A Tutorial

On

Advanced Analysis

For

Cadence Spectre

Prepared By:

Rishi Todani

rishi@32mosfets.com

Web: http://www.32mosfets.com

With Guidance, Encouragement and Blessing From

Dr. Ashis Kumar Mal

Associate Professor, ECE Department,

NIT Durgapur

CONTENTS

CHAPTER

ONE

TEST BENCH SETUP

Simple test benches to perform analysis covered in this tutorial are discussed here. For a single ended circuit, say operational amplifiers, a sample test circuit is shown in Fig. 1.1. An instance named “port”, found in analoglib is connected at the input and output nodes.

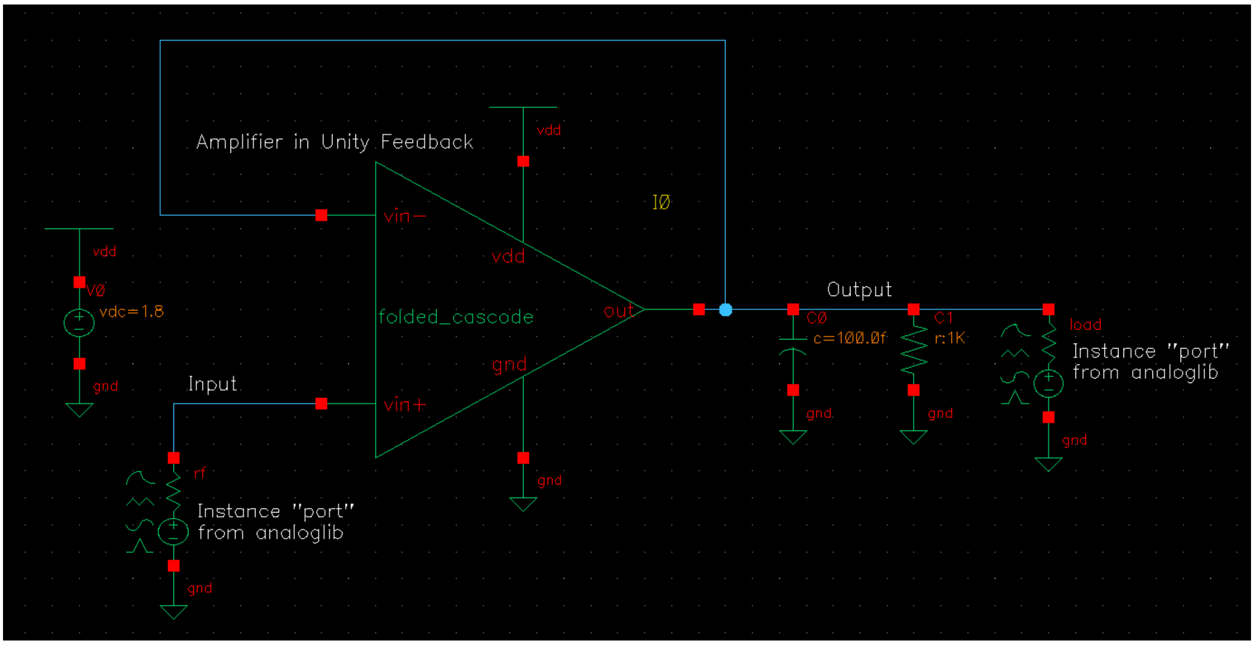


Figure 1.1: Test Bench for single ended circuits

If the circuit under test is a fully differential circuit, the test bench slightly changes. An instance called “balun” from rflib is also used. Balun is capable of converting single ended signals to differential and visa versa and maintains the common mode level during the conversion. The rflib library is already included inside cadence installation directory and can easily be added to the library manager. The rflib library can be found in the location:
<CDS INSTALL DIR>/tools/dfII/samples/artist

or for example, if cadence IC5141 is intalled inside a folder called IC51, at the location /cad/cadence, then the absolute path to rflib could be
/cad/cadence/IC51/tools/dfII/samples/artist

It is required that the designer adds this library to his library manager and use balun for these simulations.

Few other handy libraries can also be found in this location, like ahdlLib, aExamples, rfExamples, corners, monteCarlo etc., which can be explored.

A sample circuit for testing a fully differential operational amplifier is given in Fig. 1.2.

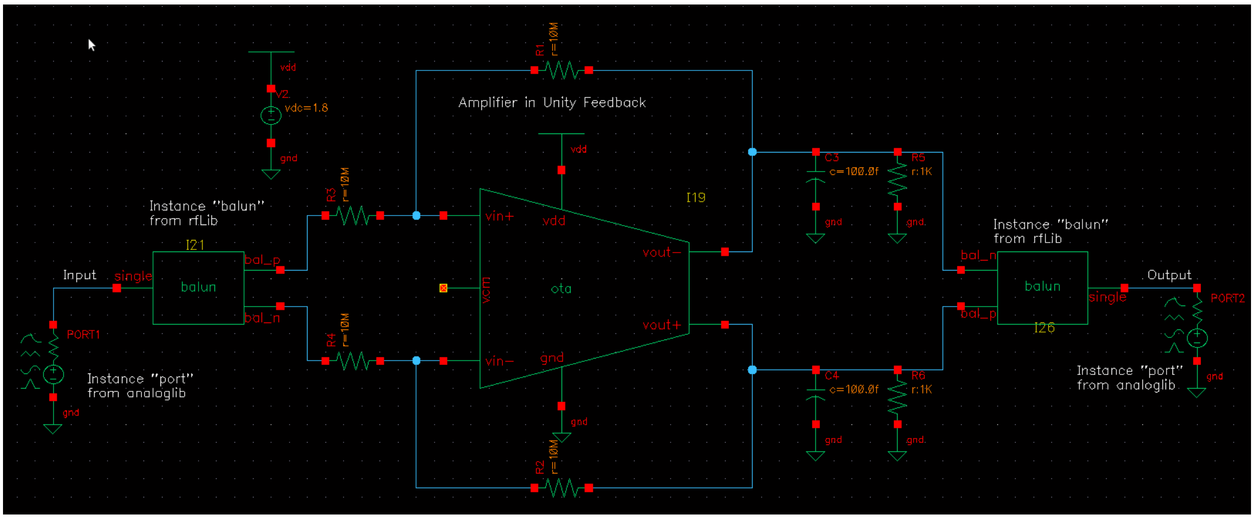


Figure 1.2: Test Bench for fully differential circuits

Some designers may want to connect the load to the circuit in a different way. Another way to connect the load would be to connect the common mode dc level to the negative terminal. This reduces the current requirement to drive the load. A sample is given in Fig. 1.3.

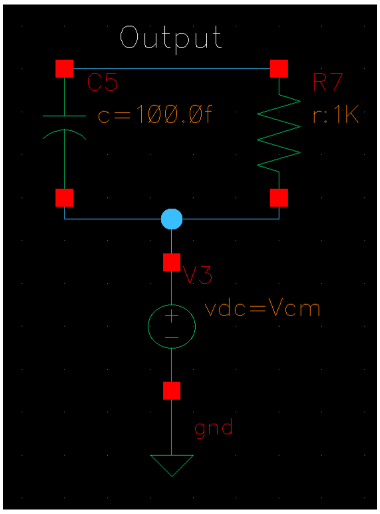


Figure 1.3: Alternate technique of connecting load

CHAPTER

TWO

S-PARAMETER ANALYSIS

The S-parameter or SP analysis is a linear small signal analysis.

2.1 Setup of S-Paramter Analysis

For performing these analysis, following setup is to be done.

1. Setup test schematic. If differential input/output are present, use "**balun**" from "**rflib**" and convert to single ended.
2. Use instance "**port**" at input and output node. It can be found in **analoglib**.
3. Let input port be called **rf** and output port be called **load** for easy reference.
4. Setting of port **rf** (Refer Fig. 2.1) Resistance - 50 ohm Port number - 1 DC Voltage - input common mode level source type - dc
5. Set port load to type **dc**. Check and save schematic and open ADE.
6. in ADE setup **sp** analysis. In sp analysis window (Refer Fig. 2.2), in ports field, click **select** button. Then schematic opens, click on port rf (input port) and then on port load (output port). Ports "**rf load**" should get listed in sp analysis window.
7. In "**sweep range**" field enter start-stop values of frequency. Choose sweep type as linear and set number of steps (say 50).
8. In "**Do Noise**" select **yes** and set output port to "**/load**" and input port to "**/rf**"

   NOTE: By choosing yes under "Do Noise", noise analysis is setup. We can obtain small signal noise when input power level is low and circuits are considered to be linear.
9. The port creates two variables: frf: fundamental frequency (enter a value) prf: input power (say -50)
10. Run Simulation

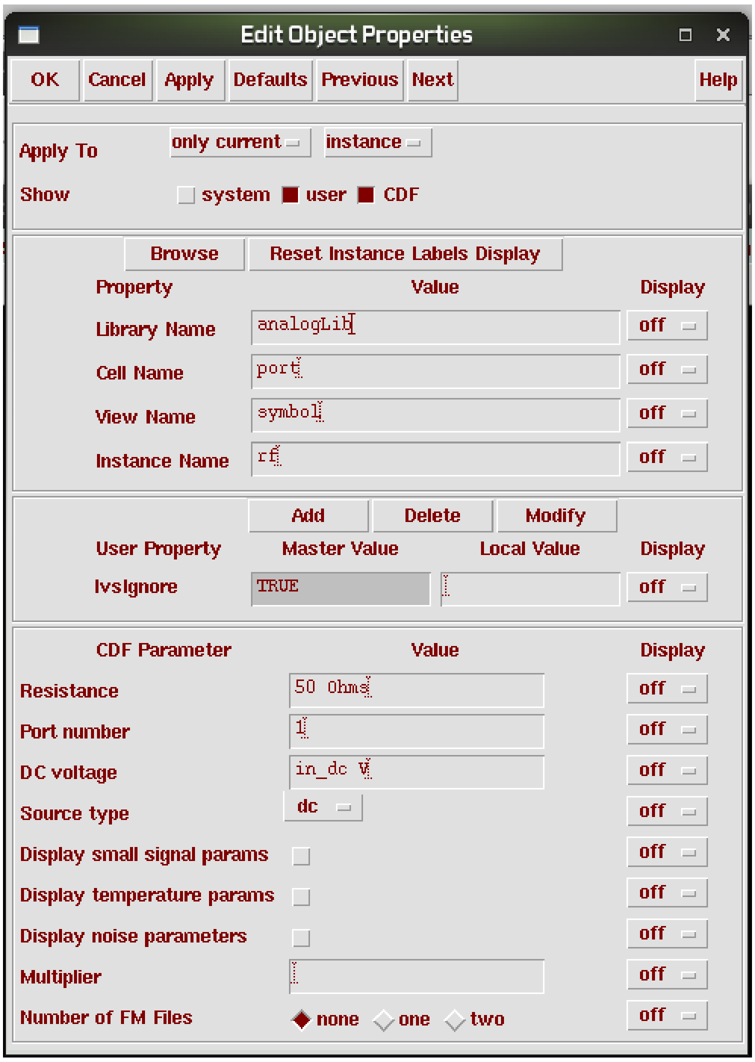


Figure 2.1: Input port setting

2.2 Plotting Transducer Gains (G$_T$, G$_A$, G$_P$, G$_{msg}$, G$_{max}$ and G$_{umx}$.)

- G$_T$ - Transducer power gain
- G$_A$ - Available power gain
- G$_P$ - Operating power gain of a two port network
- G$_{msg}$ - Maximum stability Gain
- G$_{max}$ - Maximum Transduver gain
- G$_{umx}$ - Maximum Unilateral Transducer power gain

Where,

$$G_T = \frac{\text{Average Power Delivered to load}}{\text{Maximum available average power at source}} \tag{2.1}$$

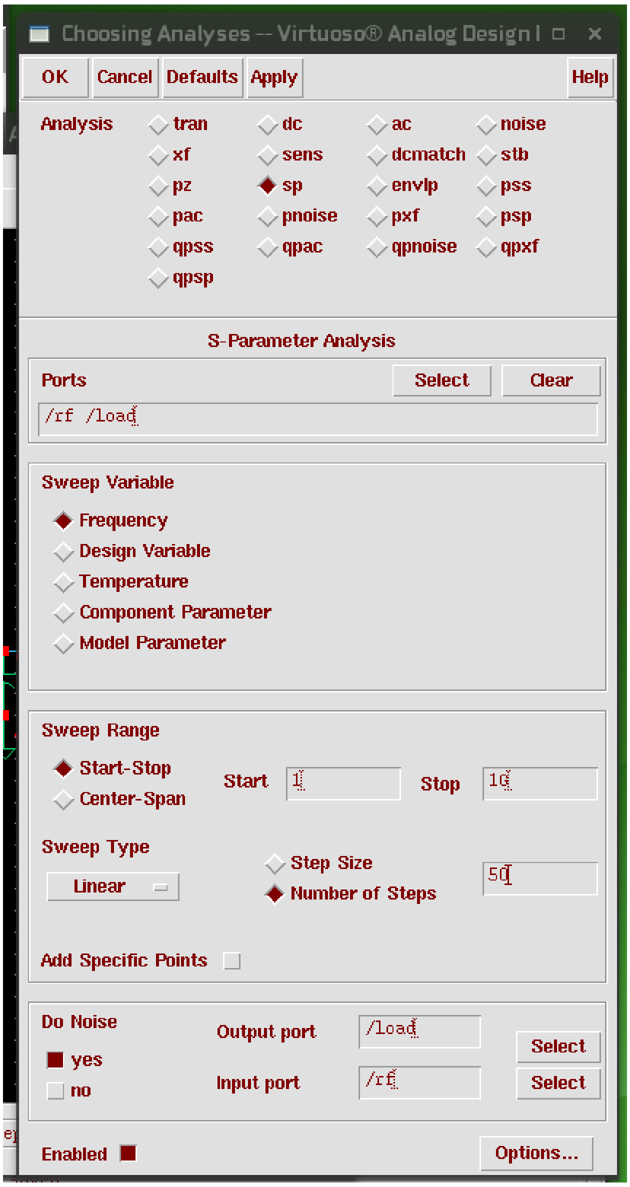


Figure 2.2: SP analysis setup

$$G_P = \frac{\text{Maximum Power Delivered to load}}{\text{Average power entering the network}} \tag{2.2}$$

$$G_A = \frac{\text{Maximum available average power at load}}{\text{Maximum power available at source}} \tag{2.3}$$

Setup S-Parameter analysis as described above in 2.1. Run Simulation.

