

Date: _____

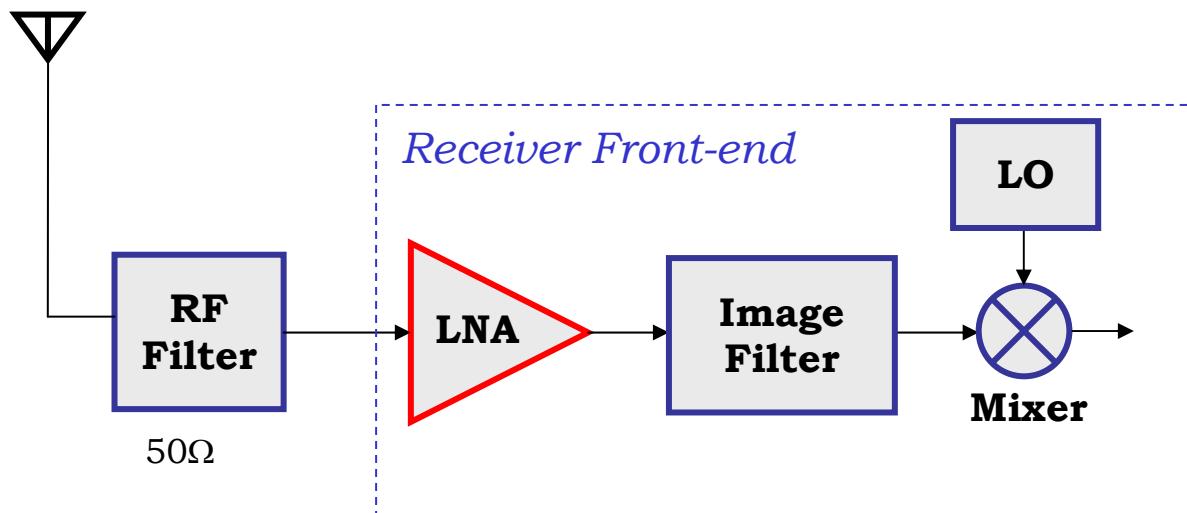
Student Name: _____ Lab Supervisor: _____

Personal Number: - Signature: _____

Notes:

LAB-2 (Tutorial) Simulation of LNA (Cadence SpectreRF)

Prepared By
Rashad.M.Ramzan
rashad@isy.liu.se



Introduction:

This tutorial describes how to use SpectreRF in Analog Design Environment to simulate parameters which are important in design and verification of Low Noise Amplifiers (LNAs). To characterize the LNA following figure of merits are usually measured.

1. Power Consumption and Supply Voltage
2. Gain
3. Noise
4. Input and Output Impedance Matching
5. Reverse Isolation
6. Stability
7. Linearity

We will use S-Parameters (SP), Periodic Steady State Analysis (PSS), Periodic AC (PAC) and Pnoise analysis available in SpectreRF to simulate above parameter of LNA. Usually there is more than one method available to simulate the desired parameter; we will use the procedure recommended by Cadence and takes less simulation time.

1. S-Parameter Analysis
 - Small Signal Gain (S21, GA, GT, GP)
 - Small Signal Stability (Kf and Δ or Bif)
 - Small Signal Noise (SP and Pnoise)
 - Input and Output Matching (S11, S22, Z11, Z22)
2. Large Signal Noise Simulation (PSS and Pnoise)
3. Gain Compression, 1dB Compression Point (Swept PSS)
4. Large Signal Voltage Gain and Harmonic Distortion (PSS)
5. IP3 Simulation (Swept PSS)
6. Conversion Gain and Power Supply Rejection Ratio (PSS and PXF)

Instructions

- If LAB is not finished in scheduled time slot, you can complete in your own time, if there is any problem, send an email or show up in the office of the TA. You must answer the questions in the LAB compendium before you start the tutorial, this will help you to effectively comprehend the tutorial material and simulations methodology.

Cadence Setup Guidelines

1. Please read the complete manual before you start the software. You will be using AMS 0.35 μ m CMOS (c35b4) process for these LABs.
 - Remove any previously loaded Cadence modules (Type **module on** command prompt and read the instruction. These instructions will guide you how to list, load and remove the modules)
 - Create a new directory **myrfdir** where your simulation data will be stored.
 - cd myrfdir, do rest all the steps from this directory
 - Load the Cadence and technology file using
 - **module add cadence/5.0.33**
 - **module add ams/3.60**
 - Start cadence by typing: **myrfdir > ams_cds –tech c35b4 –mode fb&**

- Make a new library **rf_Lna** in Cadence Library Manager
 - Create and draw the Schematics, LNA_testbench a as shown in Fig-1 and LNA as shown in Fig-2.
2. Use the RF NMOS transistors from library **PRIMLIBRF** valid up till 6GHz. The models provided in **PRIMLIB** are valid up till 1GHz. The maximum allowable size of NOMS in SpectreRF is 200 μ m (20 fingers of 10um or 40 fingers of 5um), if you need bigger transistor, use two transistors in parallel.
 3. Use **analogLib** for other active and passive components. In Library Manager click on Show Categories box on the top of window, this will show you the categories of components.
 4. There are many views available when you place the symbol in schematic, use **Symbol** or **Spectre** view only.
 5. *If Balun is used in your testbench*, you may find this in the Library **rfLib**. If you do not have this library in path. In icfb window, Click Tools → Library Path Editor and add the in Library field: **rfLib**
Library path: /sw/cadence/5.0.33/tools.sun4v/dfl/samples/artist/rfLib
 6. From Schematic view the balun model might not be accessible to simulator. Use the **config** view of testbench for simulation.
 7. To get to the **config** view you can use following procedure
 - Complete the testbench schematic → save and close the window
 - From **icfb** window
 - File → New → Cell view
 - Tool → Hierarchy Editor
 - View name → config
 - Select the appropriate Library and type the cell Name
 - In New Configuration window
 - Use template → SpectreSverilog and press OK
 - New Configuration window fields will be automatically filled.
 - Press OK
 - In **Hierarchy Editor** window
 - Right click on View found (balun) → Select view → veriloga
 - Save and exit the Hierarchy Editor
 - In Library Manger, you will find the config view of your test bench.
 - Open this config view and use for simulation

1. Back Ground Preparation (LNA)

Please read the Application Note “LNA Design Using Spectre RF” and answer the following questions before you attend the LAB.

