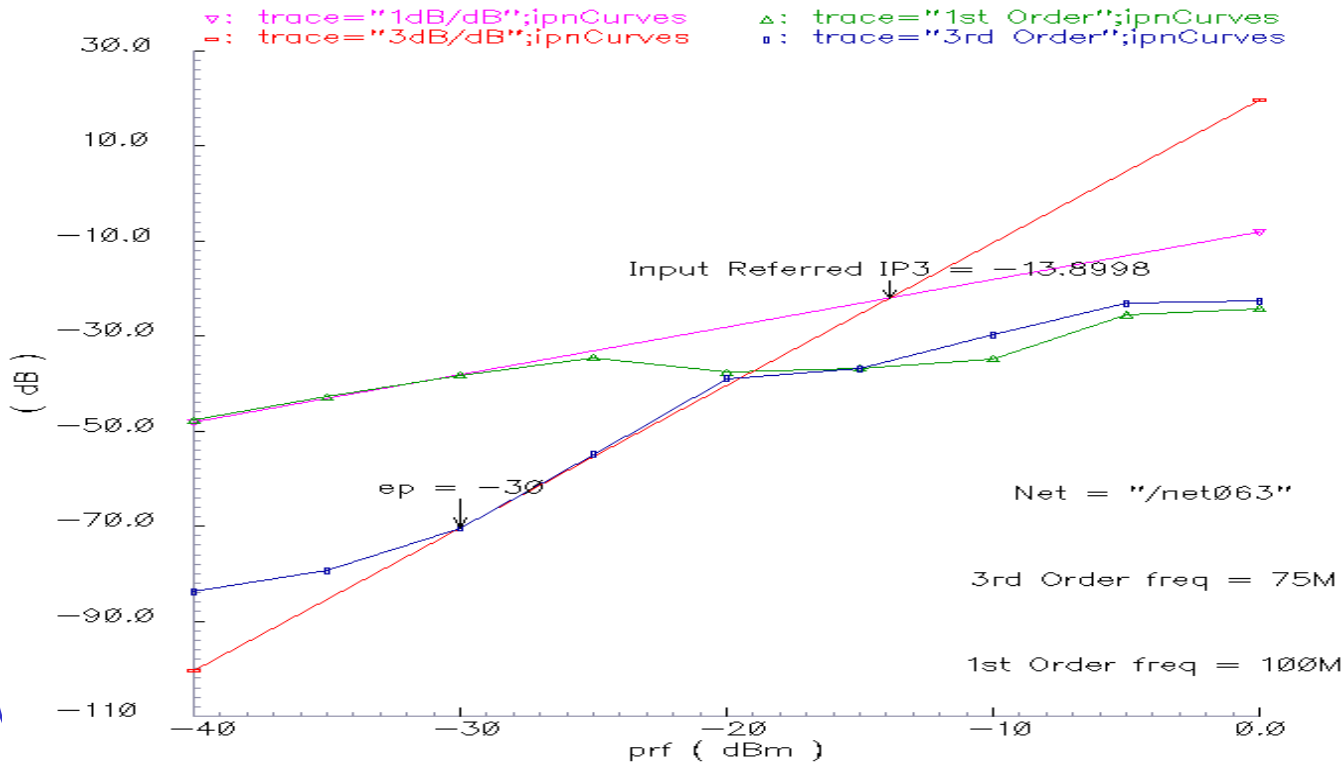
Frequency name: F1  
 Second frequency name: F2  
 Noise file name:   
 Number of noise/freq pairs: 0  
 Resistance: 50 Ohms  
 Port number:   
 DC voltage:   
 Source type: sine  
 Delay time:   
 Sine DC level:   
 Amplitude:   
 Amplitude (dBm): prf1  
 Initial phase for Sinusoid:   
 Frequency: frf Hz  
 Amplitude 2:   
 Amplitude 2 (dBm): prf1  
 Initial phase for Sinusoid 2:   
 Frequency 2: frf+25M Hz



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



**PSS Results**

OK Cancel Help

Plot Mode  Append  Replace

Analysis Type

pss

Function

- Voltage
- Power
- Current Gain
- Transconductance
- Compression Point
- Power Contours
- Reflection Contours
- Current
- Voltage Gain
- Power Gain
- IPN Curves
- Beat Frequency

Circuit Input Power  Single Point  Variable Sweep ("prf")

"prf" ranges from -40 to 0

Extrapolation Point (dBm)

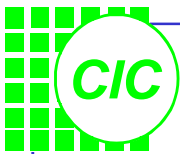
Input Referred IP3  Order

3rd Order Harmonic		1st Order Harmonic	
1	25M	0	0
2	50M	1	25M
3	75M	2	50M
4	100M	3	75M
5	125M	4	100M

Add To Outputs  Replot

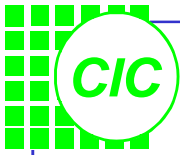
> Select port or net on schematic ...





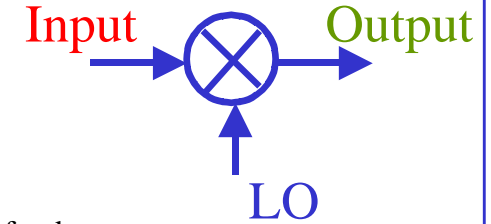
## 5. PAC Analysis

- 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.



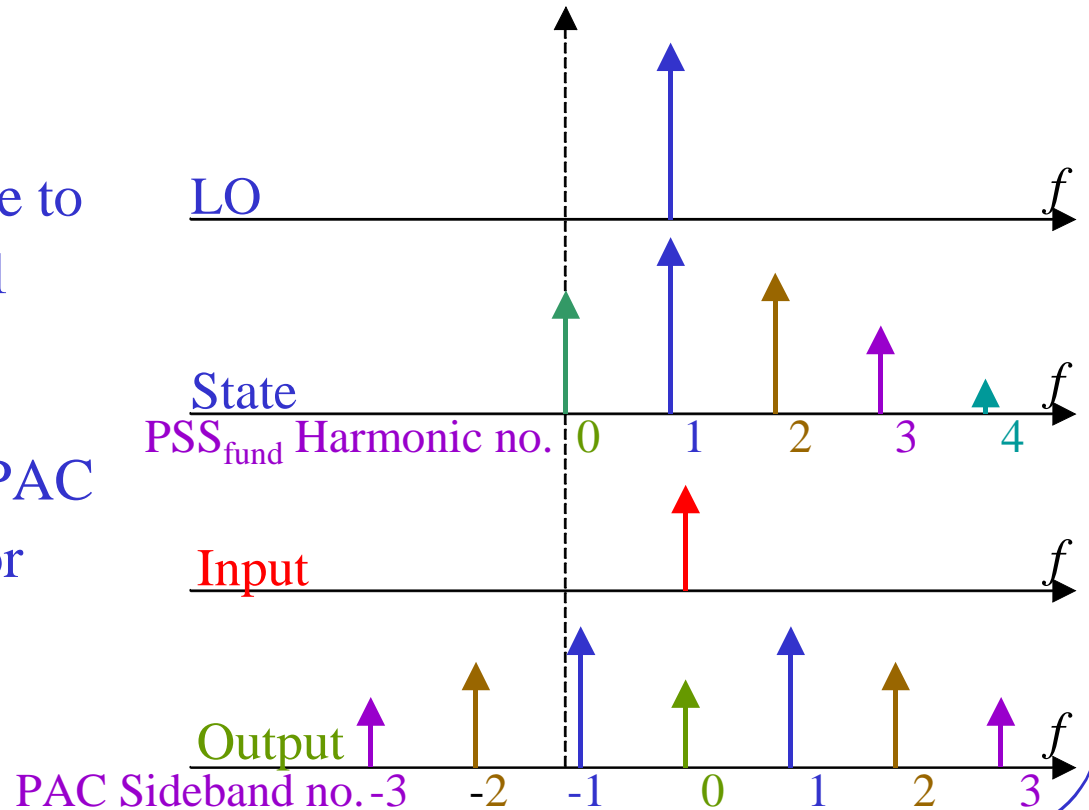
# PAC Analysis Overview

- 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.



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

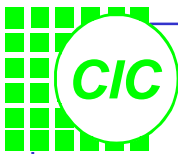
where  $f_{in}$  represents the input frequency, and  $Ki$  are the PAC sidebands





# 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 *maxxfreq* 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.

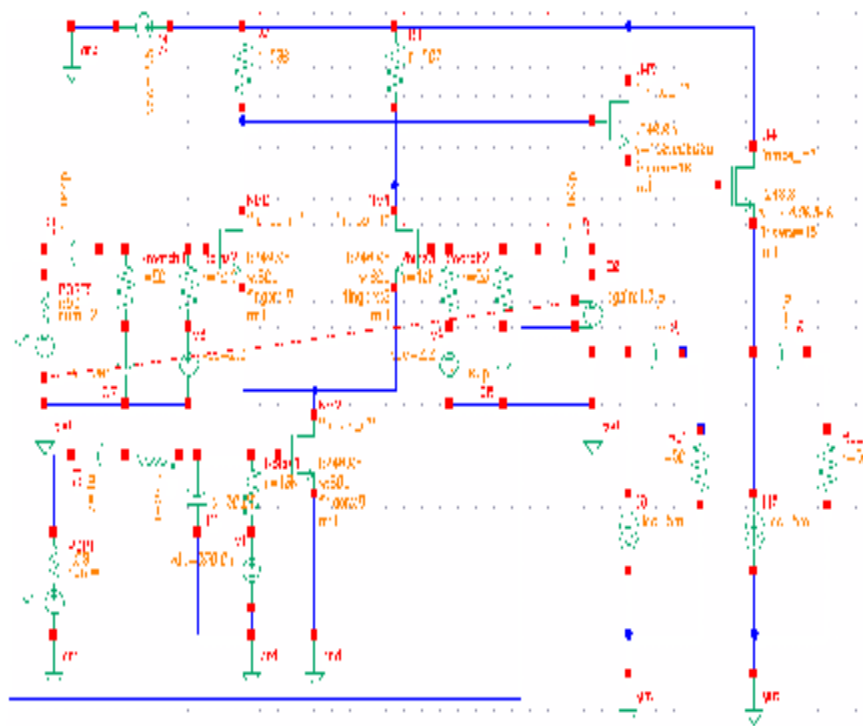
- \* When setting Output harmonics less than the PAC harmonics, be sure to set the *maxacfreq* parameter to assure that the simulator takes sufficient time points to accurately characterize the output waveform in the PSS analysis.

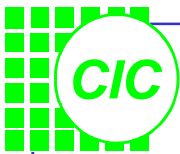
# Lab4 : PAC Analysis

- 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.

#	Name	Expr	Value	Signal	SrcId
3	F2	flo	2.3G	Moderate	PORT2

- 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 3<sup>rd</sup> harmonic of the LO.

- Select **Netlist and Run**.

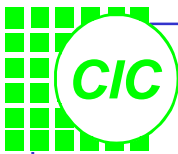
PSS Beat Frequency (Hz) 2.3G

Sweep Type Linear

Frequency Sweep Range (Hz) Start-Stop Start 2.35G Stop 2.45G

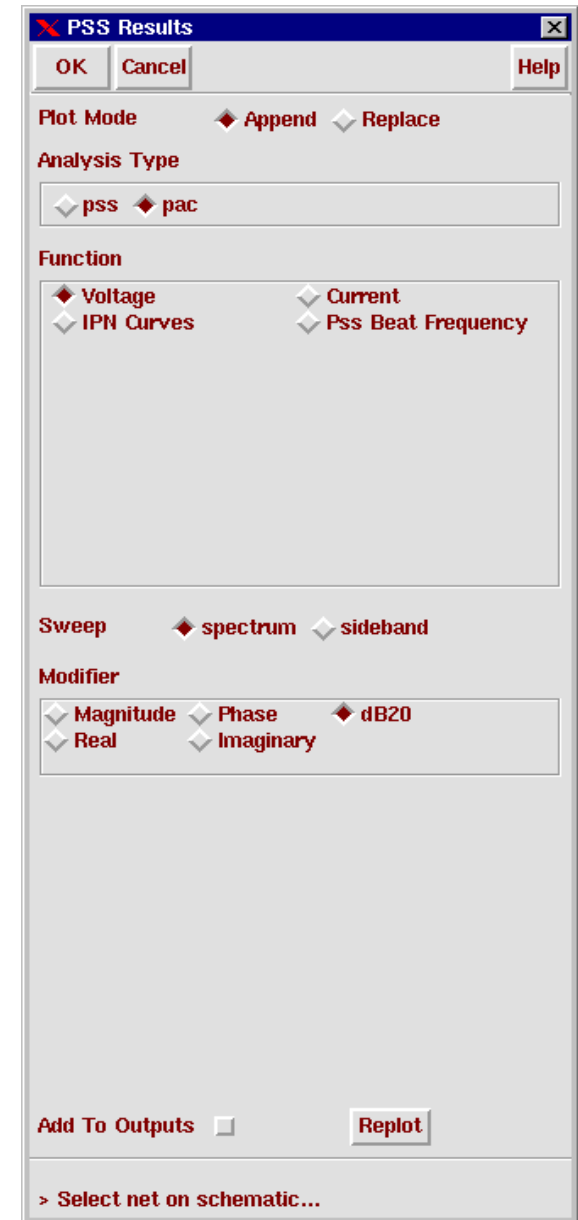
Number of Steps 40

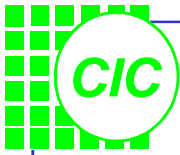
Maximum sideband 0



# 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:

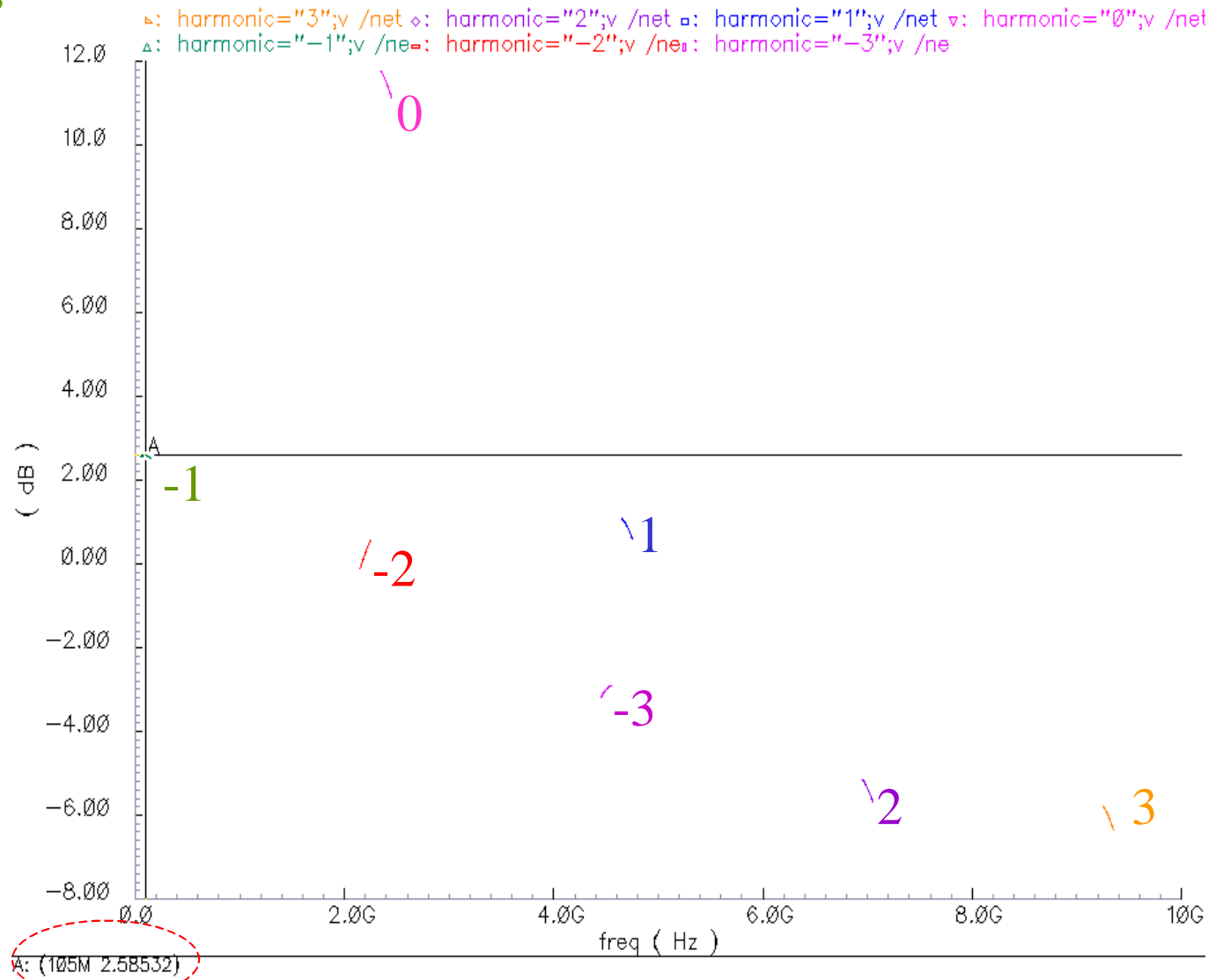




# Periodic Steady State Response

- 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**.