1. Click **Results** > **Direct Plot** > **Main Form**. Set plotting mode to **append**. In analysis field, select **sp**. In function select G_T (transducer gain). In modifier select dB10. Click on **Plot** button to plot G_T (refer Fig. 2.4).

2. Similarly we can plot G_A, G_P, G_{msg}, G_{max} and G_{umx}.

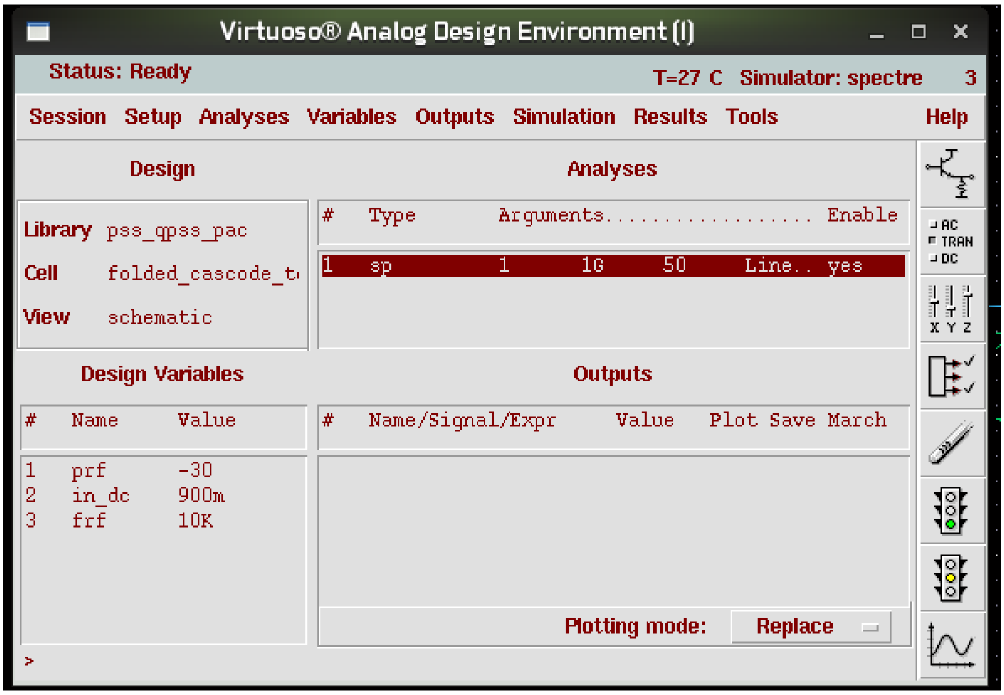


Figure 2.3: ADE for sp analysis

2.3 Plotting GAC, GPC, Kf, B1f, LSB and SSB

Setup S-Parameter analysis as described above in 2.1. Run Simulation.

1. Click **Results** > **Direct Plot** > **Main Form**. Set plotting mode to **append**. In analysis field, select **sp**. In function select **G**$_{AC}$ (available gain circle). Plot type choose **Z-smith**. Sweep gain level (dB) at frequency = fundamental freq from xdB to ydB (say 14 to 18dB) with steps in dB (say 0.25 dB). Plot.

2. Similarly plot **GPC** (Power gain Circle). The two contours are plotted for fundamental frequency. Plot **Kf** and **B1f**.

3. In function choose **LSB** (Load stability circle). **Plot type = z-smith**. Specify frequency range covering fundamental frequency and give step size. Plot.

4. Similarly plot **SSB** (Source stability circle).

2.4 Noise Figure, Noise Circle, VSWR, S11, S12, S21, S22

Setup S-Parameter analysis as described above in 2.1. Run Simulation. Click **Results** > **Direct Plot** > **Main Form**.

1. In Function select **Noise Figure (NF)**. In modifier select **dB10** and click plot (Refer Fig. 2.5).

2. In function select **Noise Circle (NC)**. Plot type - **Z-Smith**. Select Sweep **Noise Level (dB)** (at fundamental Freq). Enter Frequency = Fundamental Freq = frf in ADE. **Level Range (dB)** = 1dB to 3dB in steps of 0.25dB (say)

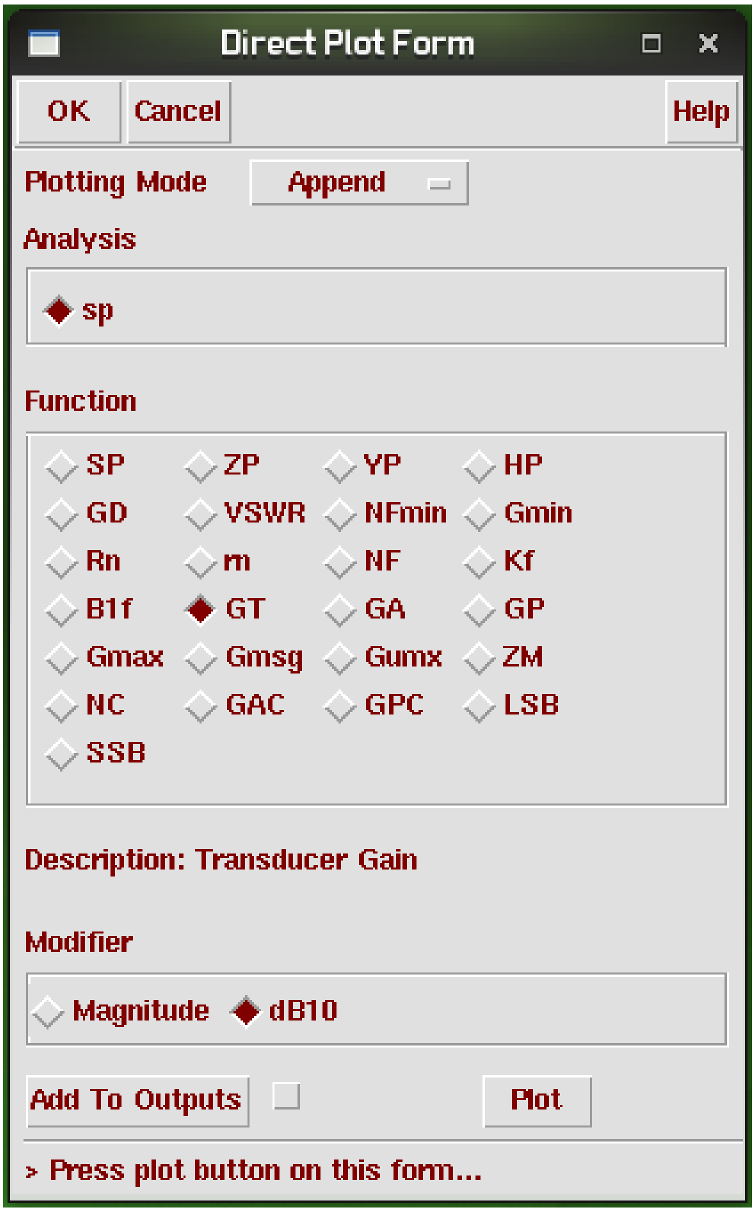


Figure 2.4: Plotting G_T

frf = 2.4 GHz (say)
Frequency (Hz) = fundamental freq = frf = 2.4 GHz
Level Range (dB), Start = 1.5 dB, Stop = 2.5 dB, step = 0.25 dB
Plot

3. Function = **VSWR**, Modifier = **dB20**, Press on **VSWR1** and then on **VSRW2**.

4. Function = **SP**, type = **Rectangular**, Modifier = **dB20**. Press **S11, S12, S21 and S22**.

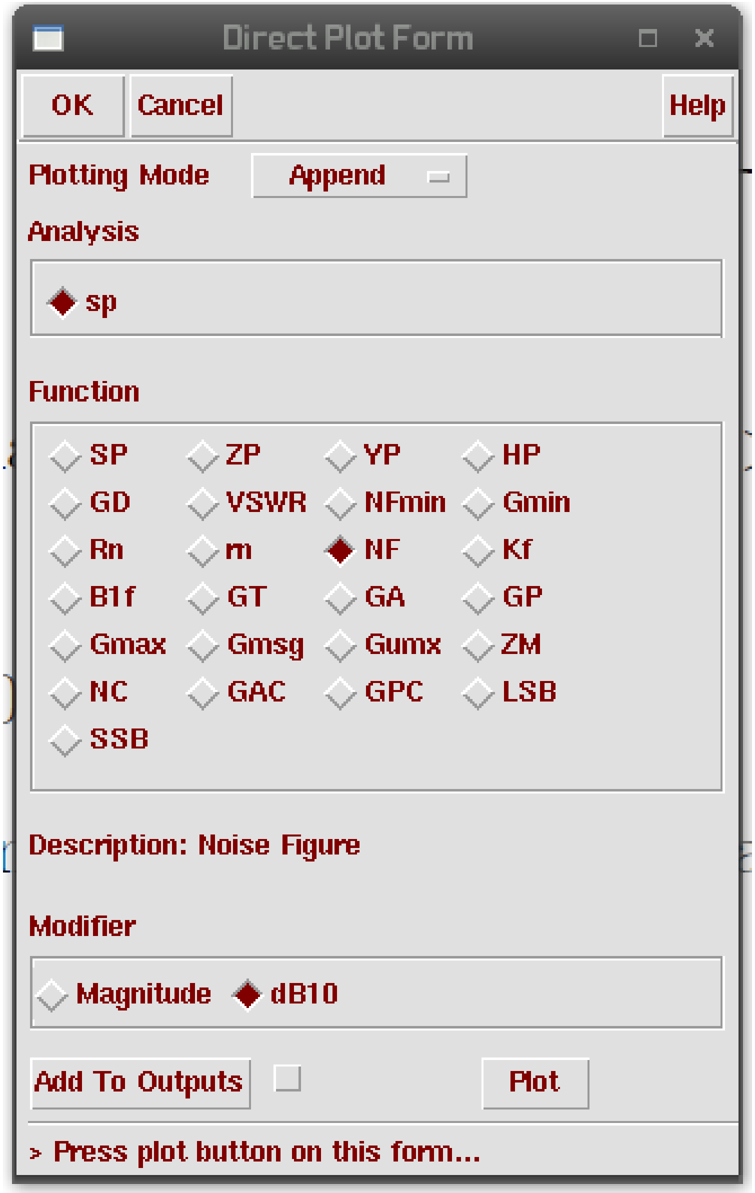


Figure 2.5: Plotting Noise Figure (NF)

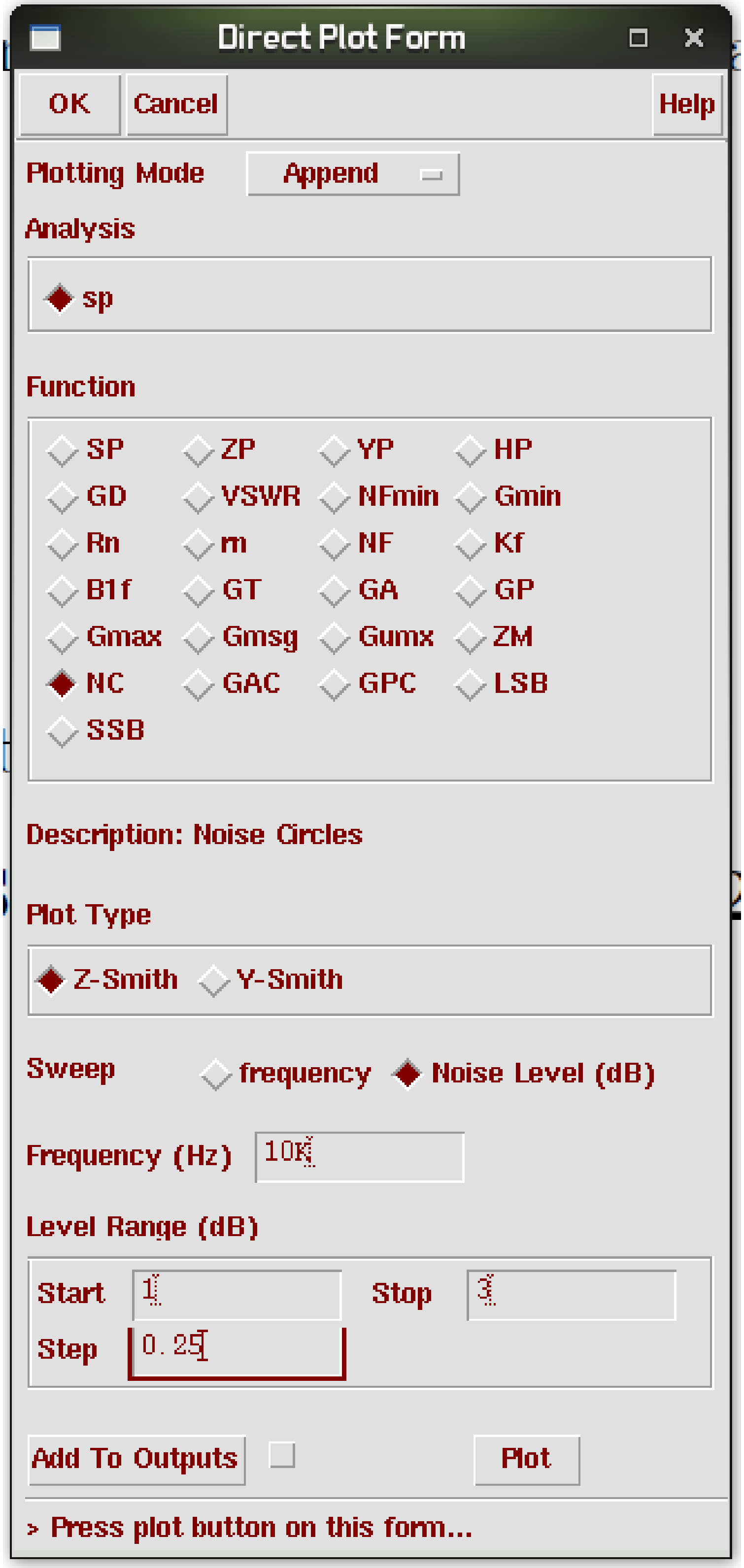


Figure 2.6: Plotting Noise Circle (NC)

CHAPTER

THREE

LARGE SIGNAL NOISE ANALYSIS (PSS AND PNOISE)

Use PSS and PNoise analysis for large signal and non-linear noise analysis, when the circuits are linearised around the periodic steady state operating point.