- Define Transducer Power Gain (G_T), Operating Power Gain (G_P) and Available Power Gain (G_A) for a two port network?

- How we can relate the S-Parameters to the gain, input impedance and output impedance of any two-port network?

- Why is the reverse isolation gain important in the LNA design? Which S-parameter directly characterizes the reverse isolation gain?

- What is stern stability factor? What is minimum condition of stability for LNA?

- Define the Power Supply Rejection Ratio (PSRR)? Look at the circuit diagram of LNA, what is your guess about the PSRR of this LNA?

2. LNA Simulation

2.1. Circuit Simulation Setup:

- We will be using AMS 0.35 μ m CMOS (c35b4) process for these LABs.
- Load the Cadence and technology file using
 - **module add cadence/5.0.33**
 - **module add ams/3.60**
- Start cadence by typing **ams_cds –tech c35b4 –mode fb&**
- Make a new library RF_LAB1 in Cadence Library Manager
- Create and draw the Schematics, LNA_testbench a as shown in Fig-1 and LNA as shown in Fig-2. The components values are listed below for your convenience.
- Input Port in Schematic LNA_testbench
 - 50 Ohms in *Resistance*
 - 1 in *Port Number*
 - Sine in *Source Type*
 - *frf1* in *Frequency name 1* field
 - *frf* in *Frequency 1* field
 - *prf* in *Amplitude1(dBm)* field
- Output Port in Schematic LNA_testbench
 - 500 Ohms in *Resistance*
 - 2 in *Port Number*
- Component Values in Schematic LNA_testbench
 - **Vdd = 3.3V**, C1, C2= 10nF, CL= 500fF
- Component Values in Test Bench Schematic
 - C1, C2= 10nF, CL= 500fF
- Component Values in LNA Schematic
 - M1, M2 = 200 μ m/0.35 μ m , Mbias = 60 μ m/0.35 μ m
 - Ls = 700 pH, Lg = 12 nH, Ld = 6 nH, Rd = 700 Ω

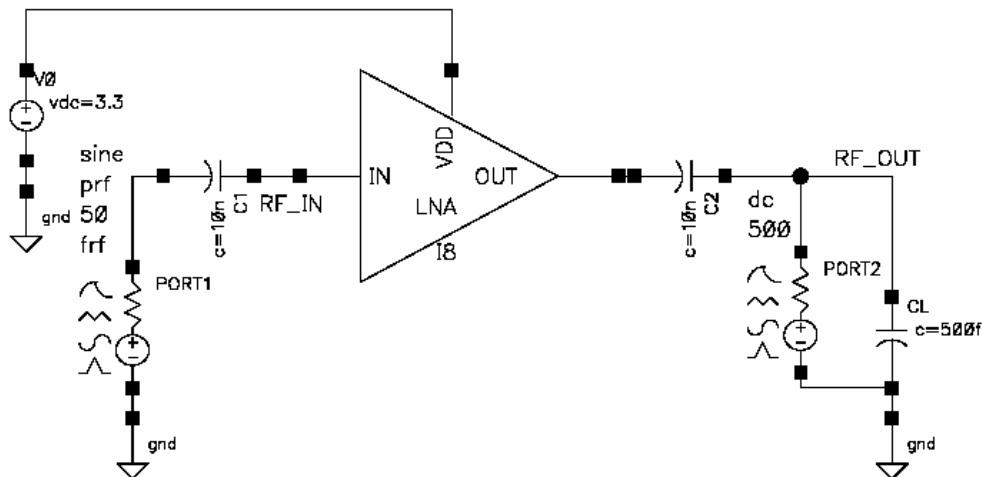


Fig1: Test Bench of LNA

- Open the Schematic LNA_testbench and Select Tools → Analog Environment
- Variable values in **affirma Design Variable** window (variables → Copy from Cellview)
 - $f_{rf} = 2.4 \text{ GHz}$ and $prf = -20 \text{ dBm}$
- In Simulation Environment Window (**affirma** window) choose *Setup* → *Environment*
- In field Analysis Order fill the following: dc pss pac pnoise (Important, if this field is not set PXF, Pnoise and PAC analysis may not work at all!)

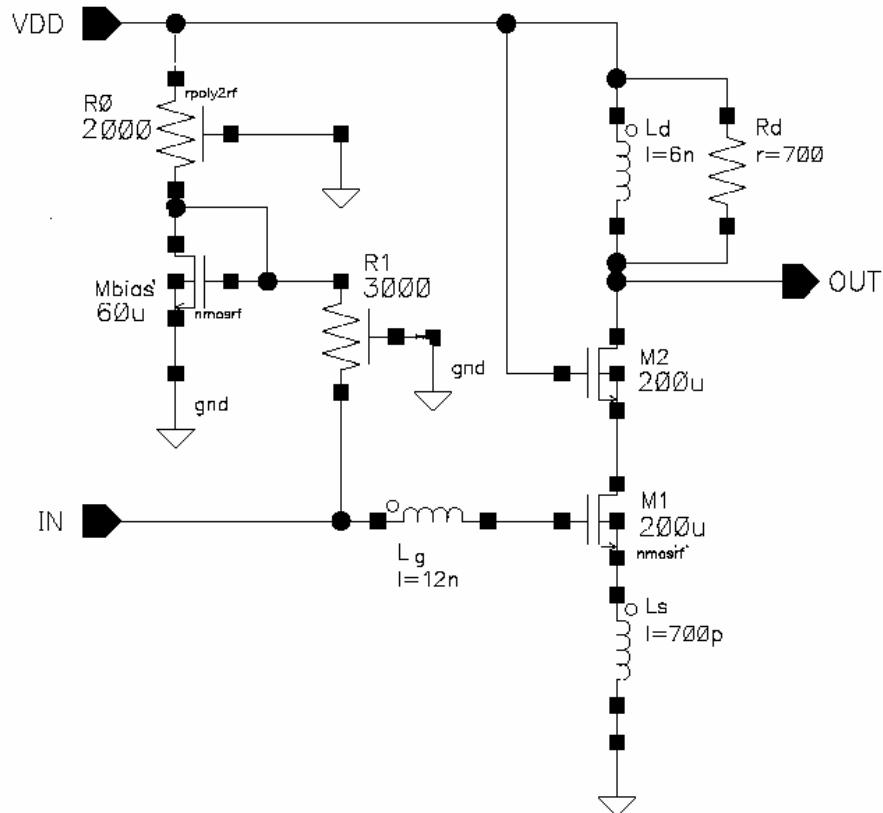


Fig2: Circuit Diagram of Source Inductor Degenerated LNA

Notes: Capacitor can be added in parallel with the L_d to make the gain and NF response more selective and narrow. R_d models the series resistance of ideal inductor. You can use the fixed Q inductor from cadence menu and remove this resistor.