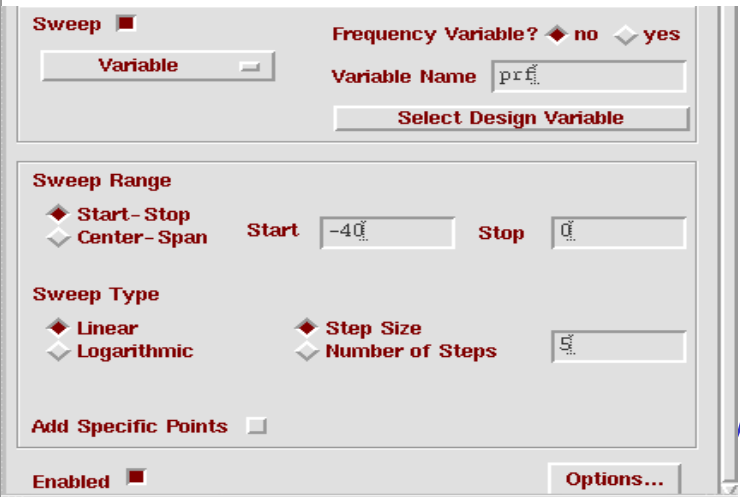
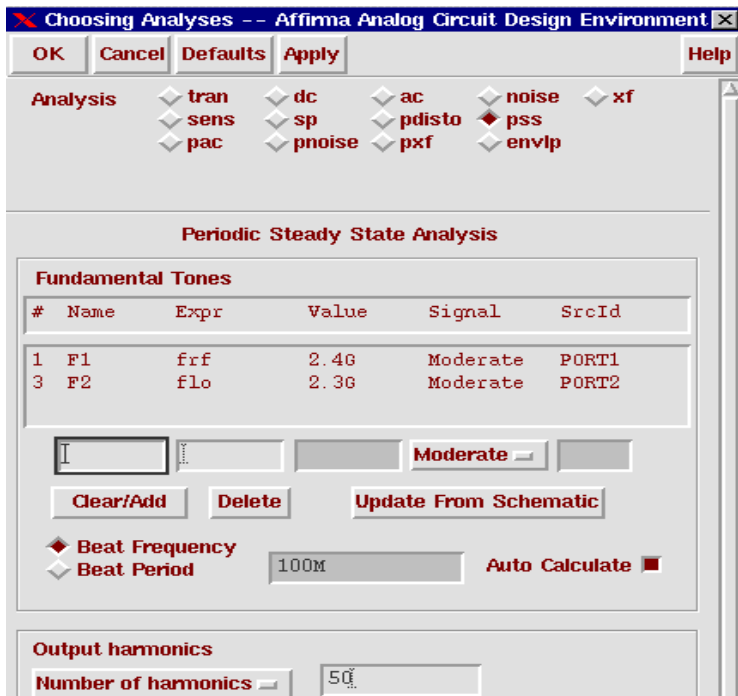




# Running a Swept Small-Signal IP3 Simulation

- 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**.

Choosing Analyses -- Affirma Analog Circuit Design Environment

OK Cancel Defaults Apply Help

Analysis

<input type="checkbox"/> tran	<input type="checkbox"/> dc	<input type="checkbox"/> ac	<input type="checkbox"/> noise	<input type="checkbox"/> xf
<input type="checkbox"/> sens	<input type="checkbox"/> sp	<input type="checkbox"/> pdisto	<input type="checkbox"/> pss	
<input checked="" type="checkbox"/> pac	<input type="checkbox"/> pnoise	<input type="checkbox"/> pxf	<input type="checkbox"/> envlp	

Periodic AC Analysis

PSS Beat Frequency (Hz) 100M

Sweeptype

Frequency Sweep Range (Hz)

Single-Point [] Freq 2.425G

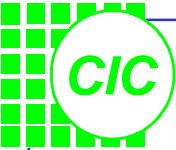
Because the sweep section of the PSS analysis is active, only a single point for this analysis is currently supported.

Sidebands

Maximum sideband 5G

Enabled

Options...



# IP3 Results

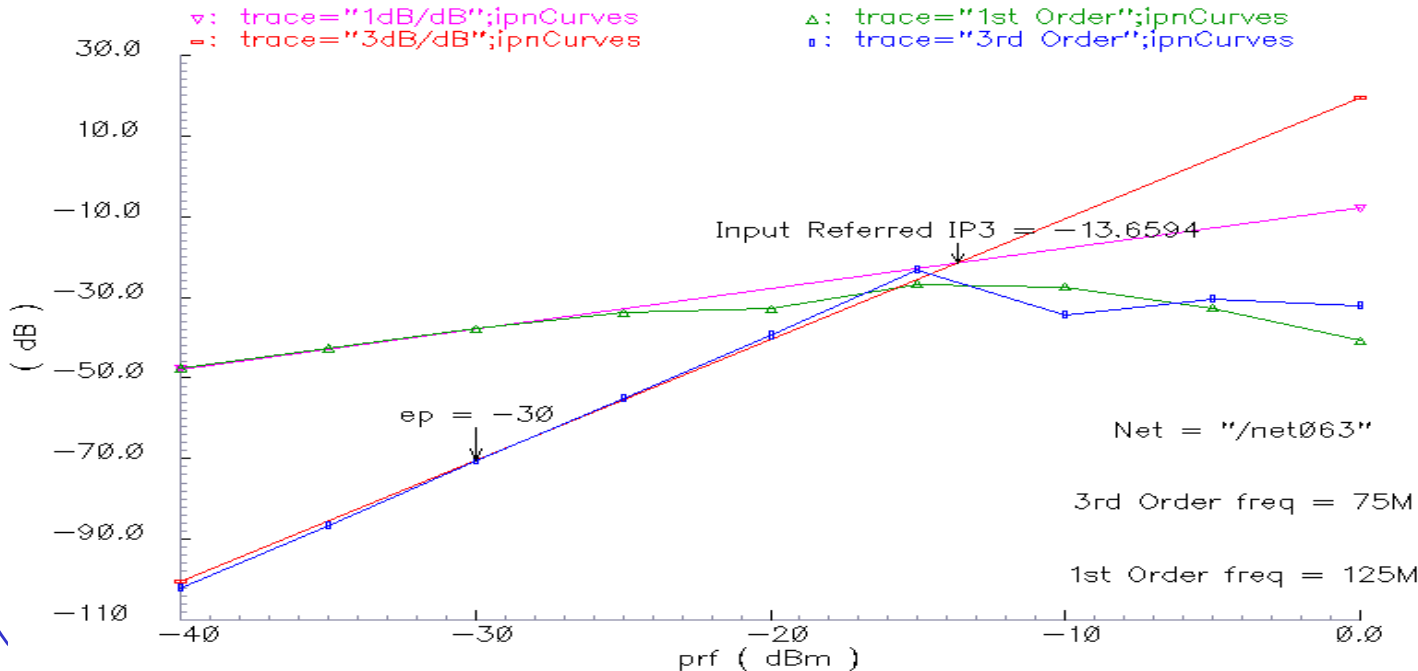
LO: 2.3 G

RF: 2.4 G & 2.425G

1<sup>st</sup> order harmonics: 100M & 125M

3<sup>rd</sup> order harmonics: 75M & 150M

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



**PSS Results**

OK Cancel Help

Plot Mode  Append  Replace

Analysis Type

pss  pac

Function

Voltage  Current

IPN Curves  Pss Beat Frequency

Circuit Input Power  Single Point  Variable Sweep ("prf")

"prf" ranges from -40 to 0

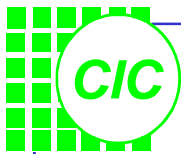
Extrapolation Point (dBm)

Input Referred IP3  Order

3rd Order sideband		1st Order sideband	
-26	175M	-27	275M
<b>-25</b>	<b>75M</b>	-26	175M
-24	25M	-25	75M
-23	125M	-24	25M
-22	225M	<b>-23</b>	<b>125M</b>

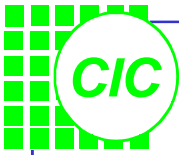
Add To Outputs  Replot

> Select port or net on schematic ...



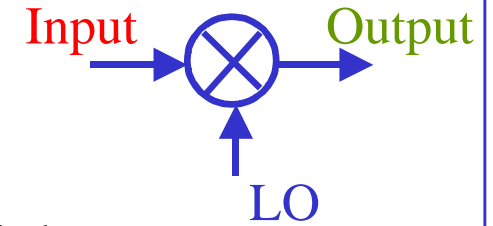
## 6. PXF Analysis

- 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.



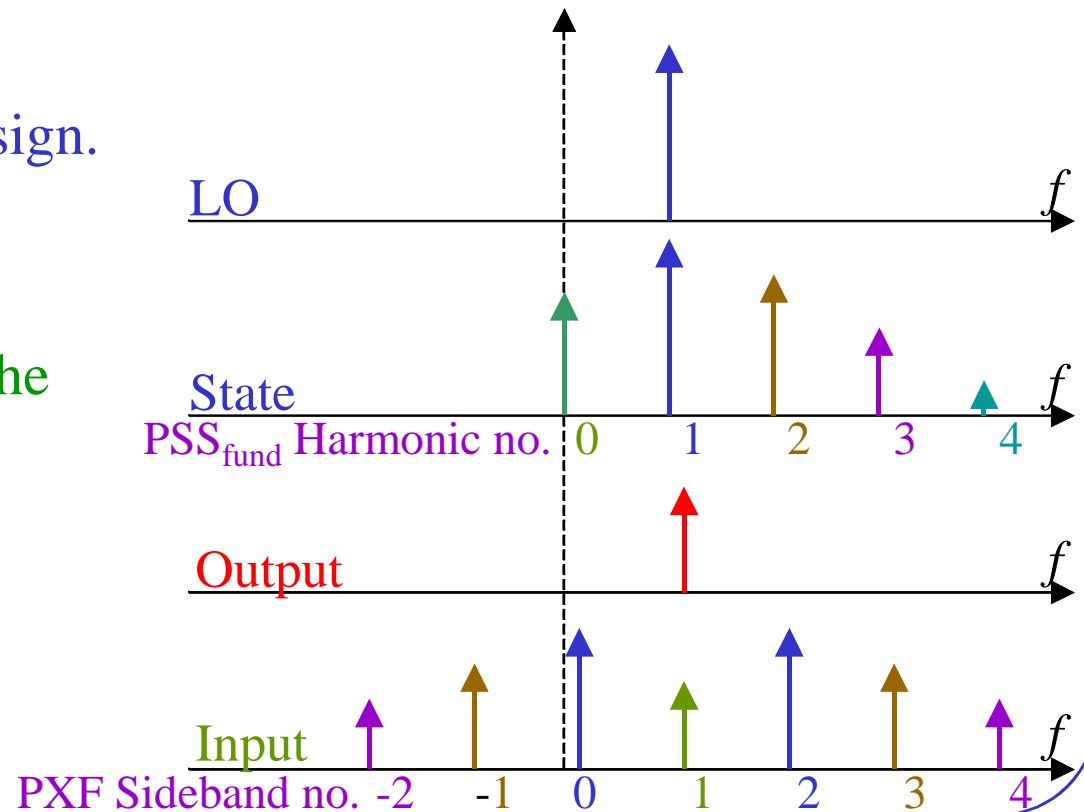
# PXF Analysis Overview

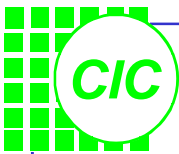
- The **PXF** analysis computes the energy contributions from all source harmonic frequencies to a signal or swept output frequency. In this way, a single output response is the combination of all possible frequency components in the design.
- Set the *maxsideband*, or the sidebands parameters, to select the periodic small-signal input frequencies of interest, while sweeping the selected output frequency.



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

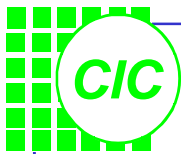
where  $f_{out}$  represents the output signal frequency;  $k$  is the PXF sidebands number





# 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.

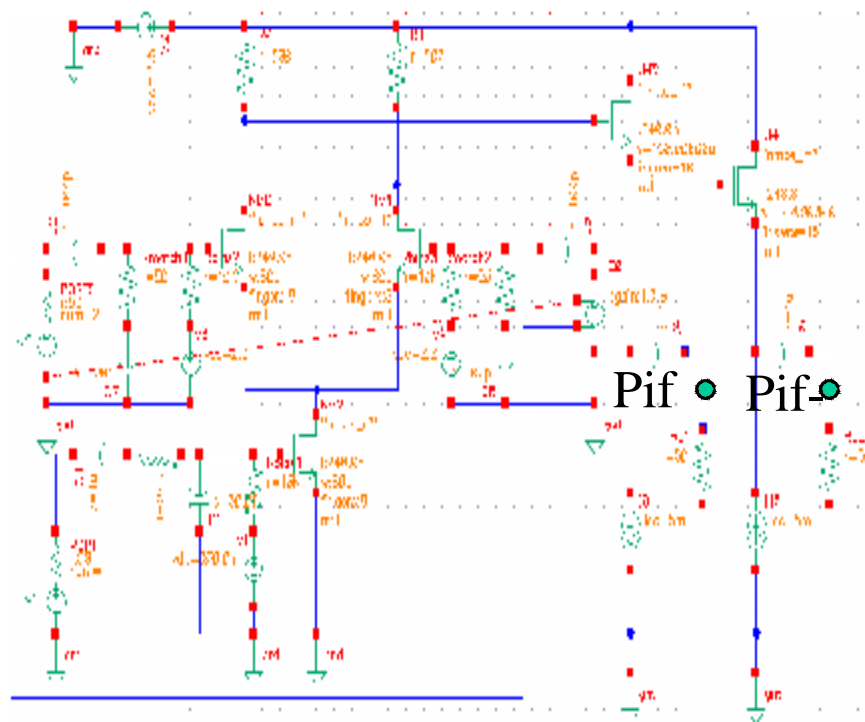
PSS fundamental	The number of harmonics should be no less than the PXF harmonics. *
Output net (v) or Voltage source (i)	Specify in form (To measure current, put a 0v battery in series with the branch. )
Output sweep frequency	Sweep, array or single point
Input frequencies of interest	Sidebands
Results 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.

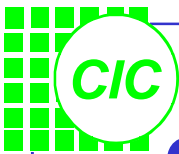
# Lab5 : PXF Analysis

- 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.

Choosing Analyses -- Affirma Analog Circuit Design Environment

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  xf  
 sens  sp  pdisto  pss  
 pac  pnoise  pxf  envlp

Periodic Steady State Analysis

Fundamental Tones

#	Name	Expr	Value	Signal	SrcId
3	F2	flo	2.36	Moderate	PORT2

Clear/Add Delete Update From Schematic

Beat Frequency  
 Beat Period 2.36 Auto Calculate

Output harmonics  
Number of harmonics 0

Accuracy Defaults (errpreset)  
 conservative  moderate  liberal

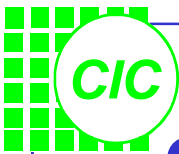
Additional Time for Stabilization (tstab)

Save Initial Transient Results (saveinit)  no  yes

Oscillator

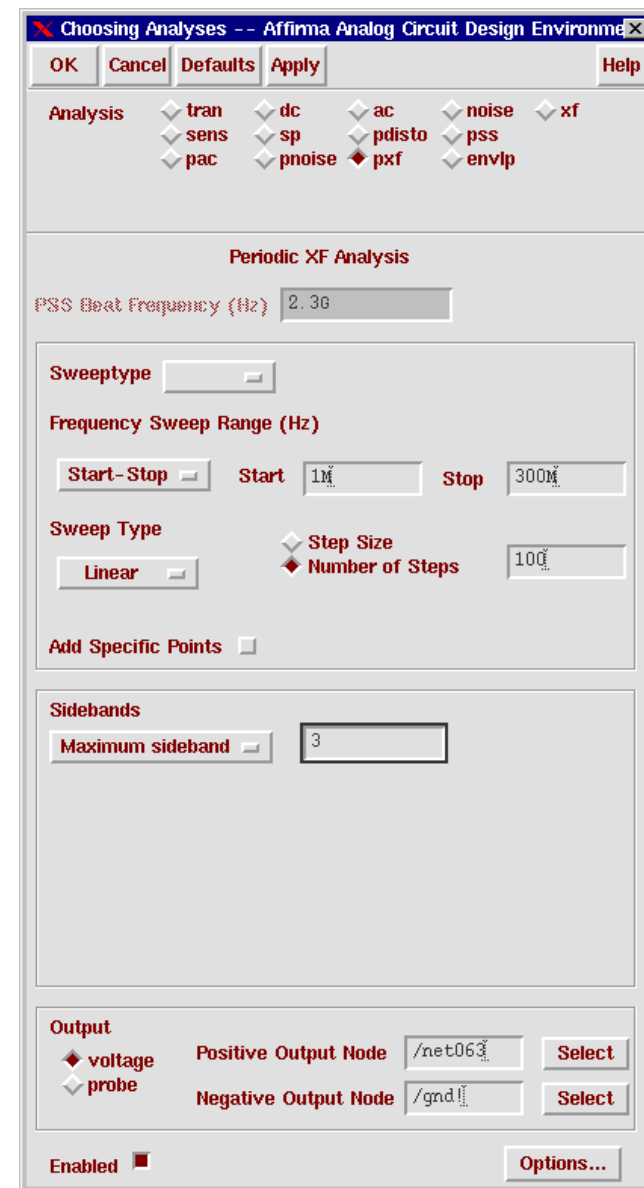
Sweep

Enabled  Options...



