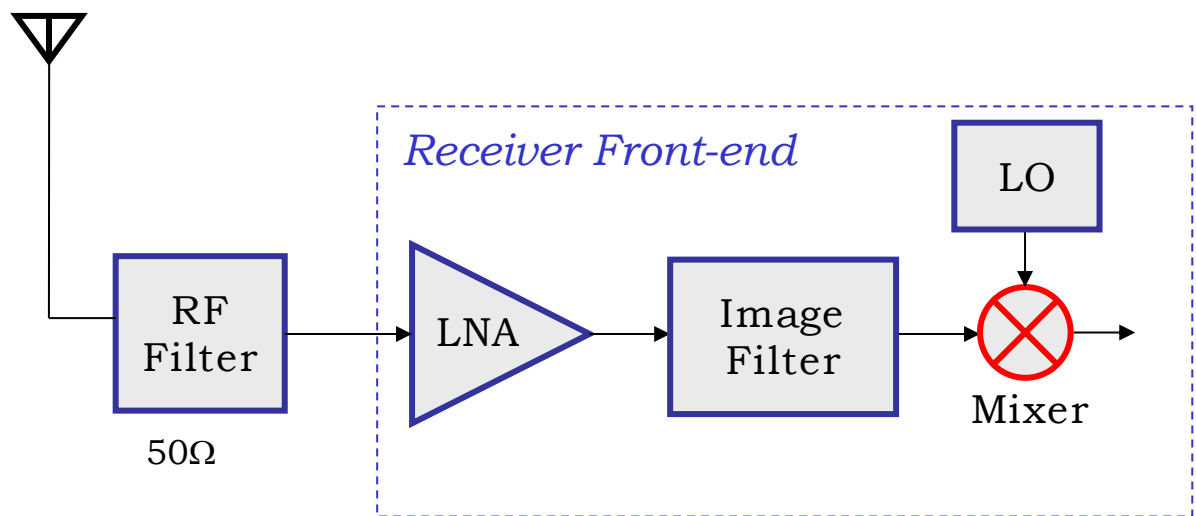# TSEK03 LAB 3:
Gilbert mixer simulation
using Cadence SpectreRF

1.   Introduction

This tutorial LAB describes how to use SpectreRF in Analog Design Environment to simulate the parameters that are important in the design and verification of a mixer. To characterize a mixer, the following figure of merits are usually simulated and measured.

1. Power Consumption
2. RF to IF Conversion Gain
3. Noise and NF
4. Input and Output Impedance Matching
5. LO to RF and LO to IF Isolation
6. Linearity

The analyses listed below are used to characterize the mixer for the above-mentioned parameters:

1. Conversion Gain
   - Voltage Conversion Gain Versus LO Signal Power (Swept PSS with PAC)
   - Voltage Conversion Gain Versus RF Frequency (PSS and Swept PAC)
   - Voltage Conversion Gain Versus RF Frequency (PSS and Swept PXF)
   - Power Conversion Gain Versus RF Frequency (QPSS)
2. Port-to-Port Isolation among RF, IF and LO Ports (PSS and Swept PAC)
3. Power Dissipation (QPSS)
4. S-Parameters (PSS and PSP)
5. Total Noise and NF, SSB and DSB Noise Figures (PSS and Pnoise)
6. Intermodulation Distortion and Intercept Points (PSS and Swept PSS)
7. Mixer Performance with a Blocking Signal (QPSS, QPAC, and QPnoise)

Lab instructions

If the lab is not finished in the scheduled time slot, you can complete it in your own time. If there is any problem, send an email or show up at the instructor's office. You must answer the questions in the lab manual before you start the tutorial. This will help you to comprehend the tutorial material and the simulations methodology.

2.   Background Preparation

Please read the Application Note "**Spectre RF Workshop**, **Mixer Design Using SpectreRF**" and your class lecture material and answer the following questions before you attend the lab.

- List the major categories (Active/Passive, single/double balanced) of the mixers, one advantage and disadvantage of each type?

- Passive mixers have better IP3 but they have conversion loss rather than gain and hence degraded NF. Gilbert Mixer is double balanced active mixer with differential topology. Please comment about the isolation, gain, NF, and IP3 characteristic of a Gilbert Mixer compared to passive mixers. Why is higher LO strength needed for a Gilbert mixer?

- Define the SSB and DSB Noise Figure of a mixer. In case of Zero-IF architecture which type of NF should be simulated and measured?

- The RF-LO, RF-IF, and LO-IF feedthrough create problems in the receiver design. Please specify one problem for each one.

  RF-LO:

  RF-IF:

  LO-IF:

- What is meant by the "desensitization" in a radio receiver?

3. Gilbert Mixer Simulation

3.1 Simulation Environment Setup

- We will be using AMS 0.35μm CMOS (c35b4) process for this Lab and Cadence version 5.
- Open a terminal window and establish a ssh connection to the ixtab server through the command: `ssh ixtab`, then input your credentials.
- Create a new directory **'tsek03_lab3'** where your simulation data will be stored: mkdir **tsek03_lab3**.
- cd **tsek03_lab3**, then do the rest of the steps from this directory.
- Load the Cadence and technology file using
    - **module add cadence/MMSIM10.1**
    - **module add cadence/IC5141_USR6**
    - **module add ams/3.80**
    - **setenv IUSDIR .**
- Start cadence by typing **ams_cds -tech c35b4 -mode fb**
- You must now choose a technology the first time you run this: choose C35B4C3 - PIP VG5 HIRES

- When creating schematics, use the RF NMOS transistors from library **PRIMLIBRF**. The transistor models are valid up till 6 GHz. The models provided in **PRIMLIB** are only valid up till 1 GHz. The maximum allowable size of NMOS in SpectreRF is 200 um (20 fingers of 10 um or 40 fingers of 5 um). **If you need larger transistors, use two transistors in parallel.**
- **BUG IN PDK 3.80:** To be able to update the total width when entering a multiple fingers, change the parameter *Width Stripe* in the Properties of the transistor cell to *5 um* and back to *10 um*. Check that the Width on the line above has the value given by Width Stripe * Number of gates.
- There are many views available when you place the symbol in the schematic, use *Symbol* or *Spectre* view only.
- 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.
- To use a balun in the schematics, you have to add **rfLib** to the Library Manager. Open the Library Path Editor from the Library Manager Window (Edit, Library Path...) and then add a new line with **rfLib** in the Library column and **/sw/cadence/IC5141_USR6/tools/dfII/samples/artist/rfLib** in the path. If the link is correct, it will turn to **blue**, otherwise, **red**. Save the new cds.lib and continue.

- Make a new library **lab3** (you can put your own name or as you like) in Cadence Library Manager and attach this library to the TECH_C35B4 technology file.
- Create and draw the Schematics **mixer** as shown in Fig. 1.
- The component values are listed in the next section for your convenience.
- Copy the **mixer_testbench** (Fig. 2) from the course library (it takes too long time to enter!), enter in the terminal window

```
cp -r /site/edu/eks/TSEK03/2017/laboration/LAB3/mixer_testbench
~/myrfdir/lab3
```

- Close and open the Cadence Library Manager. You now have two cells in your library **lab3**.