Small Signal Gain, NF, Impedance Matching and Stability (S-Parameter)

- In the **affirma** window, select *analysis-choose*, the **analysis choose** window shows up
 - Select *sp* for Analysis
 - In port field click on select and then activate the schematic (if not activated automatically), choose the input port first and then the output port. The names of two selected ports will appear in *Ports* field.
 - *Sweep Variable* → frequency
 - *Sweep Range (start--stop)* → 1G to 5G
 - *Sweep Type* → Automatic
 - *Do Noise* → Yes
 - Select Input and output ports accordingly by clicking Select and then clicking at the appropriate *Port* in Schematic
 - Make sure that *Enabled Box* is checked then click OK.
- In the **affirma** window click on *Simulation* → Netlist and Run to start the simulation, make sure that simulation completes without errors.
- Now in the **affirma** window click on the *Results* → *Direct plots* → *Main Form*
- The *S-parameters* results window appears.
- **Impedance Matching**
 - In the *S-parameters Results* window...
 - *Select Function* → SP
 - *Plot Type* → Rectangular
 - *Modifier* → dB20
 - Click S11 {S12, S22 and S21} press the PLOT button.
 - Change the waveform window setting to make the plot look like Fig-3.

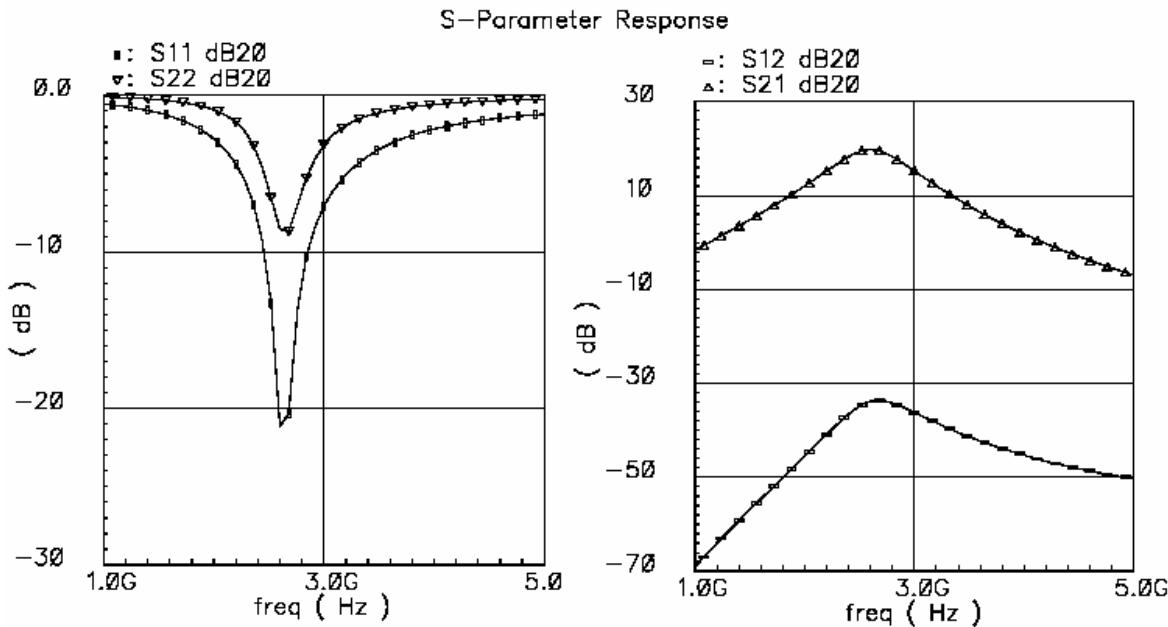


Fig3: S-Parameters of LNA

- **GT, GA and GP (Different Type of Gains)**

- In the *S-parameters Results* window....
- *Select Function* → *GT, GA and GP (one by one)*
- *Plot Type* → *Rectangular* and *Modifier* → *dB10*
- Press the *PLOT* button, the results are shown in Fig-4.

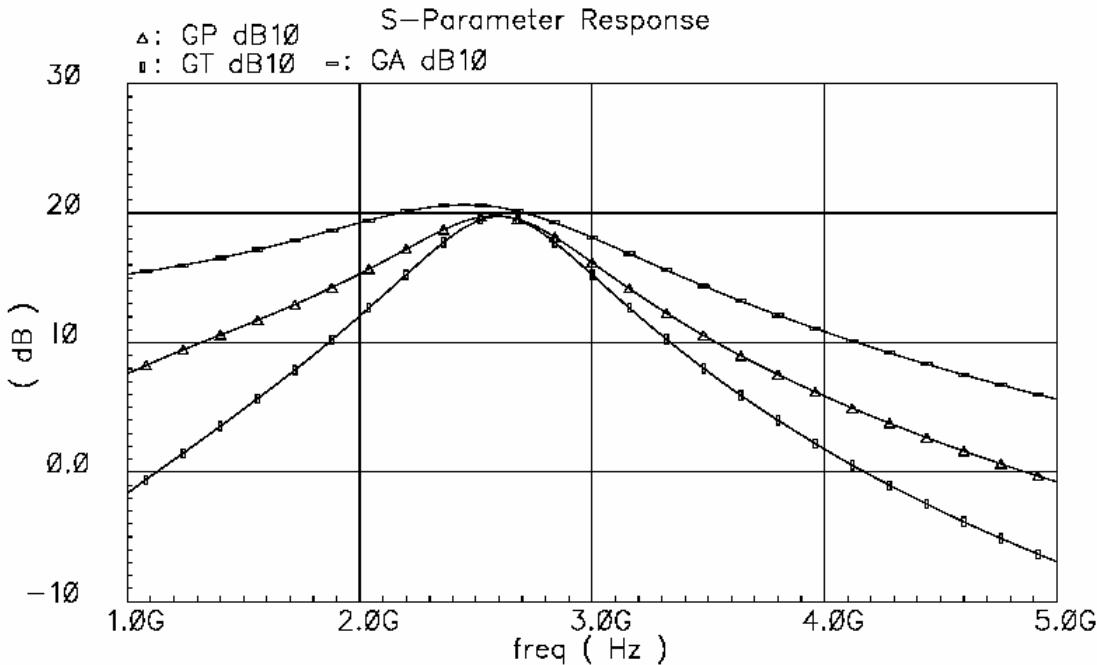


Fig4: GT, GA and GP

The power gain *GP* is closer to the transducer gain *GT* than the available gain *GA* which means the input matching network is properly designed. That is, *S₁₁* is close to zero.