# Setting Up the PXF Simulation(2)

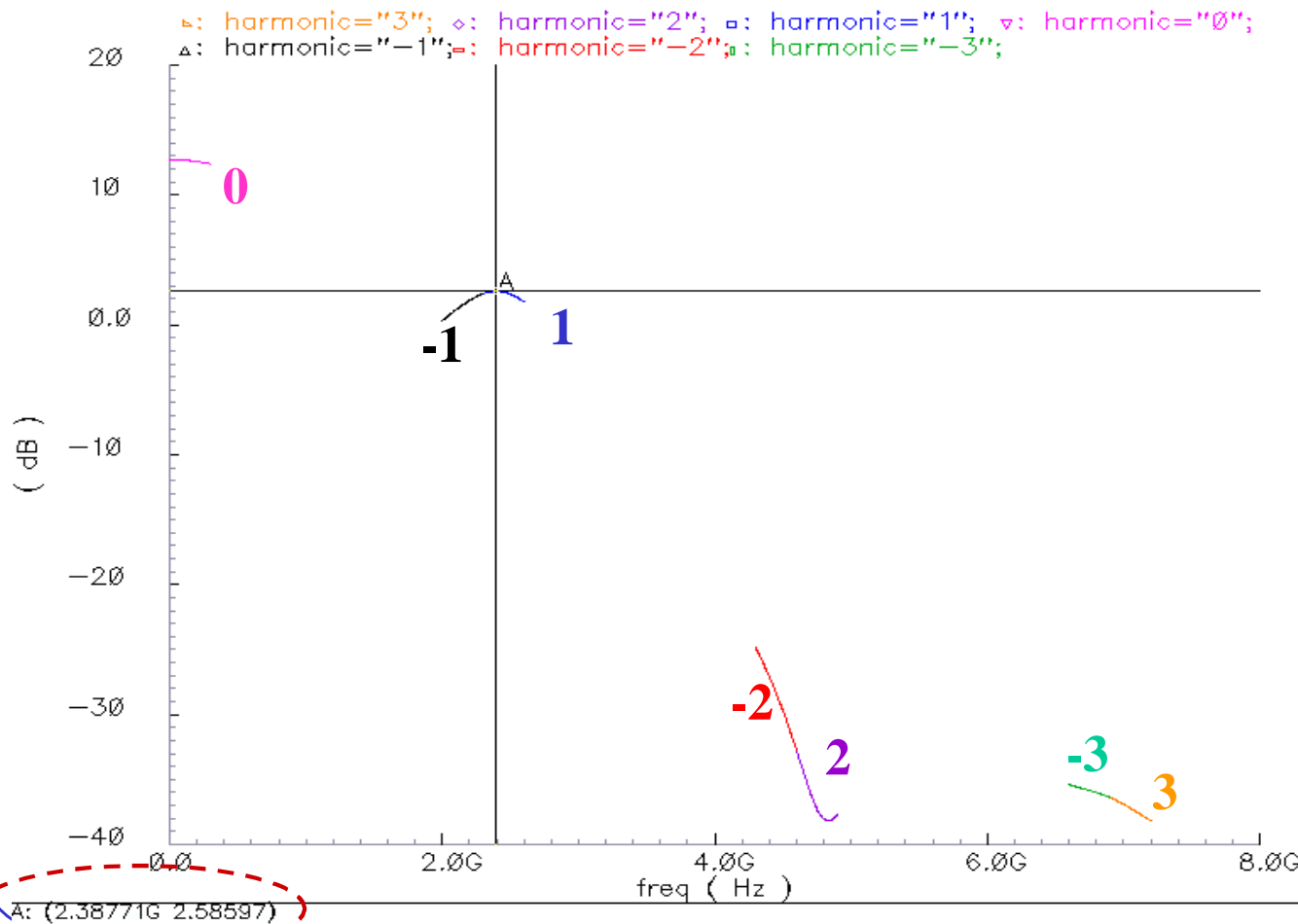
- Click on **pxf** in the *Choosing Analyses* form, and setup the form as left; then click **OK** .
- The *Frequency Sweep Range* is specified from 1MHz to 300 MHz. The **PXF** analysis will calculate all inputs that produce this range of frequencies at the **Pif** port.
- To set the *Positive Output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



**PSS Results**

OK Cancel Help

Plot Mode  Append  Replace

Analysis Type

pss  pxf

Function

Voltage Gain  Transimpedance

Pss Beat Frequency

Sweep  spectrum  sideband

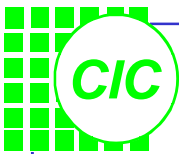
Modifier

Magnitude  Phase  dB20

Real  Imaginary

Add To Outputs  Replot

> Select port or voltage source on 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.

Output

◆ voltage Positive Output Node /net063 Select

◇ probe Negative Output Node /net061 Select

- Run the simulation.

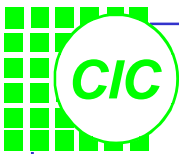
Affirma Analog Circuit Design Environment (1)  
Status: Ready T=27 C Simulator: spectre 4

Session Setup Analyses Variables Outputs Simulation Results Tools Help

Design		Analyses			
Library	test	#	Type	Arguments.....	Enable
Cell	mixer1	1	pxf	3 1M 300M 100 ..	yes
View	schematic	2	pac	50 2.4016	no
		3	pss	2.3G 0	yes

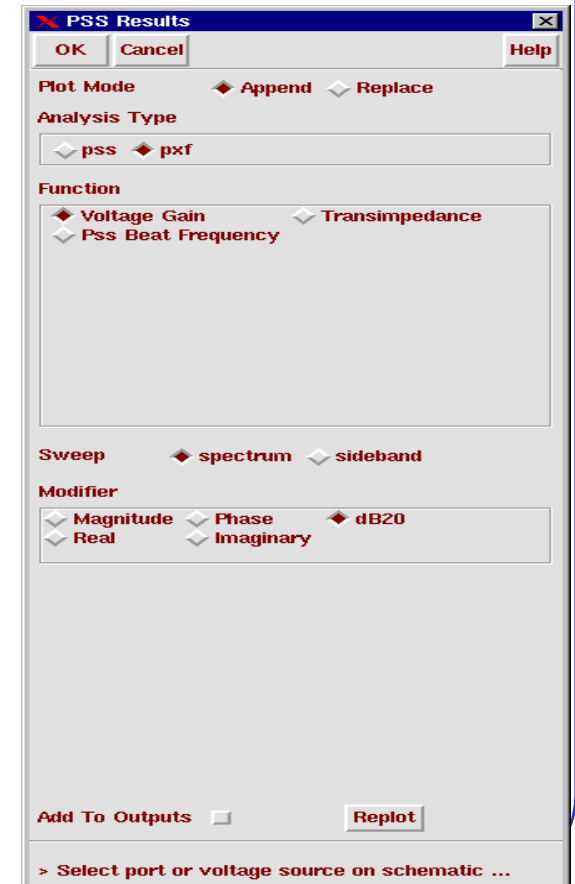
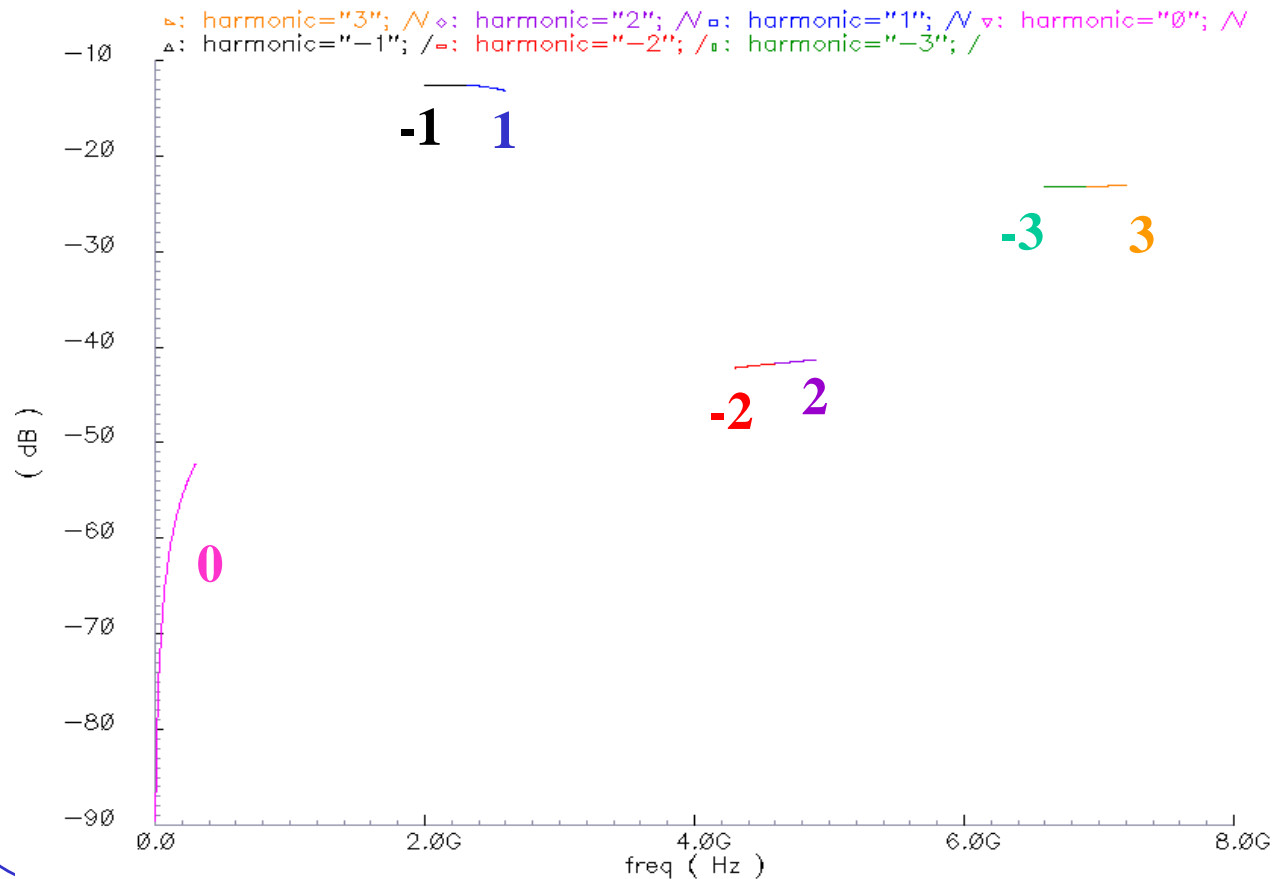
Design Variables			Outputs			
#	Name	Value	#	Name/Signal/Expr	Value	Plot Save March
1	prf	-40	1	PORT1/PLUS		yes all no
2	frf	2.4G	2	PORT1/MINUS		yes all no
3	flo	2.3G	3	RL1/MINUS		yes all no
			4	RL1/PLUS		yes all no

> Results in /users2/cic/ovid/simulation/mixer1/spectre/schematic



# Plotting the Power Supply Rejection

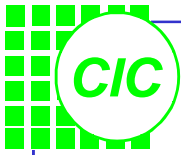
- Use **Direct Plot** function to see the results. In the PSS Results form, select pxf button. Follow the prompts at the bottom of the form, and select the DC supply (vdc=2.5v) in the schematic





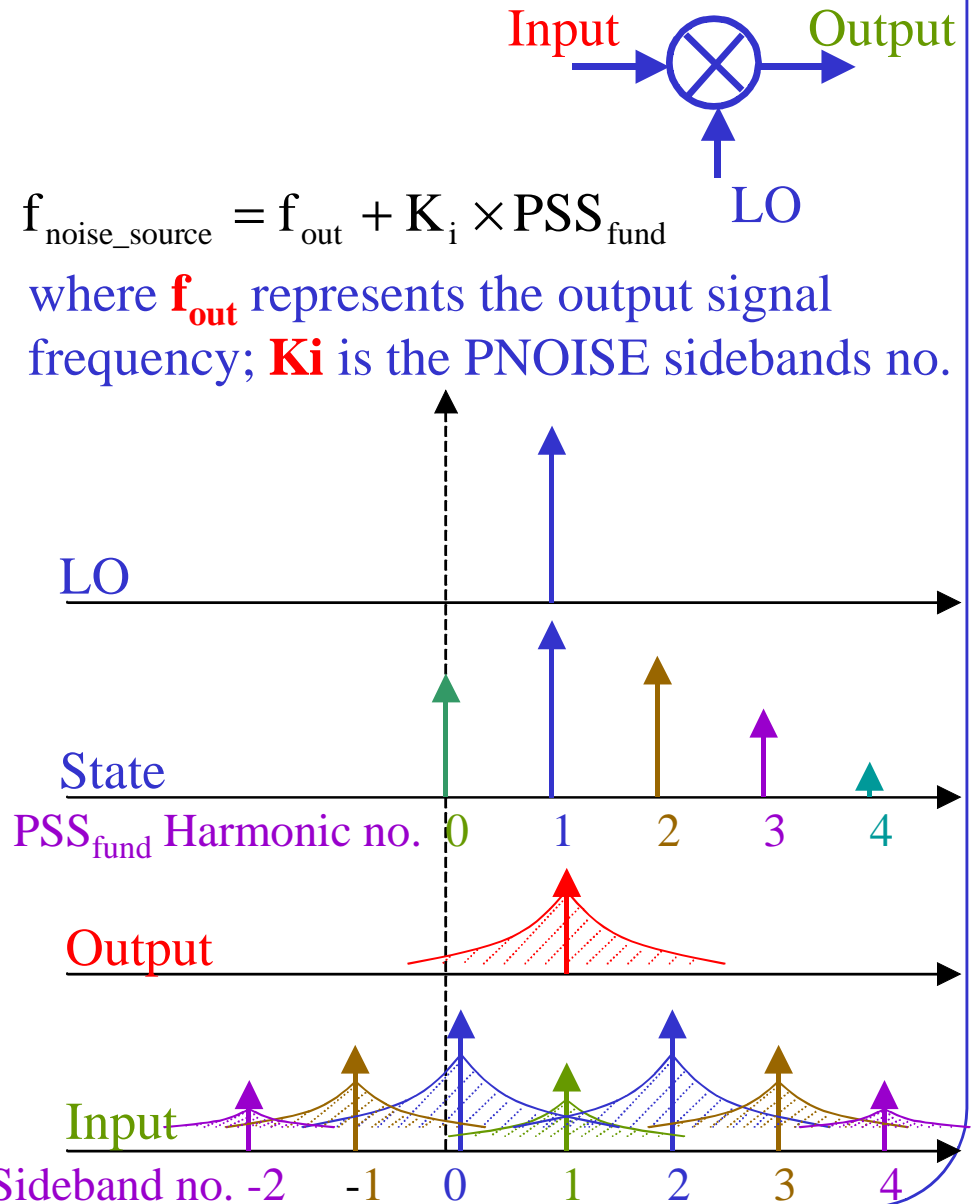
# 7. PNOISE Analysis

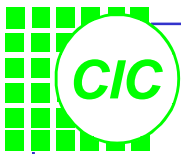
- PNOISE analysis, unlike conventional noise analysis, computes **frequency convention effects**, noise folding, **aliasing**.
- For noise sources that are bias dependent, such as **shot noise sources**, the time-varying operating point acts to modulate the noise sources. The transfer function from the 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.
- 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.

