

SmartSpiceRF

Low Noise Amplifier Simulation

Tutorial

Introduction

Information presented in this Tutorial is based on the assumption that the user is familiar with the following:

- Basics of the computer operating system and hardware employed
- Basic terminology of semiconductor process and device operation
- Circuit design, schematic capture, and simulation

The simulation flow described in Tutorial uses Silvaco Process Design Kit (PDK) based on a fictional process called SBCD (Silvaco Bipolar Cmos Dmos). SBCD process includes the following devices: two low-voltage Cmos, two high-voltage Dmos, four resistors, capacitor, and inductor. SBCD libraries provide all the data needed to demonstrate a real design flow.

This Tutorial presents front end design flow of Low Noise Amplifier (LNA) circuit and guides you through the following circuit design steps:

- Schematic Capture with Gateway
- Simulation with SmartSpiceRF
- Results postprocessing with SmartView

The following Silvaco EDA tools are needed to work with Tutorial:

- GATEWAY
- SMARTSPICERF
- SMARTVIEW

1: GATEWAY Schematic Capture

Start Gateway and load the workspace file LNA.workspace by selecting **File→Open→Workspace**. When the browser window appears, navigate to the directory where RF Demo PDK Examples were installed, descent into ./LNA directory, and select LNA.workspace file.

1.1: Loading LNA Schematic

To load LNA example circuit schematic, select **File→Open→ Schematic**. When the file browser appears, navigate to the ./LNA directory and select the file LNA.schl.r. The LNA example circuit schematic will appear in the window (Figure 1).

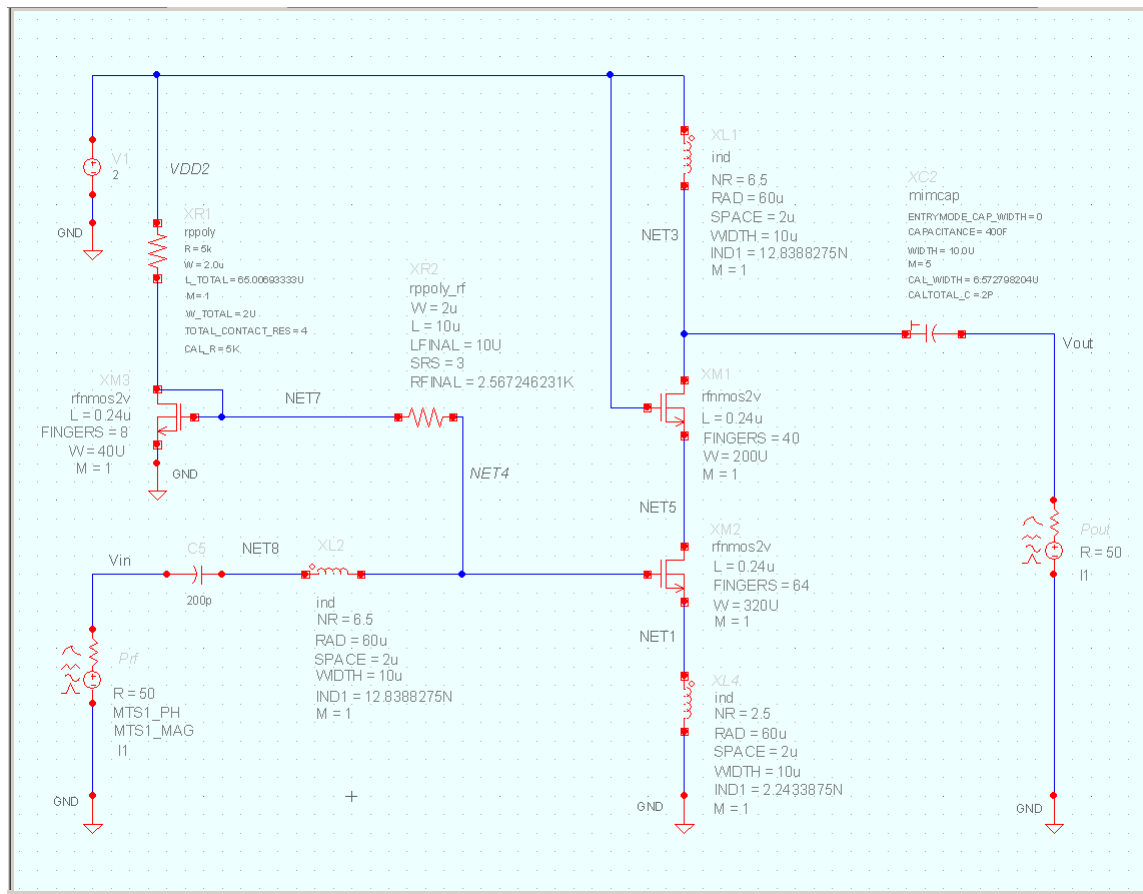


Figure 1: LNA Schematic

1.2: Generating Netlists

Netlists are ASCII files used to describe device connectivity and element properties of the circuit. GATEWAY can create two different netlist formats: SMARTSPICE for circuit simulation, or GUARDIAN for layout design. See the GATEWAY USER'S MANUAL for more information on these formats.

To generate a netlist in SMARTSPICE format, either select **Simulation→Create Netlist**, or click on **SmartSpice Netlist** icon (Figure 2) in **Tool Bar**.



