Date: ______________

Student Name: ___________________________ TA Name: ___________

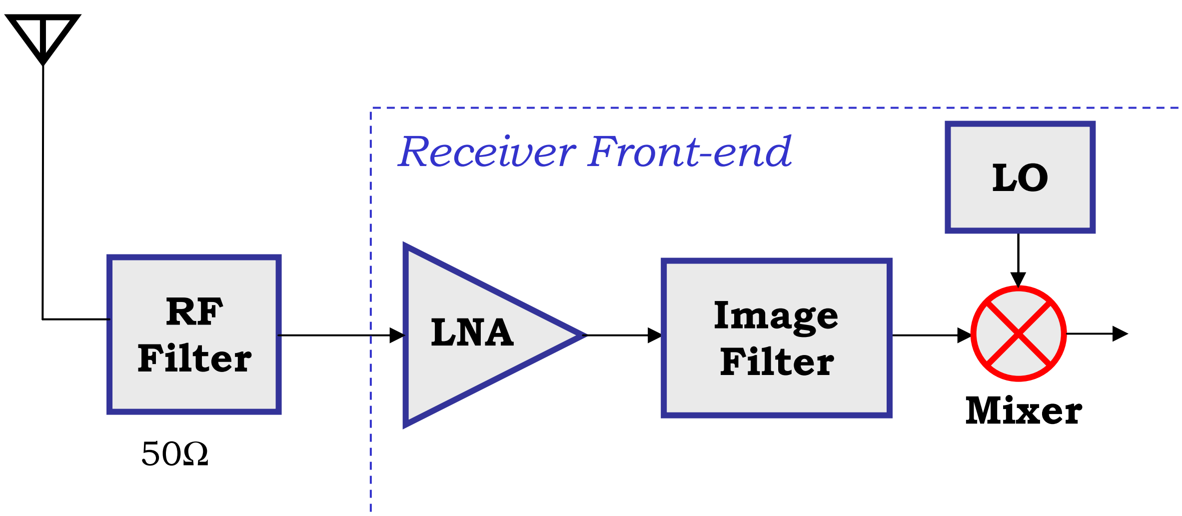
Personal Number: - TA Signature: ________________

Notes:

(The mixer tutorial and assignment at the end of tutorial is optional)

LAB-2 (Tutorial)
Gilbert Mixer Simulation
(Cadence SpectreRF)

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



Introduction:

This Tutorial LAB describes how to use SpectreRF in Analog Design Environment to simulate the parameters which are important in design 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)

Instructions

- You can complete this tutorial in your own time, if there is any problem please 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 comprehend the tutorial material and simulations.

Cadence Setup and Guidelines

Please read the "*Cadence Setup and Guidelines*" section LNA Tutorial.

1. Background Preparation

Please answer the following questions before the LAB. For answers look at the lecture notes and text books for this course.

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

- How can we relate the blocker strength (both in case of in band and out of band blockers) allowed in certain standard with LO leakage?

- Why can we not use AC and SP analysis for Mixer circuits, why are periodic and quasi periodic types of analysis mandatory to simulate these circuits?

- 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 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 problem in receiver design. Please specify one problem for each of them.

RF-LO:

RF-IF:

LO-IF:

- What is meant by the “desensitization” in radio receiver?

- Why are even order distortions “fatal” for ZeroIF receiver designs and are ignorable for superheterodyne receivers? Is differential LNA and Mixer a remedy to this problem?

2. Gilbert Mixer Simulation

2.1. Simulation Environment Setup

- We will be using AMS 0.35μm 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 **new_lab** (you can put your own name or as you like) in Cadence Library Manager
- Create and draw the Schematics, mixer_testbench a as shown in Fig-1 and mixer as shown in Fig-2. The components values are listed in next section for your convenience.
- ***For details of simulation setup please read the Cadence Setup Guidelines section of LNA Tutorial***