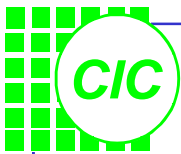




## PNOISE Analysis Overview(2)

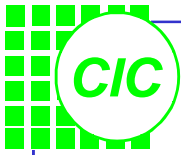
- When the reference sideband has any value other **0**, Single Sideband (**SSB**) NF is calculated. To determine the reference sideband, run a **PXF** analysis.
- The *Noise Summary Table* displays the following data:
  - Noise contribution (value and %) for each component in the circuit
  - Total output 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

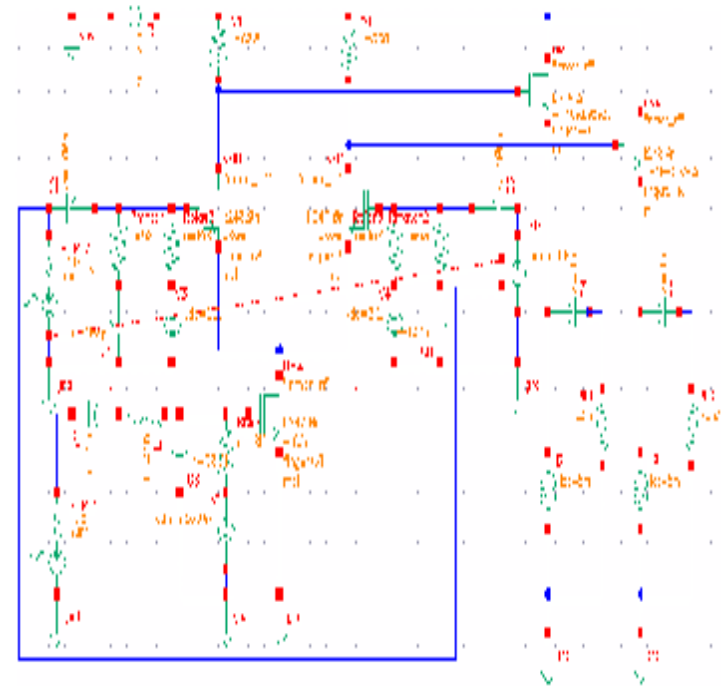
- Specify the information in this table when running a PNOISE analysis.

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*.

INTEGRATION METHOD PARAMETERS

method  euler  trap  traonly  
 gear2  gear2only

---

ACCURACY PARAMETERS

relref  pointlocal  alllocal  sigglobal  allglobal

iteratio

steadyratio

maxacfreq

Choosing Analyses -- Affirma Analog Circuit Design Environment

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  xf  
 sens  sp  pdisto  pss  
 pac  pnoise  pxf  envlp

Periodic Steady State Analysis

Fundamental Tones

#	Name	Expr	Value	Signal	SrcId
3	F2	flo	2.3G	Moderate	PORT2

Moderate

Clear/Add Delete Update From Schematic

Beat Frequency  Beat Period  Auto Calculate

Output harmonics

Number of harmonics

Accuracy Defaults (errpreset)  
 conservative  moderate  liberal

Additional Time for Stabilization (tstab)

Save Initial Transient Results (saveinit)  no  yes

Oscillator

Sweep

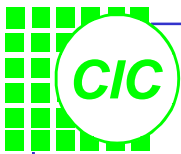
Enabled  Options...



# Setting Up the PNOISE Simulation(2)

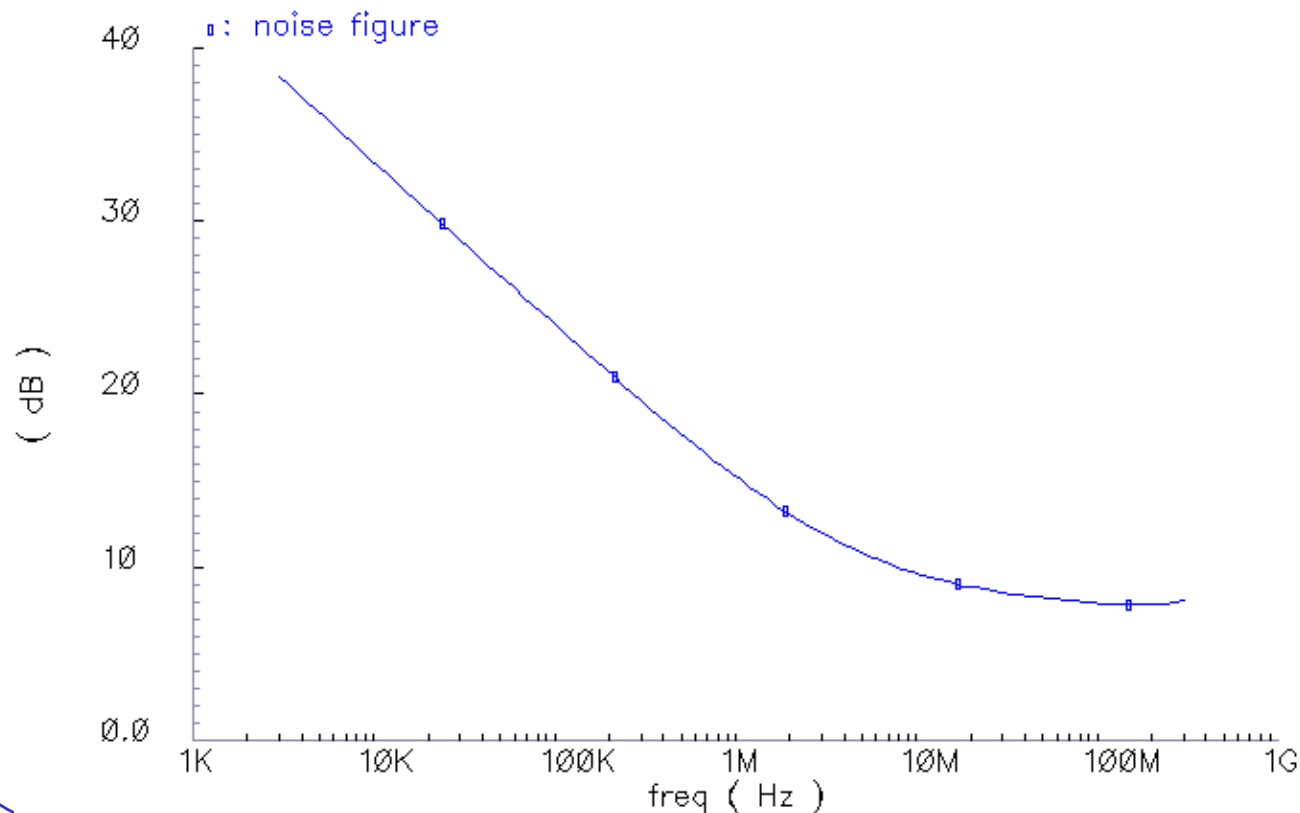
- Click on **pnoise** in the *Choosing analyses form*, and set up the form as right:
- A **Maximum sideband** of 8 implies PNOISE will calculate the noise out to 8 harmonics of the PSS<sub>fund</sub>, or 18.4 GHz.
- To set the **Positive/Negative Output Node**, click the **Select** button, and select 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**.

The screenshot shows the 'Choosing Analyses' dialog box for the Affirma Analog Circuit Design Environment. The 'Analysis' section has 'pnoise' selected. The 'Periodic Noise Analysis' section is configured with 'PSS Beat Frequency (Hz)' at 2.3G, 'Sweeptype' as a dropdown, 'Frequency Sweep Range (Hz)' from 3K to 300M, 'Sweep Type' as Logarithmic, 'Points Per Decade' at 20, and 'Number of Steps' at 20. The 'Sidebands' section has 'Maximum sideband' set to 8. The 'Output' section has 'voltage' selected, with 'Positive Output Node' and 'Negative Output Node' both set to '/net06'. The 'Input Source' section has 'port' selected, with 'Input Port Source' set to '/PORT1'. The 'Reference Side-Band' is set to 1. The 'Noise Type' section has 'sources' selected. The 'Enabled' checkbox is checked, and there is an 'Options...' button.



# 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.



**PSS Results**

OK Cancel Help

Plot Mode  Append  Replace

Analysis Type

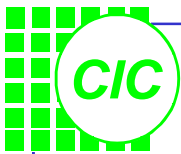
pss  pnoise

Function

Output Noise  Input Noise  
 Noise Figure  Noise Factor  
 Transfer Function  Phase Noise  
 Pss Beat Frequency

Currently, only frequency data is available

Add To Outputs  Plot

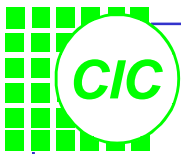


# Printing the Noise summary Report

- 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 screenshot shows the 'Noise Summary' dialog box with the following settings:

- Buttons:** OK, Cancel, Defaults, Apply, Help
- Type:**  spot noise,  integrated noise
- noise unit:** V<sup>2</sup>
- Frequency Spot (Hz):** 1K
- From (Hz):** 3K, **To (Hz):** 300M
- weighting:**  flat,  from weight file
- FILTER:**
  - include:** All Types, None
  - Component list:** diode, port, resistor
  - include instances:** [text box], Select, Clear
  - exclude instances:** [text box], Select, Clear
- TRUNCATE & SORT:**
  - truncate:**  none,  by number,  by rel. threshold,  by abs. threshold
  - top:** 3, **noise %:** 50, **noise value:** 0.0
  - sort by:**  noise contributors,  composite noise,  device name



# The Noise Summary Table

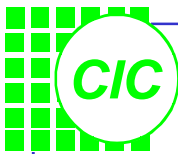
- Click **OK** in the *Noise Summary form*, and the *Noise Summary Table* displays.
- Note what are the main contributions of noise.

The screenshot shows a 'Results Display Window' with a table of noise contributions. The table has four columns: 'Device', 'Param', 'Noise Contribution', and '% Of Total'. The data is sorted by noise contribution in descending order. The top contributor is '/PORT1' with a noise contribution of 7.90045e-10, accounting for 25.76% of the total. Other significant contributors include 'NM2.rg' (7.06%), 'NM0.mcore' (6.98%), 'NM1.mcore' (6.96%), '/R1' (6.88%), and '/R0' (6.77%).

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

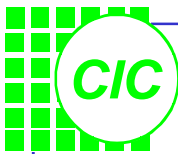
Integrated Noise Summary (in V<sup>2</sup>) Sorted By Noise Contributors  
Total Output Noise = 3.06722e-09  
Total Input Referred Noise = 4.38152e-10





# 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 not 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:

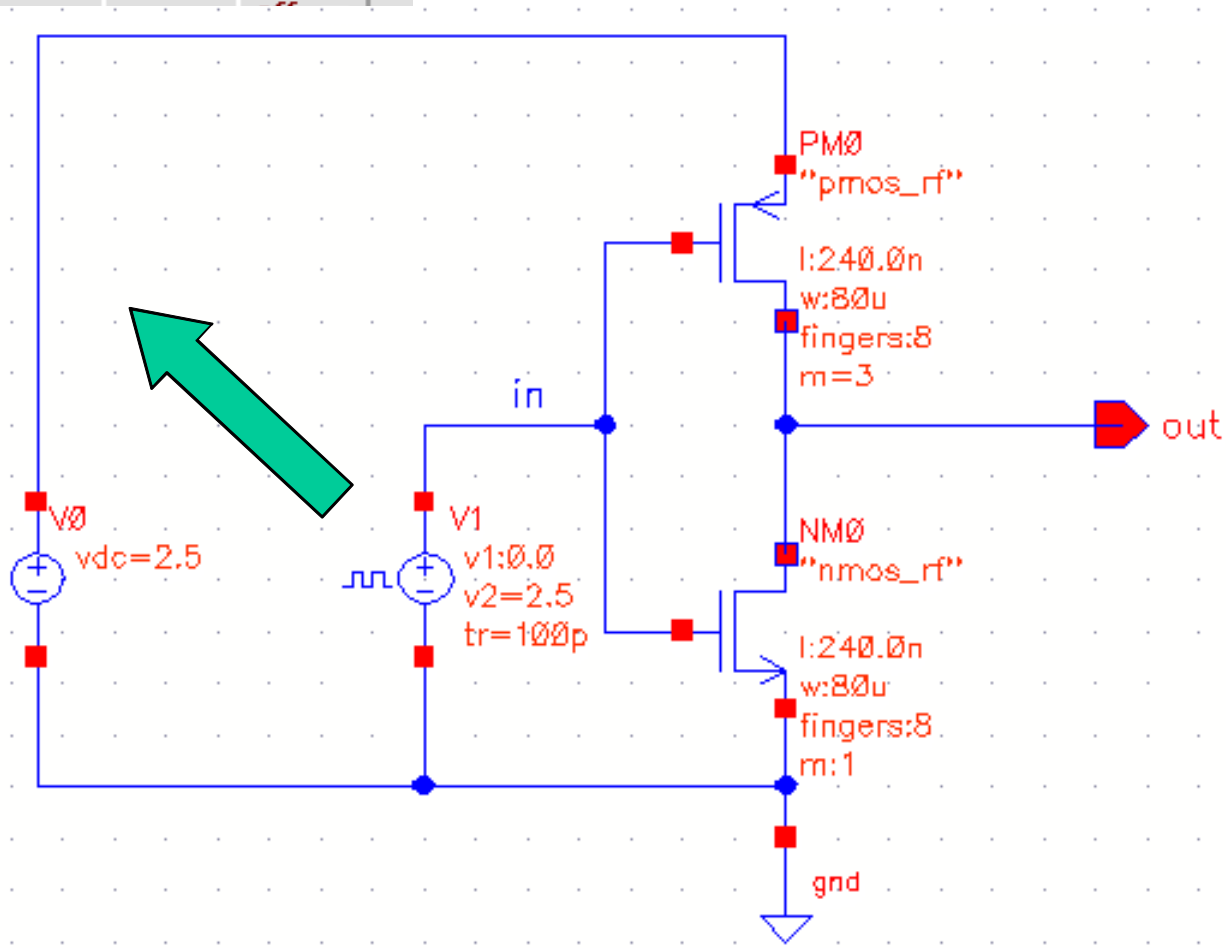
- Calculate correlations in noise at different ports of a multiport circuit



# Lab7: Calculating Time-Varying Instantaneous Noise Power

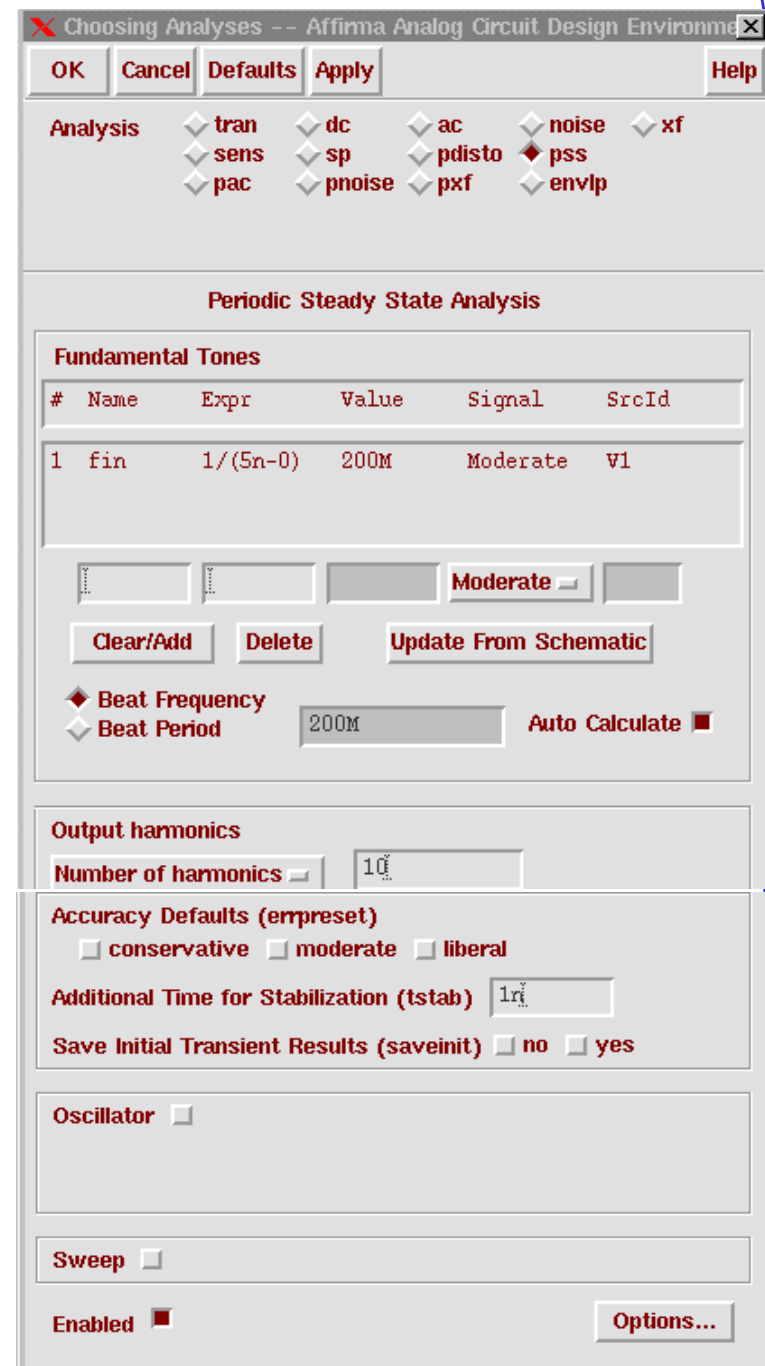
Voltage 1	0.0 V	off
Voltage 2	2.5 V	off
Delay time		
Rise time	100p	
Fall time		
Pulse width		
Period	5n	
Frequency name for 1/period	fir	

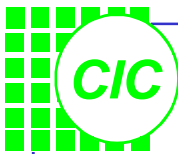
- Create a new schematic view.
- Use library “**analogLib**” & “**tsmc25rf**” to draw the scheme.
- After drawing, **Check and Save!**



# 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.



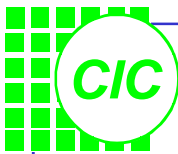


# Setting Up the PNOISE Simulation(2)

- 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.