Figure 2: Create Netlist Icon

LNA netlist then will appear in a new window:

```

File Edit View Bookmarks Spice Help
[Icons] AC DC [Symbol] [Symbol] H[Symbol] [Symbol] [Symbol] [Symbol] [Symbol] [Symbol] [Symbol]
* Schematic name: LNA
C5 NET8 Vin 200p
Pout Vout GND R=50
Prf Vin GND ACMAG=prf ACPHASE=0 R=50 MAG1=prf PHASE1=0
V1 VDD2 GND DC 2
XC2 NET3 Vout mimcap LT=6.5728U M=5
XL1 VDD2 NET3 ind WIDTH=10u RAD=60u SPACE=2u NR=6.5 M=1
XL2 NET8 NET4 ind WIDTH=10u RAD=60u SPACE=2u NR=6.5 M=1
XL4 NET1 GND ind WIDTH=10u RAD=60u SPACE=2u NR=2.5 M=1
XM1 NET3 VDD2 NET5 NET5 rfnmos2v LR=240N WR=5U NR=40 M=1
XM2 NET5 NET4 NET1 NET1 rfnmos2v LR=240N WR=5U NR=64 M=1
XM3 NET7 NET7 GND GND rfnmos2v LR=240N WR=5U NR=8 M=1
XR1 VDD2 NET7 rpoly W=2U L=65.007U nc=3 M=1
XR2 NET7 NET4 rpoly_rf L=30U W=2U NC=2 M=1

* Global Nodes Declarations
.GLOBAL GND VDD VDD2

* End of the netlist

```

Figure 3: LNA Netlist

1.3: Simulator Setup

Check GATEWAY that your preferred simulator is set to SMARTSPICERF. In GATEWAY click on **Edit -> Preferences** to open the preferences setup window (Figure 4). Choose **Tools > Simulator** and set (check) Simulator to SMARTSPICERF. Version number is set to Default (the latest installed version) in **Version** window. You can define any specific version too.

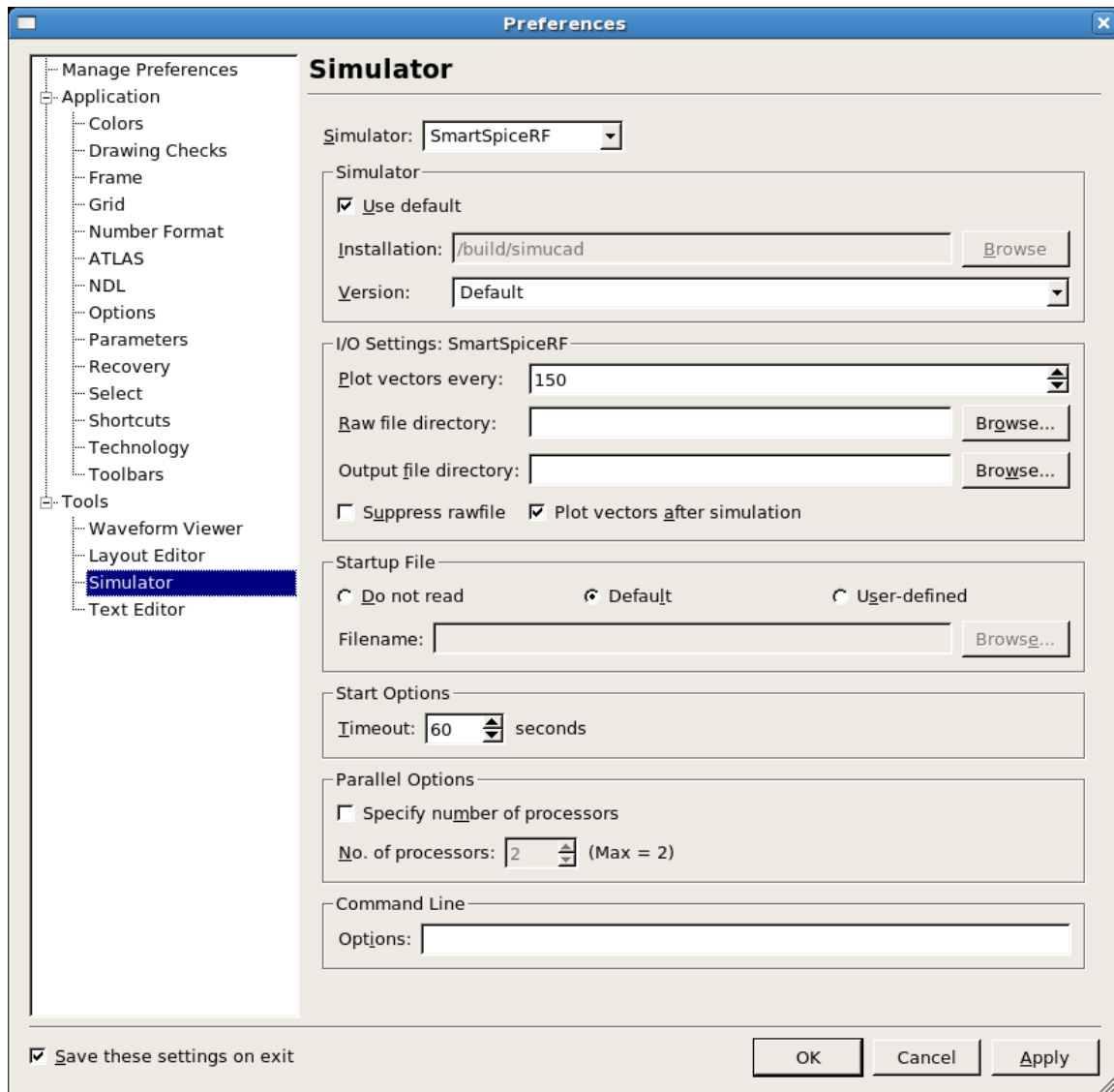


Figure 4: Simulator Preferences Window

1.4: Control File

SMARTSPICERF needs more than just a circuit netlist perform a meaningful simulations. It needs Circuit Netlist, Voltage or Current stimulus, set of Options, Analysis Statements, and active devices Model parameters or Libraries. All this information comes together in the form of Input Deck file (*.in, *.inp, *.cir, *.sp, *.sm, *.scs, etc.), which provides SMARTSPICERF with all needed information to run simulation and generate output data.