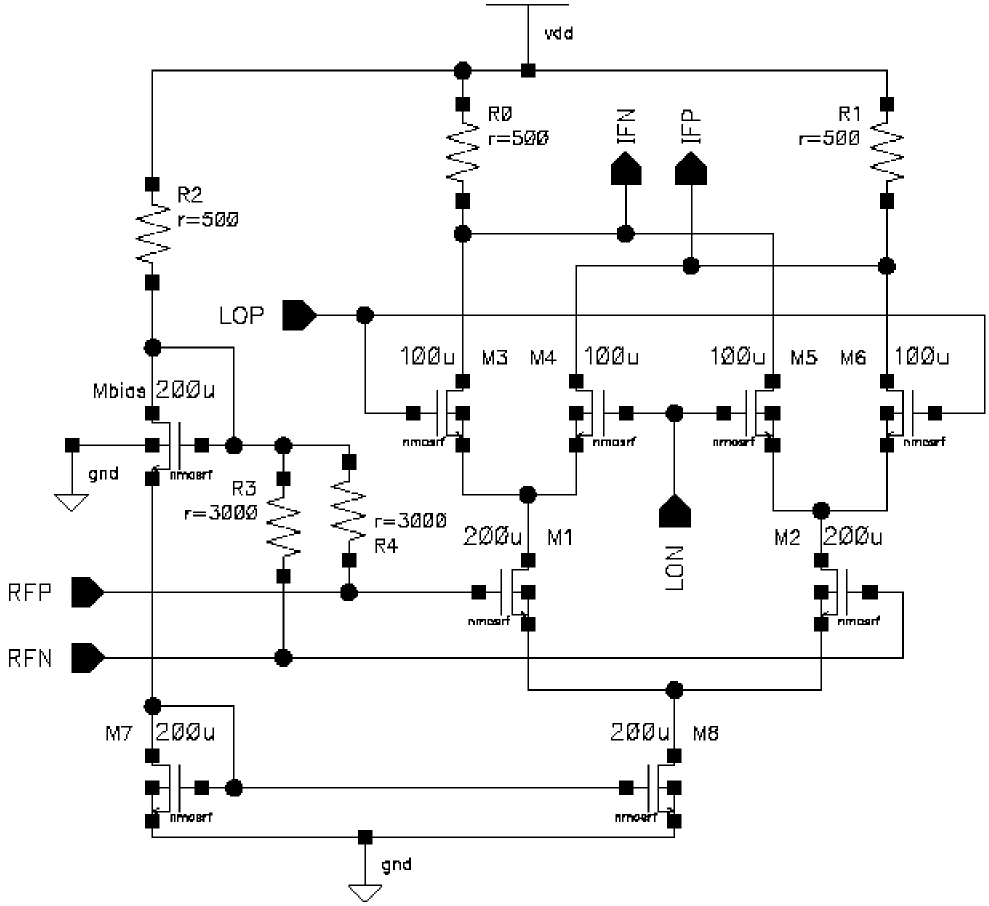


Fig 1: Gilbert Mixer Schematic

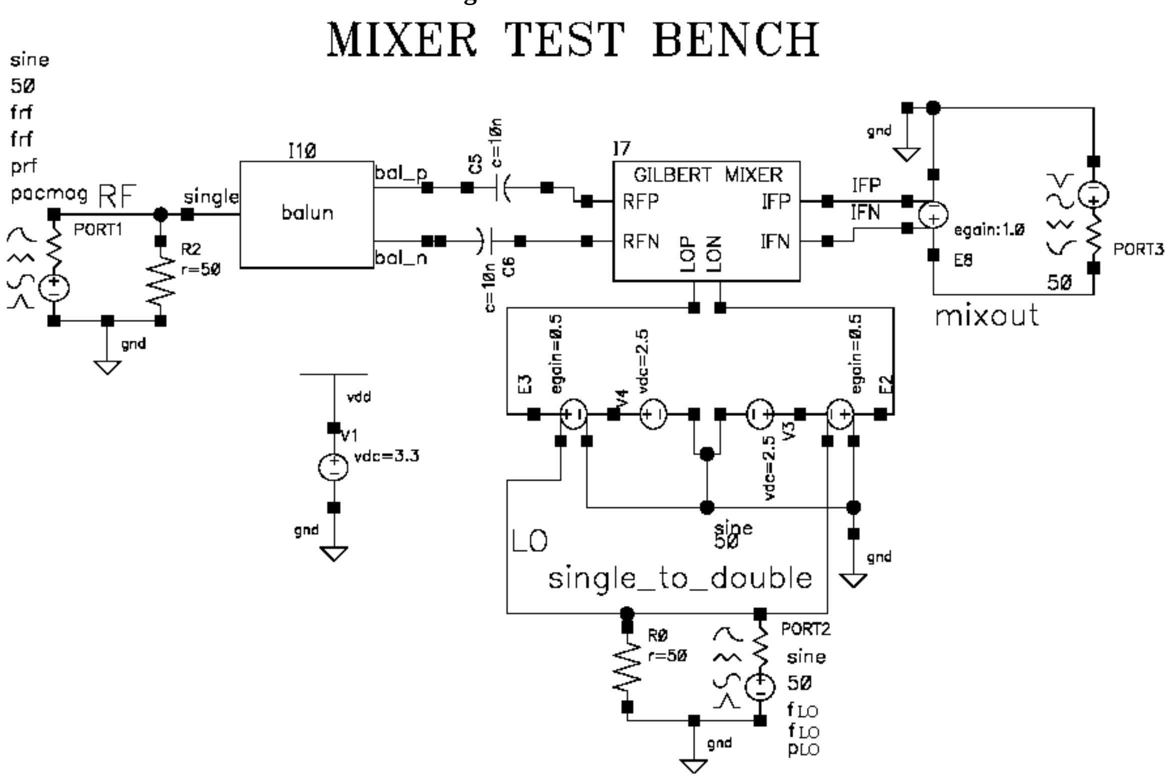


Fig 2: Test Bench of Gilbert Mixer

2.2. 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 = 0db)
  - All LO port VCVS (Type → linear, Gain=0.5, gain =0.5)
  - IF port VCVS (Type → linear, Gain=1, 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

2.3. 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*
- Verify the Design variables values in the **affirma** window are
  - *frf = 2.4 GHz, flo = 2.4 GHz*
  - *prf = -50* and *plo = 10* both in dbm field
  - *pacmag = 1 , not in dbm range*
- In the **affirma** window, select *Analysis* → *Choose*
- The *Choose Analysis* window shows up
  - Select *PSS* for Analysis
  - Uncheck the *Auto Calculate* Box
  - *Set fundamental tone* → *flo flo* 2.4GHz (press update from schematic button) , look like

    flo flo 2.4G Large PORT2
  - *Beat Frequency* → 2.4G, *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 → 2.4GHz
  - Sideband → Max Sideband →2
  - Enable and apply