Use Noise and SP analysis for small signal and linear noise analysis when the circuits are linearized around DC operating point.

As input power increases, the circuit becomes non-linear, the harminocs are generated and the noise spectrum is folded. Therefore, we should use PSS and PNoise analysis.

When Input power level remains low, the Noise Figure calculated from PNoise, PSP, Noise and SP analysis should all match.

3.1 Setup PSS and PNOISE analysis

Add "port" to input and output of schematic and do the following settings

1. In schematic, select input port "rf"
   Port no. = 1
   DC volt = 0.9 or Vdd/2 or VCM
   source type = sine
   frequency name = rf
   freq 1 = frf (this is the fundamental frequency)
   amplitude 1 (dBm) = prf

2. In ADE, copy variables from schematic and enter values of frf and prf.
   frf = 100K (say)
   prf = -40

3. Setup PSS analysis in ADE.
   Select **Auto calculate** for beat frequency. It automatically takes it as frf.
   Set **number of harmonics to 10**. This allows us to look at in the frequency domain results with 10 harmonics of beat frequency.
   Select **Moderate** accuracy.

4. In ADE, under analysis, choose PNOISE. Refer Fig. 3.1
   Specify noise source and number of side bands. The larger the number of side bands, the more accurate the results.
   Set reference sideband as 0 if your circuit has no frequency conversion from input to output (amplifier).
   Sweep type - default
   Give a start - stop range covering the fundamental frequency (say 95K to 105K)
   Specify number of steps = 20 (say)
   Maximum sidebands = 10
   Output source - Probe /load (output port)
   Input source - Probe /rf (input port)
   Reference sideband = 0 (for amplifiers)
   Click OK and run Simulation

3.2 Ploting Noise Figure (NF)

To plot Noise Figure (NF), open Direct Plot > Main Form.
Select Analysis - PNOISE50

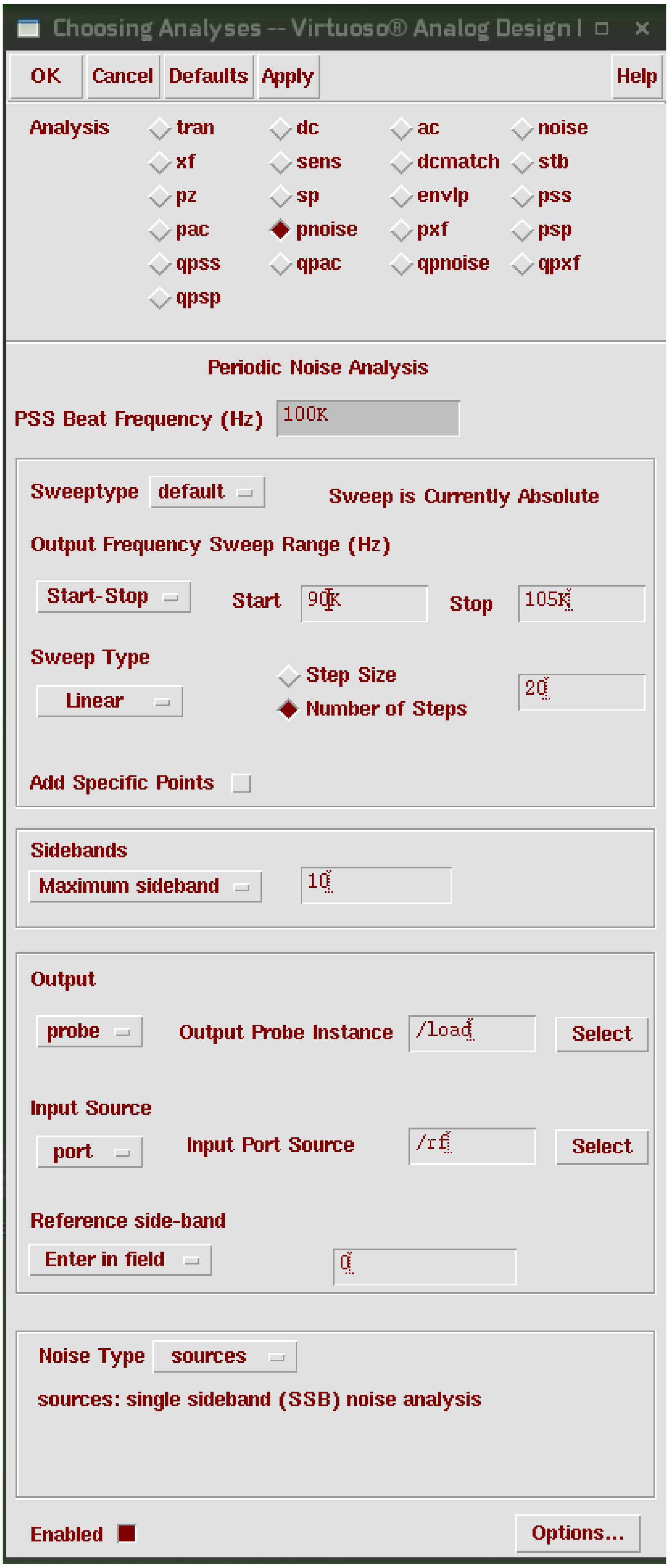


Figure 3.1: PNOISE analysis setup

CHAPTER

FOUR

GAIN COMPRESSION & TOTAL HARMONIC DISTORTION (THD) (SWEPT PSS)

Setup the test bench by connecting **port** at input and output. Let input port be called **rf** and output be called **load**.

Let output port be of type dc. Choose **rf input port** and set its properties (Refer Fig. 4.1).
Resistance = 50 Ohm
Port no. 1
DC Voltage = Input common mode ($V_{DD}/2$)
Source Type = sine
Freq Name 1 = RF
Freq 1 = frf
Amplitube (dBm) = prf

4.1 1dB Compression

Follow these steps to plot **1 dB Gain Compression** Point.

1. Setup testbench as above.
   In ADE Copy Variables. Set frf = 10KHz (say), prf=-10dB (say)

2. Setup PSS analysis. (Refer Fig. 4.2)
   Auto Calculate Beat Frequency.
   Number of Harmonics = 10.
   Accuracy = **Moderate**.
   Sweep prf from ay -30 to 30 Number of steps = 30.
   Netlist and Run

3. Results > Direct Plot > Main Form.
   Analysis = PSS.
   Function = Compression Point.
   Select - **Port (Fixed)**.
   Format = **Output Power**.
   Input Power extrapolation point - (blank)

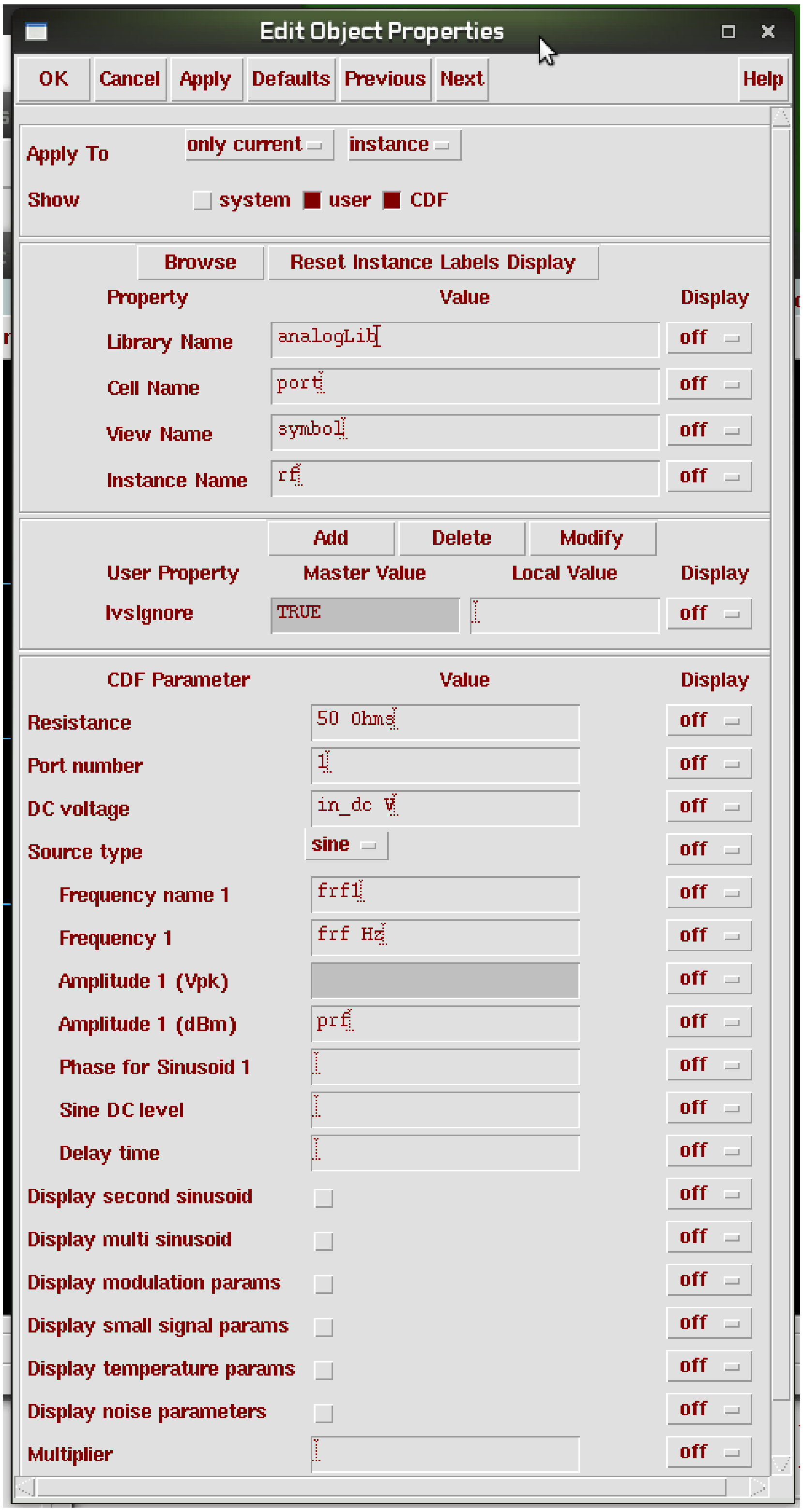


Figure 4.1: Port setup for PSS

Select **Input referred 1dB Compression** from drop down menu.
1st order harmonics - **Select Fundamental freq**.
Select Load port to plot (Refer Fig. 4.3)