Click on the simulation **Tool Bar** icon **Edit Control File** (Figure 5):



Figure 5: Edit Control File Icon

The control file will be opened in the text editor *Sedit* window. The control file *LNA.ctr* (Figure 6) consists of path to SILVACO RF Demo PDK model library file as well as SMARTSPICERF .OP, HNET, and .HARM Analyses statements. The control file combined with the circuit netlist and list of vectors to be saved creates an Input Deck for SMARTSPICERF. There are two RF simulation runs for this example to provide S-parameters, Noise Figure, and 1dB compression point calculations.

```

File Edit View Bookmarks Spice Help
AC DC IN W HES IN OP PZ X+1 FREQ
*
.inc ../../models/sbcd_rf.lib
.OP
*
.PARAM PRF=-40_dBm
.OPTIONS EXPERT=777

|#com
*****
* SIMULATION 1 (HNET) *
*****
.HNET PRF POUT
+ LIN 600 10MegHz 60Hz
+ FUND=10GHz
+ ANNOTATE = 4
+ WAVES
+ NOISE
+ LOAD_STAB_CIRCLES
+ SOURCE_STAB_CIRCLES
+ GAIN_CIRCLES
+ POWER_GAIN_CIRCLES
#endcom

|#com
*****
* SIMULATION 2 (P1db) (PARAMETRIC SWEEP)*
*****
.HARM
+ FUND=2400MegHz
+ NHARM=10
+ ANNOTATE = 3
+ WAVES
+ SWEEP prf LIN 11 -50 0
.MEASURE HARM_SP dbOUT EXPR VAL='@Pout[pout1]' ; at 2400MegHz
.MEASURE MEAS r_1dbCompression COMPR1DB dbOUT EP=-45
#endcom

```

Figure 6: LNA Circuit Control File

2: SmartSpiceRF Simulation

2.1: .HNET Analysis

Periodic Steady-State HNET Analysis is a two-port network small-signal analysis, which is used to compute scattering parameters (S-parameters) of the circuit.

2.1.1: Uncomment the .HNET analysis statement section in LNA circuit control file (Figure 7).

```


*#com
*****
* SIMULATION 1 (HNET) *
*****
.HNET PRF POUT
+ LIN 600 10MegHz 6GHz
+ FUND=10GHz
+ ANNOTATE = 4
+ WAVES
+ NOISE
+ LOAD_STAB_CIRCLES
+ SOURCE_STAB_CIRCLES
+ GAIN_CIRCLES
+ POWER_GAIN_CIRCLES
*#endcom

#com
*****
* SIMULATION 2 (P1db) (PARAMETRIC SWEEP)*
*****
.HARM
+ FUND=2400MegHz
+ NHARM=10
+ ANNOTATE = 3
+ WAVES
+ SWEEP prf LIN 11 -50 0
.MEASURE HARM_SP dbOUT EXPR VAL='@Pout[pout1]' ; at 2400MegHz
.MEASURE MEAS_r_idbCompression COMPR1DB dbOUT EP=-45
#endcom

```

Figure 7: .HNET Analysis Statement

.HNET will first perform a PSS analysis with the 10GHz fundamental. Although there are no signals at 10GHz in this circuit, using .HNET will allow us to make use of the extended s-parameter measurements for Noise, Load Stability, Source Stability, Gain and Power circles that HNET provides.

2.1.2: Click on  (Gateway -> Simulation -> Run) to begin the SMARTSPICERF simulation.

2.2: .HNET Analysis Simulation Results

SmartSpiceRF outputs simulation results of the Large-Signal PSS and Small-Signal HNET analyses and statistic information into Output window and creates number of plots with Waveforms, Spectra, Hnet Data, Measurement results, etc. .HNET analysis outputs Large-signal steady-state waveforms and spectra of circuit variables.

2.2.1: SmartSpiceRF Plots and Vectors

.HNET analysis produces the number of plots:

- **hnet** plot consists of the two-port network analysis results.
- **hop_sp** plot consists of the frequency-domain spectra. The results are calculated at the frequency points $k \cdot fund$, where $fund$ is the circuit fundamental frequency, and $k=0, 1, \dots, nharm$.
- **hop_wf** plot consist of the waveforms at unified time grid with the number of time points defined by a specified number of harmonics $nharm$. This plot is output, if keyword WAVES was specified in analysis statement. If not, only the spectra plot hop_sp will be output.

.PRINT, .PROBE, .SAVE and .MEASURE statements must use the analysis and/or specific plot names HNET, HOP_WF, HOP_SP and so forth, to make any measurement and output separate results. The output statements without the analysis type name will output all types of results for all types of analysis.

3: SmartView Graphic Postprocessor

Simulation results in form of plots will be loaded into SMARTVIEW, and SMARTVIEW **Data Browser** window will be opened.