- 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 click on the *Results* → *Direct plot (main form)* → *PSS*
- The *PSS* results window appears.
  - *Analysis Type* → *PAC*
  - *Function* → *Voltage, Select* → *net*
  - *Sweep* → *Variable, Signal Level* → *Peak*
  - *Modifier* → *dB20, Output Harmonics* → *-1*
  - *Select **mixout** node in schematics*
  - *You will see the plot as shown in Fig-3.*

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.

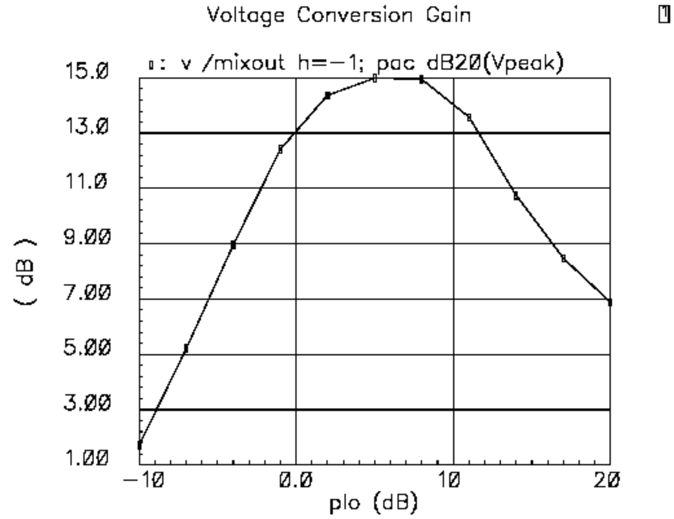


Fig 3: Voltage Conversion Gain versus the LO Signal Power

Note-2: The plo for maximum gain is 5dBm 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 → 2.4GHz to 2.41GHz*
  - *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* 2.4GHz (press update from schematic button) , look 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 **affirma** window click on *Simulation*→ Netlist and Run to start the simulation, make sure that simulation completes without errors.
- In the **affirma** window click on the *Results→ Direct plot (main form)→ PSS*
- The *PSS* results window appears.
  - *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-4.*