- **NF (Noise Figure)**

- In the *S-parameters Results* window...
- *Select Function* → *NF (and NFmin)*
- *Plot Type* → *Rectangular*
- *Modifier* → *dB10*
- Press *PLOT*.
- The results are shown in Fig-5.

- **Stability Factor Kf and Bif (Δ)**

- In the *S-parameters Results* window...
- *Select Function* → *Kf and Bif (one at a time)*
- *Plot Type* → *Rectangular*
- Press the *PLOT* button.
- The results are shown in Fig-6.

The Stern stability factor *K* and Δ can be plotted in two ways. The stability curves for *K* and Δ are plotted with respect to frequency sweep as shown in Fig-6 or they can be plotted as load stability circle (LSB) and source stability circle (SSB).

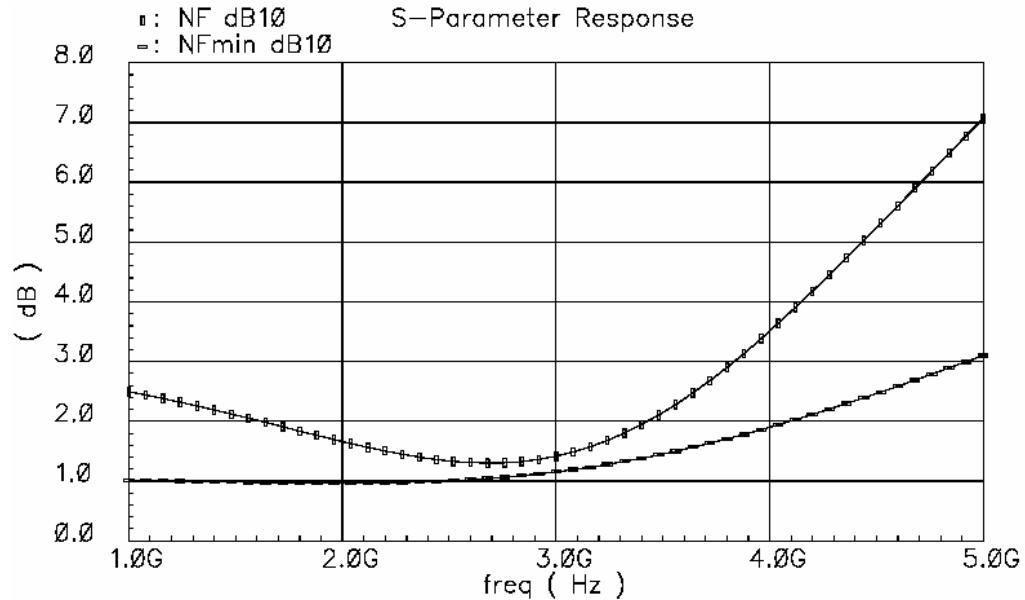


Fig5: NF, NFmin using S-Parameters

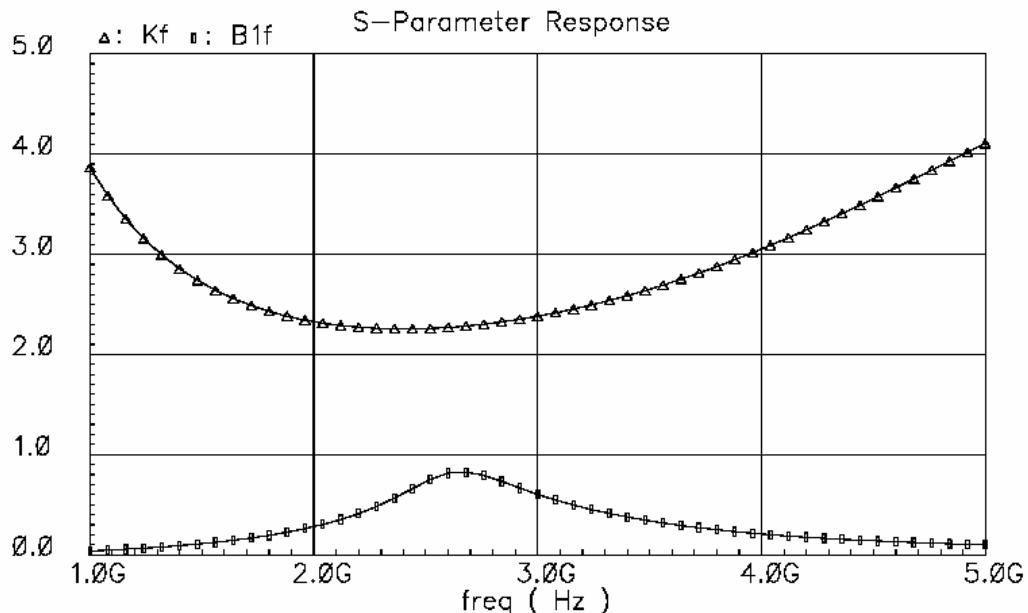


Fig6: Kf and Delta of LNA

Note: You can also measure the Z-parameters like Z_{11} and Z_{22} . This might help in the input and output impedance matching circuit design. S_{11} or input matching can be improved by changing the source degeneration inductor (L_s)

2.2. NF by Large Signal Noise Simulation (PSS and Pnoise Analysis)

Use the PSS and Pnoise analyses for large-signal and nonlinear noise analyses, where the circuits are linearized around the periodic steady-state operating point. (Use the Noise and SP analyses for small-signal and linear noise analyses, where the circuits are linearized around the DC operating point.) As the input power level increases, the circuit becomes nonlinear, the harmonics are generated and the noise spectrum is folded. Therefore, you should use the PSS and Pnoise analyses. When the input power level remains low, the NF calculated from the Pnoise, PSP, Noise, and SP analyses should all match.