3.1: Open `hnet1` plot, then select vectors `s11_h0h0`, `s12_h0h0`, `s21_h0h0`, `s22_h0h0`, and click **Plot** (see Figure 8).

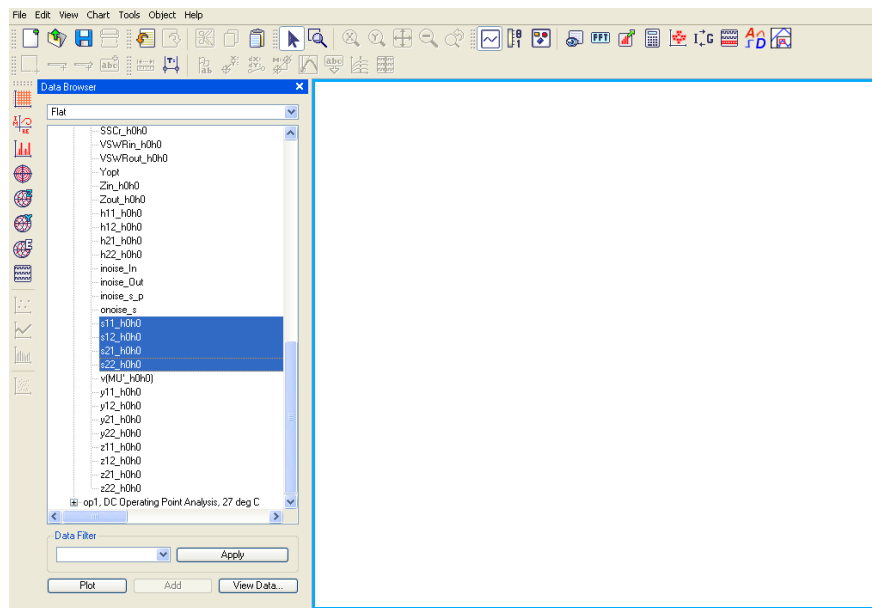


Figure 8: SmartView Data Browser Window

3.2: Then select **View->Layout->Tile**. Right-click on the picture and select **Properties** and select dB(MAG) as the **Data Map Type** (Figure 9).

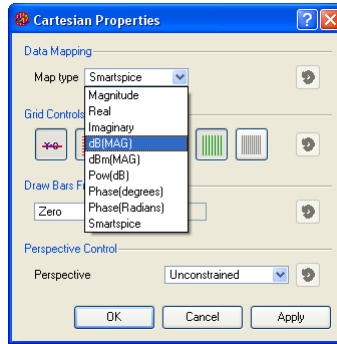


Figure 9: Cartesian Properties Window

3.3: Then select **Chart->Split->Split-all/Selected** and **View->Layout->Tile** to get the plot shown in Figure 10. You can also change the grid details, colors, and line width by selecting **Edit->Preferences**.

The result shows that at 2.4 GHz LNA has $S_{11} = -13.1$ dB, $S_{22} = -18.4$ dB, $S_{21} = 16.9$ dB, $S_{12} = -28.8$ dB.

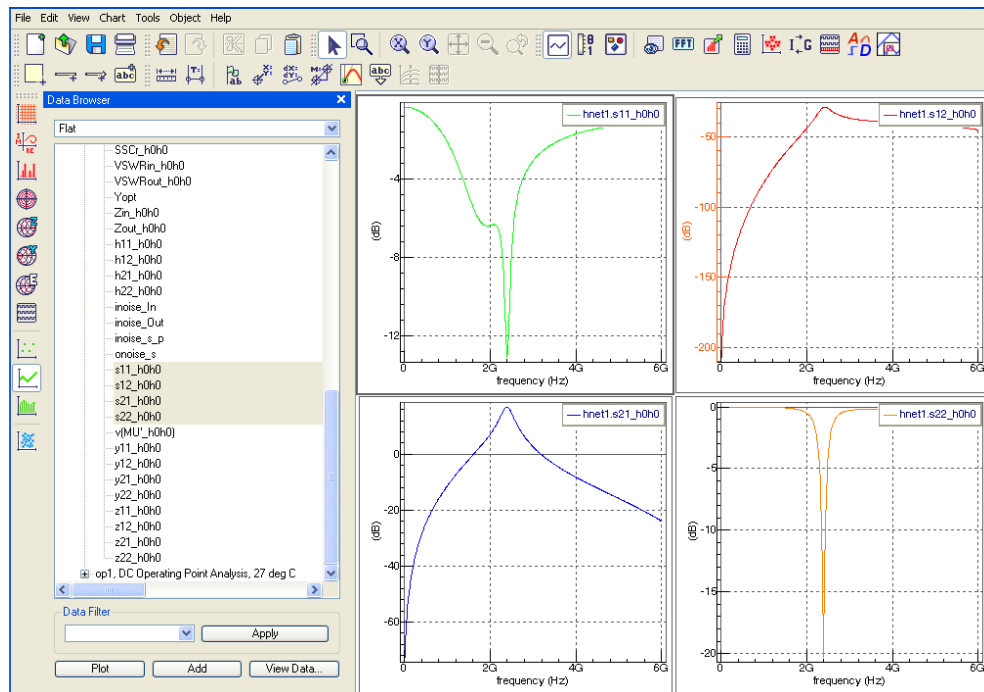



Figure 10: Cartesian Chart of S11, S22, S21), and S12

3.4: Press  on the left-side of SmartView window, choose S11_h0h0 and S22_h0h0, and click **Plot** to draw its on the E-Smith chart (see Figure 11).