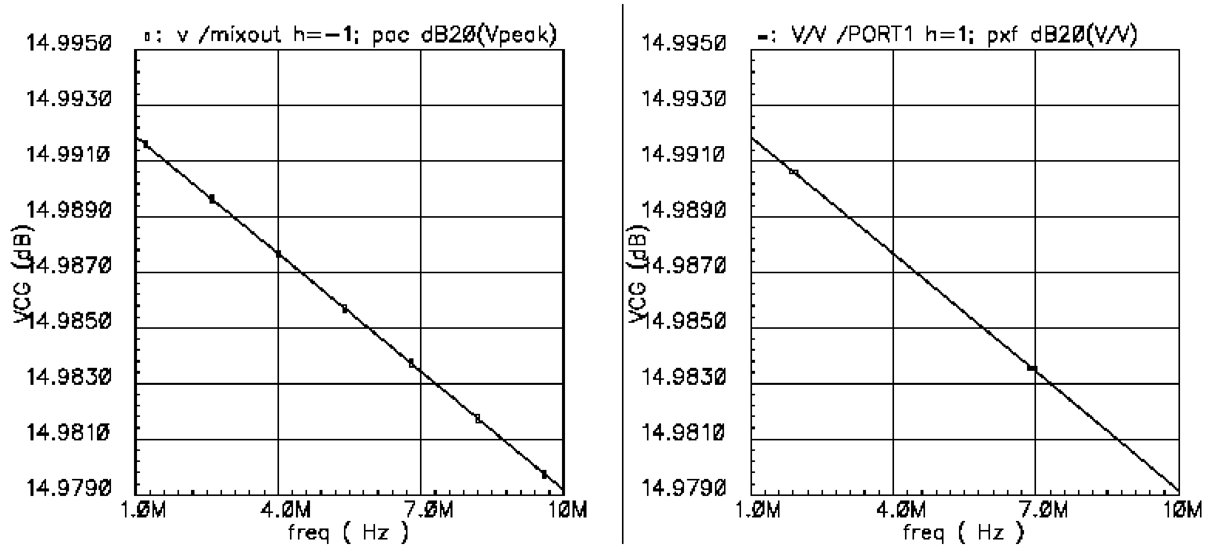


Fig 4: Voltage Conversion Gain versus RF frequency using PAC and PXF

c) Voltage Conversion Gain versus RF Frequency (PSS with swept PXF)

- Test Bench Parameters same as part a)
- In Design variables
  - Change 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* → (press update from schematic button) , it should look like

    flo flo 2.4G Large PORT2
  - *Beat Frequency* → *2.4G*
  - *Output Harmonics* → *10, Accuracy Default* → *Moderate*
  - *Sweep option*→ *off, Enable and apply*
- Now at the top of **choosing Analysis** window
  - *Select PXF for Analysis*
  - *Frequency Range* → *1KHz to 10MHz*
  - *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 **affirma** window click on *Simulation*→ Netlist and Run to start the simulation, make sure that simulation completes without errors.

- In the **affirma** window click on the *Results→Direct plot (main form)→PSS*
- The *PSS* results window appears.
  - *Analysis Type → PXF*
  - *Function →Voltage, Sweep →Sideband, Modifier →dB20*
  - *Output Harmonics→*1 1K -10M
  - *Select* RF port *in schematics*
  - *You will see the plot as shown in Fig-4 above.*

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

- Test bench is same as ***voltage conversion gain*** analysis.
- Make sure ***plo = 5*** in 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 → 2.4GHz to 2.41GHz
  - Sideband → Max Sideband →2, Sweep Type → Automatic
  - Enable and apply
- Now at the top of **choosing Analysis** window
  - *Select PXF for Analysis*
  - *Frequency Range →*2.4GHz to 2.43GHz
  - *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 **affirma** window click on *Simulation→* Netlist and Run to start the simulation, make sure that simulation completes without errors.