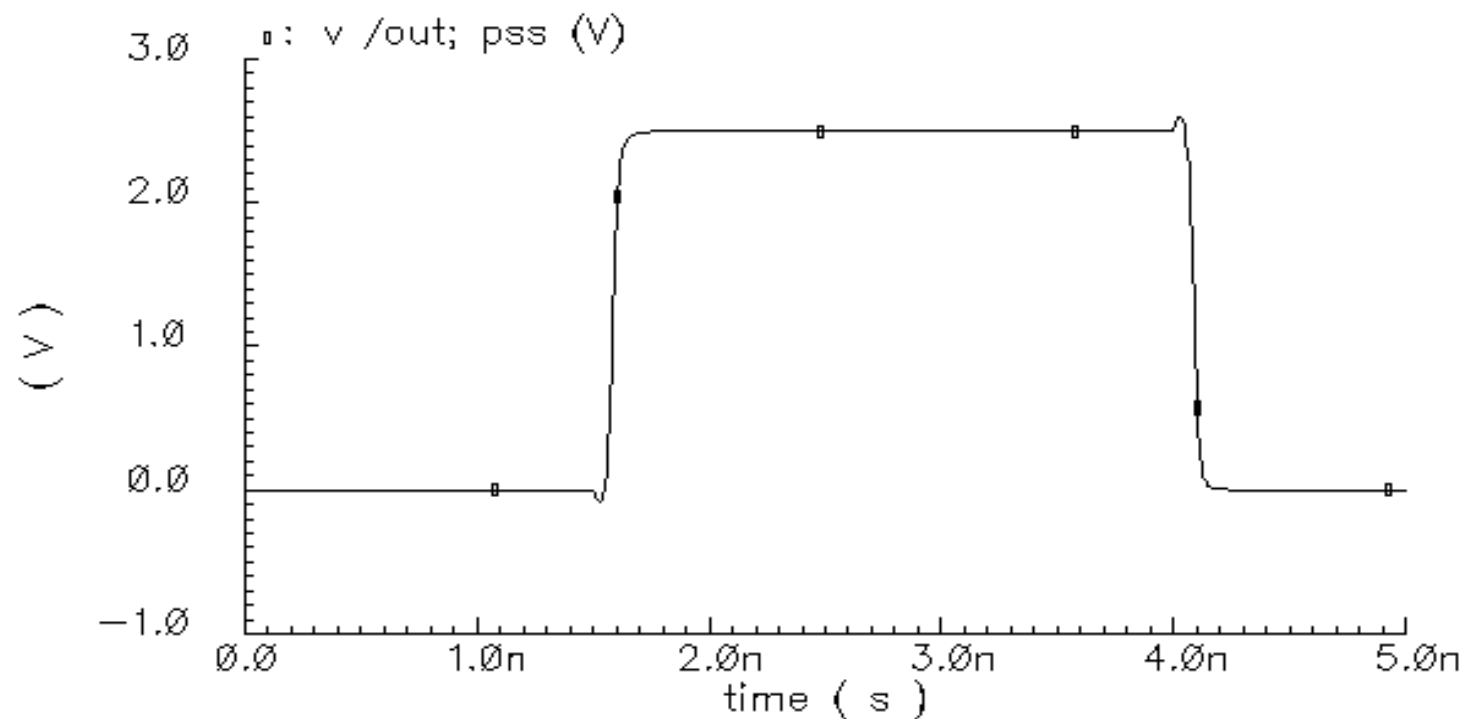
The screenshot shows the PNOISE analysis configuration window. At the top, there is a list of analysis types: tran, dc, ac, noise, xf, sens, sp, pdisto, pss, pac, pnoise, pxf, and envlp. The 'noise' and 'pnoise' options are selected. Below this is the 'Periodic Noise Analysis' section. The 'PSS Beat Frequency (Hz)' is set to 200M. The 'Sweeptype' is set to a dropdown menu. The 'Frequency Sweep Range (Hz)' is set to 'Start-Stop' with 'Start' at 2 and 'Stop' at 200M. The 'Sweep Type' is set to 'Logarithmic'. The 'Points Per Decade' is set to a dropdown menu, and the 'Number of Steps' is set to 100. There is an 'Add Specific Points' checkbox. Below this is the 'Sidebands' section with 'Maximum sideband' set to 10. The 'Output' section has 'voltage' selected for the output type. The 'Positive Output Node' is set to '/out' and the 'Negative Output Node' is set to '/gnd'. The 'Input Source' is set to 'none'. The 'Noise Type' is set to 'timedomain' and the 'Noise Skip Count' is set to 0. There is another 'Add Specific Points' checkbox. At the bottom, there is an 'Enabled' checkbox and an 'Options...' button.

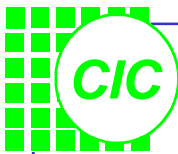


# Plotting Time Domain Results

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

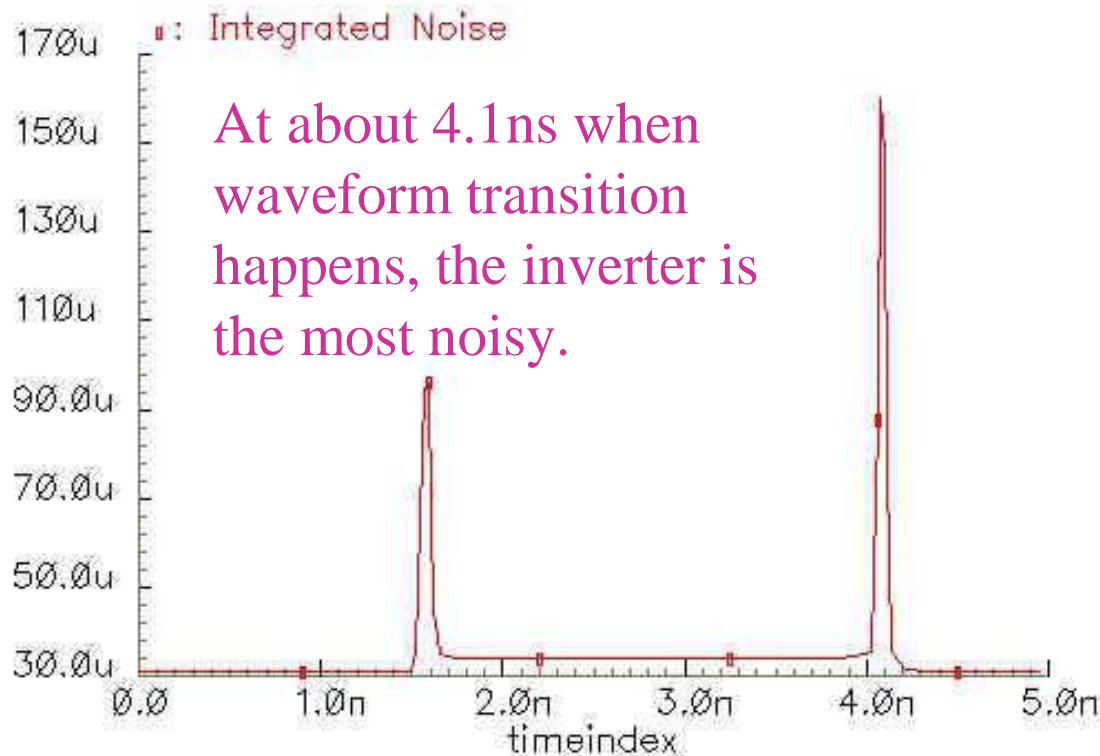
test inverter schematic : Jun 18 14:33:37 2002





# Plotting Time Domain Noise Results

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



**PSS Results**

OK Cancel Help

Plot Mode  Append  Replace

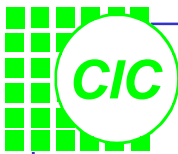
Analysis Type  
 pss  tdnoise

Function  
 Output Noise  Integrated Output Noise

Modifier  
 Magnitude  dB20

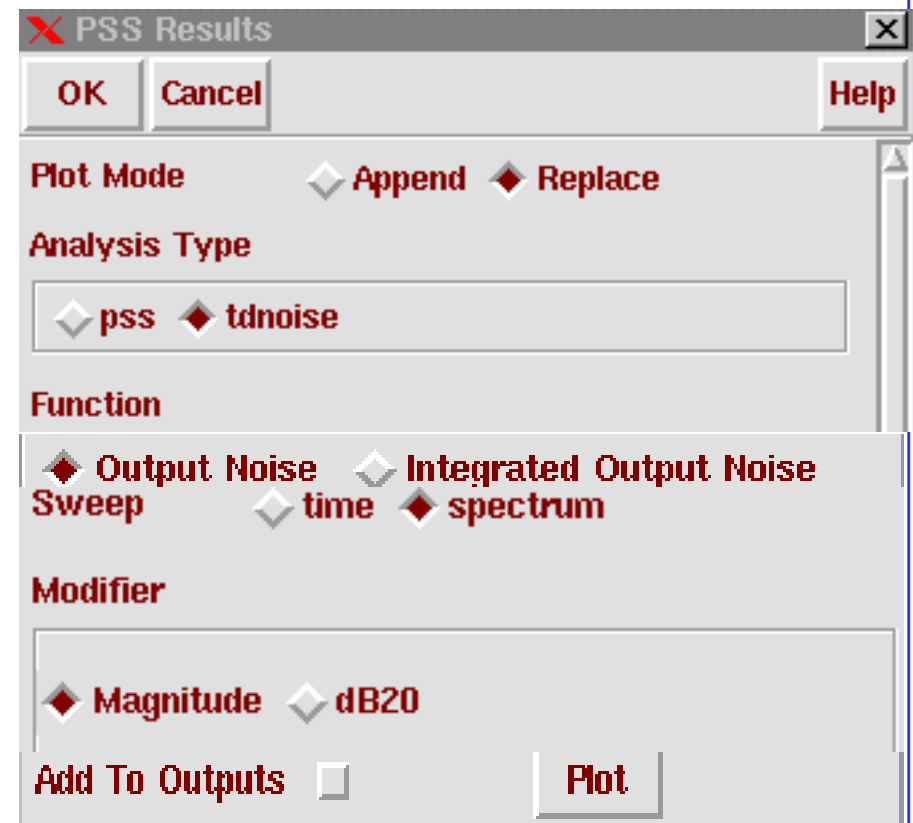
Time Domain Frequency Integration Range  
Start Frequency   
Stop Frequency

Add To Outputs  Plot



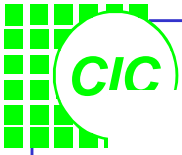
# Plotting Time Domain Noise Results on Spectrum(1)

- 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.



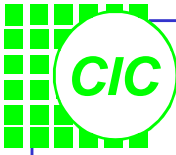






## 8. Periodic Distortion Analysis

- **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.

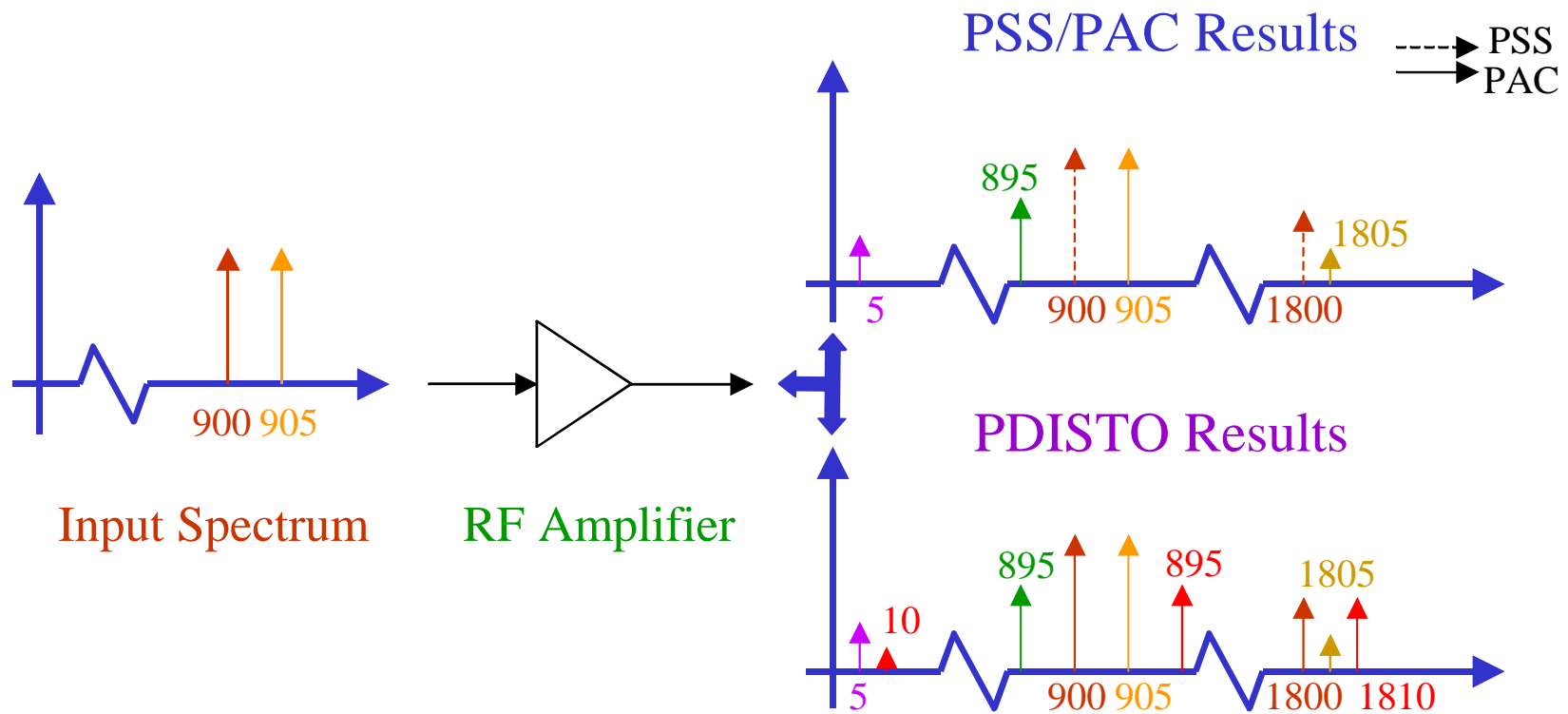


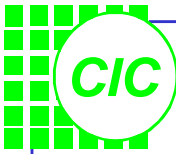
# PNOISE Analysis Overview

- 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 **P<sub>-1dB</sub>** point)
- The ability to **sweep P<sub>DISTO</sub>** provides a way to perform intermodulation distortion calculations with multiple input signals, considered as large signals.