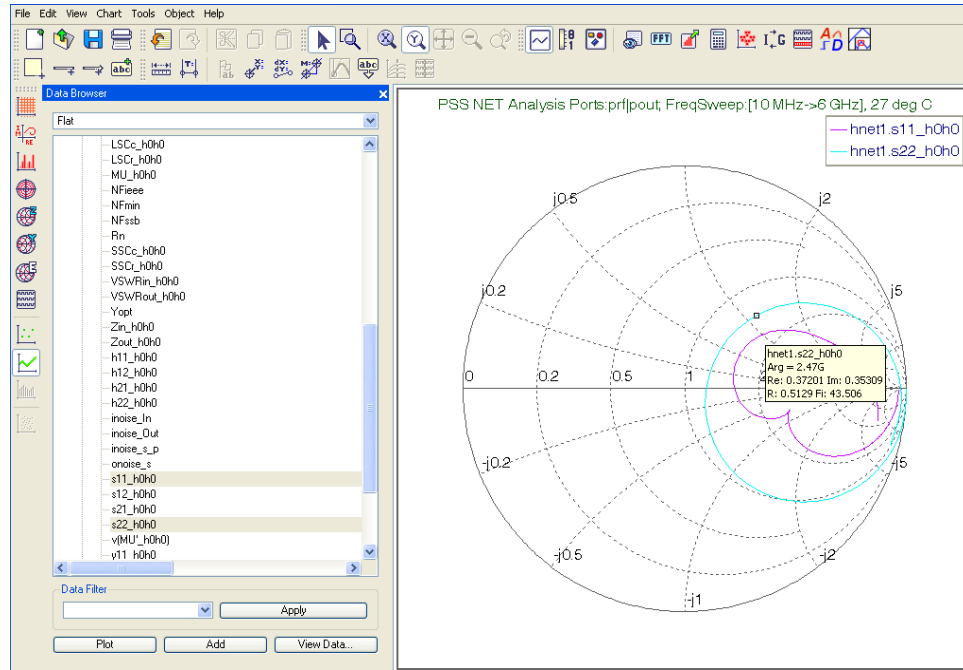


Figure 11: Smith Chart for S11 and S22

3.5: To display the Noise Figure plot, select the vector NFssb and click **Plot**. The LNA noise figure at 2.4 GHz is 5.3 dB (Figure 12).

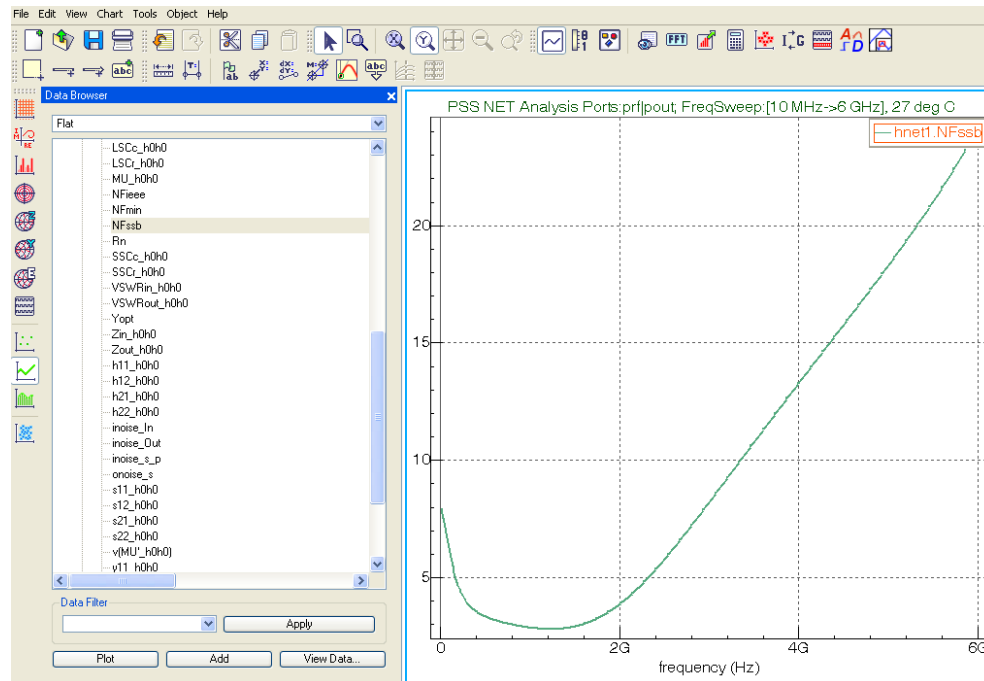


Figure 12: Noise Figure Plot

3.6: To plot the stability factor, select the vector K_h0h0 and click **Plot** (Figure 13).