- ***For details of simulation setup please read the Cadence Setup Guidelines section of LNA Tutorial***
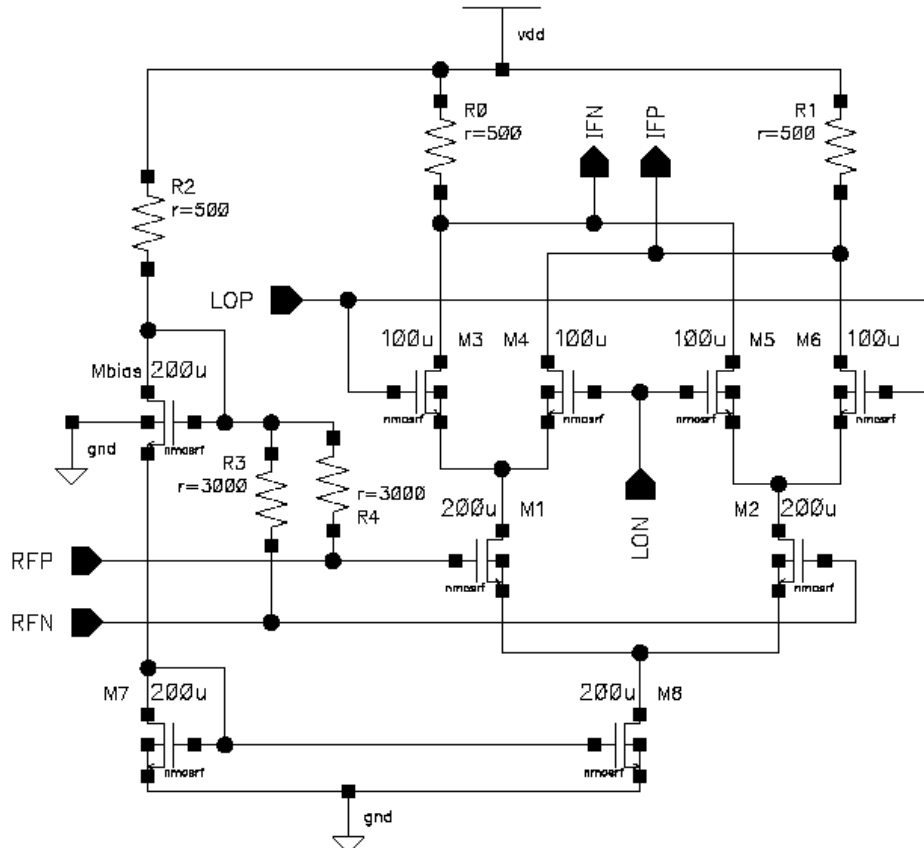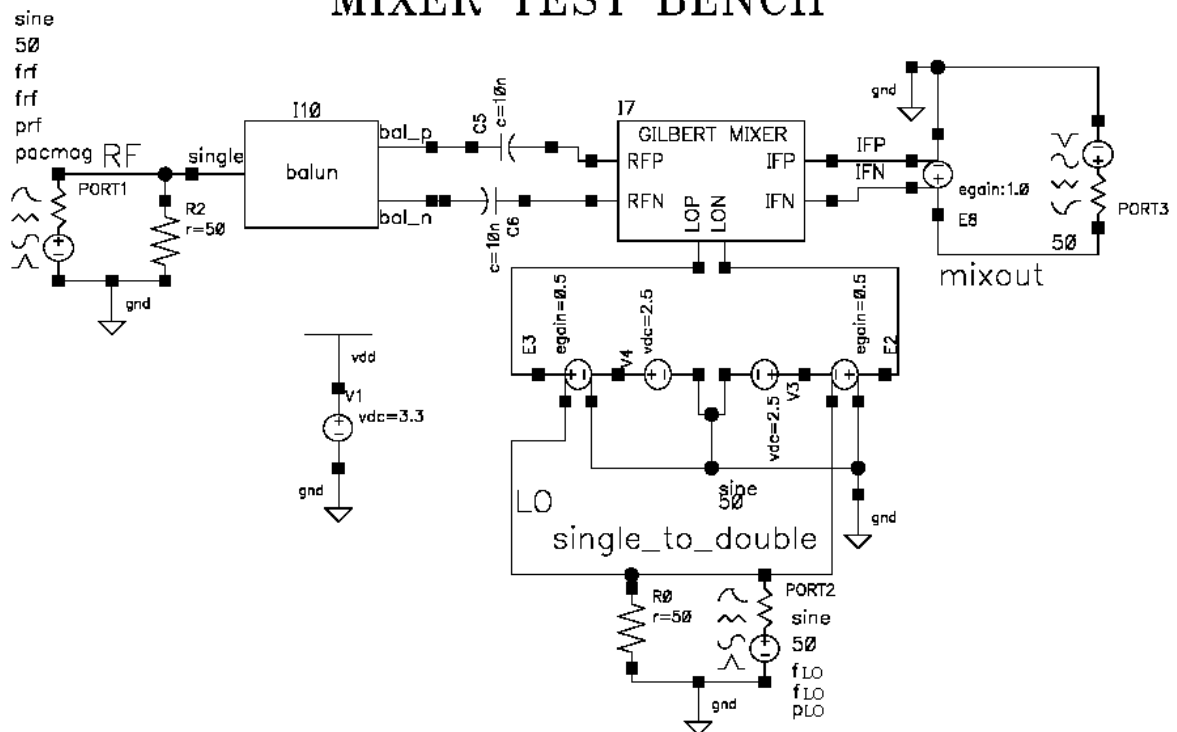
Fig 1: Gilbert Mixer Schematic

MIXER TEST BENCH

Fig 2: Test Bench of Gilbert Mixer

3.2   Circuit Simulation Setup

- RF Port in mixer_testbench Schematic
  - 50 Ohms in *Resistance*
  - 1 in *Port Number*
  - Sine or dc in *Source Type* depending upon the analysis you choose
  - Type *frf* **in** *Frequency name 1* field (choose sine for this)
  - Type *frf* in *Frequency 1 field*
  - Type *prf* in *Amplitude1 in dBm* field
  - *display small signal parameter* → check Box
  - Type *pacmag* in *PAC Magnitude field*
- LO Port in mixer_testbench Schematic
  - 50 Ohms in *Resistance*
  - 2 in *Port Number*
  - Sine in *Source Type*
  - Type *flo* in *Frequency name 1* field
  - Type *flo* in *Frequency 1 field*
  - Type *plo* in *Amplitude1(dBm)* field
- IF Port in Schematic mixer_testbench
  - 50 Ohms in *Resistance*
  - 3 in *Port Number*
  - dc in *Source Type*
- Component Values in Schematic mixer_testbench
  - Vdd = 3.3V, Coupling Capacitors= 10nF
  - RF and LO external port matching resistors = 50 Ω
  - Balun (Single input Impedance = 50 Ω, Balanced output Impedance = 50 Ω, Insertion loss = 0 dB)
  - All LO port VCVS (Type → linear, Gain=0.5)
  - IF port VCVS (Type → linear, Gain=1)
  - V3 and V4 → DC Voltage → 2.5V
- Component Values in MIXER Schematic
  - M1, M2, M7, M8, Mbias = 200µm/0.35µm
  - M3, M4, M5, M6 = 100µm/0.35µm
  - R0, R1, R2 = 500 Ω and R3, R4 = 3000 Ω

3.3   Voltage Conversion Gain

A mixer's frequency converting action is characterized by conversion gain (active mixer) or loss (passive mixer). ***The voltage conversion gain is the ratio of the RMS voltages of the IF and RF signals.*** The ***power conversion gain is the ratio of the power delivered to the load and the available RF input power.*** When the mixer's input impedance and load impedance are both equal to the source impedance, the power and voltage conversion gains, in decibels, are the same. Note that when you load a mixer with a high impedance filter, this condition is not satisfied.