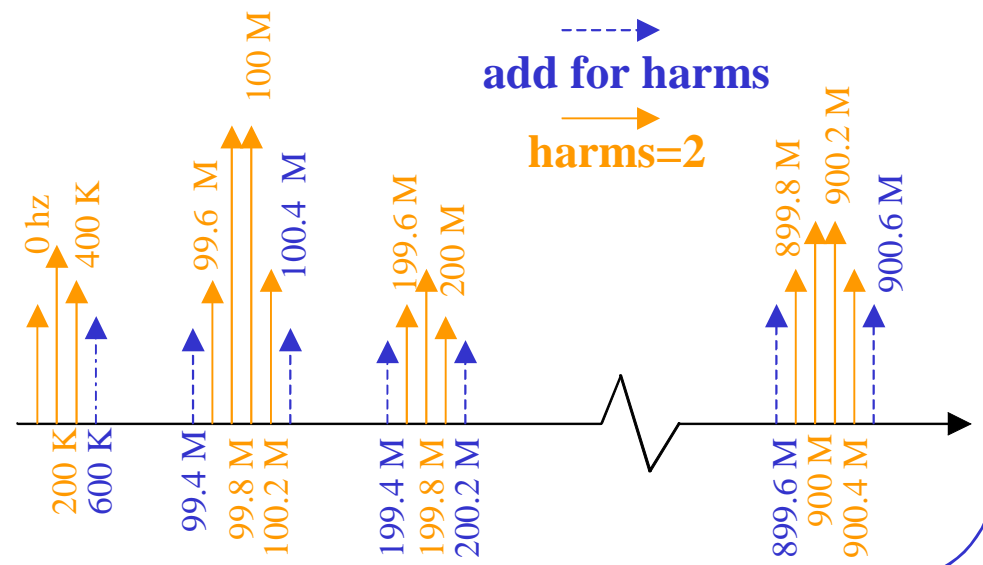
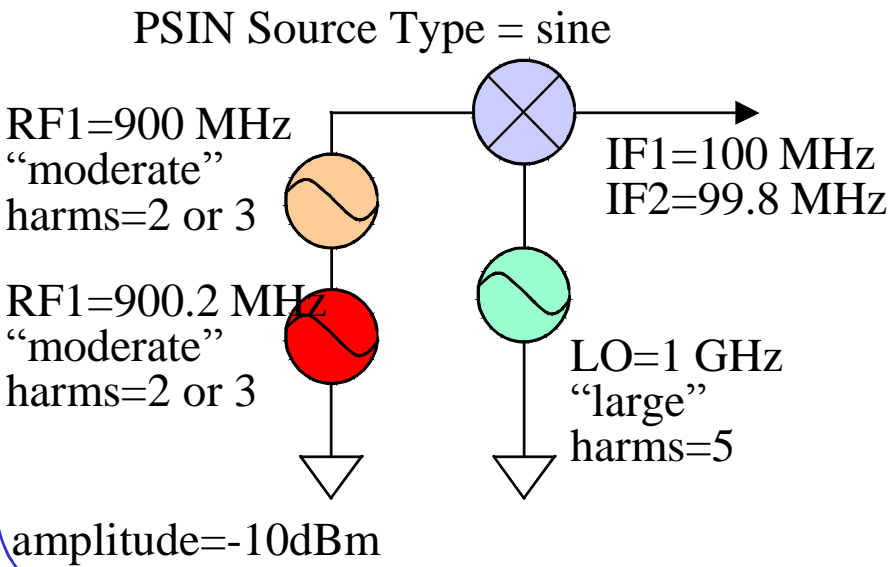
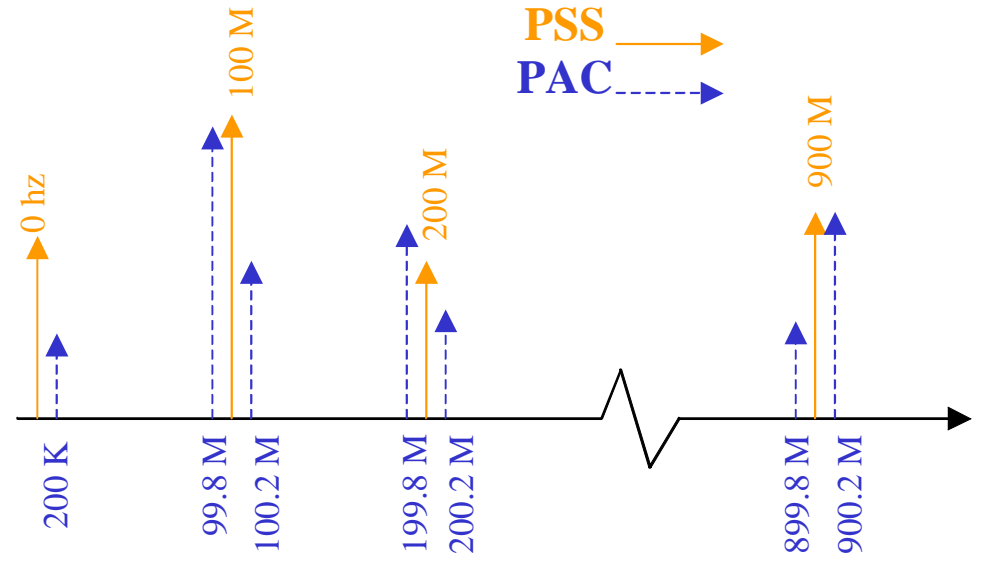
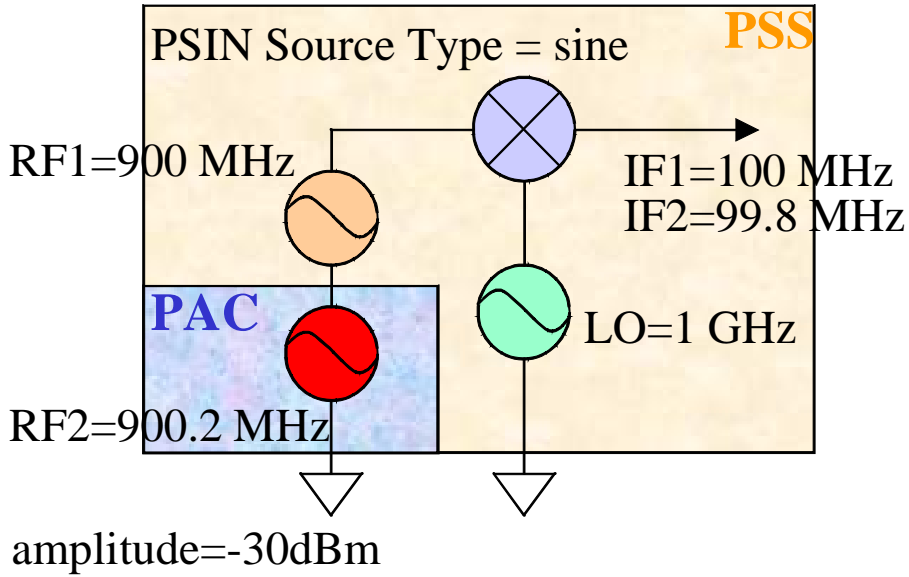
# PDISTO v.s. PAC

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





# Comparing PDISTO and PAC(1)

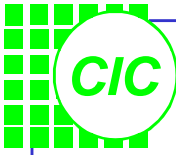


## Comparing PDISTO and PAC(2)

- 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

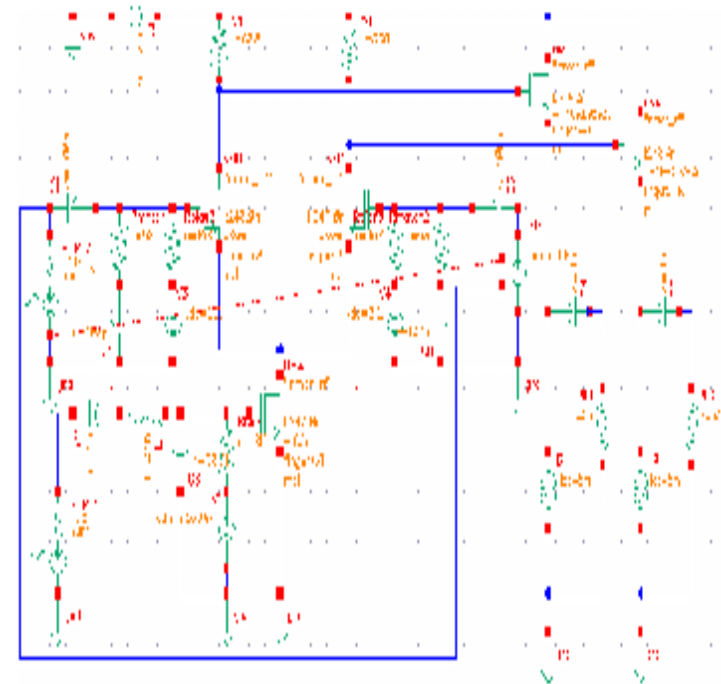
- Unlike **PSS**, **PDISTO** does not required multiple inputs be commensurate or coperiodic. However, they still must be periodic.
- For coperiodic, well separated signals, use **PSS**.
- For signals that are closely spaced or not coperiodic, use **PDISTO**.
- For circuits driven by 2 or more moderate 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  $1e-4$ .

**Simulator Options**

OK Cancel Defaults Apply Help

TOLERANCE OPTIONS

reltol

vabstol

iabstol

TEMPERATURE OPTIONS

temp

**Choosing Analyses -- Affirma Analog Circuit Design Environme**

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  xf  
 sens  sp  pdisto  pss  
 pac  pnoise  pxf  envlp

Periodic Distortion Analysis

Fundamental Tones

#	Name	Expr	Value	Signal	SrcId	Harms
1	F1	frf	2.4G	Moderate	PORT1	2
2	F1_1	frf + 1M	2.401G	Moderate	PORT1	2
3	F2	flo	2.3G	Large	PORT2	5

F1 frf 2.4G Moderate PORT1 2

Clear/Add Delete Update From Schematic

Accuracy Defaults (errpreset)  
 conservative  moderate  liberal

Additional Time for Stabilization (tstab)

Save Initial Transient Results (saveinit)  no  yes

Sweep

Enabled

Options...

# 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

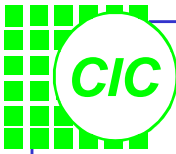
The screenshot shows the Affirma Analog Circuit Design Environment (1) window. The status bar indicates "Status: Ready", "T=27 C", and "Simulator: spectre 4". The menu bar includes Session, Setup, Analyses, Variables, Outputs, Simulation, Results, Tools, and Help. The main window is divided into several sections:

- Design:** Library: test, Cell: mixer1, View: schematic.
- Analyses:** A table with columns #, Type, Arguments, and Enable. Row 1 is highlighted: #1, Type: pdisto, Arguments: (empty), Enable: yes.
- Design Variables:** A table with columns #, Name, and Value.
 

#	Name	Value
1	prf	-30
2	frf	2.4G
3	flo	2.3G
- Outputs:** A table with columns #, Name/Signal/Expr, Value, Plot, Save, and March.
 

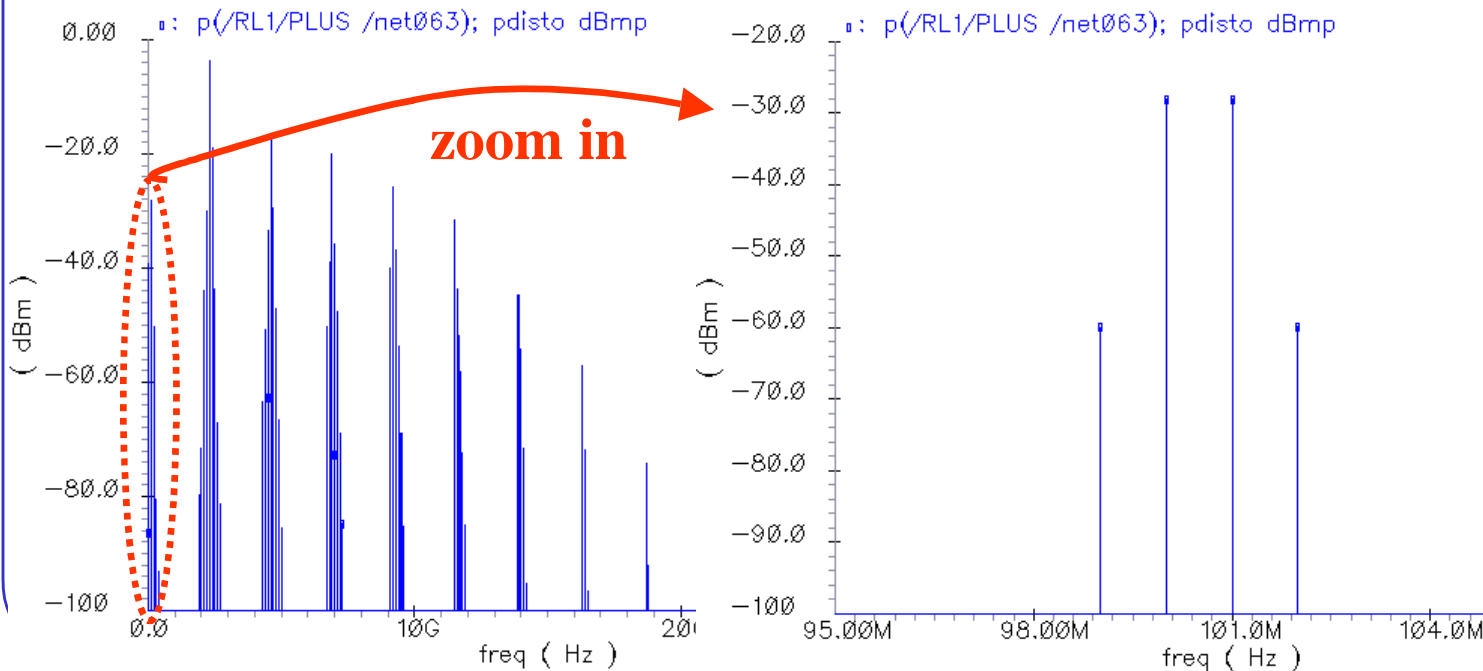
#	Name/Signal/Expr	Value	Plot	Save	March
1	PORT1/PLUS		yes	all	no
2	PORT1/MINUS		yes	all	no
3	RL1/MINUS		yes	all	no
4	RL1/PLUS		yes	all	no
5	RL2/MINUS		yes	all	no
6	RL2/PLUS		yes	all	no

The status bar at the bottom indicates: > Results in /users2/cic/lovid/simulation/mixer1/spectre/schematic



# 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



**PDISTO Results**

OK Cancel Help

Plot Mode  Append  Replace

Analysis Type

pdisto

Function

Voltage  Current

Power  Voltage Gain

Current Gain  Power Gain

Transconductance  Transimpedance

Compression Point  IPN Curves

Currently, only spectrum data is available

Modifier

Magnitude  dB10  dBm

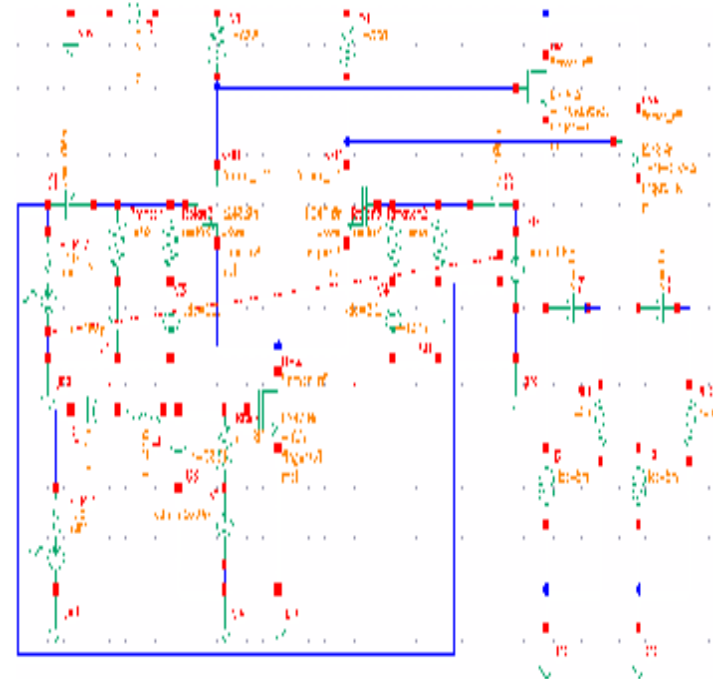
Add To Outputs  Replot

> Select instance terminal on 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.

Choosing Analyses -- Affirma Analog Circuit Design Environment

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  xf  
 sens  sp  pdisto  pss  
 pac  pnoise  pxf  envlp

Periodic Distortion Analysis

Fundamental Tones

#	Name	Expr	Value	Signal	SrcId	Harms
2	F1	frf	2.4G	Moderate	PORT1	2
6	F1_1	frf+25M	2.425G	Moderate	PORT1	2
3	F2	flo	2.3G	Large	PORT2	3

F1 frf 2.4G Moderate PORT1 2

Clear/Add Delete Update From Schematic

Accuracy Defaults (errpreset)  
 conservative  moderate  liberal

Additional Time for Stabilization (tstab)

Save Initial Transient Results (saveinit)  no  yes

Sweep  Frequency Variable?  no  yes  
 Variable Name   
 Select Design Variable

Sweep Range  
 Start-Stop Start  Stop   
 Center-Span

Sweep Type  
 Linear  Step Size   
 Logarithmic  Number of Steps

Add Specific Points

Enabled  Options...