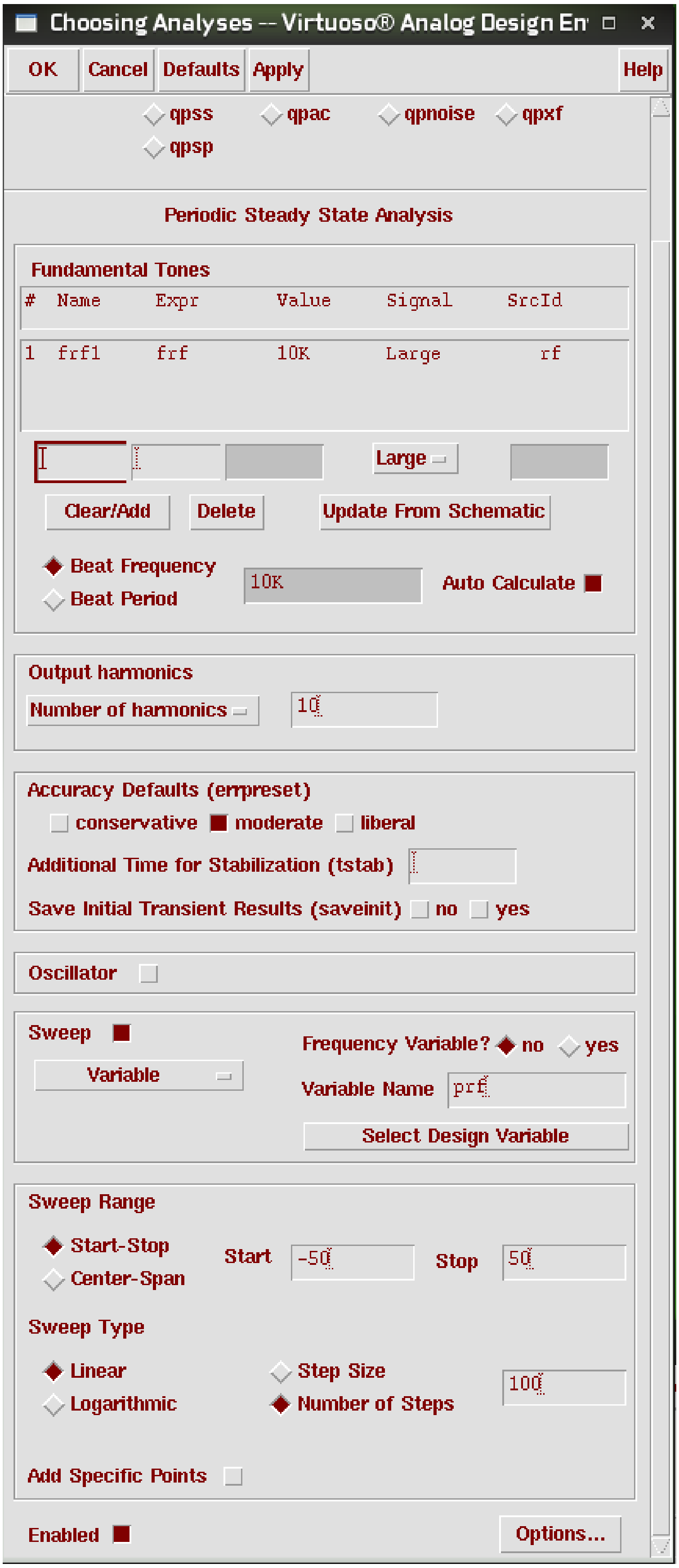


Figure 4.2: Setup PSS analysis

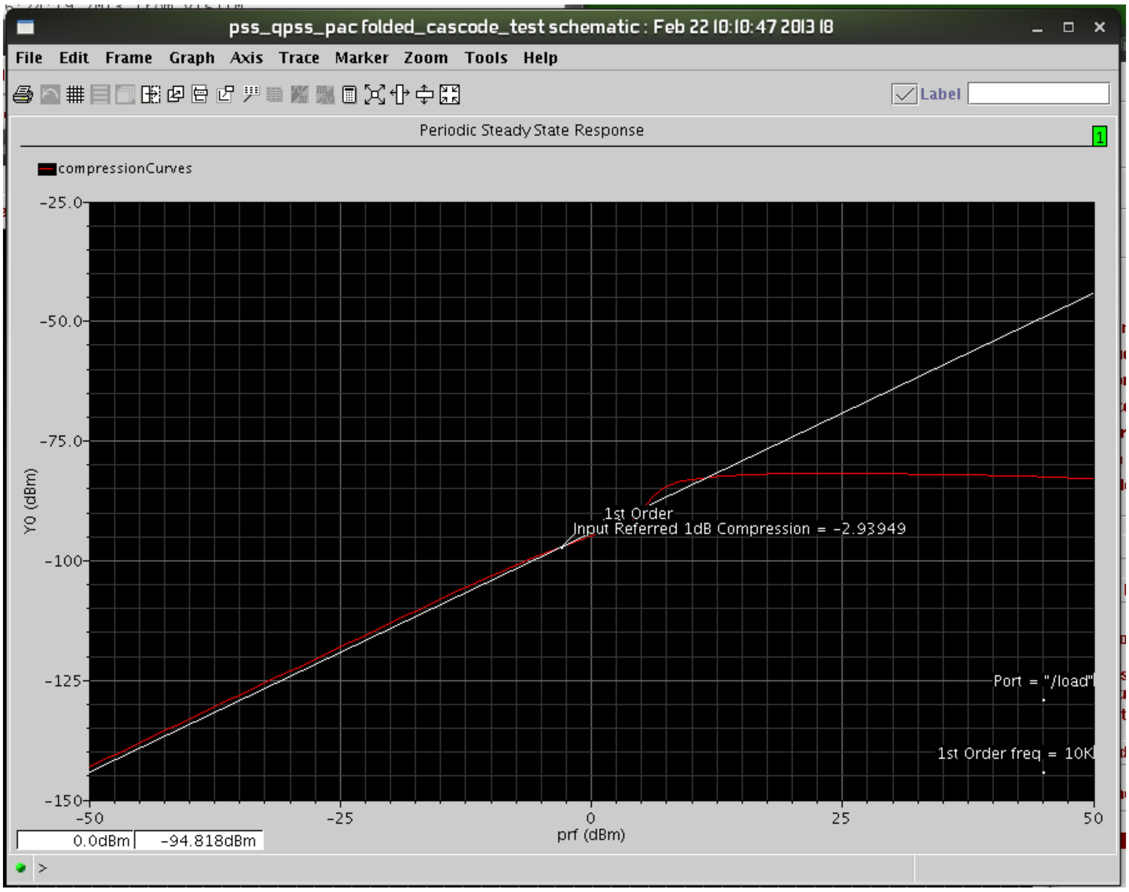


Figure 4.3: 1dB compression output

NOTE: 1 dB compression point is the point where the two curves on the graph intersect. The power axis (x axis) given the input power at which the output power reduces by 1 dB

4.2 Harmonic Distortion

Harmonic distortion can be observed by plotting the spectrum of any node voltage. Harmonic distortion is characterised as the ratio of the power of the fundamental signal divided by the sum of the power at the harmonics.

To plot harmonic distortion,

1. Setup testbench as above. In ADE Copy Variables. Set frf = 10KHz (say), prf=-10dB (say)

2. Setup PSS analysis.
   Auto Calculate Beat Frequency.
   Number of Harmonics = **10**.
   Accuracy = **Moderate**.
   Sweep prf from ay -30 to 30 with number of steps = 10.
   Netlist and Run

3. Results > Direct Plot > Main Form. (Refer Fig. 4.4)
   Analysis = **PSS**.

Function = **Voltage**.
Select "**net**" from drop down menu
sweep = **spectrum**
Signal level = **peak**
modifier = **dB20**
Variable value = **select power at fundamental freq**
Select output Net on schematic to plot (Refer Fig. 4.5)

4.3 Total Harmonic Distortion

Perform PSS analysis just like Harmonic distortion (Refer Fig. 4.6).
In **Direct Plot > Main Form**, select **THD** as the function.
Choose Fundamental frequency from the frequency sweep list.
Click on Output net to plot THD.
Plot of percentage THD over input power appears (Refer Fig. 4.5).