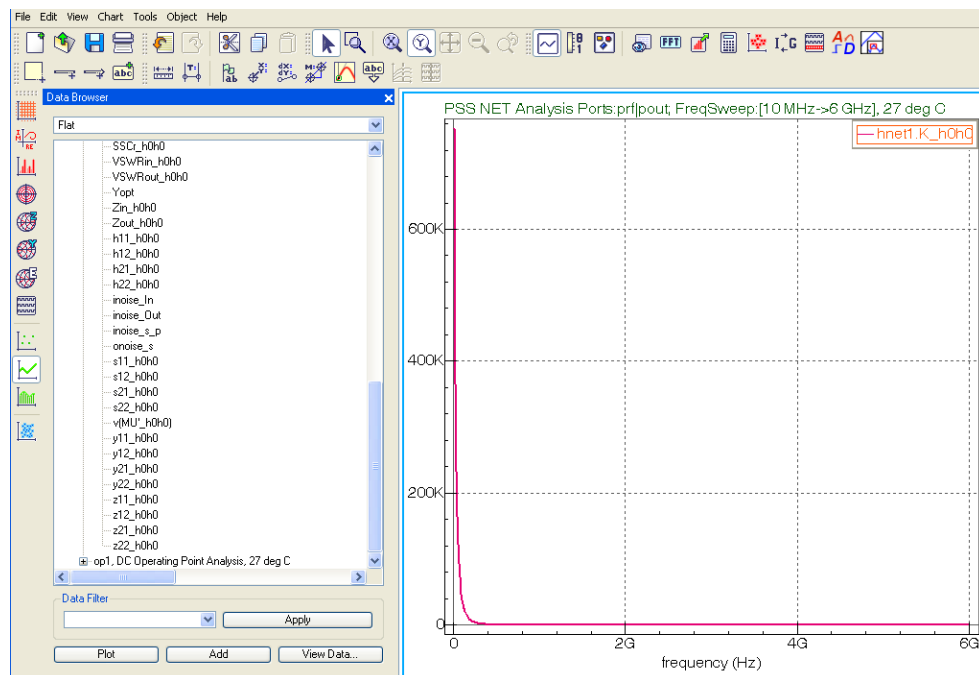


Figure 13: Stability Factor K Plot

3.7: To display the Gain, Power Gain, and Stability Circles over a single frequency, use the **Data Filter** to filter out the desired single frequency and click **Plot**. On Figure 14 LNA circles for the frequency 2.4GHz are shown.

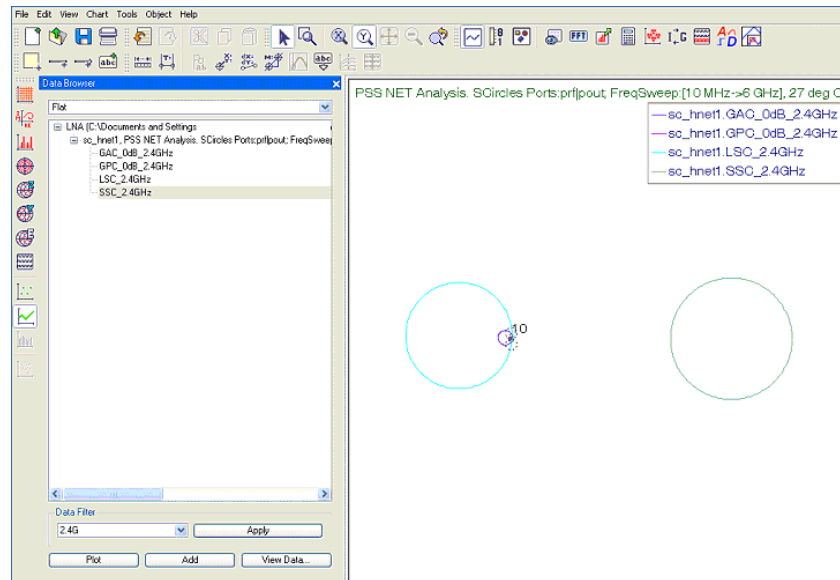


Figure 14: Gain, Power Gain, and Stability Circles

4: 1dB Compression Point Calculation (P1dB)

The Input Referred 1dB Compression Point is the input signal level at which the actual gain departs from the theoretical gain by -1dB. SMARTSPICERF allows you to compute Input Referred 1dB Compression Point using the measurement statement COMPR1DB.


4.1: P1dB Simulation Setup

4.1.1: To calculate P1db, comment the .HNET analysis and uncomment the .HARM analysis in LNA.ctr (Figure 15).

```
#com
*****
* SIMULATION 1 (HNET) *
*****
.HNET PRF POUT
+ LIN 600 10MegHz 6GHz
+ FUND=10GHz
+ ANNOTATE = 4
+ WAVES
+ NOISE
+ LOAD_STAB_CIRCLES
+ SOURCE_STAB_CIRCLES
+ GAIN_CIRCLES
+ POWER_GAIN_CIRCLES
#endcom

*#com
*****
* SIMULATION 2 (P1db) (PARAMETRIC SWEEP)*
*****
.HARM
+ FUND=2400MegHz
+ NHARM=10
+ ANNOTATE = 3
+ WAVES
+ SWEEP prf LIN 11 -50 0
.MEASURE HARM_SP dbOUT EXPR VAL='@Pout[pout1]' ; at 2400MegHz
.MEASURE MEAS r_1dbCompression COMPR1DB dbOUT EP=-45
*#endcom
```

Figure 15: .HARM Analysis Statement

4.1.2: Click on  (Gateway->Simulation->Run) to begin the SMARTSPICERF simulation.

4.2: P1dB Simulation Results

Simulation results in form of plots will be loaded into SMARTVIEW, and SMARTVIEW **Data Browser** window will be opened.

4.2.1: To display the P1dB simulation results, from SMARTVIEW Data Browser window open `meas1` plot, choose both vectors `dbout` and `prf` for and press **Plot** (Figure 16).