- Change the Input Port Parameters in the Schematic
 - 50 Ohms in *Resistance*, 1 in *Port Number*, DC in *Source Type*
- Verify the variable values in the **affirma** window
 - $frf = 2.4 \text{ Ghz}$
 - $prf = -20$ (This value is meaningless in this simulation)
- In the **affirma** window, select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
 - Select *pss* for Analysis
 - Uncheck the *Auto Calculate* Box
 - *Beat Frequency* → 2.4G
 - *Output Harmonics* → 20
 - *Accuracy Default* → Moderate
 - Make sure that *Enabled* Box is checked then click OK.
- Now at the top of **choosing Analysis** window (This is another analysis)
 - Select *pnoise* for Analysis
 - *PSS Beat Frequency(Hz) = 2.4GHz*
 - *Sweep Type* → *Absolute*
 - *Frequency Sweep Range* → Start: 1G Stop: 5G
 - *Sweep Type* → *Automatic*
 - *Maximum Sidebands* → 20
 - In output Section
 - Select *Voltage*
 - *Positive Output Node* → Select net RF_OUT from Schematic
 - *Negative Output Node* → Leave Empty , it means GND
 - *Input Sources* → Select *PORT*
 - *Input PORT Source* → Select PORT1 from Schematic
 - *Reference Side Band* → 0
 - *Noise Type* → *Sources*
 - *Enable* Box in the bottom should be checked.
 - Click OK
- In the **affirma** window click on *Simulation* → Netlist and Run to start the simulation, make sure that simulation completes without errors.
- Now in the **affirma** window click on the *Results* → *Directplot* → *Main Form*

- The PSS results window appears.
 - Plot mode → Append*
 - Analysis Type → pnoise*
 - Function → Noise Figure*
 - Add to Output → Box Unchecked*
 - Click on PLOT Button , results are shown in Fig-7.*

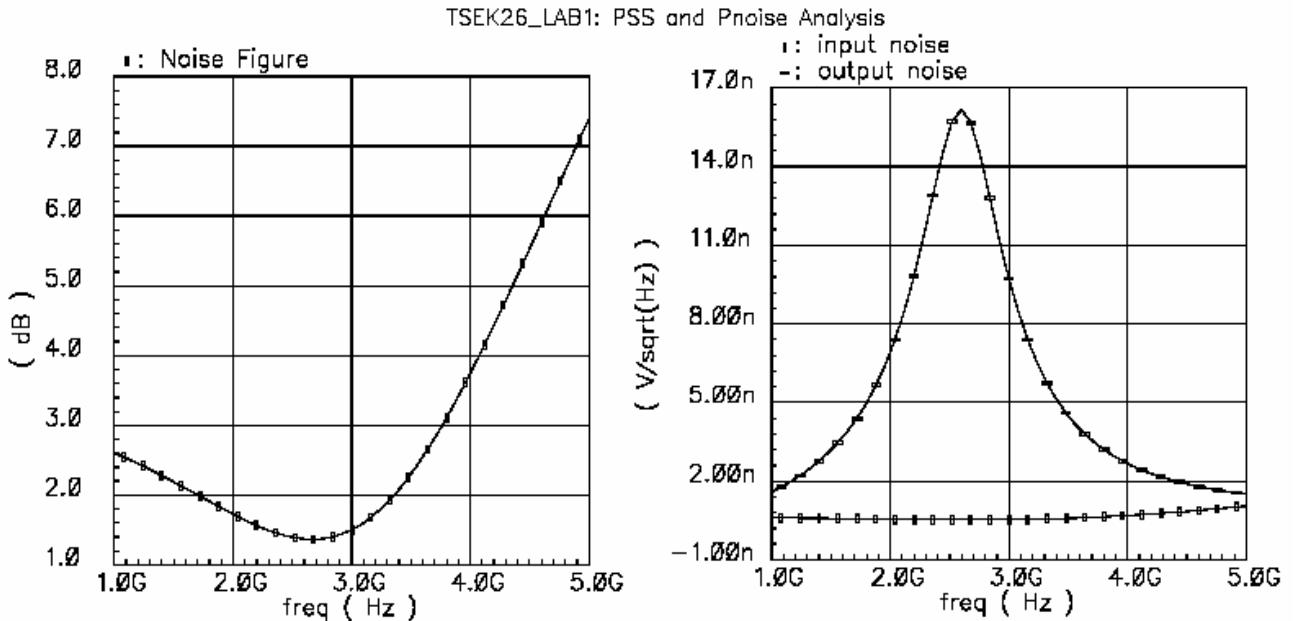


Fig7: NF, Input and Output Noise using Pnoise Analysis

- The Pnoise analysis summary shows you the contributions of different noise sources in the total noise. This is very powerful feature to focus the effort to improve the noise performance of the device which contributes the maximum noise.
- Now to see noise contribution in the **affirma** window click on the *Results → Print → (PSS) Noise Summary*
 - Type → Spot Noise*
 - Frequency Spot → 2.4G*
 - Click on *ALL TYPES* button so that all entries are highlighted
 - Truncate → None*
 - Leave all other field as it is and press *APPLY*
 - The Noise Contribution of Different Sources appears in new window
- Fill up the Table below indicating the noise contribution of different components.**