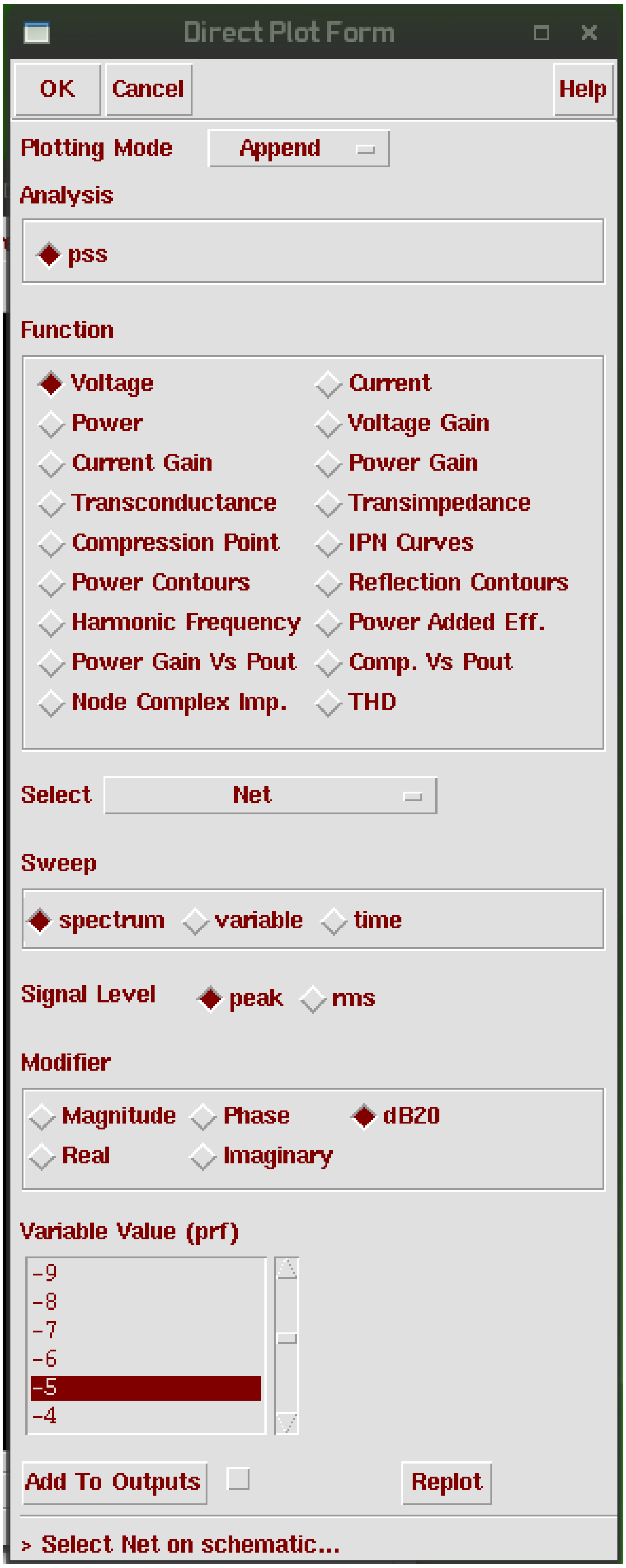


Figure 4.4: Harmonic Distortion plotting window

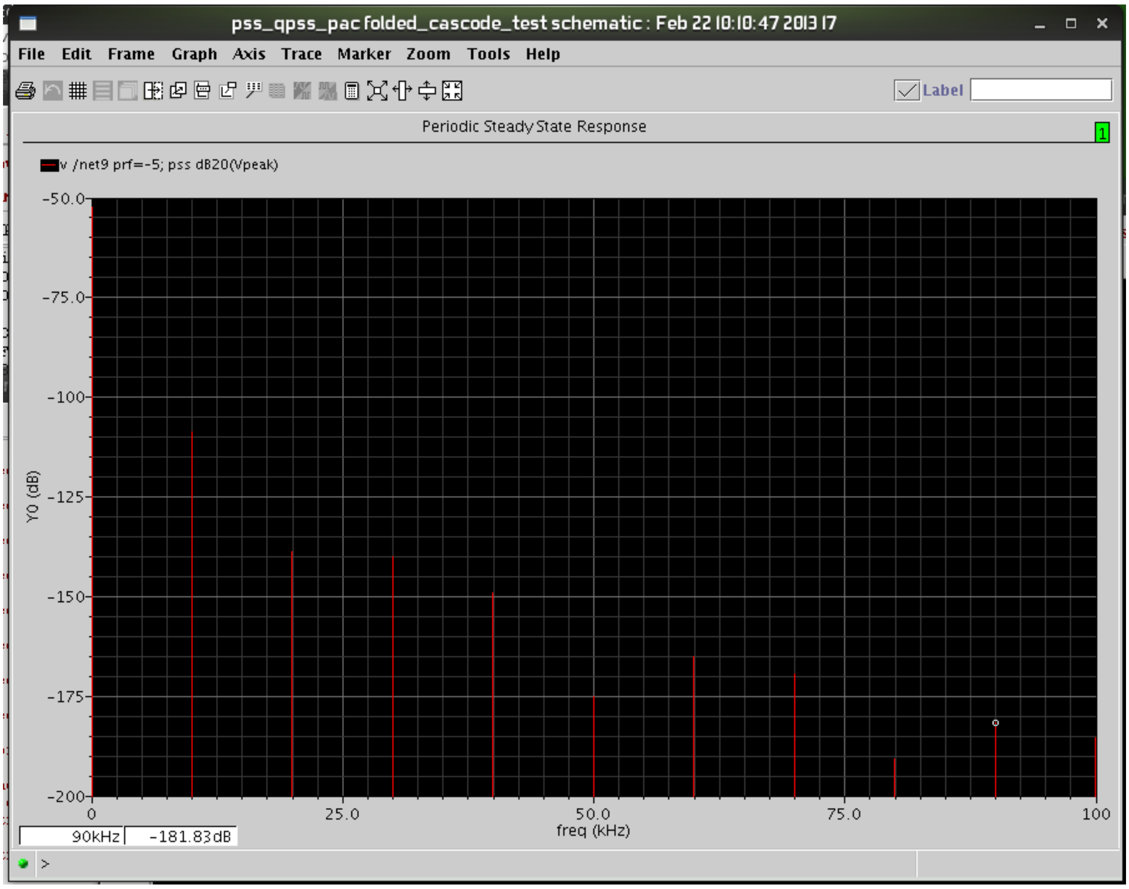


Figure 4.5: Harmonic Distortion output

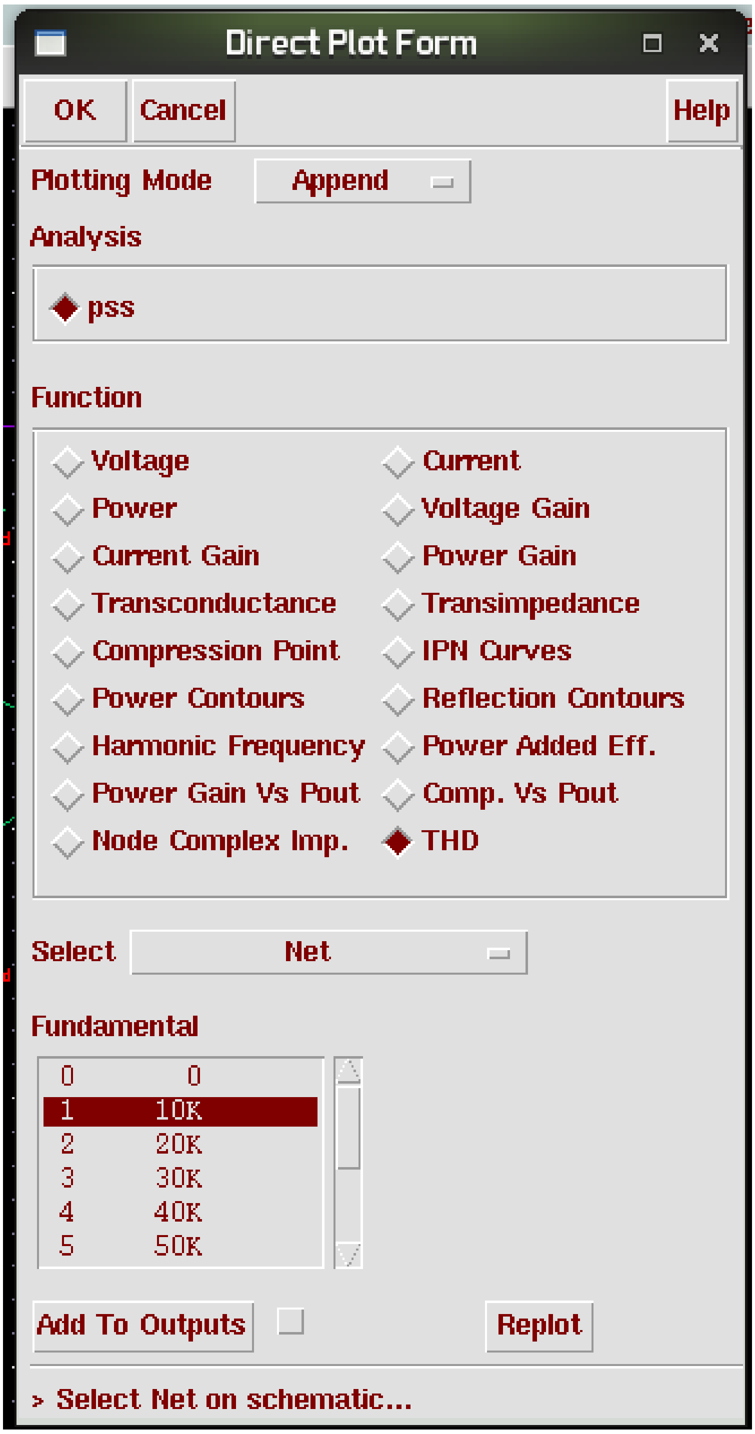


Figure 4.6: Total Harmonic Distortion plotting window

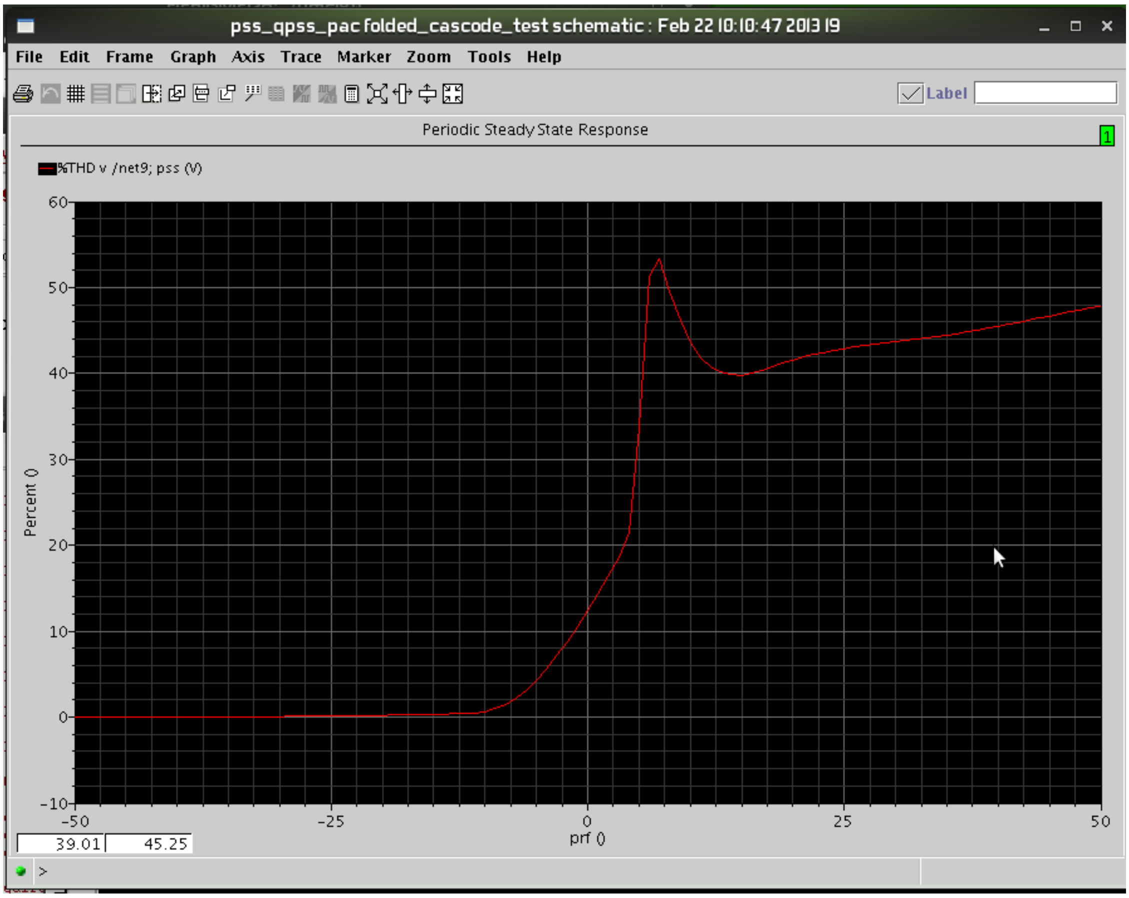


Figure 4.7: Total Harmonic Distortion output

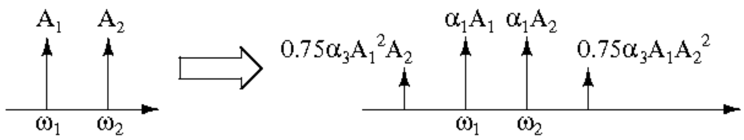
CHAPTER

FIVE

IP3 MEASUREMENT (PSS AND PAC)

5.1 What is IP3

IP3 is defined as the cross point of the power for the first order tones, ω_1 and ω_2, and the power for the third order tones $2\omega_1 - \omega_2$ and $2\omega_2 - \omega_1$ on the load



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

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

In this method, we first treat one signal, say ω_1 as large signal and perform pss analysis on it. The other tone, say ω_2, is treated as small signal and PAC analysis is performed based on linear time invariant systems obtained after PSS.

The IP3 point is the intercept between the power of the signal ω_2 and power of the signal $2\omega_1 - \omega_2$. Since the magnitude of this component is $0.75\alpha_3 A_1^2 A_2$, it has linear relationship with power level of ω_2. Thus ω_2 can be treated as small signal. It is necessary to set power level of both tones the same.

5.2 Setup PSS and PAC analysis

Follow these steps to setup PSS and PAC analysis.

1. Setup test bench with *port* as input and output instance. For input port
   Resistance = 50 ohms
   Port num = 1
   DC volt = 0.9 V or Vdd/2
   source type = sine
   frequency name 1 = rf
   freq1 = frf

Amplitude 1 (dBm) = prf

Select "Display small signal parameters"
pac magnitude (dBm) = prf

2. Start ADE and copy variables from schematic. Enter value of frf and prf. frf is ω_1 which is the fundamental frequency. Enter prf in negative range say -50 (typically).

3. Setup pss analysis as discussed in earlier chapters. Auto calculate beat frequency (equal to value of frf put in ADE). This is ω_1.
Set accuracy to moderate.
Activate sweep. Sweep power prf from say -50 to -5 dBm in say 20 steps.
click OK.

4. Setup PAC analysis.
Enter input frequency sweet range. Choose single point and give value of ω_2.
ω_2 should be very slightly larger than ω_1 so that $2\omega_1 - \omega_2$ is nearly ω_1.
Maximum sideband = 2 (for 3rd order)

5. Simulate. Results > Direct Plot > Main Form.

5.3 Plotting IPN Curves

Select the following under Direct Plot > Main form.
Analysis = PAC
Function = IPN Curves
Drop down menu Select - Port (Fixed R(port))
Circuit Input Power - Variable Sweep ("prf")
Input power extrapolation point (dBm) = -40
Select "Input referred IP3" in drop down menu
Under 3rd order harmonic, select the frequency which is equal to $2\omega_1 - \omega_2$. i.e. If 10K was entered as frf and 10.025K as the frequency in pac, then 3rd order harmonic becomes 9.975K.
Under First order harmonic select the frequency equal to value of ω_2 or 10.025K in this case.
Click on output port to plot the input referred IP3 in dBm.

CHAPTER

SIX

IP3 AND IM3 MEASUREMENT (QPSS)

This method treats both the tones ω_1 and ω_2 as large signal and performs QPSS analysis.

Both the methods, PSS with PAC and QPSS, are equivalent because of linear dependence of output components magnitude $2\omega_1 - \omega_2$ on the input component magnitude ω_2.

However, for IP3, recommended method is PSS with PAC analysis because it is more efficient than QPSS.

6.1 Setup QPSS Analysis

Follow the given steps to setup QPSS analysis to plot IP3 and IM3.

1. Setup test bench with *port* as input and output instance. For input port
   Resistance = 50 ohms
   Port num = 1
   DC volt = 0.9 V or Vdd/2
   source type = sine
   frequency name 1 = rf
   freq1 = frf
   Amplitude 1 (dBm) = prf
   frequency name 2 = rf2
   freq2 = frf + df (fd is a small delta f)
   amplitude 2 (dBm) = prf

2. in ADE, copy variable from schematic and set values.
   prf = -50 (say)
   frf = 100K (say)
   df = 0.2k (say)

3. Setup QPSS analysis. Refer Fig. 6.1
   Set frf1 to large signal and frf2 to moderate, since Only one large signal is allowed in QPSS
   Set accuracy to Moderate.

4. Simulate and choose Direct Plot.

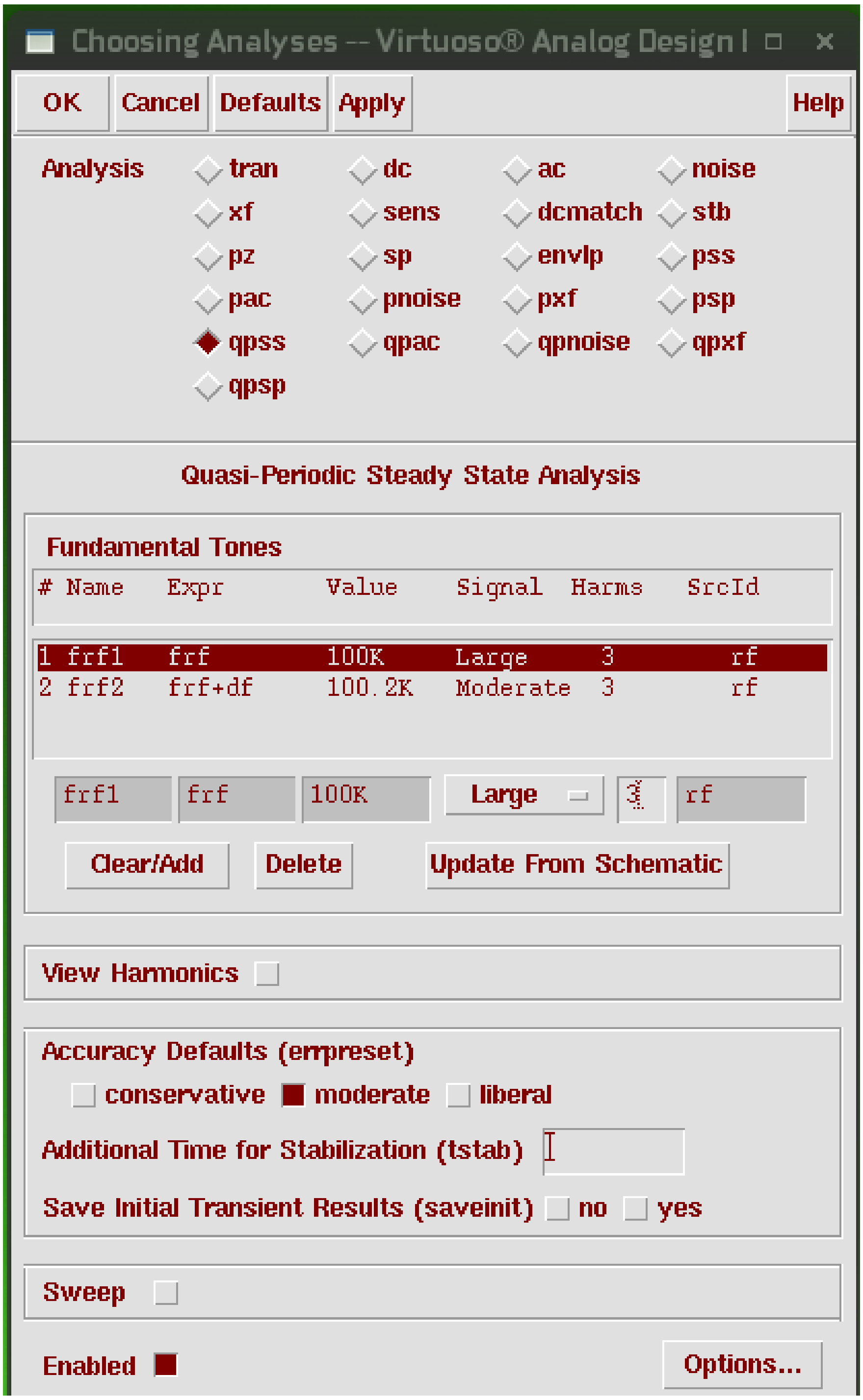


Figure 6.1: QPSS analysis setup

6.2 Plotting IP3 Curves

Under Direct Plot > Main form, select Analysis = QPSS
Function = IPN Curves
Select = Port (fixed R (port))
Single value input power value = -40 (say)
From drop down menu, select: Input referred IP3 ; 3rd order

In 3rd order harmonic we need to select $2\omega_1 - \omega_2$
i.e. 2rf1 - rf2
In this example = 2x100K - 100.2K = 99.8K

In first order harmonic we select ω_2, i.e. 100.2K.

Click on output port to plot IP3 curves

6.3 Plotting IM3 Spectrum

The analysis and simulation required for IM3 is same as IP3 using QPSS analysis. Perform the above same analysis and select the following in Direct Plot > Main form option.
Analysis = QPSS
Function = Power
Select = Port (fixed R (port))
Sweep = Spectrum
Modifier = dB10 or dBm
Click on output port to plot

The plot gives the IM3 power plots for the circuit.

CHAPTER
SEVEN

CORNER ANALYSIS

Corner Analysis can be performed using the Analog Corner Analysis tool inside Analog Design Environment. Corner analysis is capable of checking the circuit performance at all the process corners with control over temperature and any other variable at the same time. Running Corner analysis successfully requires some knowledge about the model library files of the technology under use. The designer must know the path where the model files are stored and also the corners provided by the foundry. The corners can be found by reading the library files. They are typically marked by sections in the model library file.

7.1 Locate Your Model Libraries

The technology files are typically saved in a folder named *designkits* in the cadence installation directory. In a typical UNIX system, designkit foler can be found in **\cad\cadence\designkits**. Inside this directory, more folders can be found corresponding to available technologies. Inside each of the technology folder, a directory named **Models** can be found. Inside this, model libraries for Spectre and HSPICE can be found in different folders. Spectre model files are typically suffixed with extention .scs. Thus a sample path to model libraries may be as

\<cadence_install_dir>\designkits\<technology_dir>\Models\Spectre

or

\cad\cadence\designkits\UMC180\Models\Spectre

In this path, *.scs files can be found which contains the device parameters included in the technology. Find out of the file which contains the MOSFET parameters. In UMC180, the MOSFET parameters are given in file MM180_REG18_V124.mdl.scs and .lib.scs. MDL file contains all BSIM parameters and LIB file contains the variation in these parameters over the process corners.

7.2 Know Your Process Corners

The five process corners usually available are given in Table 7.1.