# 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<sup>st</sup> order harmonics: 100M & 125M

3<sup>rd</sup> order harmonics: 75M & 150M

**PDISTO Results**

OK Cancel Help

Plot Mode  Append  Replace

Analysis Type

pdisto

Function

Voltage  Current  
 Power  Voltage Gain  
 Current Gain  Power Gain  
 Transconductance  Transimpedance  
 Compression Point  IPN Curves

Circuit Input Power  Single Point  Variable Sweep ("prf")

"prf" ranges from -40 to -20

Extrapolation Point (dBm)

Input Referred IP3  Order

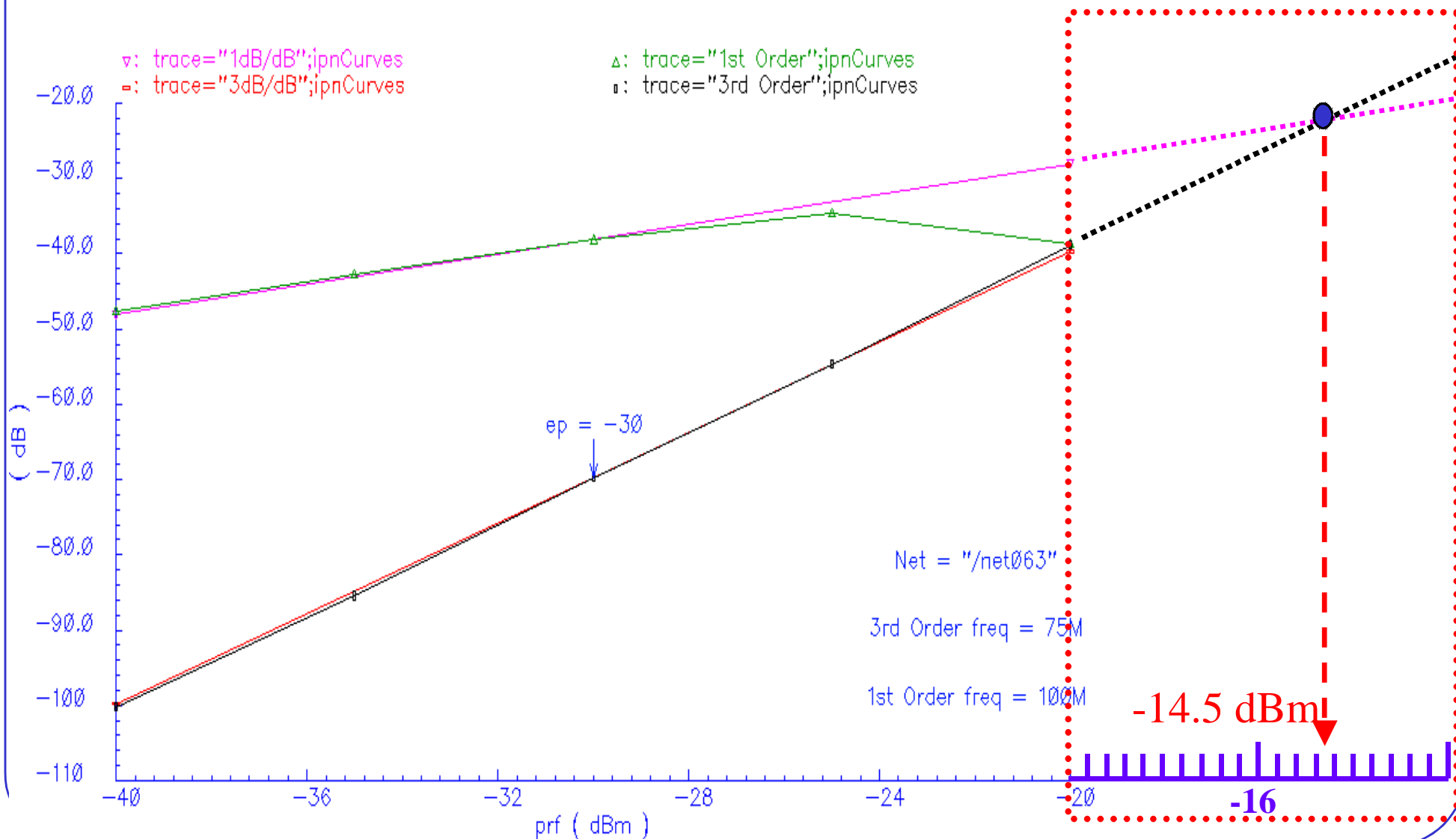
	Freq. (Hz)	F2	F1	F1_1
3rd Order Harmonic	50M	0	-2	2
	75M	-1	2	-1
	100M	-1	1	0
	125M	-1	0	1
1st Order Harmonic	150M	-1	-1	2
	0	0	0	0
	25M	0	-1	1
	50M	0	-2	2
	75M	-1	2	-1
	100M	-1	1	0

Add To Outputs  Replot

> Select port or net on schematic ...



# Displaying the IP3 Plot(2)





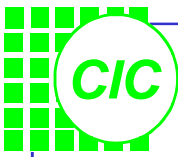
## 9. Oscillator and Phase Noise Analysis

- 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:**

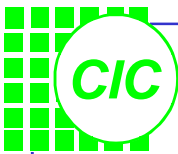
The circuit is simulated repeatedly while the length of the period and the initial conditions are adjusted to achieve a periodic steady state solution.



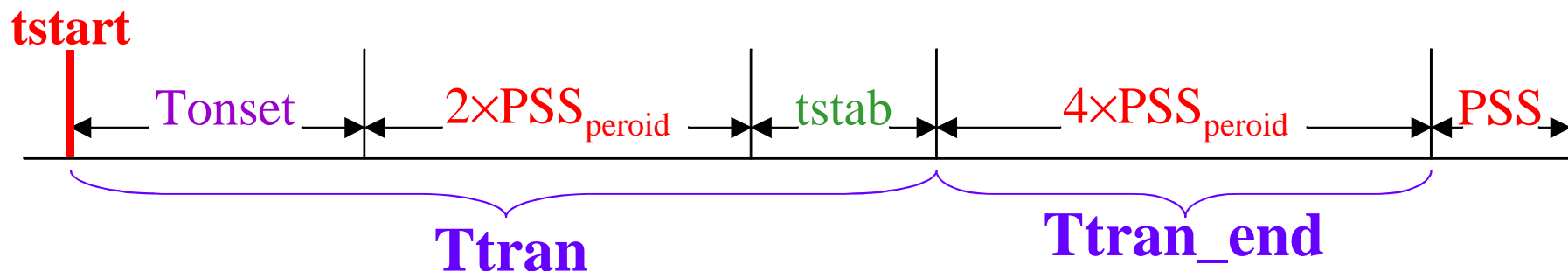


# Troubleshooting Oscillators

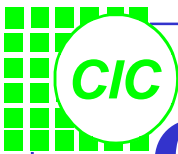
- Does not converge – increase *tstab*
- Improve the estimate of the period. Be especially carefully that the period specified is not too short.
- Change the value of the method parameter from *gear2only* to *trap* or *traponly*.
- Does not converge – increase *maxperiods*
- If the shooting iteration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



# Oscillator PSS Algorithm



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



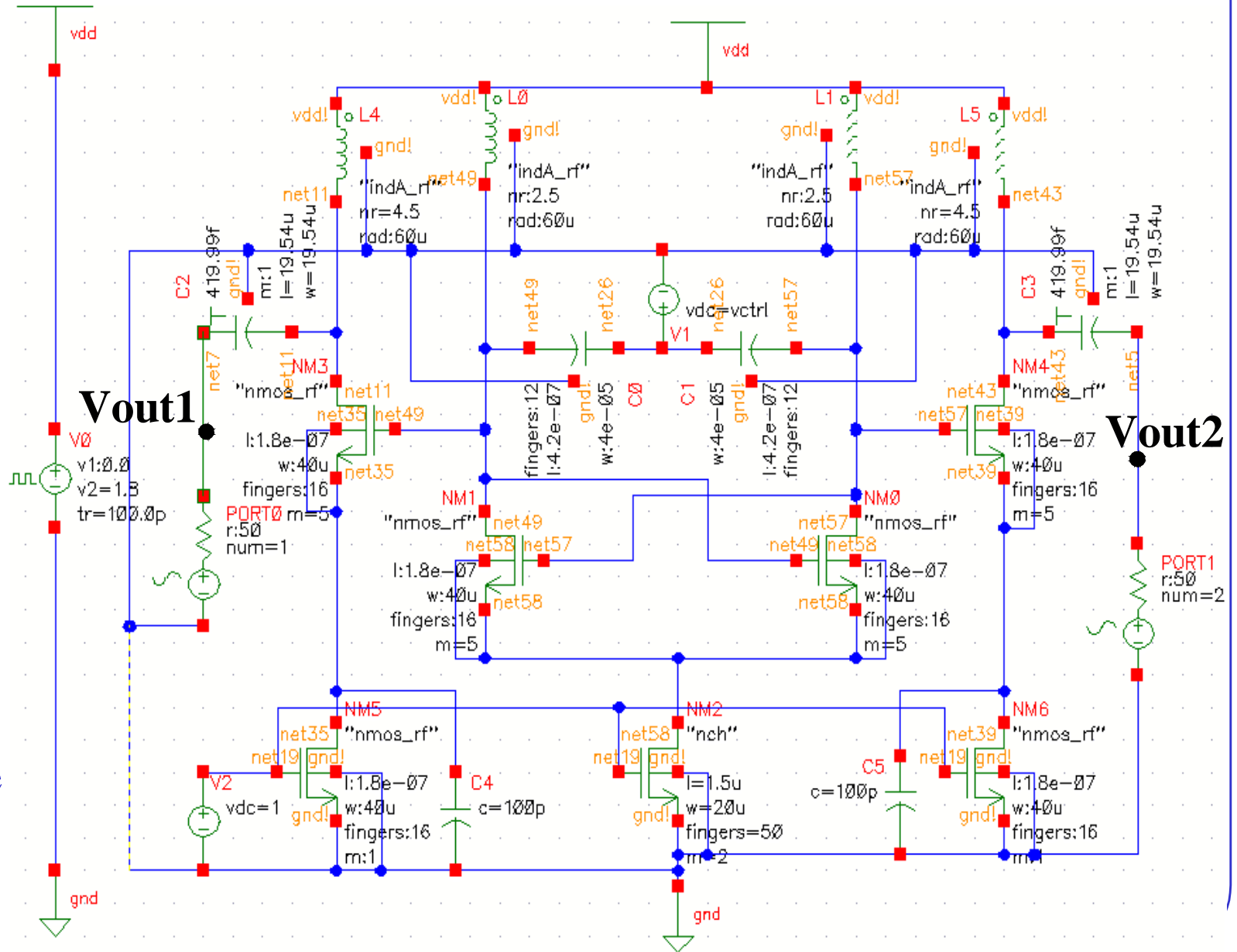
# Oscillator Algorithm and maxstep

- Default *maxstep* > period if  $T_{tran} > 50$  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* <  $1/(200 \times \text{FreqOsc})$
- In PSS shooting iterations stage, maxstep is the smallest of:
  - *maxstep* manual entry
  - $\text{PSSperiod}/(\text{maxharm} \times 40)$
  - $1/(\text{maxacfreq} \times 5)$
  - $\text{PSS Beat Frequency}/200$
- 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.

The screenshot shows the Design Environment software interface. The main window displays the Design Environment form with the following data:

Design			Analyses			
Library	Cell	View	#	Type	Arguments	Enabl
test	osc	schematic	1	tran	0 200n	yes

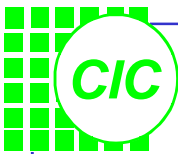
  

Design Variables			Outputs					
#	Name	Value	#	Name/Signal/Expr	Value	Plot	Save	March
1	vctrl	900m	1	PORT0/PLUS		yes	yes	no
			2	PORT0/MINUS		yes	yes	no

The Model Library Setup dialog box is open, showing the following table:

Model Library File	Section
/users2/cic/ovid/181p6m_pdk/models/mm018.scs	tt
/users2/cic/ovid/181p6m_pdk/models/mm018.scs	res
/users2/cic/ovid/181p6m_pdk/models/rf018.scs	rf_macro
/users2/cic/ovid/181p6m_pdk/models/rf018.scs	tt_rfmos

The dialog box also includes a section for 'Model Library File' and 'Section (opt.)' with an empty table and buttons for 'Add', 'Delete', 'Change', 'Edit File', and 'Browse...'. The main window has a status bar showing 'Status: Ready' and 'T=27 C Simulator: spectre 6'.



# 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.

Choosing Analyses -- Affirma Analog Circuit Design Environment

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  xf  
 sens  sp  pdisto  pss  
 pac  pnoise  pxf  envlp

Transient Analysis

Stop Time

Accuracy Defaults (empreset)  
 conservative  moderate  liberal

Enabled

Options...

writefinal

ckptperiod

INTEGRATION METHOD PARAMETERS

method  euler  trap  traponly  
 gear2  gear2only  trapgear2

ACCURACY PARAMETERS

relref  pointlocal  alllocal  sigglobal  allglobal

Iteratio

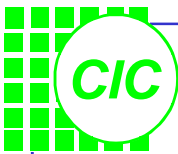
ANNOTATION PARAMETERS

stats  yes  no

annotate  no  title  sweep  status  steps

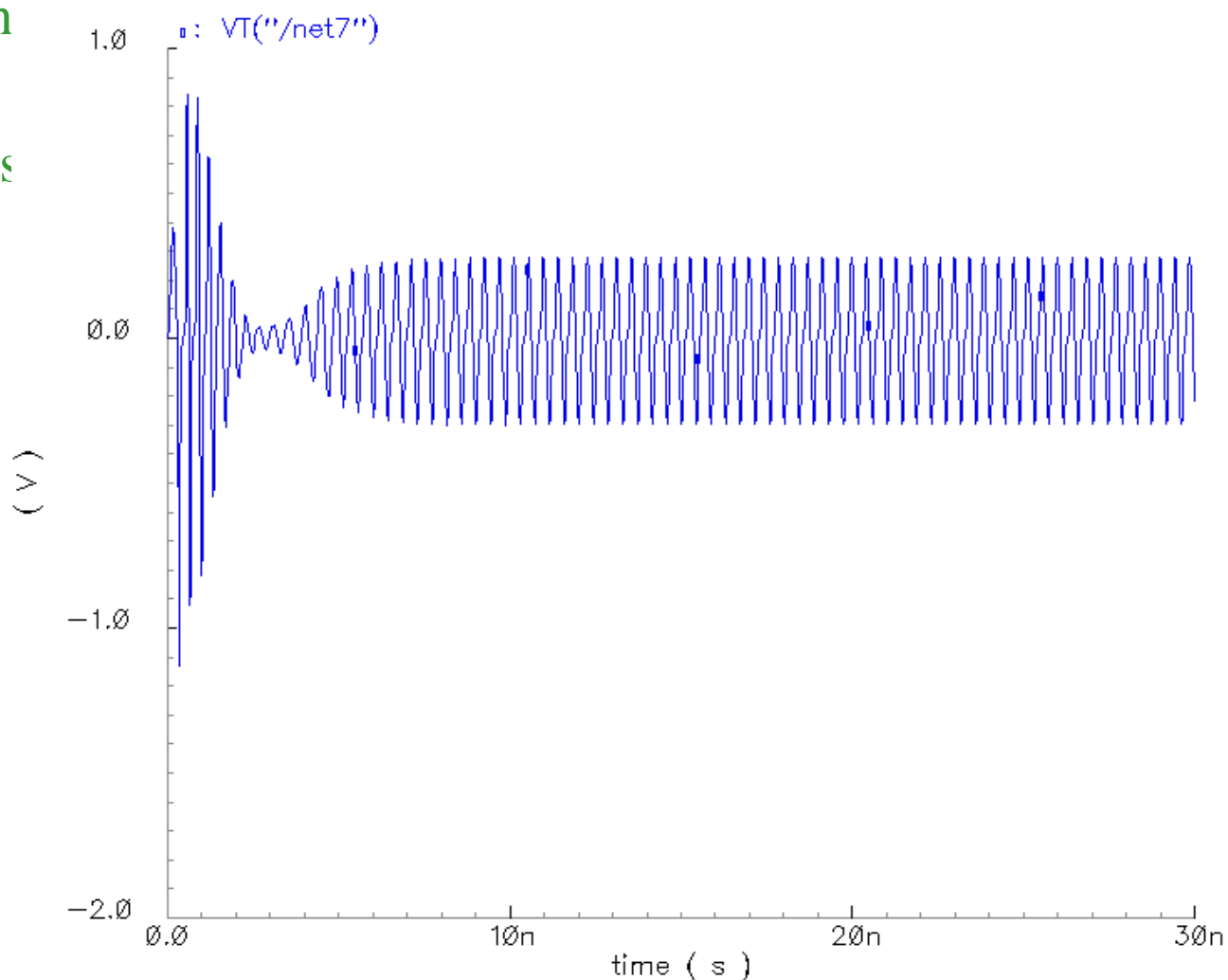
OUTPUT PARAMETERS

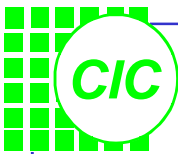
save  selected  lvlpub  lvl  allpub  all



# 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.

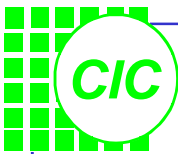




# Oscillator Notes

- 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.