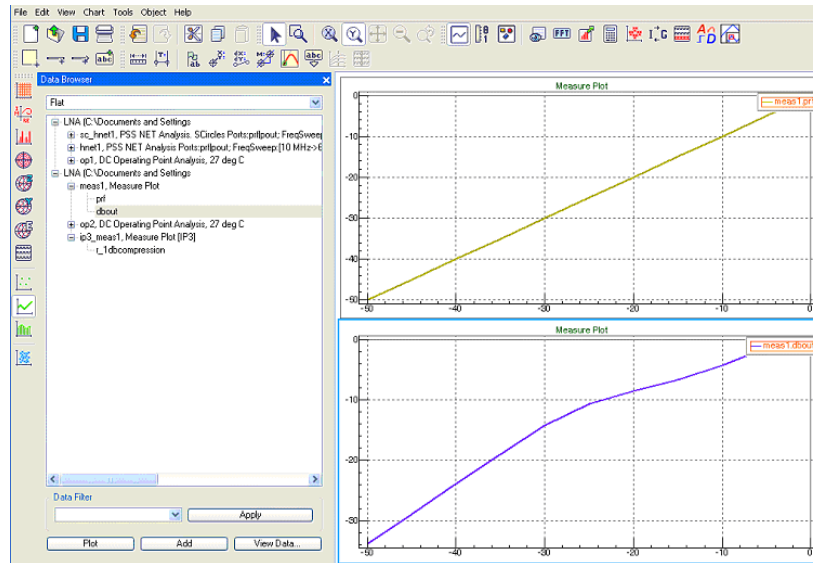


Figure 16: P1dB Simulation Plot

4.2.2: To view the value of P1dB result, select vector `r_1dbcompression` for `meas1` from SMARTVIEW and press **View Data** (Figure 17). The measured input referred 1dB compression point is -27.2 dBm.

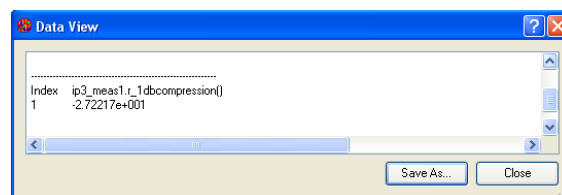


Figure 17: 1dB Compression Point Value

5: Third-Order Intercept Point Calculation

It is the nature of third-order products in the output signal to emerge from the noise at certain input levels and increase as the cube of the input levels. Thus, the slope of the third-order line increases 3dBm for every 1-dBm increase in the response to fundamental signal. The gain line can be continued to a point where it intersects with the gain line of the fundamental signal. This point is the third-order intercept point. The input signal level corresponding to this point is Input Referred IP3 Point.

SMARTSPICERF allows you to compute Input Referred IP3 Point with the formula:

$$IP3 = EP + \frac{P1 - P3}{GAIN}$$

using the measurement statement IP3.

5.1: Loading LNA Schematic

To load LNA circuit schematic for IP3 calculation select **File->Open->Schematic**. When the file browser appears, navigate to the ./LNA directory and select the file LNA_iip3.schl. The LNA circuit will appear in the window (Figure 18).