Thus, tt refers to a corner where NMOS and PMOS, both exhibit typical characteristics. The value of process parameters at different corners can be read from the model library file included in spectre model library list.

Table 7.1: Process Corners in UCM 180 nm

NMOS	**PMOS**	**Corner**
Typical	Typical	**tt**
Fast	Fast	**ff**
Slow	Slow	**ss**
Slow	Fast	**snfp**
Fast	Slow	**fnsp**

Comment Specific to UMC 180 nm: In UMC 180, the model library for MOSFETS are broken into two files. One file, *.mdl.scs, lists all the BSIM parameters including some delta variations in few parameters which are corner dependent. Another file *.lib.scs contains the values of these delta variations for all the five corners. For Example, in mdl file, the threshold voltage is given as vth0=Some constant + dvth0. Now, dvth0 takes different values at the different corners, which are given in lib.scs file. The different corner definitions are written in different sections of the lib.scs file.

7.3 Running Corner Analysis

It is required to first setup ADE for running the analysis over which corner analysis is to be run. For example, if the designer wants to check the UGF of the amplfier at different corners, he should set up AC analysis and create expression to print/plot the UGF of the circuit. Run normal simulation once to check if everything works.

Now open Corner analysis window by clicking **Tools** > **Corners**, in ADE window. A window opens up. We first need to add a process, in this example UMC180.

In corner analysis window, click **Setup** > **Add Process**. In Add Process window which opens up, enter a logical name to refer to the process, say UMC. In Model Style, select Single Model Library if definitions of both NMOS and PMOS are in same model file. If not select Multiple Model file. For UMC180, since both NMOS and PMOS parameters are in same model file, select Single Model Library. For base directory give the complete path to the directory where spectre model files are saved. for example
\cad\cadence\designkits\UMC180\Models\Spectre

Under model file, given the name of the file where process corner definitions are given (lib.scs file). For example MM180_REG18_V124.mdl.scs. Process variables can be left blank and click OK.

It will now be seen that process is added to Corner analysis window. Now we need to add the corners and any veriables if required. To add corners, click **Edit** > **Corner Definitions** > **Add Corner**. In Enter Corner Name window which pops up, enter the corner **(tt,ff,ss,fnsp,snfp)** and click OK. It would be seen that a corner is added. Similarly we can add the remaining four corners. A veriable called Temp (temperature) is preadded. Assign some value to this for every corner. If you wish to vary any other variable over the five corners, that can be added by clicking **Add Variable** button. Corners can also be added by clicking **Add Corner** button. The performance measurements are taken up from the outputs setup in ADE. Once done, click **Run** to start Corner Analysis. Under performance measurement, you can select plot or print as your output mode. A typical corner analysis window after setup is shown in Fig. 7.1. Outputs are shown in Fig. 7.2 and 7.3.

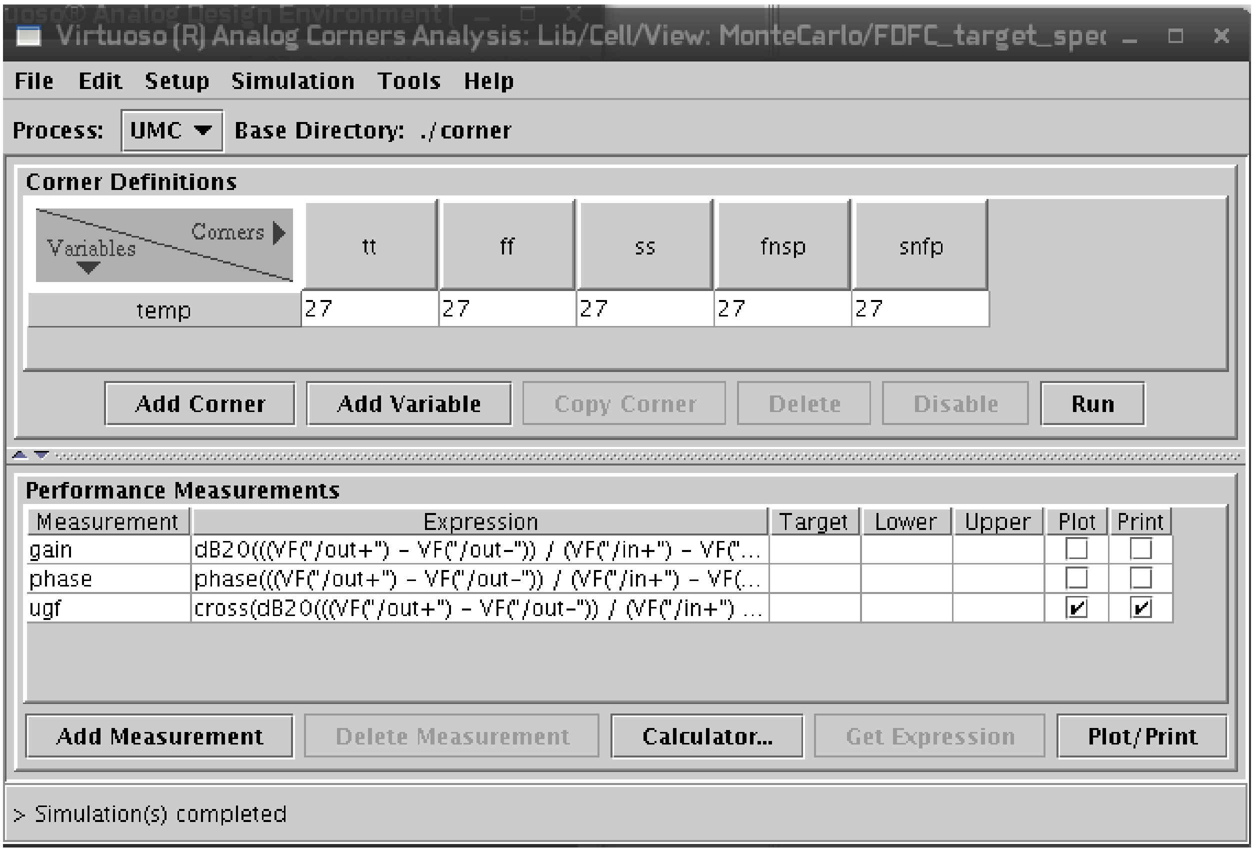


Figure 7.1: Corner Analysis Window

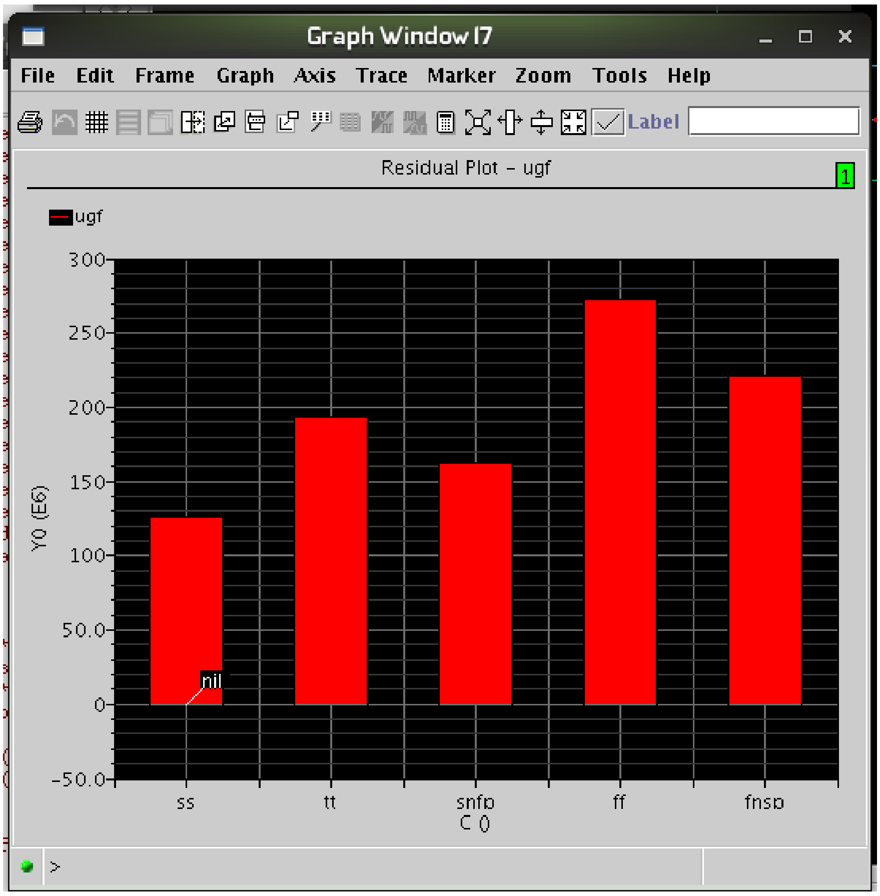
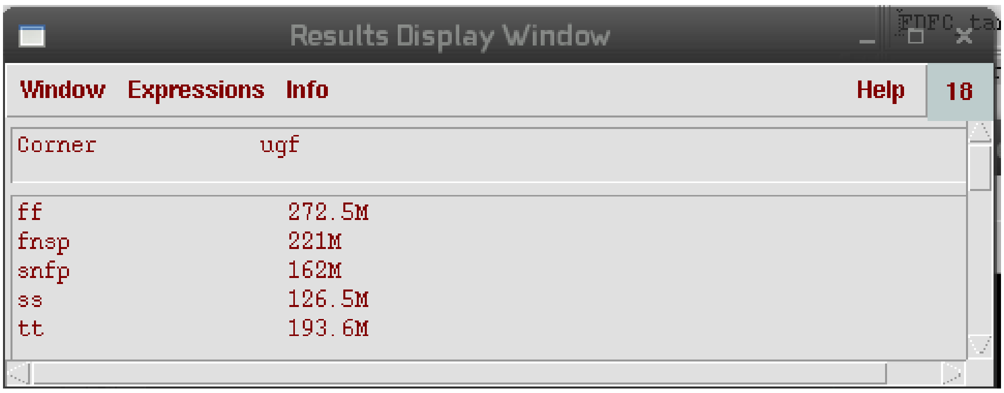


Figure 7.2: A plot from corner analysis



Corner	ugf
ff	272.5M
fnsp	221M
snfp	162M
ss	126.5M
tt	193.6M

Figure 7.3: Result of Corner Analysis in table (print) form

CHAPTER

EIGHT

MONTE CARLO ANALYSIS

The Monte Carlo analysis is a swept analysis with associated child analyses similar to the sweep analysis. The Monte Carlo analysis refers to "statistics blocks" where statistical distributions and correlations of netlist parameters are specified. For each iteration of the Monte Carlo analysis, new pseudo-random values are generated for the specified netlist parameters (according to their specified distributions) and the list of child analyses (like DC gain or unity gain frequency or slew rate of an amplifier) are then executed.

Expressions are associated with the child analyses. These expressions, which are constructed as scalar calculator expressions by the user during Monte Carlo analysis set up, can be used to measure circuit metrics, such as the slew-rate of an op-amp. During a Monte Carlo analysis, these expression results will vary as the netlist parameters vary for each Monte Carlo iteration. The Monte Carlo analysis therefore becomes a tool that allows you to examine and predict circuit performance variations, which affect yield.

The statistics blocks allow you to specify batch-to-batch (process) and per-instance (mismatch) variations for netlist parameters. These statistically-varying netlist parameters can be referenced by models or instances in the main netlist and may represent IC manufacturing process variation, or component variations for board-level designs for example.

8.1 Key Requirements to Perform Monte Carlo Simulation

The Monte Carlo Simulation requires some understanding of the model libraries, the associated files and the device parameters involved. The example covered here is given with respect to UMC 180 nm CMOS technology. Same steps are to be followed even with other technologies. However, correct model file has to be identified.

The key requirements for running a Monte Carlo (MC) simulation are:

1. Circuit designed in Virtuoso and its functionality verified.
2. Analysis setup for which Monte Carlo is to be carried out. For example, set up AC analysis and setup required output if you wish to perform MC simulation to plot the unity gain frequency of an amplifier.
3. Understanding model file being used for simulation.
4. Creation of Monte Carlo library

5. Running Monte Carlo Simulation

8.2 Writing and Including Libraries

The Monte Carlo library is made up of three sections or parts

1. Parameter section
2. Statistical section
3. Device model section

8.2.1 Parameter Section

The parameter section contains a definition of all the parameters which may be varied in Monte Carlo Simulation. The initial of the typical value of the parameter is also given here. It is not necessary that all the parameters mentioned in this section should be varied.

Syntax of declaring a parameter is given below. The words in italics are keywords and words inside <, > are to be replaced.

parameters **<parameter name>=<mean value>**

<parameter name> is replaced by the name of a parameter and its mean value is given on the right hand side of the equal to sign.

A typical example of parameter section can be given as follows:

```
section param
       simulator lang=spectre
              parameters dvth0_n_18_mm=0
              parameters dvth0_p_18_mm=0
              <More parameters>
endsection param
```

Here, the initial or the mean values of two parameters are defined as zero. Now, we are in position to vary these two parameters statistically.

8.2.2 Statical Section

The statistical section gives a list of parameters which are varied in Monte Carlo Simulatiom. It details the distribution type and the deviation about the mean position in the form on absolute value or in percentage form.

The syntax of declaring statistical section is

vary **<parameter name>** ***dist=*****<distribution type> {*****std=*****<deviation> |*****N=*****<value> }** ***percent=*****<no or yes>**

A typical example of statistical section can be given as follows:

```
section stats
       simulator lang=spectre

       process {
               vary dvth0_n_18_mm dist=gauss std=0.1 percent=no
```

```
                vary dvth0_p_18_mm dist=gauss std=0.1 percent=no
                <More lines to vary parameters>
                }

        mismatch{
                vary dvth0_n_18_mm dist=gauss std=0.1 percent=no
                vary dvth0_p_18_mm dist=gauss std=0.1 percent=no
                <More lines to vary parameters>
                }

endsection stats
```

The parameters listed inside the process field are varied once per Monte Carlo Simulation. Whereas, parameters listed inside mismatch field are varied per every instance in the circuit.

8.2.3 Model Section

This section gives all the remaining process/device parameters necessary to carry out any simulation. Care should be taken that the parameters mentioned in param and stats section should not appear in models section. If this is not taken care, MC simulation may not run.