You can calculate the voltage conversion gain in two ways:
- Using a small signal analysis, like PSS with PAC or PXF. The PSS with PAC or PXF analyses supply the small-signal gain information. A second method is to use a two-tone large-signal QPSS analysis, which is more time-consuming.
- The power conversion gain also requires two-tone large-signal QPSS analysis.

a)   Voltage Conversion Gain versus the LO Signal Power (swept PSS with PAC)

- RF Port Parameters in the Schematic
  - *Resistance* → 50 Ω, *Source Type* → *DC*
- LO Port Parameters in the Schematic
  - *Resistance* → 50 Ω, *Source Type* → *sine (flo, flo, plo)*
- IF Port Parameters in the Schematic
  - *Resistance* → 50 Ω, *Source Type* → *DC*

- Start the **Virtuoso Analog Design Environment (ADE)** window (Tools, Analog Environment) and enter the Design variables values.
  - *frf = 2.4G, flo = 2.4G*
  - *prf = -50* and *plo = 10* both in dbm field
  - *pacmag = 1*

- Select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
  - Select *PSS* for Analysis
  - Check the *Auto Calculate* Box
  - Check that *Fundamental tone* looks like
    - flo   flo   2.4G   Large   PORT2
  - *Output Harmonics* → 10
  - *Accuracy Default* → Moderate,
  - Sweep → variable (plo), Sweep Range → -10 to 20, Sweep Type → Linear
  - No of steps → 10,
    Enable and Apply.

- Now at the top of **choosing Analysis** window
  - Select *PAC* for Analysis

- Frequency Sweep Range → 2.4G
- Sideband → Max Sideband →2
- Enable and Apply/close

Before you can run any simulations, you have to do the following:

1. In the ADE Window, menu Setup, Environment... Switch View List, add *"veriloga"* at the end of the list.

2. **ANOTHER BUG IN THE PDK 3.80** for the transistor models, but if you switch to the model in PDK 3.70, it will work! In the ADE Window, menu Setup, Model Libraries, carefully edit first line to "ams-hit-3.70" at the line referring to the cmos53.scs model file:
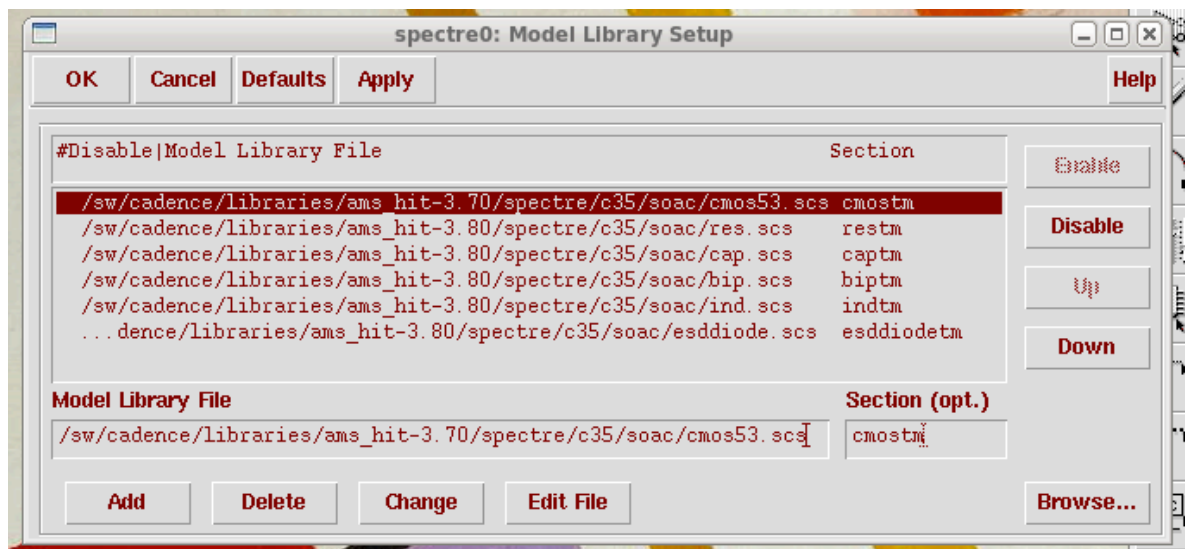

Fig 3:   Change model reference for cmos53.scs

In the **ADE** window click on *Simulation*→ Netlist and Run to start the simulation, make sure that the simulation completes without errors.

- In the **ADE** window click on the *Results→ Direct plot (main form)*
- The *PSS* results window appears.
    - *Analysis Type → PAC*
    - *Function → Voltage, Select → net*
    - *Sweep → Variable, Signal Level → Peak*
    - *Modifier → dB20, Output Harmonics →-1*
    - *Select network **mixout** node in schematics*
    - *You will see the plot as shown in Fig 4.*

Note-1: The PAC analysis calculates the gain directly when the *pacmag* parameter is 1V. If this is not the case take the ratio of input and output.
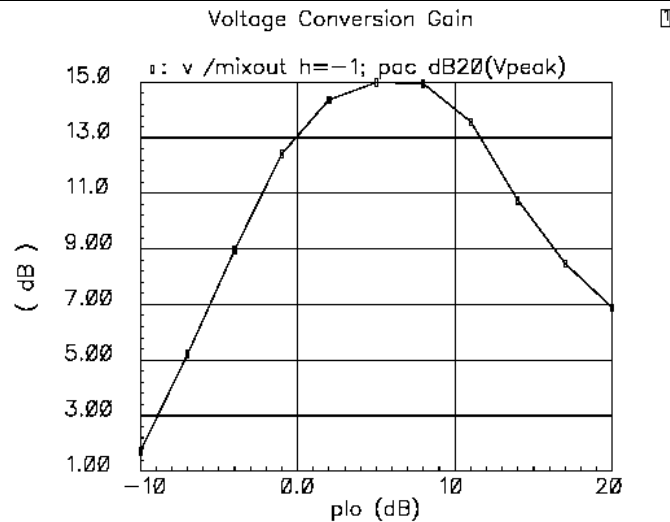
Fig 4: Voltage Conversion Gain versus the LO Signal Power

Note-2: The *plo* for maximum gain is 5 dBm in this case. We will use this value in the subsequent simulations.

b)   Voltage Conversion Gain versus RF Frequency (PSS with swept PAC)

- Test Bench Parameters same as part (a)
- In Design variables
  - Change plo = 5
- Now at the top of **choosing Analysis** window
  - *Select PAC for Analysis*
  - *Frequency Sweep Range → 2.4G to 2.41G*
  - *Sideband → Max Sideband → 2*
  - *Enable and apply*
- The *Choose Analysis* window shows up
  - *Select PSS for Analysis*
  - *Uncheck the Auto Calculate Box*
  - *Set fundamental tone → flo   flo   2.4G (press update from schematic button), it looks like*
    - flo   flo   2.4G   Large   PORT2
  - *Beat Frequency → 2.4G, Output Harmonics → 10*
  - *Accuracy Default → Moderate*
  - *Switch off the sweep option*
  - *Enable and apply*

- In the **ADE** window click on *Simulation → Netlist and Run to start the simulation, make sure that simulation completes without errors.