Comp	%Contribution	Comp	%Contribution	Comp	%Contribution
Port1					
M1					

2.3. Large Signal Voltage Gain and Harmonic Distortion (PSS)

- Change the Input Port Parameters in Schematic Window
 - 50 Ohms in *Resistance*
 - 1 in *Port Number*
 - **Sine** in *Source Type*
 - *frf1* in *Frequency name 1* field
 - *frf* in *Frequency 1* field
 - *prf* in *Amplitude1(dBm)* field
 - *Check and save the schematic*
- Verify the variable values in the **affirma** window
 - *frf* = 2.4 Ghz
 - *prf* = -20dBm
- In the **affirma** window, select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
 - Select *pss* for Analysis
 - In Fundamental Tones section, the following line should be visible


```
1 frf1 frf 2.4G Large PORT1
```
 - Check the *Auto Calculate* Box
 - *Beat Frequency* → 2.4G (Automatically appears)
 - *No of Harmonics* → 10
 - *Accuracy Default* → Moderate
 - *Enable* Box in the bottom should be checked.
 - Click OK
- In the **affirma** window click on *Simulation* → Netlist and Run to start the simulation, make sure that simulation completes without errors.
- In the **affirma** window, select *Results* → *Direct Plot* → *Main Form*
- The **analysis choose** window shows up
 - Select PSS for analysis
 - Select *Function* as *Voltage Gain*
 - *Modifier* → *dB20, Input Harmonics* → *2.4G*
 - *Select* → *Output* and then activate the schematic window and select *RF_OUT*;
 - *Select* → *Input* then activate the schematic window and select *RF_IN*
 - At the top of PSS result window change the plot mode to append.
 - Now Select *Function* as *Voltage*
 - *Sweep* → *Spectrum, Signal Level* → *peak*, *Modifier* → *dB20*
 - *Select* → *net* and then point to *RF_OUT* net in schematic
- Modify the display window. The results are shown in Fig-8.

After the PSS analysis, we can observe the harmonic distortion of the LNA by plotting the spectrum of any node voltage. Harmonic distortion is characterized as the ratio of the power of the fundamental signal divided by the sum of the power at the harmonics.

2.4. 1dB Compression Point(Swept PSS)

- Change/Check the Input Port Parameters in Schematic Window
 - 50 Ohms in *Resistance*
 - 1 in *Port Number*
 - **Sine** in *Source Type*
 - *frf1* in *Frequency name 1* field
 - *frf* in *Frequency 1* field
 - *prf* in *Amplitude1(dBm)* field
- Verify the variable values in the **affirma** window
 - *frf* = 2.4 Ghz
 - *prf* = -20dBm
- In the **affirma** window, select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
 - Select *pss* for Analysis
 - In Fundamental Tones following line shold be visible
1 frf1 frf 2.4G Large PORT1
 - Uncheck the *Auto Calculate* Box
 - *Beat Frequency* → 200M
 - *No of Harmonics* → 12 (as $12 \times 200\text{M} = 2.4\text{GHz}$)
 - *Accuracy Default* → Moderate
 - Highlight the *Sweep* Button
 - Click the “*Select Design Variable*” Button, small window appears, choose *prf* in it
 - *Sweep Range* → Choose the start : -40dBm and Stop: 0dBm (Do not write the units just enter numeric values)
 - *Sweep Type* → *Liner* and *No of Steps* =12
 - *Enable* Box in the bottom should be checked and Click OK
- In the **affirma** window click on *Simulation* → Netlist and Run to start the simulation, make sure that simulation completes without errors.
- In the **affirma** window, select *Results* → *Direct Plot* → *Main Form*
- The **analysis choose** window shows up
 - Select *Function* → *Compression Point*
 - *Select Port (Fixed R (Port))*
 - *Gain Compression* → *1dB*
 - *Extrapolation Point* → *-40dB*
 - *Ist Order Harmonic* → *2.4G*
 - Activate the Schematic Window and click on Output PORT to view the results as shown in Fig-9.

A PSS analysis calculates the operating power gain. That is, the ratio of power delivered to the load divided by the power available from the source. This gain definition is the same as that for *GP*. Therefore, the gain from PSS should match *GP* when the input power level is low and nonlinearity is weak. In case of differential LNA the even mode disturbances will be suppressed.