NOTE: When including these three sections in the model library list inside Analog Design Environment, make sure param section in included first and model section in included last. It simply means that in the list of model libraries, param should be on top and model should be at the bottom. A snapshot is shown in Fig. 8.1.

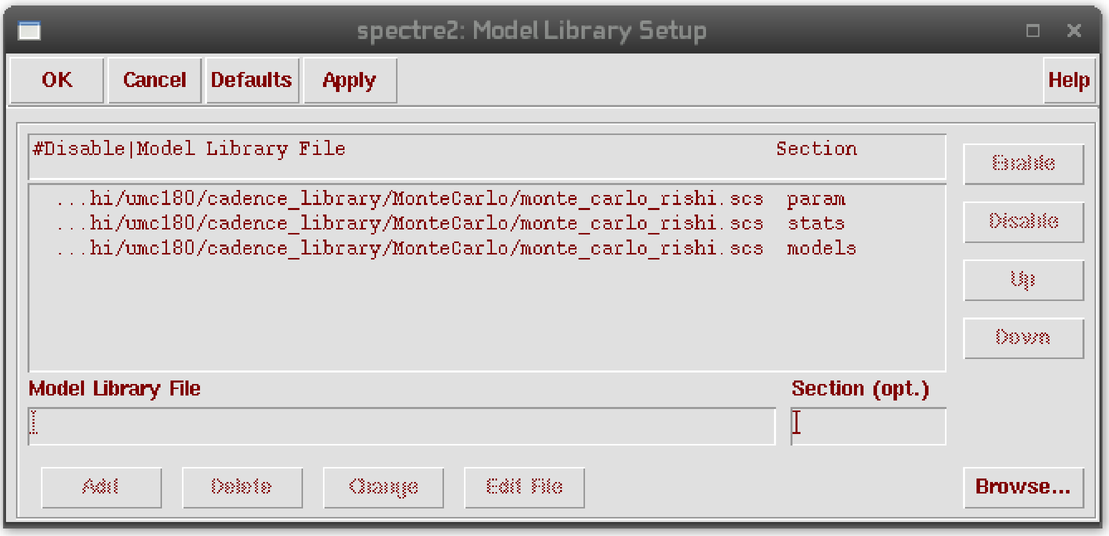


Figure 8.1: Snapshot of Model Libraries Window

A complete Monte Carlo library file is given at the end of this chapter.

8.3 Running Monte Carlo Simulation

Start virtusoso and set up the circuit for which MC simulation in to be done. Set up Analog Design Environment (ADE) with all the analysis (child analysis, say ac or transient or any other analysis) and outputs which you wish to observe after MC simulation.

Let us consider we wish to run MC simulation and plot the unity gain frequency (UGF) of an amplifier. We therefore setup an AC analysis in ADE and setup the expression for getting a scalar value of UGF. A snapshot of ABE is given in Fig. 8.2. Additional outputs for getting bode plot are also setup in this image.

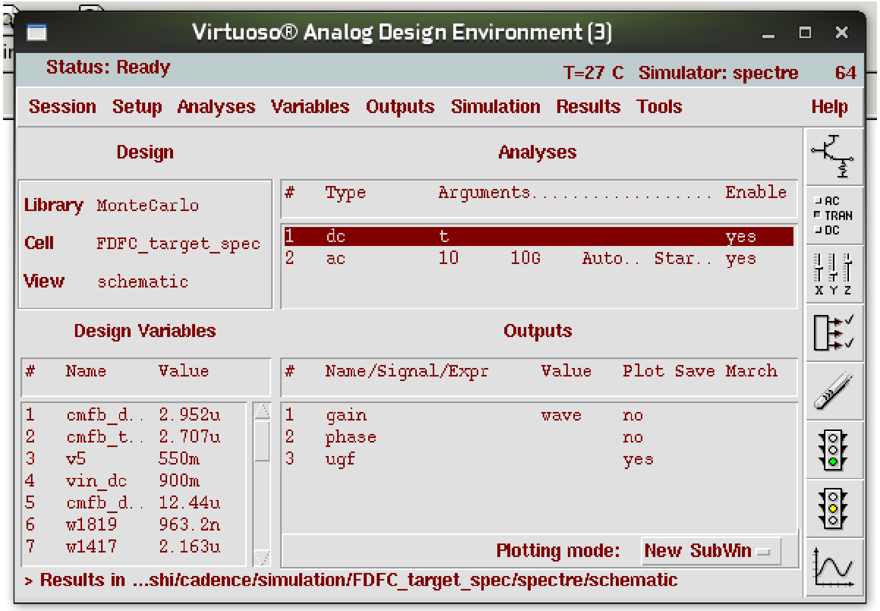


Figure 8.2: Snapshot of Analog Design Environment

Once this is set up, it is recommended to run normal simulation (netlist and run) just to clarify that model libraries are properly setup and all other setting in ADE is correct. If normal simulation runs successfully, means now Monte Carlo Analysis can be run.

Now Monte Carlo Analysis window is invoked by clicking **Tools** > **Monte Carlo** in ADE window. In the window that appears, we need to select whether we wish to vary only Process or only mismatch or both. The number of runs can be left as 100 starting at 1. Options to save data can be marked. Make sure output expressions appear on the outputs section. If not, they can be brought from ADE by choosing **Outputs** > **Retrieve Outputs**. Make sure the plot option is set to yes for the output we wish to plot. A snapshot is given in Fig. 8.3.

We can now run Monte Carlo Analysis by Clicking **Simulation** > **Run**. MC analysis may take few minutes to run depending on your circuit, number of parameters varied and number of runs. A possible output of monte carlo simulation for possible values of UGF is shown in Fig. 8.4.

8.4 Additional Information

The above given example is good for getting the Monte Carlo Simulation running. However, there are several other options which can be explored.

8.4.1 Specifying Distributions

Three different distribution types are supported by Spectre.

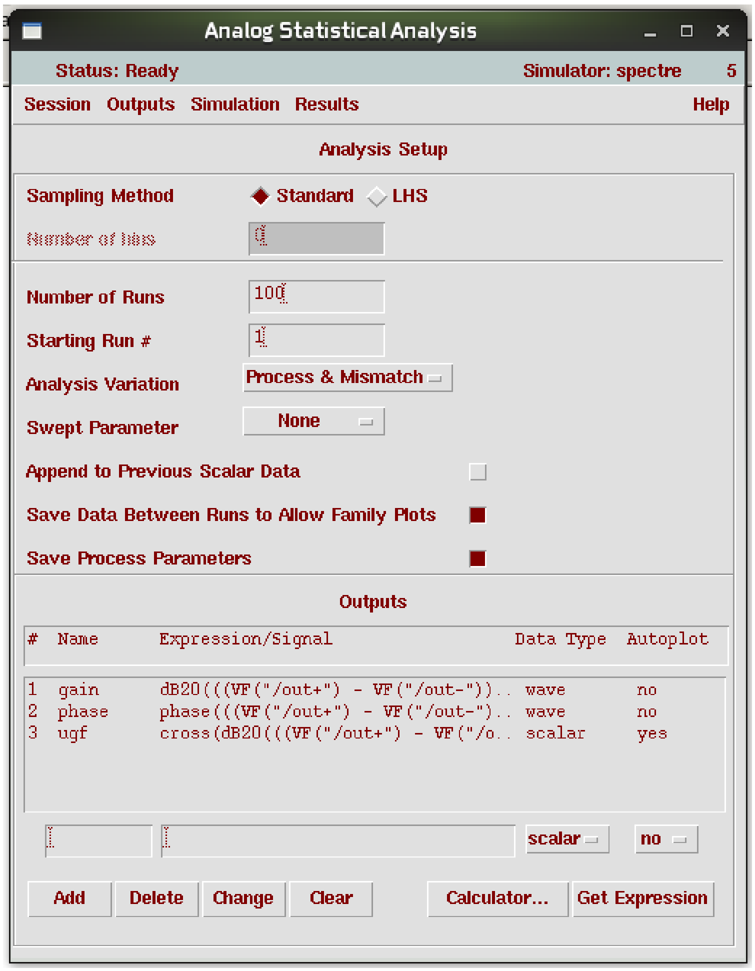


Figure 8.3: Snapshot of Monte Carlo Setup Window

1. Gaussian Distribution represented by keyword **gauss**
2. Uniform Distribution represented by keywork **unif**
3. Lognormal Distribution represented by keywork **lnorm**

Few of the possible examples/ways to represent the variation is given below

```
vary dvth0_n_18_mm dist=unif N=10 percent=yes \\Deviation in percent
vary dvth0_n_18_mm dist=lnorm std=12 percent=no \\lognormal distribution
```

With the nuniform distribution, standard deviation is not specified but instead N is defined. The range of of deviation is set by ± N. Thus, x varies from mean-N to mean+N where mean value of x is defined in parameters section.

x=unif(mean-N,mean+N)

The *percent* flag indicates whether the standard deviation *std* or uniform range *N* are specified in absolute terms (*percent=no*) or as a percentage of the mean value (*percent=yes*). The line

vary dvth0_n_18_mm dist=unif N=10 percent=yes

means that the parameter dvth0_n_18_mm is varied by 10% above and below its mean value with uniform distribution. If mean value of dvth0_n_18_mm = 100, then dvth0_n_18_mm is varied as (100±10%), i.e. from 90 to 110.

NOTE: It is not advised to use percent with lognormal distribution.

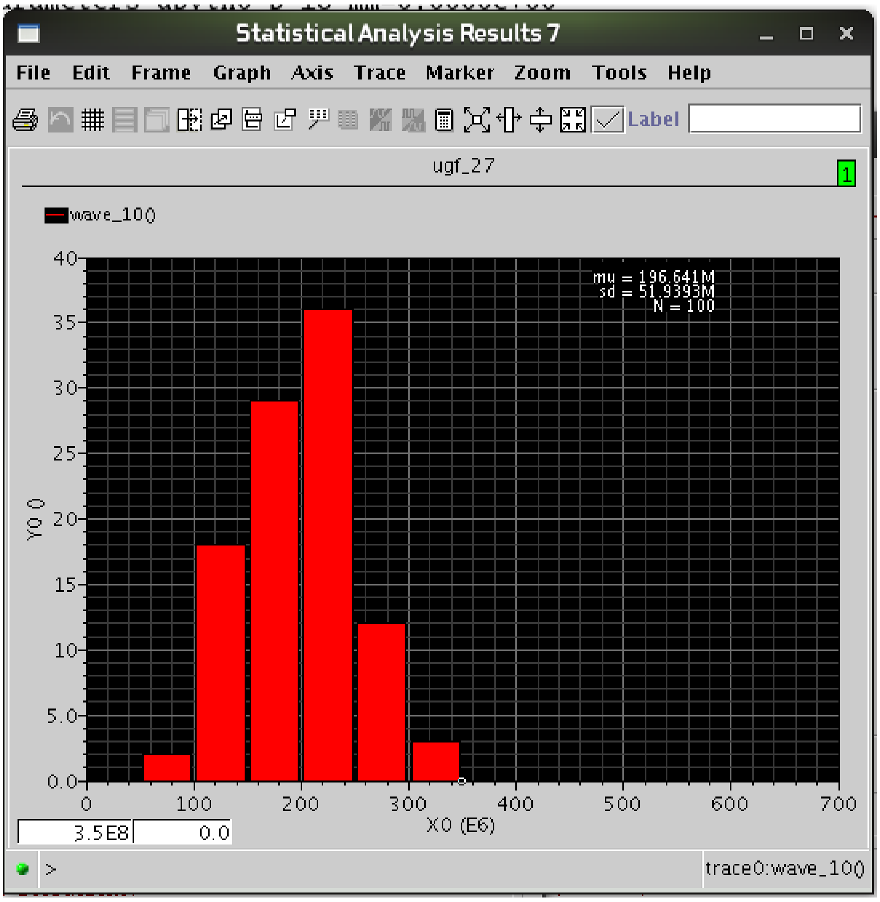


Figure 8.4: Possible output of Monte Carlo Simulation

8.4.2 Correlation Statements

There are two types of correlation statements that you can use:

- Process parameter correlation statements
- Instance correlation statements

Process Parameter Correlation

The syntax of the process parameter correlation statement is:
***correlate param*=[<list of parameters>] *cc*=<value>**

This allows us to specify a correlation coefficient between multiple process parameters. We can specify multiple process parameter correlation statements in a statistics block, to build a matrix of process parameter correlations. During a Monte Carlo analysis, process parameter values will be randomly generated according to the specified distributions and correlations.

The process parameter correlation is written inside the same statistics block where process and mismatch distribution is given. The following lines may be added after closing the brackets for mismatch distibution. For example:
correlate param=[dvth0_n_18_mm dvth0_n_18_mm] cc=0.6

Mismatch Correlation (Matched Devices)

The syntax of the instance or mismatch correlation statement is:
correlate dev=[**<list of subcircuit instances>**] ***param***=[**<list of parameters>**] ***cc***=**<value>**

where the device or subcircuit instances to be matched are listed in the list of subcircuit instances, and the list of parameters specifies exactly which parameters with mismatch variations are to be correlated.

The instance mismatch correlation statement is used to specify correlations for particular subcircuit instances. If a subcircuit contains a device, you can effectively use the instance correlation statements to specify that certain devices are correlated (i.e. matched) and give the correlation coefficient. You can optionally specify exactly which parameters are to be correlated by giving a list of parameters (each of which must have had distributions specified for it in a mismatch block), or specify no parameter list, in which case all parameters with mismatch statistics specified are correlated with the given correlation coefficient. The correlation coefficients are specified in the ¡value¿ field and must be between ± 1.0, not including 1.0 or -1.0.

The device correlation is written in a separate statistics block from one constituting distribution and process correlation, but inside the same section (stats). For example

```
statistics {
           correlate dev=[M1 M2] param=[dvth0_n_18_mm dvth0_n_18_mm] cc=0.8
           }
```

NOTE: correlation coefficients can be constants or expressions, as can *std* and *N* when specifying distributions.