- In the **ADE** window click on the *Results→ Direct plot (main form)*
  - *Analysis Type → PAC*
  - *Function → Voltage, Select → net*
  - *Sweep → Sideband, Signal Level → Peak, Modifier → dB20*
  - *Output Harmonics→ -1 0   -10M*
  - *Select **mixout** node in schematics*
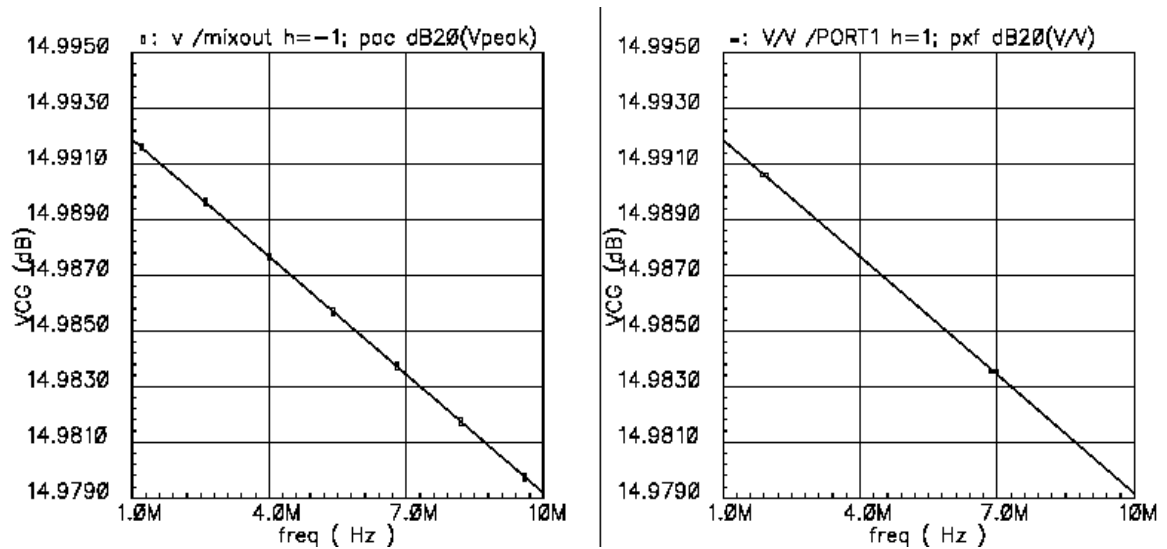 - *You will see the plot as shown in Fig 5.*

Fig 5:   Voltage Conversion Gain versus RF frequency using PAC and PXF
(PXF simulation right removed from in this lab.)

3.4   Port-to-Port Isolation among (PSS, Swept PAC and Swept PXF)

The PAC and PXF analysis can be combined to produce the transfer function from different ports to each other. Here, we will simulate the RF-LO, RF-IF and LO-IF feedthrough.

RF-LO feedthrough affects the LO if a strong blocker is present at the RF input. RF-IF feed-through creates even order distortion for Zero-IF receivers. LO-IF feedthrough must be limited to avoid the desensitization problem in the stage following the mixer.

- The test bench is the same as in the *voltage conversion gain* analysis.
- Make sure *plo = 5* in the design variables
- RF port type: *Resistance* → 50 Ω, *Source Type* → *sine (as in earlier analysis)*
- Now at the top of **choosing Analysis** window
- The *Choose Analysis* window shows up
  - Select *PSS* for Analysis
  - Uncheck the *Auto Calculate* Box
  - *Set fundamental tone* → (press update from schematic button , it looks like

    | flo | flo | 2.4G | Large | PORT2 |
    | frf | frf | 2.4G | Large | PORT1 |

  - *Beat Frequency* → 2.4G
  - *Output Harmonics* → 10, *Accuracy Default* → Moderate
  - Switch off the sweep option
  - Enable and apply
- Now at the top of choosing **Analysis** window
  - *Select PAC for Analysis*
  - Frequency Sweep Range → 2.4G to 2.41G
  - Sideband → Max Sideband → 2,
  - Sweep Type → Automatic
  - Enable and apply
- Now at the top of **choosing Analysis** window
  - *Select PXF for Analysis*
  - *Frequency Range* → 2.4G to 2.43G
  - *Sideband* → *Max Sideband* → *2,*
  - *Sweep Type* → *automatic*
  - Output → voltage,
  - Positive output node → mixout (from schematic)
  - Negative output node → gnd (from schematic); Enable and apply

- In the **ADE** window click on *Simulation* → Netlist and Run to start the simulation, make sure that the simulation completes without errors.

RF-to-LO Feedthrough:

- In the **ADE** window click on *Results → Direct plot*
- The results window appears.
    - *Analysis Type → PAC*
    - *Function → Voltage*
    - *Select → net*
    - *Sweep → Sideband*
    - *Signal Level → peak*
    - *Modifier → dB20*
    - *Output Harmonics → -1 0 to 10M* (This represents the down converted RF signal at LO port)
    - *Select LO port, see the results in Fig 6 (left)*

RF-to-IF Feedthrough:

- Now just change.
    - *Output Harmonics → 0 2.4G - 2.41G* (This represents the RF signal to IF port without down conversion)
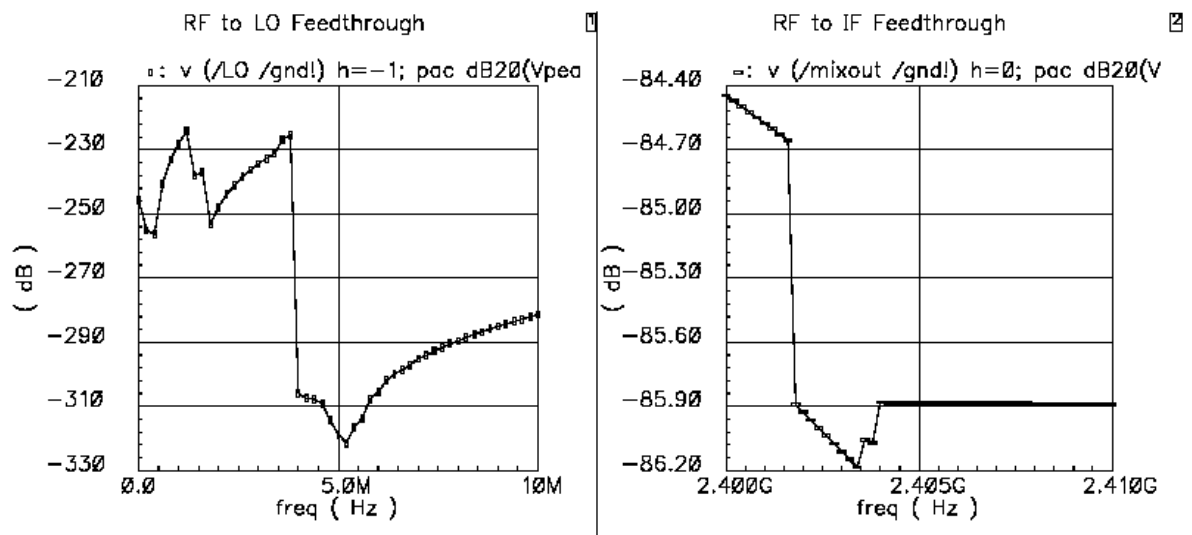   - *Select IF port, see the results in Fig 6 (right)*

Fig 6: RF-to-LO & IF Feedthrough

LO-to-IF Feedthrough:

- In the **ADE** window click on *Results* → *Direct plot (main form)* → *PSS*
- The *PSS* results window appears.
  - *Analysis Type* → *PXF*
  - *Function* → *Voltage, Sweep* → *Sideband, Modifier* → *dB20*
  - *Output Sideband* → *0 2.4G -2.43G*
  - *Select the* **LO port** *in the schematic, see the results in Fig 7 (left).*