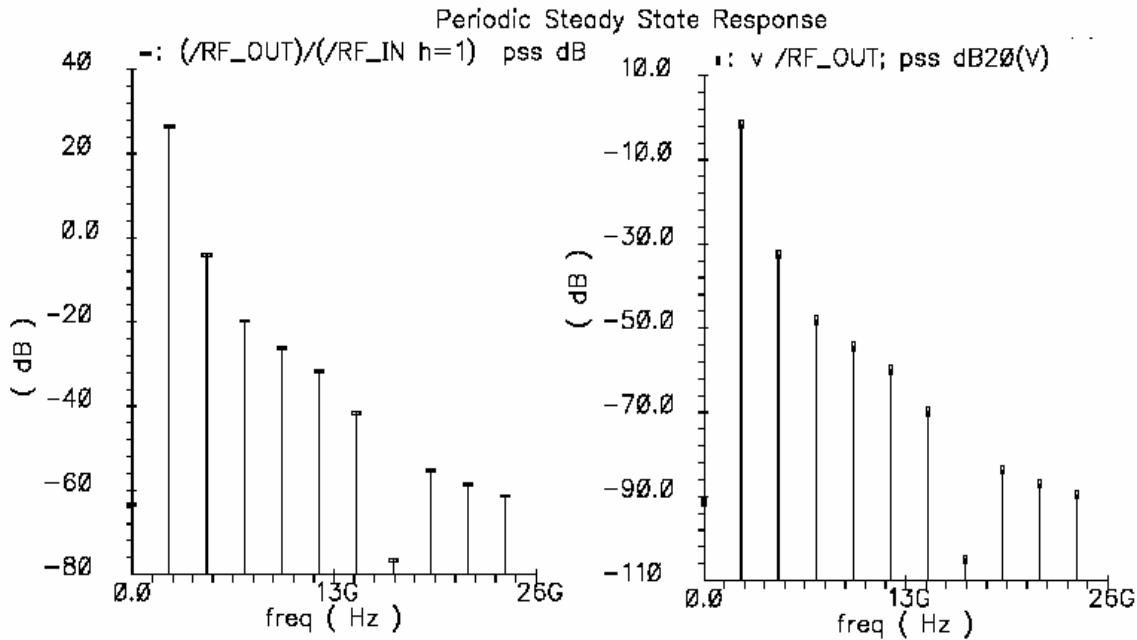


Fig8: Voltage Gain and Harmonic Distortion

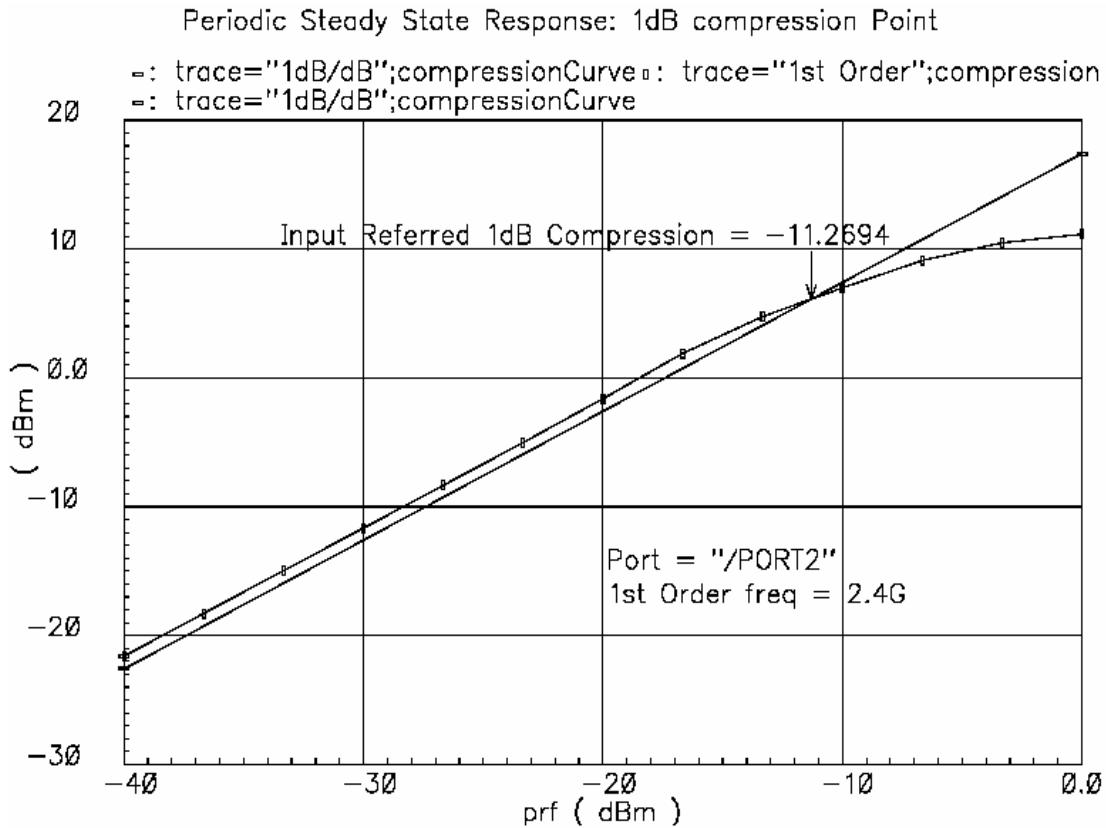


Fig9: 1dB Compression Point

2.5. IIP3 (Swept PSS)

A two-tone test is used to measure an IP3 curve where the two input tones are ω_1 and ω_2 . Since the first-order components grow linearly and third-order components grow cubically, they eventually intercept as the input power level increases as shown in Fig-10. The IP3 is defined as the cross point of the power for the 1st order tones, ω_1 and ω_2 , and the power for the 3rd order tones, $2\omega_1 - \omega_2$ and $2\omega_2 - \omega_1$, on the load side.

- There are three ways to Simulate IIP3, Using Swept PSS, PSS and PAC and QPSS. We will use Swept PSS Analysis.
- Change the Input Port Parameters in Schematic Window
 - 50 Ohms in *Resistance*
 - 1 in *Port Number*
 - **Sine** in *Source Type*
 - *frf1* in *Frequency name 1* field
 - *frf* in *Frequency 1* field
 - *prf* in *Amplitude1(dBm)* field
 - Click on the Box **Display Second Sinusoid**
 - *frf2* in *Frequency name2* field
 - *frf+40M* in *Frequency2* field
 - *prf* in *Amplitude2(dBm)* field
- Verify the variable values in the **affirma** window
 - *frf* = 2.4 Ghz
 - *prf* = -20dBm
 - Click "Apply", Close the window, Check and save Schematic
- In the **affirma** window, select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
 - Select *pss* for Analysis
 - In Fundamental Tones, the following lines should be visible


```
1 frf1 frf      2.4G Large PORT1
          2 frf2 frf+40M 2.44G Large PORT1
```
 - Check the *Auto Calculate* Box
 - *Beat Frequency* → 40M (Automatically appears)
 - *No of Harmonics* → 65 (as $65 \times 0.04\text{GHz} = 2.6\text{GHz}$)
 - *Accuracy Default* → Moderate
 - High light the *Sweep* Button
 - Select Design Variable, small window appears, choose *prf* in it
 - *Sweep Range* → Choose the start : -30dBm and Stop: 0dBm (do not write the units)
 - *Sweep Type* → *Liner* and *No of Steps* =12
 - *Enable* Box in the bottom should be checked.
- Click OK In the **affirma** window click on *Simulation* → Netlist and Run to start the simulation, make sure that simulation completes without errors.
- In the **affirma** window, select *Results* → *Direct Plot* → *Main Form*
- The **analysis choose** window shows up

- Highlight the *Replace in Plot Mode*
- Select *Function as Compression Point (IPN Curves)*
- *Analysis → PSS*
- *Function → IPN Curves*
- *Select Port (Fixed R (Port))*
- Highlight *variable Sweep Prf*
- *Extrapolation Point → -30dB*
- Highlight *Input Referred IP3*
- *Order → 3rd*
- *1st Order Harmonic → 2.4G*
- *3rd Order Harmonic → 2.48G*
- Activate the Schematic Window and click on Output port to view the results as shown in Fig-10.