8.5 Sample Monte Carlo Library File

```
library monteLib



section param
simulator lang=spectre
        parameters dtox_n_18_mm=0.0000e+00
        parameters dxl_n_18_mm=0.0000e+00
        parameters dxw_n_18_mm=0.0000e+00
        parameters dvth0_n_18_mm=0.0000e+00
        parameters du0_n_18_mm=0.0000e+00
        parameters dlvth0_n_18_mm=0.0000e+00
        parameters dwvth0_n_18_mm=0.0000e+00
        parameters dwu0_n_18_mm=0.0000e+00
        parameters dpvth0_n_18_mm=0.0000e+00
        parameters dpvsat_n_18_mm=0.0000e+00
        parameters dcgdo_n_18_mm=0.0000e+00
        parameters dcgso_n_18_mm=0.0000e+00
        parameters dcj_n_18_mm=0.0000e+00
        parameters dcjsw_n_18_mm=0.0000e+00
        parameters dcjgate_n_18_mm=0.0000e+00
```

```
        parameters dtox_p_18_mm=0.0000e+00
        parameters du0_p_18_mm=0.0000e+00
        parameters dxl_p_18_mm=0.0000e+00
        parameters dxw_p_18_mm=0.0000e+00
        parameters dvth0_p_18_mm=0.0000e+00
        parameters dlvth0_p_18_mm=0.0000e+00
        parameters dwvth0_p_18_mm=0.0000e+00
        parameters dpvth0_p_18_mm=0.0000e+00
        parameters dpvsat_p_18_mm=0.0000e+00
        parameters dcgdo_p_18_mm=0.0000e+00
        parameters dcgso_p_18_mm=0.0000e+00
        parameters dcj_p_18_mm=0.0000e+00
        parameters dcjsw_p_18_mm=0.0000e+00
        parameters dcjgate_p_18_mm=0.0000e+00
endsection param



section stats
simulator lang=spectre
statistics {
        process  {
                 vary dvth0_n_18_mm dist=gauss std=0.1 percent=no
                 vary dvth0_p_18_mm dist=gauss std=0.1 percent=no
                 }
        mismatch {
                 vary dvth0_n_18_mm dist=gauss std=0.1 percent=no
                 vary dvth0_p_18_mm dist=gauss std=0.1 percent=no
                 }
           }
endsection stats



section models
simulator lang=spectre
model n_18_mm bsim3v3 type=n
+ version=3.2000e+00        binunit=1.0000e+00        mobmod=1.0000e+00
+ capmod=2.0000e+00         nqsmod=0.0000e+00
+ tox=4.2000e-09 + dtox_n_18_mm                       toxm=4.2000e-09
+ xj=1.6000e-07             nch=3.7446e+17            rsh=8.0000e+00
+ ngate=1.0000e+23          vth0=3.0750e-01 + dvth0_n_18_mm
+ k1=4.5780e-01             k2=-2.6380e-02            k3=-1.0880e+01
+ k3b=2.3790e-01            w0=-8.8130e-08            nlx=4.2790e-07
+ dvt0=4.0420e-01           dvt1=3.2370e-01           dvt2=-8.6020e-01
+ dvt0w=3.8300e-01          dvt1w=6.0000e+05          dvt2w=-2.5000e-02
+ lint=1.5870e-08           wint=1.0220e-08           dwg=-3.3960e-09
+ dwb=1.3460e-09            u0=3.1410e+02 + du0_n_18_mm
```

```
+ ua=-9.2010e-10           ub=1.9070e-18            uc=4.3550e-11
+ vsat=7.1580e+04          a0=1.9300e+00            ags=5.0720e-01
+ b0=1.4860e-06            b1=9.0640e-06            keta=1.7520e-02
+ a1=0.0000e+00            a2=1.0000e+00            voff=-1.0880e-01
+ nfactor=1.0380e+00       cit=-1.5110e-03          cdsc=2.1750e-03
+ cdscd=-5.0000e-04        cdscb=8.2410e-04         eta0=1.0040e-03
+ etab=-1.4590e-03         dsub=1.5920e-03          pclm=1.0910e+00
+ pdiblc1=3.0610e-03       pdiblc2=1.0000e-06       pdiblcb=0.0000e+00
+ drout=1.5920e-03         pscbe1=4.8660e+08        pscbe2=2.8000e-07
+ pvag=-2.9580e-01         rdsw=4.9050e+00          prwg=0.0000e+00
+ prwb=0.0000e+00          wr=1.0000e+00            alpha0=0.0000e+00
+ alpha1=0.0000e+00        beta0=3.0000e+01         xpart=1.0000e+00
+ cgso=2.3500e-10 + dcgso_n_18_mm
+ cgdo=2.3500e-10 + dcgdo_n_18_mm                   cgbo=0.0000e+00
+ cgsl=0.0000e+00          cgdl=0.0000e+00          ckappa=6.0000e-01
+ cf=1.5330e-10            clc=1.0000e-07           cle=6.0000e-01
+ dlc=2.9000e-08           dwc=0.0000e+00           vfbcv=-1.0000e+00
+ noff=1.0000e+00          voffcv=0.0000e+00        acde=1.0000e+00
+ moin=1.5000e+01          lmin=1.8000e-07          lmax=5.0000e-05
+ wmin=2.4000e-07          wmax=1.0000e-04
+ xl= - 1.0500e-08 + dxl_n_18_mm
+ xw=0.0000e-00 + dxw_n_18_mm                       js=1.0000e-06
+ jsw=7.0000e-11           cj=1.0300e-03 + dcj_n_18_mm
+ mj=4.4300e-01            pb=8.1300e-01
+ cjsw=1.3400e-10 + dcjsw_n_18_mm                   mjsw=3.3000e-01
+ tnom=2.5000e+01          ute=-1.2860e+00          kt1=-2.2550e-01
+ kt1l=-4.1750e-09         kt2=-2.5270e-02          ua1=2.1530e-09
+ ub1=-2.6730e-18          uc1=-3.8320e-11          at=1.4490e+04
+ prt=-1.0180e+01         xti=3.0000e+00           wl=0.0000e+00
+ wln=1.0000e+00           ww=7.2620e-16            wwn=1.0000e+00
+ wwl=0.0000e+00           ll=-1.0620e-15           lln=1.0000e+00
+ lw=2.9960e-15            lwn=1.0000e+00           lwl=0.0000e+00
+ llc=-2.1400e-15          lwc=0.0000e+00           lwlc=0.0000e+00
+ wlc=0.0000e+00           wwc=0.0000e+00           wwlc=0.0000e+00
+ lvth0= - 1.0000e-03 + dlvth0_n_18_mm
+ wvth0=6.027e-02 + dwvth0_n_18_mm                  pvth0=0 + dpvth0_n_18_mm
+ lnlx=-2.8540e-08         wnlx=0.0000e+00          pnlx=0.0000e+00
+ wua=-1.8800e-11          wu0=5.4000e-01 + dwu0_n_18_mm
+ pub=3.8000e-20           pw0=1.3000e-09           wrdsw=0.0000e+00
+ weta0=0.0000e+00         wetab=0.0000e+00         leta0=1.5740e-03
+ letab=0.0000e+00         peta0=0.0000e+00         petab=0.0000e+00
+ wpclm=0.0000e+00         wvoff=-4.0780e-04        lvoff=-4.2080e-03
+ pvoff=-3.7880e-04        wa0=-4.7310e-02          la0=-4.6670e-01
+ pa0=-2.6490e-02          wags=4.2420e-03          lags=3.0280e-01
+ pags=0.0000e+00          wketa=0.0000e+00         lketa=-1.9420e-02
+ pketa=0.0000e+00         wute=6.3730e-02          lute=0.0000e+00
+ pute=0.0000e+00          wvsat=5.0660e+03         lvsat=0.0000e+00
+ pvsat=0.0000e+00 + dpvsat_n_18_mm                 lpdiblc2=-4.7520e-03
```

```
+ wat=7.0670e+03            wprt=0.0000e+00            ldif=8.0000e-08
+ hdif=2.6000e-07           n=1.0000e+00               pbsw=8.8000e-01
+ cjswg=5.0000e-10 + dcjgate_n_18_mm                   ctp=9.1400e-04
+ ptp=9.2400e-04            cta=9.1900e-04             pta=1.5800e-03
+ elm=5.0000e+00            tlevc=1.0000e+00
+ noimod=2                  noia=1.3182567385564E+19   noib=144543.977074592
+ noic=-1.24515794572817E-12                           ef=0.92
+ em=41000000



model p_18_mm bsim3v3 type=p
+ mobmod=3.0000e+00         version=3.2000e+00         capmod=2.0000e+00
+ binunit=1.0000e+00        nqsmod=0.0000e+00
+ tox=4.2000e-09 + dtox_p_18_mm                        toxm=4.2000e-09
+ xj=1.0000e-07             nch=6.1310e+17             ngate=1.0000e+23
+ vth0= - 4.5550e-01 + dvth0_p_18_mm                   k1=5.7040e-01
+ k2=6.9730e-03             k3=-2.8330e+00             k3b=1.3260e+00
+ w0=-1.9430e-07            nlx=2.5300e-07             dvt0=4.8850e-01
+ dvt1=7.5780e-02           dvt2=1.2870e-01            dvt0w=-1.2610e-01
+ dvt1w=2.4790e+04          dvt2w=6.9150e-01           lint=-1.0410e-08
+ wint=-1.5250e-07          dwg=-1.1510e-07            dwb=-1.0390e-07
+ u0=1.1450e+02 + du0_p_18_mm                          ua=1.5400e-09
+ ub=2.6460e-19             uc=-9.5870e-02             vsat=5.3400e+04
+ a0=1.3500e+00             ags=3.8180e-01             b0=-3.0880e-07
+ b1=0.0000e+00             keta=1.0440e-02            a1=0.0000e+00
+ a2=1.0000e+00             voff=-1.0730e-01           nfactor=1.5350e-00
+ cit=-1.0670e-03           cdsc=7.5780e-04            cdscd=-2.8830e-05
+ cdscb=1.0000e-04          eta0=1.0710e+00            etab=-9.2910e-01
+ dsub=1.9191e+00           pclm=2.6530e+00            pdiblc1=0.0000e+00
+ pdiblc2=5.0000e-06        pdiblcb=0.0000e+00         drout=1.4570e+00
+ pscbe1=4.8660e+08         pscbe2=2.8000e-07          pvag=1.1620e+00
+ rdsw=7.9210e+02           prwg=0.0000e+00            prwb=0.0000e+00
+ alpha0=0.0000e+00         alpha1=0.0000e+00          beta0=3.0000e+01
+ cgdo=2.0540e-10 + dcgdo_p_18_mm                      cgbo=0.0000e+00
+ cgso=2.0540e-10 + dcgso_p_18_mm                      xpart=1.0000e+00
+ cf=1.5330e-10             dlc=5.6000e-08             cgsl=0.0000e+00
+ cgdl=0.0000e+00           ckappa=6.0000e-01          clc=1.0000e-07
+ cle=6.0000e-01            dwc=0.0000e+00             vfbcv=-1.0000e+00
+ noff=1.0000e+00           voffcv=0.0000e+00          acde=1.0000e+00
+ moin=1.5000e+01           lmin=1.8000e-07            lmax=5.0000e-05
+ wmin=2.4000e-07           wmax=1.0000e-04
+ xl= - 2.0000e-09 + dxl_p_18_mm
+ xw=0.0000e+00 + dxw_p_18_mm                          js=3.0000e-06
+ jsw=4.1200e-11            cj=1.1400e-03 + dcj_p_18_mm
+ mj=3.9500e-01             pb=7.6200e-01
+ cjsw=1.7400e-10 + dcjsw_p_18_mm                      mjsw=3.2400e-01
+ tnom=2.5000e+01           ute=-4.4840e-01            kt1=-2.1940e-01
```

```
+ kt1l=-8.2040e-09           kt2=-9.4870e-03           ua1=4.5710e-09
+ ub1=-6.0260e-18            uc1=-9.8500e-02           at=1.2030e+04
+ prt=0.0000e+00             xti=3.0000e+00            ww=1.2360e-14
+ lw=-2.8730e-16             ll=6.6350e-15             wl=0.0000e+00
+ wln=1.0000e+00             wwn=1.0000e+00            wwl=0.0000e+00
+ lln=1.0000e+00             lwn=1.0000e+00            lwl=0.0000e+00
+ llc=-7.4500e-15            lwc=0.0000e+00            lwlc=0.0000e+00
+ wlc=0.0000e+00             wwc=0.0000e+00            wwlc=0.0000e+00
+ lvth0=4.4000e-03 + dlvth0_p_18_mm
+ wvth0= - 1.4800e-02 + dwvth0_p_18_mm
+ pvth0=3.2000e-03 + dpvth0_p_18_mm                    lnlx=-1.5840e-08
+ wrdsw=1.0070e+01           weta0=0.0000e+00          wetab=0.0000e+00
+ wpclm=0.0000e+00           wua=2.6300e-09            lua=-8.1530e-11
+ pua=5.8550e-11             wub=0.0000e+00            lub=0.0000e+00
+ pub=0.0000e+00             wuc=0.0000e+00            luc=0.0000e+00
+ puc=0.0000e+00             wvoff=-9.8160e-03         lvoff=-9.8710e-04
+ pvoff=-9.8330e-05          wa0=-4.8070e-02           la0=-2.8100e-01
+ pa0=8.6610e-02             wags=-4.1770e-02          lags=4.4540e-02
+ pags=-4.0760e-02           wketa=0.0000e+00          lketa=-1.2000e-02
+ pketa=0.0000e+00           wute=-2.6820e-01          lute=0.0000e+00
+ pute=0.0000e+00            wvsat=-1.4200e+04         lvsat=0.0000e+00
+ pvsat= - 4.3400e+02 + dpvsat_p_18_mm                 lpdiblc2=3.0120e-03
+ cjswg=4.200e-10 + dcjgate_p_18_mm                    wat=-6.4050e+03
+ wprt=2.1660e+02            n=1.0000e+00              pbsw=6.6500e-01
+ cta=1.0000e-03             ctp=7.5300e-04            pta=1.5500e-03
+ ptp=1.2400e-03             ldif=8.0000e-08           rsh=8.0000e+00
+ rd=0.0000e+00              rsc=0.0000e+00            rdc=0.0000e+00
+ hdif=2.6000e-07            rs=0.0000e+00
+ noimod=2                   noia=3.57456993317604E+18 noib=2500
+ noic=2.61260020285845E-11 ef=1.1388
+ em=41000000
endsection models

endlibrary monteLib
```