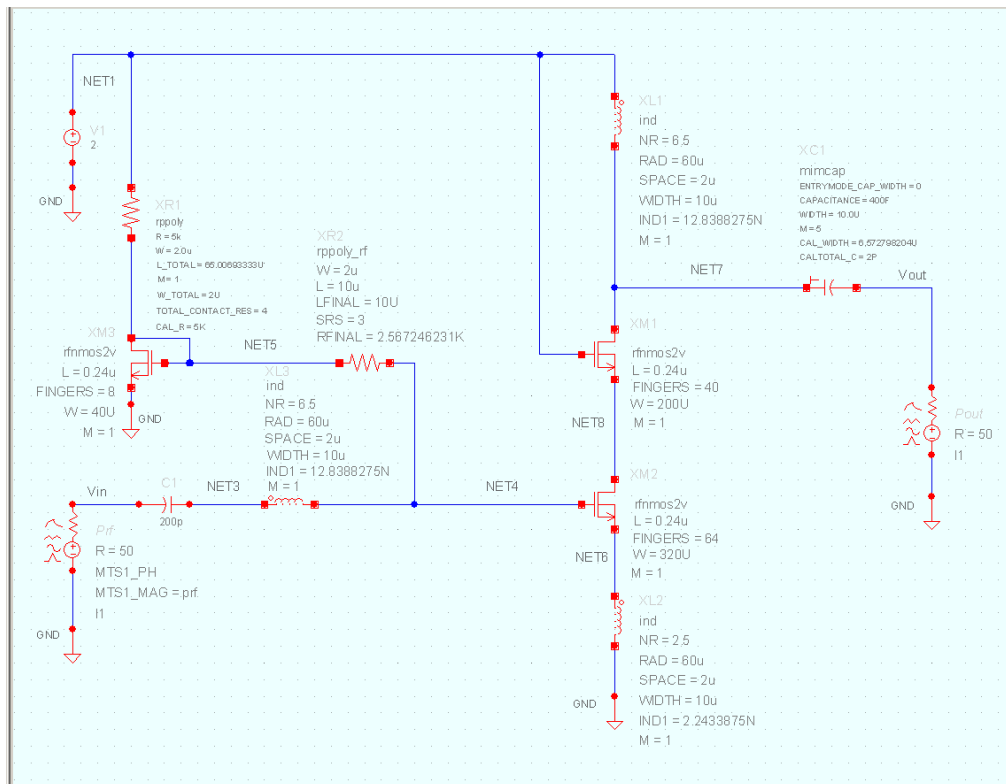



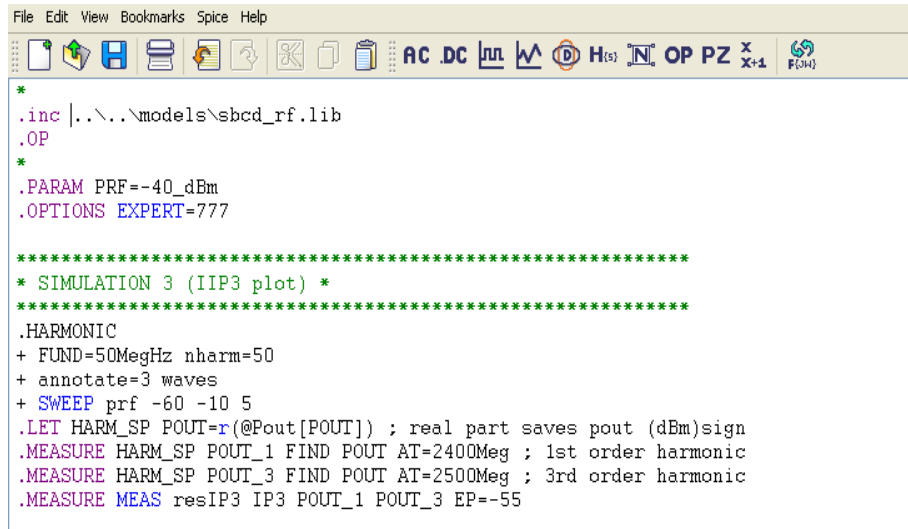
Figure 18: LNA_iip3 Schematic with Modified Input Port

In this simulation a two-tone signal is applied to the RF input in a form of multitone Port instance (PRF) as follows:

Name	Value
MTS1_MAG	prf
J1	48
MTS2_MAG	prf
J2	49

5.2: IP3 Simulation Setup

5.2.1: Click on  to generate netlist, then open the control file, LNA_iip3.ctr, to view .HARMONIC analysis and measurement statements to calculate IP3 (Figure 19).




```

File Edit View Bookmarks Spice Help
[Icons] AC DC [Waveform] [H(s)] [N] OP PZ X X+1 F{34}
*
.inc |..\models\sbcd_rf.lib
.OP
*
.PARAM PRF=-40_dBm
.OPTIONS EXPERT=777

*****
* SIMULATION 3 (IIP3 plot) *
*****
.HARMONIC
+ FUND=50MegHz nharm=50
+ annotate=3 waves
+ SWEEP prf -60 -10 5
.LET HARM_SP POUT=r(@Pout[POUT]) ; real part saves pout (dBm)sign
.MEASURE HARM_SP POUT_1 FIND POUT AT=2400Meg ; 1st order harmonic
.MEASURE HARM_SP POUT_3 FIND POUT AT=2500Meg ; 3rd order harmonic
.MEASURE MEAS resIP3 IP3 POUT_1 POUT_3 EP=-55

```

Figure 19: LNA IP3 Simulation Control File

5.2.2: Click on  (**Gateway -> Simulation -> Run**) to begin the SMARTSPICERF simulation. Simulation results in form of plots will be loaded into SMARTVIEW, and SMARTVIEW **Data Browser** window will be opened.

5.3: IP3 Simulation Results

Simulation results in form of plots will be loaded into SMARTVIEW, and SMARTVIEW **Data Browser** window will be opened.

5.3.1: To display the P1dB simulation results, from SMARTVIEW Data Browser window open meas1 plot, choose both vectors pout_1 and pout_3, and press **Plot** (Figure 20).

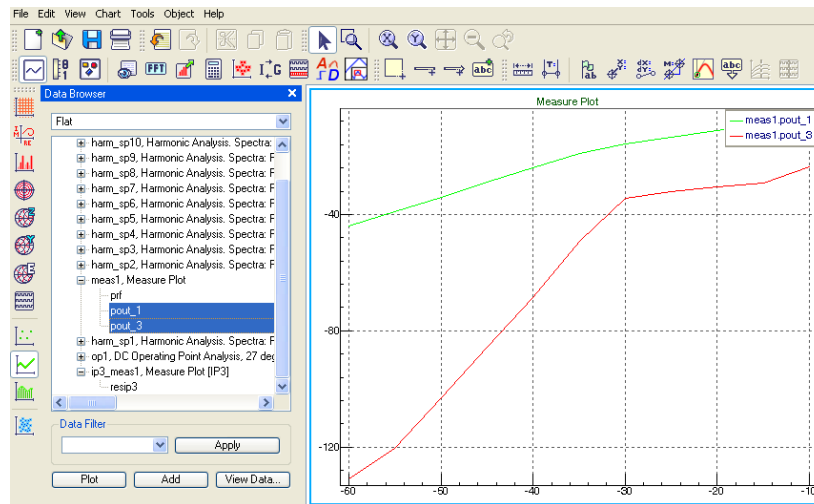


Figure 20: IP3 Simulation Plot

5.3.2: To view calculated IP3 value, select vector resip3 for ip3_meas1 from SMARTVIEW and press **View Data** (Figure 21).

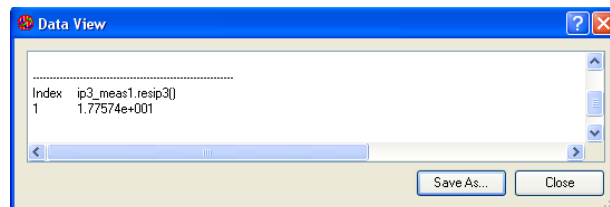


Figure 21: IP3 Point Value