LO-to-RF Feedthrough:

  - *Now select* **RF port** *instead of the LO port in the schematic, see the results in Fig 7 (right).*
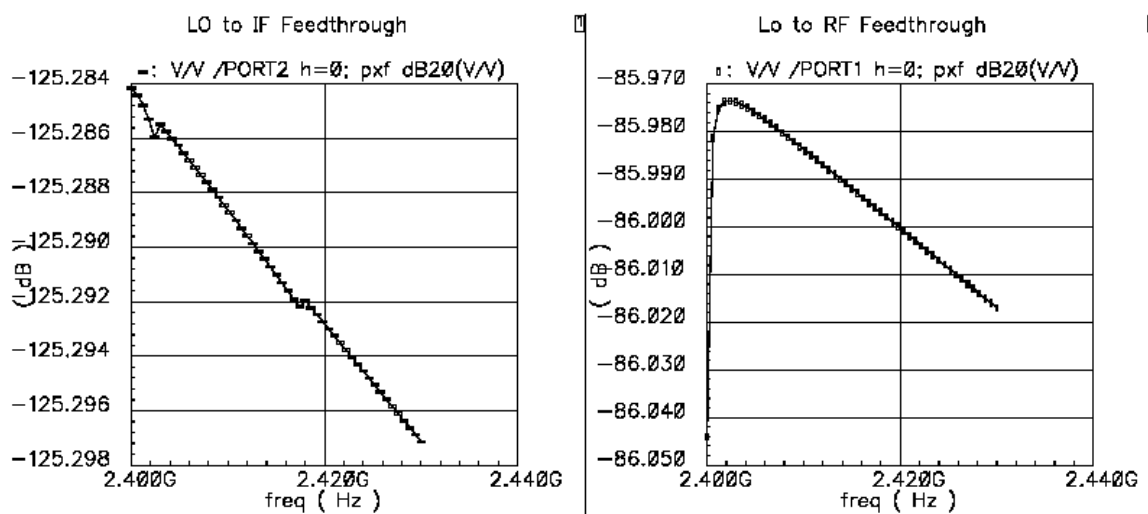

Fig 7: LO-to-IF Feedthrough

3.5   Power Dissipation, Large Signal Power (Voltage) Conversion Gain (QPSS)

QPSS (Quasi Periodic Steady State Analysis) is an analysis that invokes a series of PSS like analyses over all the input frequencies, their harmonics and the inter-modulation of the frequencies and harmonics.

QPSS allows arbitrary signal inputs, including sum of sinusoids that are not periodic, so called quasi-periodic extension of PSS. Similar to PAC (Periodic AC analysis) it calculates the responses of the circuits that exhibit the frequency translation like mixer, oscillator etc. Unlike PAC, PSS is not explicitly required before QPSS as it simulates the moderate and large signal behavior instead of small signal behavior.

- Disable all other analysis
- RF Port Parameters in the Schematic
    - *Resistance* → 50 Ω, *Source Type* → *sine (frf, frf, prf)*
- LO Port Parameters in the Schematic
    - *Resistance* → 50 Ω, *Source Type* → *sine (flo, flo, plo)*
- IF Port Parameters in the Schematic
    - *Resistance* → 50 Ω, *Source Type* → *DC*
- Verify that the Design variables values in the **ADE** window are
    - *frf = 2.41G, prf = -30, flo = 2.4G, plo = 5, pacmag = 1*

- In the **ADE** window, select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
    - Select *QPSS* for Analysis
    - Click → update from the schematic
    - You should see the lines below (change the harmonics manually to 5 and 3. Your port numbers may be different)

flo	flo	2.4G	large	port2	5
frf	frf	2.41G	moderate	port1	3

    - Accuracy → moderate
    - Enable and apply

- In the **ADE** window click on *Simulation* → Netlist and Run to start the simulation, make sure that simulation completes without errors.

- In the **ADE** window click on the *Results* → *Direct plot (main form)*
- The results window appears.
    - *Analysis Type* → *qpss, Function* → *power*
    - *Select* → *instance with two terminal, Modifier* → *dB10*
    - *Select the voltage source **vdc =3.3 V** in the schematics*
    - *You will see the plot as shown in Fig 8*

Note: QPSS and PSS provide the spectrum, not a scalar value. Summation of harmonics and sidebands gives a good estimate of the total power consumption. Most of the power is in the main output harmonics.
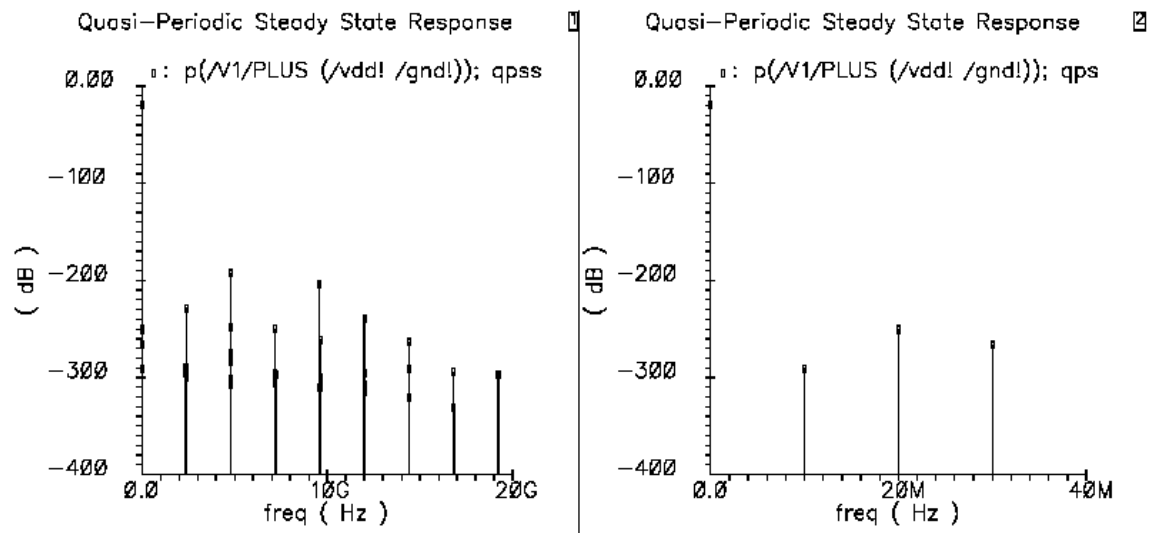
Fig 8: Large Signal Voltage Conversion Gain

3.6   S-Parameters (PSS and PSP)

QPSS (Quasi Periodic Steady State Analysis) is an analysis that invokes a series of PSS like analysis over all the input frequencies, their harmonics and the inter-modulation of the frequencies and harmonics.