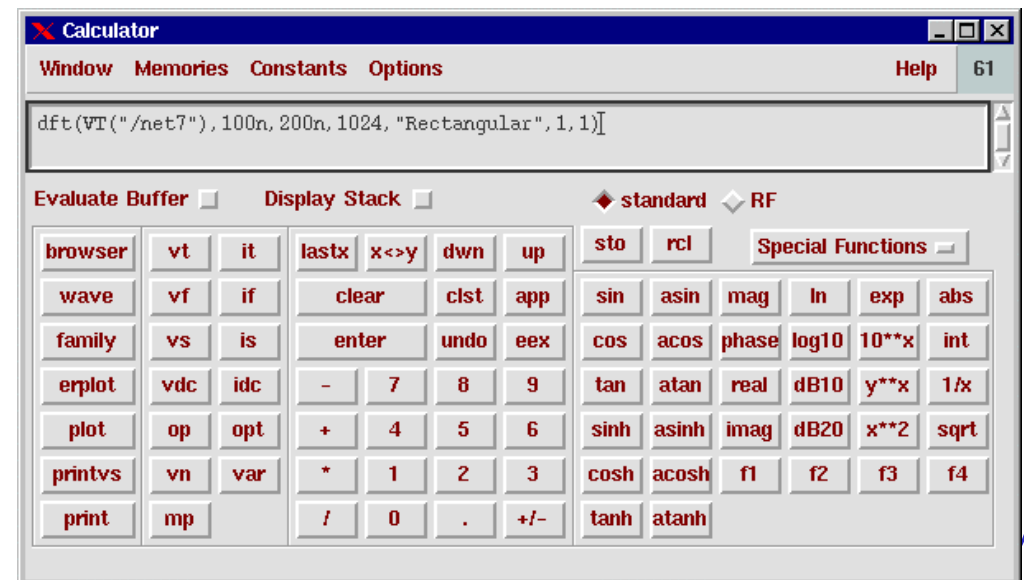
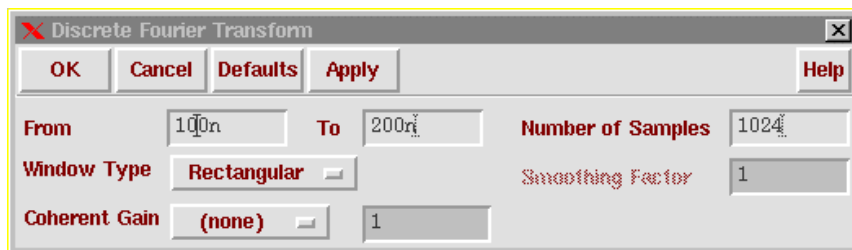
# spectre.fc file

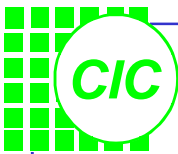
```
CIC
# CHECKPOINT_VERSION 1
# Generated by spectre from circuit file `input.scs` during analysis tran.
# 4:41:44 PM, Thur Jun 20, 2002
# Number of equations = 77
NM2:int_d      1.38143129288017
NM2:int_s      4.27510716664238e-05
net5          -0.150439862589224
net7           0.202993505669876
net11         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

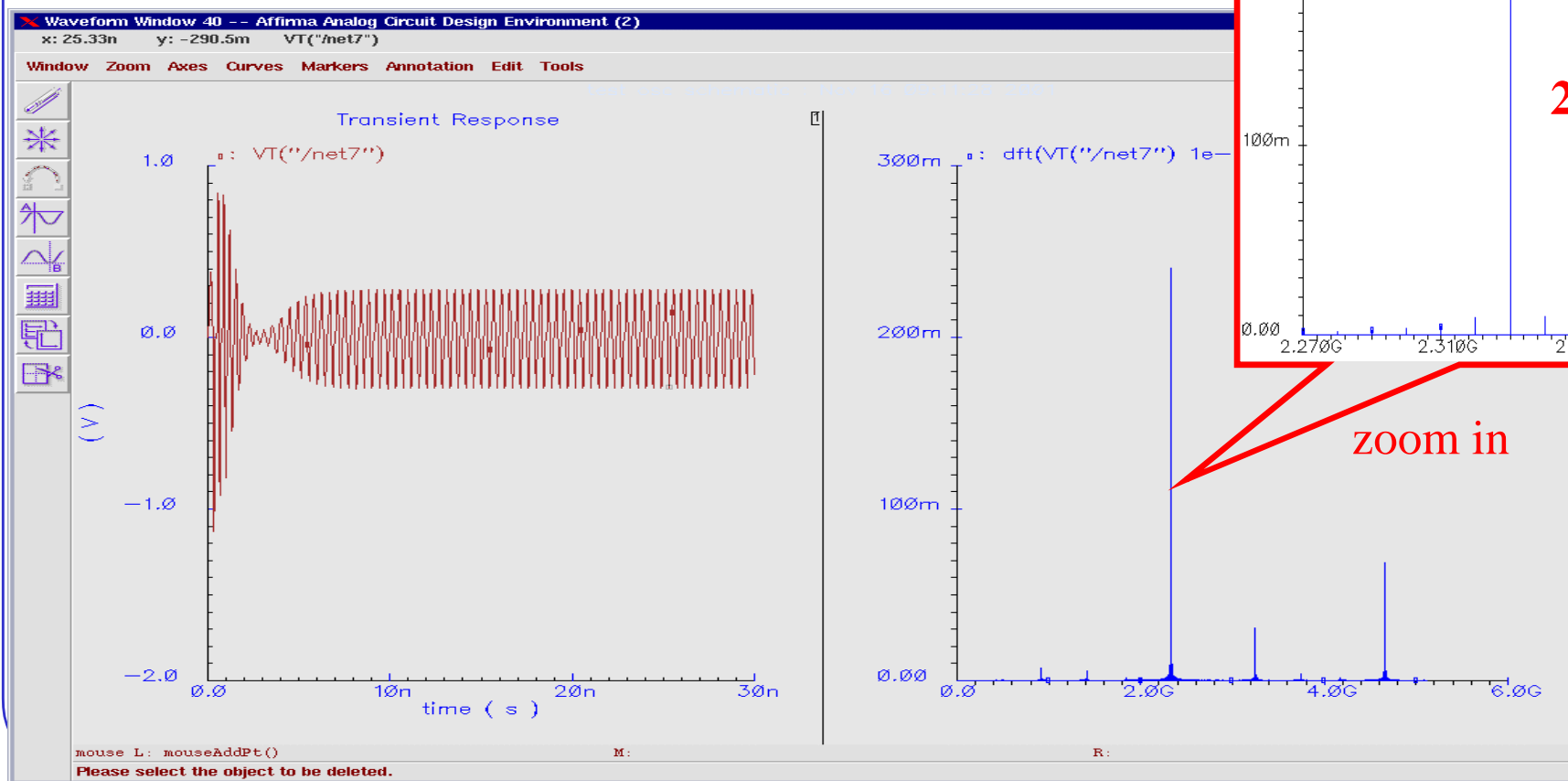
- In the Waveform window, click the **Add Subwindow** icon, then a subwindow with a label of **2** in the upper right corner is added.
- Click the **Calculator**, then the calculator appears.
- Click the **vt** button in the Calculator and follow the prompt at the bottom of the schematic window. Then 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.

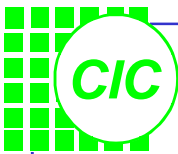




# The Frequency Domain Results

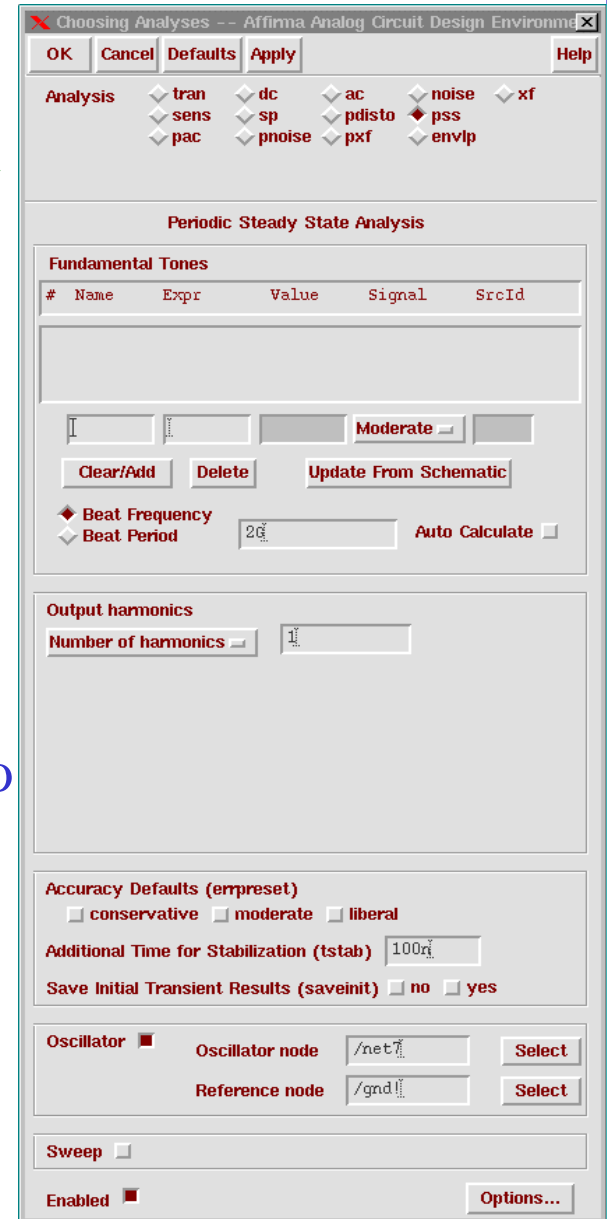
- Finally, in the Calculator, click **erplot**
- Note the initial estimate of the oscillation frequency is developed.





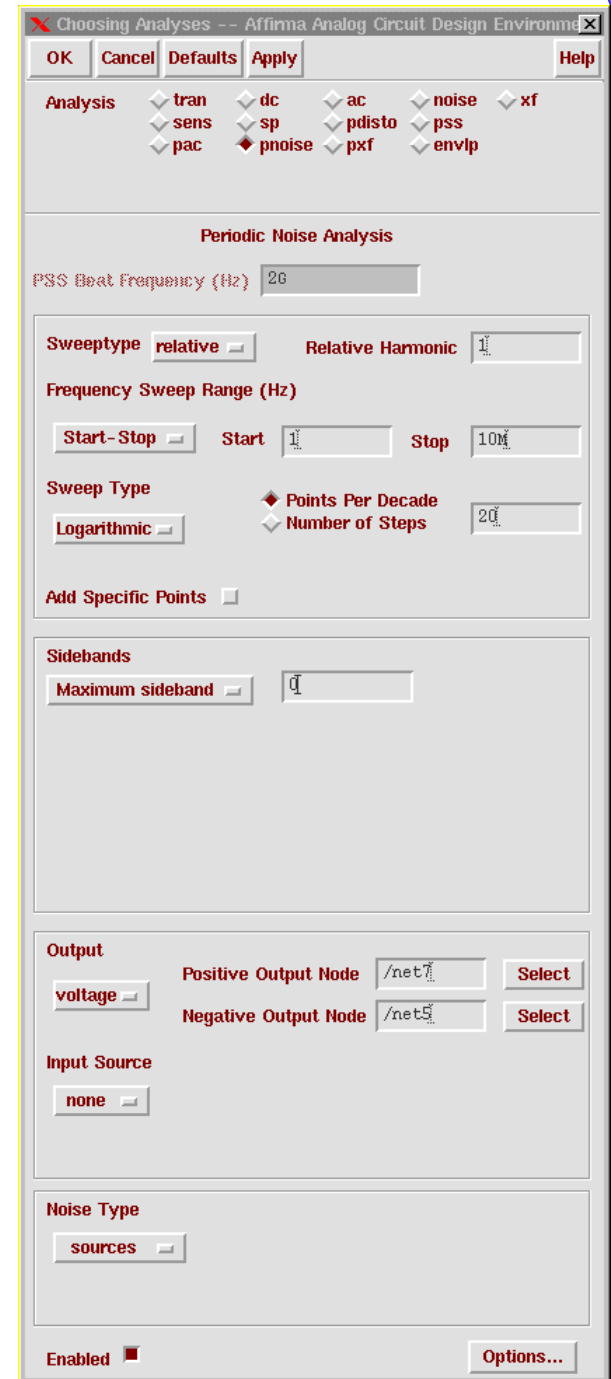
# PSS/PNOISE Analysis(1)

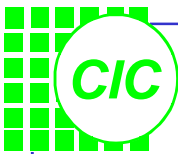
- In the *Choosing Analysis* window, turn off the transient analysis; select the **pss** analysis and set up the form as right:
- An estimate of 2GHz was selected for Beat Frequency. It's recommended to estimate a lower frequency than expected to help in the convergence.
- The value of **tstab** is set 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)

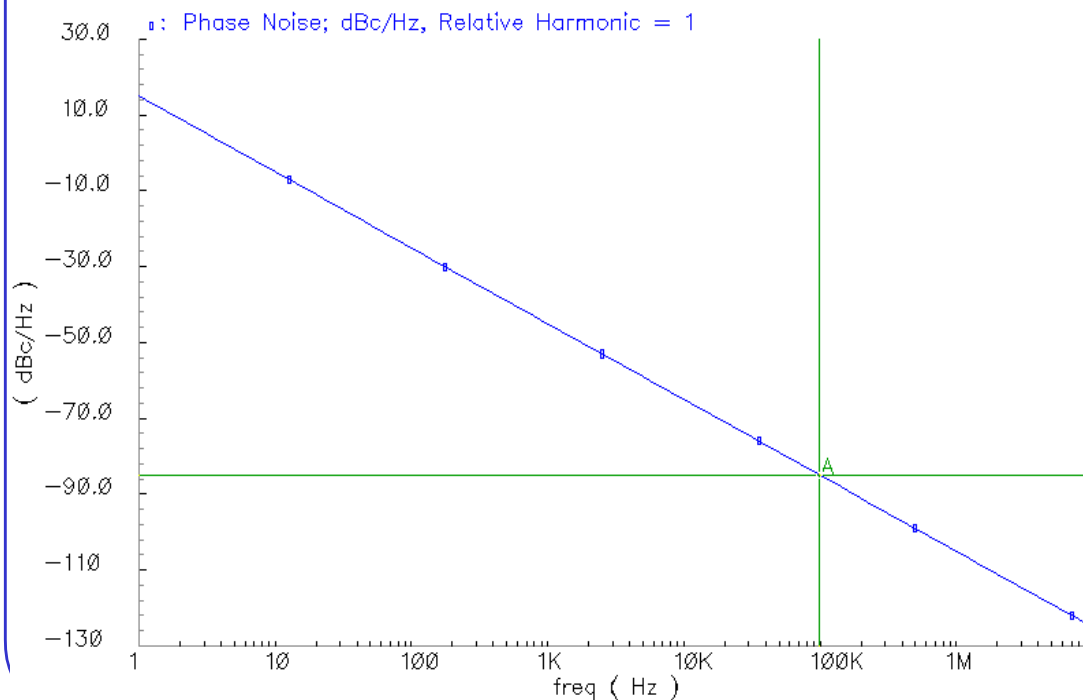
- Next, click the **pnoise** button, and set up the PNOISE analysis as right:
- The phase noise from 1 Hz to 10 MHz, *relative* to the derived oscillation frequency, will be calculated.
- The Sidebands field is set to a Maximum sideband of 0. In this case, you are interested in the upconverted  $1/f$  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.



A: (100K -84.9865)

The image shows two side-by-side screenshots of the 'PSS Results' dialog box. Both windows have 'OK', 'Cancel', and 'Help' buttons at the top. The 'Plot Mode' is set to 'Append' and 'Replace'. The 'Analysis Type' is set to 'pss' and 'pnoise'. The 'Function' section has 'Output Noise', 'Noise Figure', 'Transfer Function', and 'Pss Beat Frequency' selected. The right window shows the message 'Currently, only frequency data is available'.