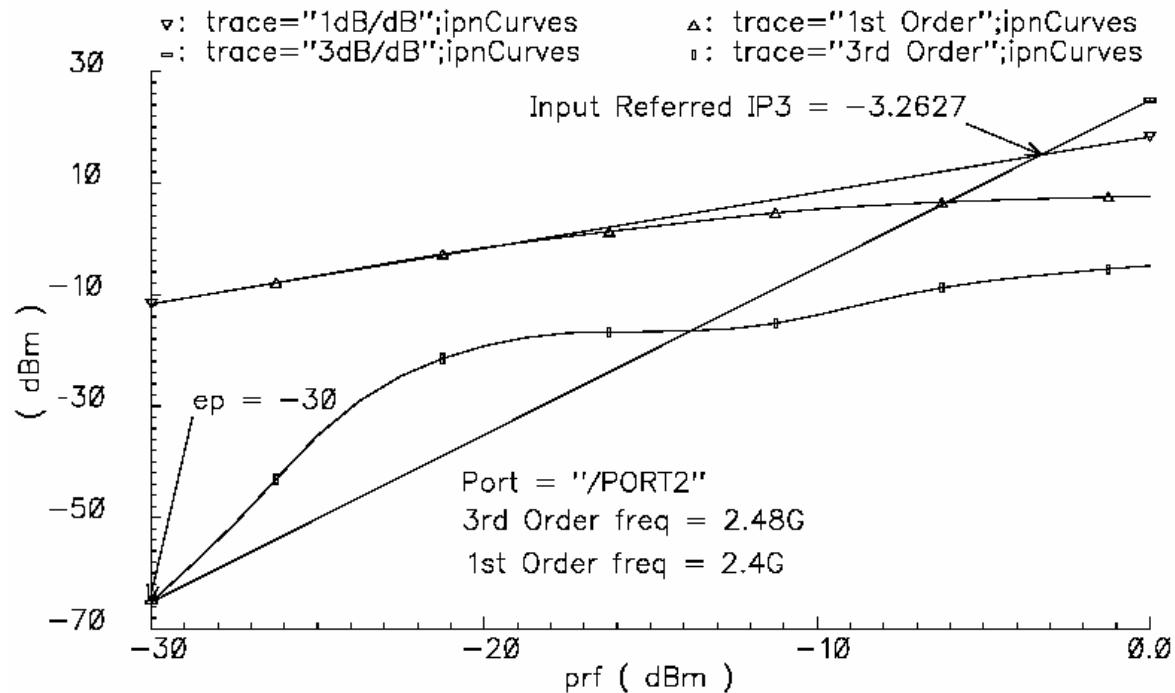


Fig10: Input Referred IP3

Note: IP3 plot above is not very nice looking, one can do more iterations and come up with better aligned 3rd order line with 3rd order plotted data.

Conversion Gain and Power Supply Rejection Ratio (PSS and PXF)

The PXF analysis provides frequency dependent transfer function from any specific source to the designated output (RF_OUT in this case). If the specific source is power supply node then we can measure the PSRR.

- Change the Input Port Parameters in Schematic
 - 50 Ohms in *Resistance*
 - 1 in *Port Number*
 - DC in *Source Type*
- Variable values in **affirma** window
 - $f_{rf} = 2.4$ Ghz
 - $prf = -20$
- In the **affirma** window, select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
 - Select *pss* for Analysis
 - **Uncheck** the *Auto Calculate* Box
 - *Beat Frequency* → 2.4G
 - *Output Harmonics* → 4
 - *Accuracy Default* → Conservative, click *Apply*
- Now at the top of **choosing Analysis** window
 - Select *pxf* for Analysis
 - *PSS Beat Frequency(Hz) = 2.4GHz* (appears automatically)
 - *Frequency Sweep Range* → Start: 1G Stop: 5G
 - *Sweep Type* → *Linear* and *Step Size* → 40M
 - *Maximum Sidebands* → 0
 - In output Section
 - Select *Voltage*
 - *Positive Output Node* → Select net RF_OUT from Schematic
 - *Negative Output Node* → Leave Empty , it means GND
 - Click *OK*
- In the **affirma** window click on *Simulation* → Netlist and Run to start the simulation, make sure that simulation completes without errors.
- Now in the **affirma** window click on the *Results* → *Direct Plot* → *Main Form*
- The *PSS* results window appears.
 - Plot mode : Append, Select *Analysis Type* → *pxf*
 - *Function* → *Voltage Gain*
 - *Sweep* → *Spectrum*
 - *Modifier* → *dB20*
 - Activate the Schematic window, click on INPUT port, OUTPUT port and VDD symbols. The Plots window pops up with plot as shown in Fig-11.
 - Please note that PSRR is extremely poor. Why?

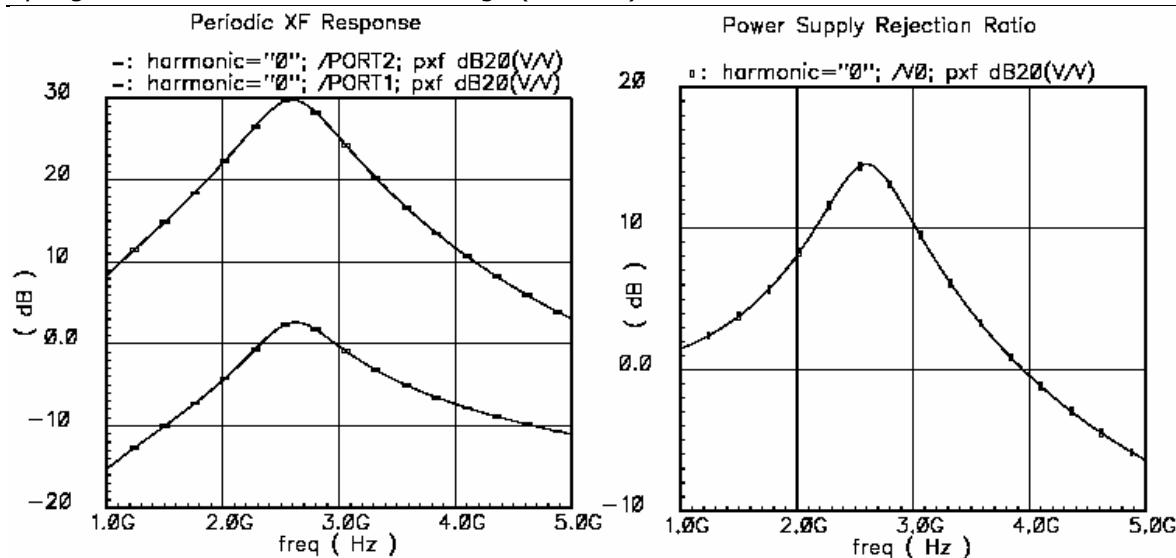


Fig11: Transfer Function and PSRR