- In Design variables
    - Change RF port → dc
- Verify the variable values in the **ADE** window
    - *flo = 2.4G (frf, prf, pcmag are meaningless in this analysis)*
    - *plo = 5*
- Disable previous QPSS analysis; Now at the top of **choosing analysis** window
- The *Choose Analysis* window shows up
    - *Select PSS for Analysis, Uncheck the Auto Calculate Box*
    - *Set fundamental tone → (press update from schematic button)*
        flo   flo 2.4G Large   PORT2
    - *Beat Frequency → 2.4G, Output Harmonics → 10*
    - *Accuracy Default → Moderate, Enable and apply*
- The *Choose Analysis* window shows up
    - *Select PSP for Analysis*
    - *Sweep type → absolute* (If you choose relative, you can see results on scale of 2.4 GHz and onward)
    - *Start-stop → 1k -- 10M, Sweep Type → Automatic*
    - *Press **Select port** button and point to the RF, IF and LO ports in the schematic, and enter the desired data*
        1 PORT1   1   1K - 10M   (RF)
        2 PORT3   0   - 2.4G - 2.39G   (IF)
        3 PORT2   1   1K - 10M   (LO)
    - The order of ports is important. In our case PORT1 (RF) is numbered 1 and PORT3 (IF) is numbered 2. These are considered as input and out ports for noise analysis respectively.
    - *Do Noise → Yes, Maximum sidebands → 10, Enable and apply*

- In the **ADE** window click on *Simulation →* Netlist and Run to start the simulation, make sure that the simulation completes without errors.

- In the **ADE** window click on the *Results → Direct plot → Main form*
- The *PSS* results window appears.
    - *Analysis Type → psp, Function → SP or NF or NFdsb*
    - *Plot Type → Rectangular, Modifier → dB20*
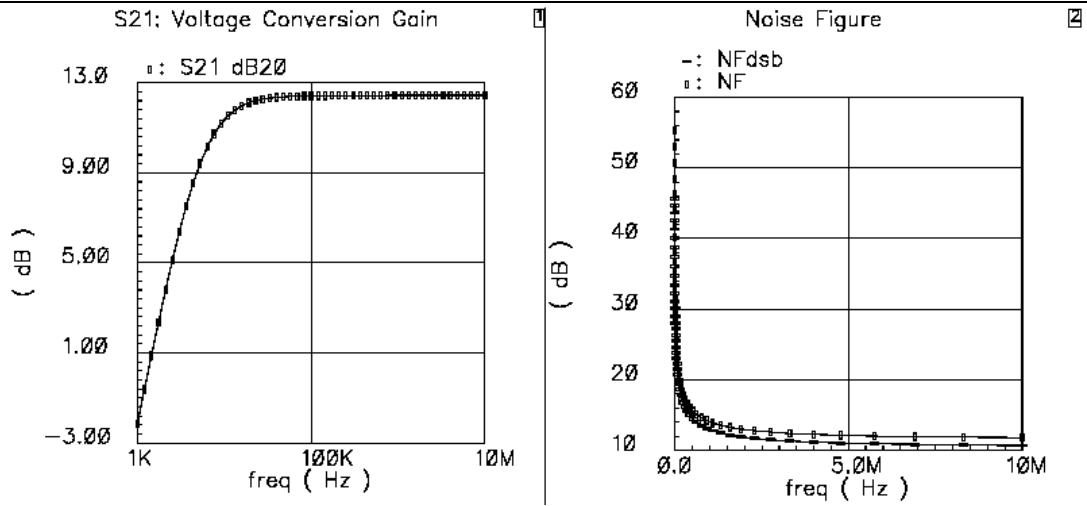   - *You will see the plot as shown in Fig 9 and Fig 10.*
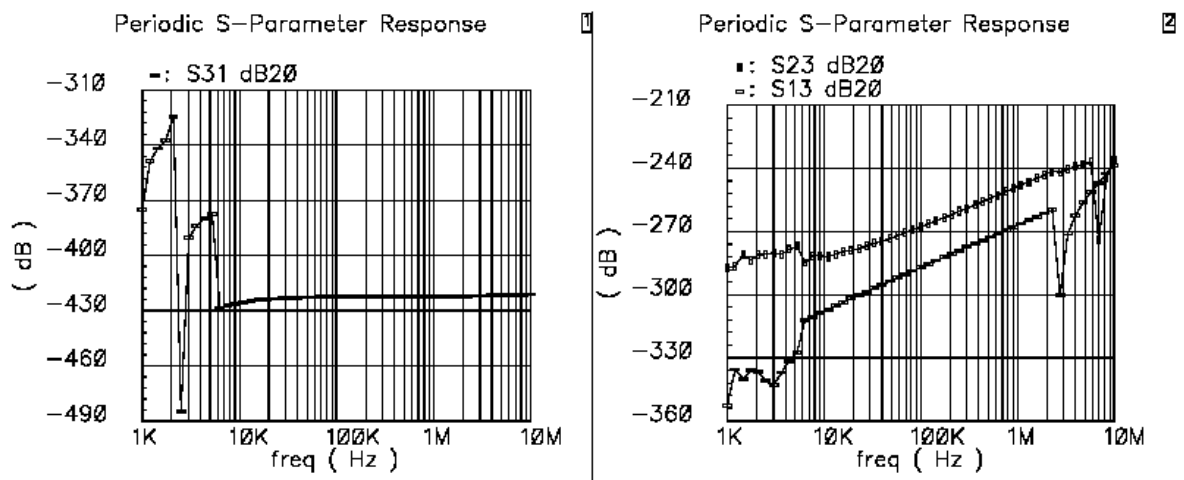
Fig 9: NF and S-Parameter Plots

Fig 10:   S-Parameters Isolation Plots

3.7 Noise Figure (PSS and Pnoise)

Typically, the signal present at the image frequency is not desired. The mixer translates both the RF and the image signals to the same IF. So for a noiseless mixer the output SNR is half the input SNR. NF_{SSB} of a noiseless mixer is 3 dB.

$$NF = 10\log\frac{SNR_{RF}}{SNR_{IF}} \quad [dB]$$

$$NF_{DSB} = 10\log\frac{(S_{RF} + S_{IM}/N_{RF} + N_{IM})}{(S/N)_{IF}}$$

$$NF_{SSB} = 10\log\frac{(S_{RF}/N_{RF} + N_{IM})}{(S/N)_{IF}}$$

$$NF_{SSB} \approx 3dB + NF_{DSB}$$

However, in some applications (direct conversion receivers) the signal present at the image frequency contains useful information, and hence the NF_{DSB} is measured and calculated.

- In schematic
  - *RF port → dc (prf, frf, pcmag are meaningless)*
  - *LO port → sine (flo, flo, plo)*
- Verify the variable values in the **ADE** window
  - *flo = 2.4 G , plo= 5*
- Now at the top of **choosing Analysis** window
- The *Choose Analysis* window shows up
  - *Select PSS for Analysis*
  - *Uncheck the Auto Calculate Box*
  - *Set fundamental tone → flo   flo   2.4G (press update from schematic button)*
  - *Beat Frequency → 2.4G*
  - *Output Harmonics → 10*
  - *Accuracy Default → Moderate*
  - Sweep → variable
  - Variable name→ plo
  - Sweep Range → -10 to 20
  - Sweep Type → Linear
  - No of steps → 10
  - Enable and apply
- The *Choose Analysis* window shows up
  - *Select Pnoise for Analysis*
  - *Sweep type → absolute*
  - *Single point → 10M* (noise is calculated at this frequency, the 1/f noise effect will be not present; to see it make this frequency 10k or 1k)
  - *Maximum sideband → 10*
  - Output → voltage → select mixout and gnd
  - Input source → port → select RF port
  - *Reference sideband → -1*

- Noise Type → sources.
- Enable and apply

- In the **ADE** window click on *Simulation* → Netlist and Run to start the simulation, make sure that the simulation completes without errors.

- Now in the **ADE** window click on the *Results* → *Direct plot (main form)* → *Pnoise*
- The *PSS* results window appears.
  - *Analysis Type* → *Pnoise*
  - *Function* → *NF or NFdsb or Output Noise*
  - *Integrated over Bandwidth,* Press *Plot.*
  - *You will see the plot as shown in Fig 11*

Note: If you select output as probe instead of voltage and point to IF port, you can get all types of NFs, noise correlation matrices and equivalent noise parameters.
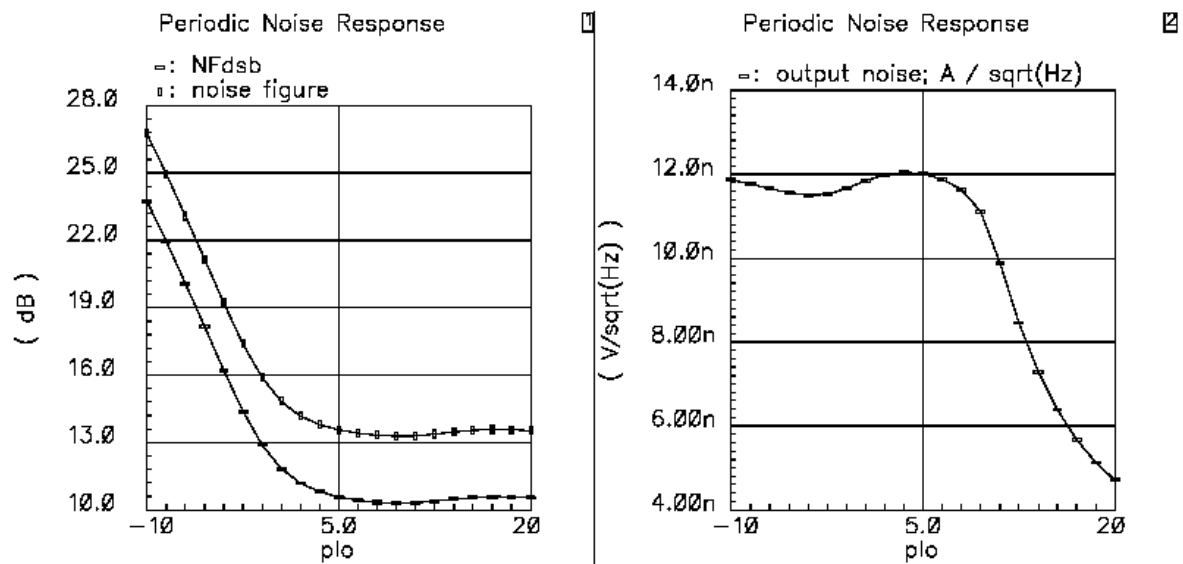

Fig 11:   Noise Figure SSB, DSB and Output Noise

3.8 1dB Compression and IIP3 (QPSS & QPAC)

In small signal conditions the output power increases linearly with increase in the input signal power. When circuits shift toward large signal operation this relation is no longer linear. The 1-dB compression point is a measure of this nonlinearity. This is power where the output of the fundamental crosses the line that represents the output power extrapolated from small signal conditions minus 1-dB.

The recommended approach to calculate the 1-dB compression point and IIP3 is to apply large LO and one medium RF tone and perform the QPSS analysis. Then you apply the second tone as a small tone close to the RF signal frequency and perform the QPAC. The power of the 2nd small signal RF tone has to be small enough that IM1 and IM3 are in their asymptotic ranges.

- Change/Check the LO Port Parameters in the Schematic Window
  - *LO port → sine (flo, flo, plo)*
  - *IF port → DC and 50 Ohms*
- Change the RF Port Parameters in the Schematic Window
  - **Sine** in *Source Type*
  - *frf* in *Frequency name 1* field
  - *frf* in *Frequency 1 field*
  - *prf* in *Amplitude1(dBm)* field

- Verify the variable values in the **ADE** window
  - *flo = 2.4G, frf = 2.401G, prf = -10, plo = 5, pacmagdb = prf*

- In the **ADE** window, select *Analysis → Choose*
- Disable previous analysis; The *Choose Analysis* window shows up
  - Select *qpss* for Analysis
  - In Fundamental Tones, the following lines should be visible (if its different please change them)

    | flo | flo | 2.4G | Large | 5 | PORT2 |
    | frf | frf | 2.401G | Moderate | 4 | PORT1 |

  - *Accuracy Default →* Moderate
  - High light the *Sweep* Button
  - Select Design Variable, small window appears, choose *prf* in it
  - *Sweep Range →* Choose start: -70 dBm and stop: 10 dBm
  - *Sweep Type →* Linear and *No of Steps* = 15
  - *Enable* Box in the bottom should be checked.
- Now at the top of **choosing Analysis** window
  - *Select QPAC for Analysis*
  - *Sweep Type → absolute, Freq → 2.4011G*
  - *Max Clock Order → 2, Enable and apply*
- Click OK in the **ADE** window click on *Simulation →* Netlist and Run to start the simulation.

- In the **ADE** window, select *Results → Direct plot → Main Form*

- *Analysis → QPSS*
- Select *Function → Compression Point*
- *Gain Compression → 1dB*
- *Extrapolation Point → -70dB*
- *1st Order Harmonic → -1 1 (1M)*
- *Input Referred 1 dB compression*
- *Select Port (Fixed R (Port)) → click IF PORT*
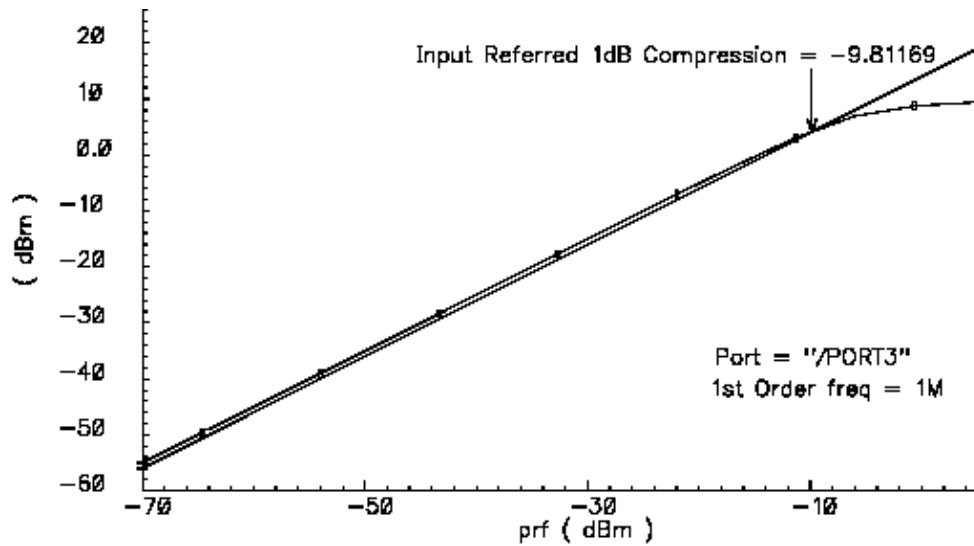- *The resulting plot is shown in Fig 12*

Fig 12:   1-dB Compression point and IIP3

- In the **ADE** window, select *Results → Direct plot (main form) → Main Form*
  - *Analysis → QPAC , Function → IPN Curves*
  - *Select Port (Fixed R (Port))*
  - *Highlight variable Sweep Prf*
  - *Extrapolation Point → -60 dB*
  - Highlight *Input Referred IP3, Order → 3rd*
  - *3ʳᵈ Order Harmonic → 1 -2 (900K)*
  - *1st Order Harmonic →-1 0 (1.1M)*
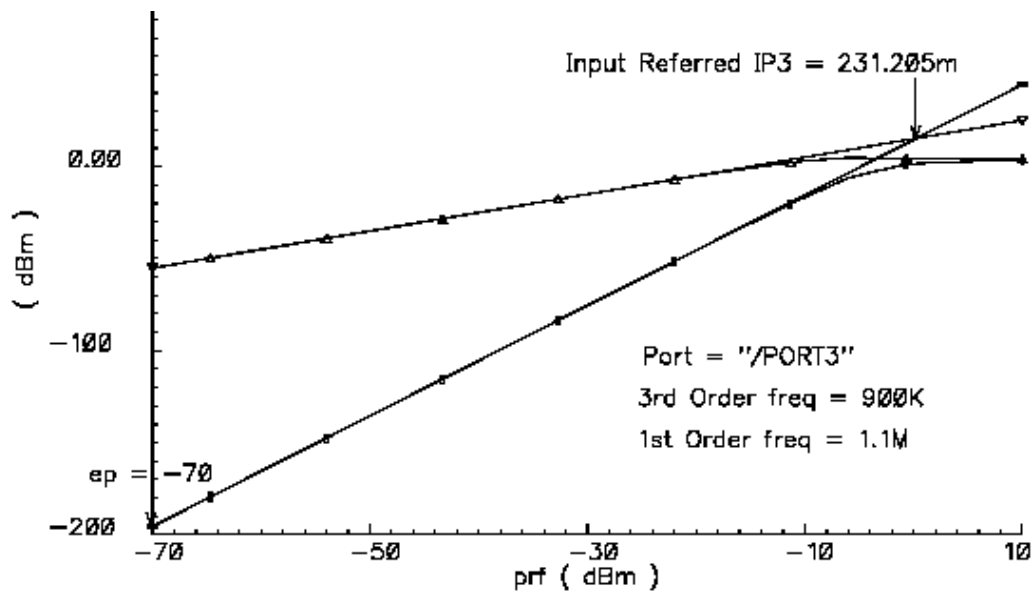 - *Activate the Schematic Window and click on IF port to view the results as shown in Fig 13*

Fig 13:   IIP3 using QPSS and QPAC

3.9 Effect of the Blocker on Gain and NF of Mixer (QPSS, QPAC and QPnoise)

In-band and out-of-band blockers are specified for all standards (GSM, DECT, etc) as discussed in class lectures. These blockers desensitize the receiver i.e. the gain and NF of the receiver because the desired signal is drastically degraded. All communication standards include the blocking requirement for both mobile terminals and base stations. The requirement defines several in-band and out-of-band blockers.

- Change/Check the LO Port Parameters in the Schematic Window
  - *LO port* → *sine (flo, flo, plo)*
  - *IF port* → *DC and 50 Ohms*

- Change the RF Port Parameters in the Schematic Window
  - **Sine** in *Source Type*
  - *frf* in *Frequency name 1* field
  - *frf* in *Frequency 1 field*
  - *prf* in *Amplitude1(dBm)* field

- Verify the variable values in the **ADE** window
  - *flo = 2.4G, frf = 2.403G*
  - *prf = -50 ,plo = 5, pacmagdb = -30*

- In the **ADE** window, select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
  - Select *qpss* for Analysis
  - In Fundamental Tones, the following lines should be visible (if its different please change them)

    | flo | flo | 2.4G | Large | 5 | PORT2 |
    | frf | frf | 2.403G | Moderate | 4 | PORT1 |

  - *Accuracy Default* → Moderate
  - High light the *Sweep* Button
  - Select Design Variable, small window appears, choose *prf* in it
  - *Sweep Range* → Choose start: -50 dBm and stop: 10 dBm
  - *Sweep Type* → *Linear* and *No of Steps* = 15
  - *Enable* Box at the bottom should be checked.
- Now at the top of **choosing Analysis** window
  - *Select QPAC for Analysis*
  - *Sweep Type* → *absolute*
  - *Freq* → *2.401G*
  - *Max Clock Order* → *2*
  - *Enable and apply*
- Now at the top of **choosing Analysis** window
  - *Select QPNoise for Analysis*
  - *Sweep Type* → *absolute, Freq* → *1M*
  - *Max Clock Order* → *10*

- ▪ *Output → Probe → select PORT3 (IF-Port)*
- ▪ *Input → Probe → select PORT1 (RF-Port)*
- ▪ *Select Reference Side Band → (1 0),*
- ▪ *Enable and apply*

- Click OK in the **ADE** window click on *Simulation →* Netlist and Run to start the simulation.

- In the **ADE** window, select *Results → Direct plot (main form) → Main Form*
  - ▪ *Analysis → qpnoise, Function → Noise Figure*
  - ▪ Select *Noise Figure, Integrated over Bandwidth,* Press *Plot.*
  - ▪ Select *NFdsb,* Press *Plot.*
  - ▪ *View the results as shown in Fig 14 (left)*

- In the **ADE** window, select *Results → Direct plot (main form) → Main Form*
  - ▪ *Analysis → qpac, Function → voltage*
  - ▪ Select *Instance with two terminals*
  - ▪ *Sweep → variable, peak, Modifier → dB20*
  - ▪ *Output Harmonic → 1M (-1 0)*
  - ▪ *Click in Schematic on IF port*
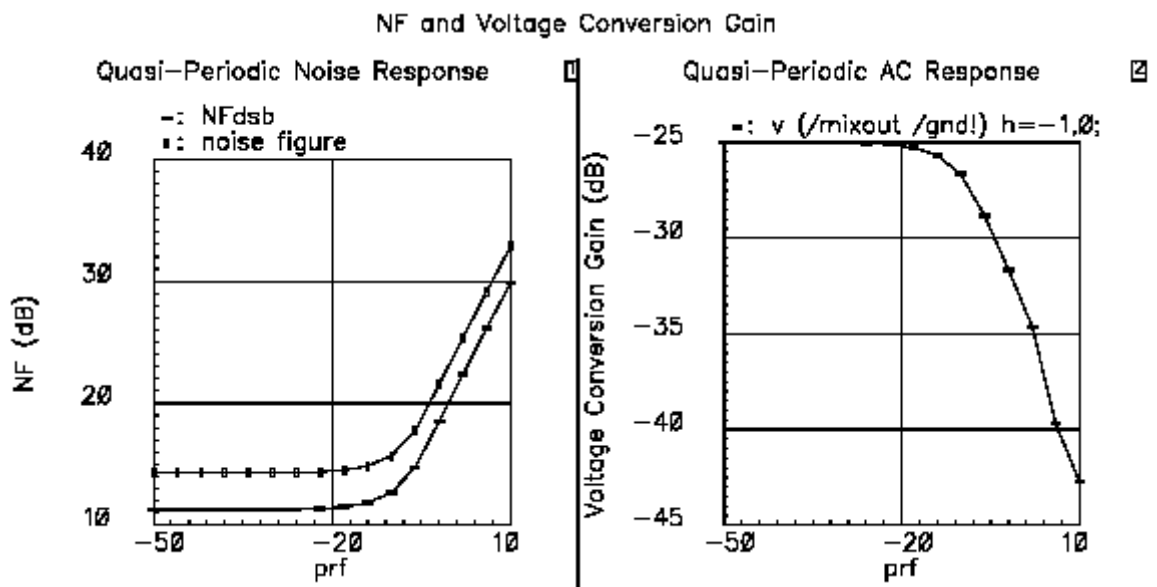 - ▪ *View the results as shown in Fig 14 (right)*

Fig 14:   Voltage Conversion Gain & NF in presence of Blocking Signal