RF-to-LO Feedthrough:

- In the **affirma** window click on the *Results→Direct plot (main form)→PSS*
- The *PSS* 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 the Fig-5*

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 the Fig-5*

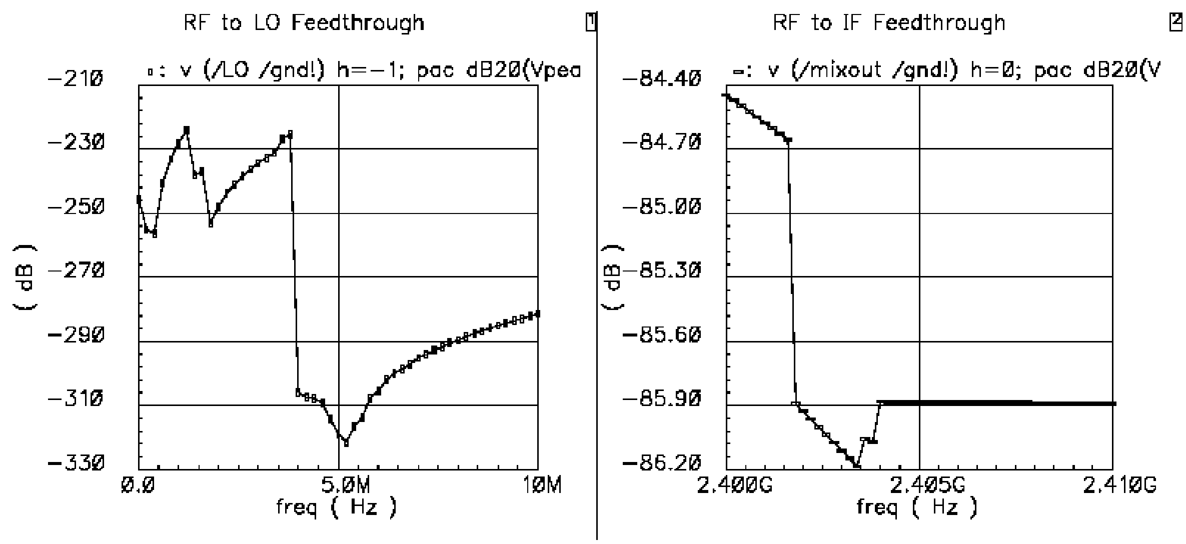


Fig 5: RF-to-LO & IF Feedthrough

LO-to-IF Feedthrough:

- In the **affirma** window click on the *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 **LO port** in schematics , see the results in the Fig-6*

LO-to-RF Feedthrough:

- *Now Select **RF port** instead of LO port in schematics (Fig-6)*

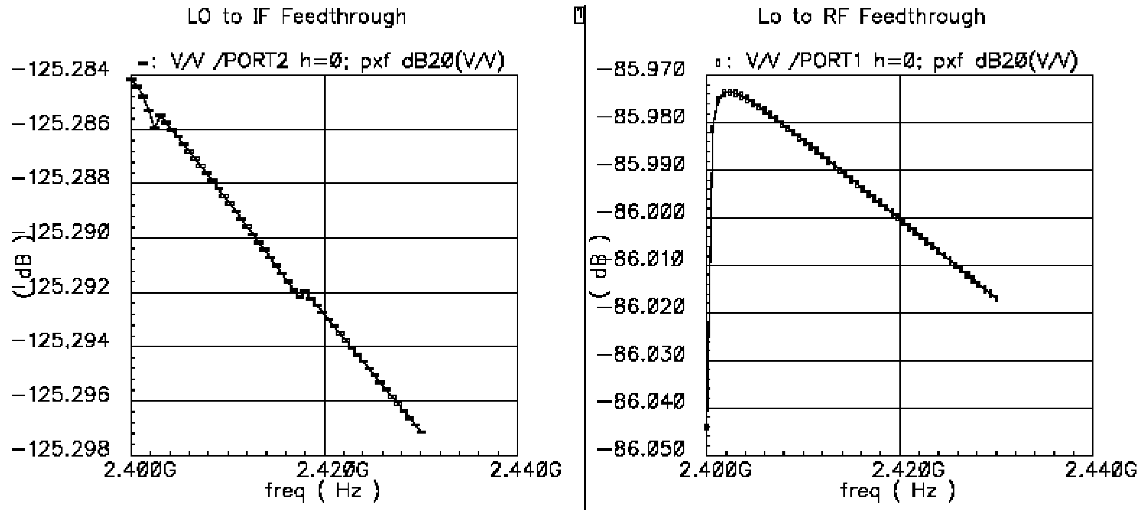


Fig 6: LO-to-IF Feedthrough

2.5. 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 which 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 **affirma** window are
    - *frf* = 2.41 GHz, *prf = -30* , *flo = 2.4GHz*, *plo = 5* , *pacmag = 1*
- In the **affirma** window, select *Analysis*→*Choose*
- The *Choose Analysis* window shows up
    - Select *QPSS* for Analysis
    - Click→ update from 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	potr2	5
frf	frf	2.41	moderate	port1	3

    - Accuracy → moderate
    - Enable and apply

- 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 click on the *Results*→*Direct plot (main form)*→*QPSS*
- The *QPSS* results window appears.
  - *Analysis Type → qpss, Function → power*
  - *Select → instance with two terminal, Modifier → dB10*
  - *Select **VDD** source terminal in schematics*
  - *You will see the plot as shown in Fig7*

Note: QPSS and PSS provide the spectrum not a scalar values. Summation of harmonics and sidebands gives a good estimate of the total power consumption. Most of the power is in the main output harmonics.

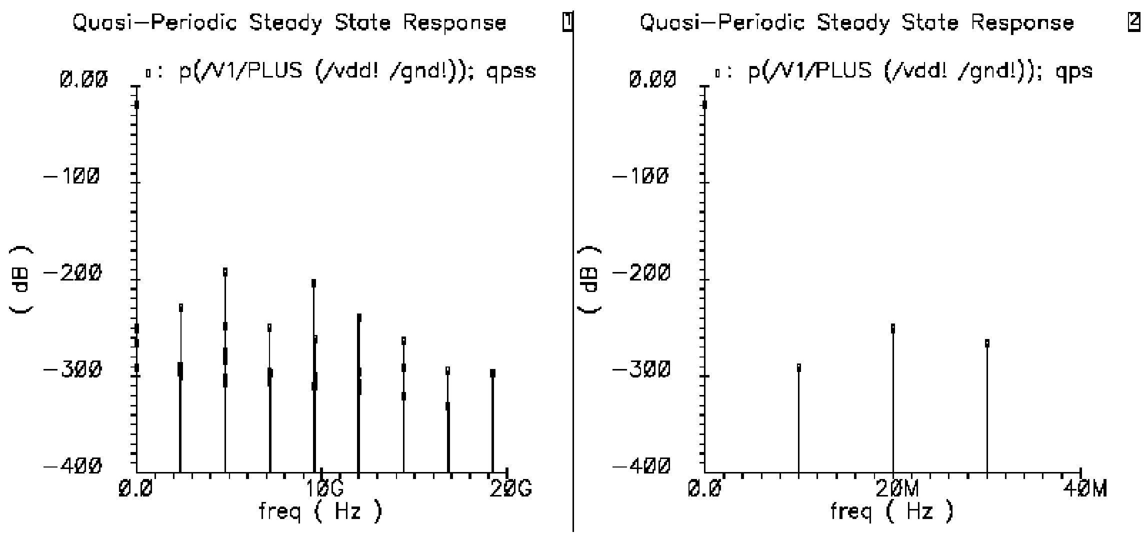


Fig 7: Large Signal Voltage Conversion Gain

2.6. 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 **affirma** window
  - *flo = 2.4 GHz (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 2.4GHz 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 2.4Ghz and onward)
- *Start-stop* → *1K -- 10M, Sweep Type* → *Automatic*
- *Press* **Select port** *button and point to the RF, IF and LO ports in schematic, and enter the desired data*

  1 PORT0 1 2.4G - 2.41G

  2 PORT3 0 2.4G - 2.41G

  3 PORT1 1 2.4G - 2.41G
- Order of ports is important, in our case Port0 (RF) is numbered 1 and port 3 (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 **affirma** window click on *Simulation*→ Netlist and Run to start the simulation, make sure that simulation completes without errors.
* In the **affirma** window click on the *Results*→ *Direct plot (main form)*→ *QPSS*
* 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 Fig8 and Fig9*

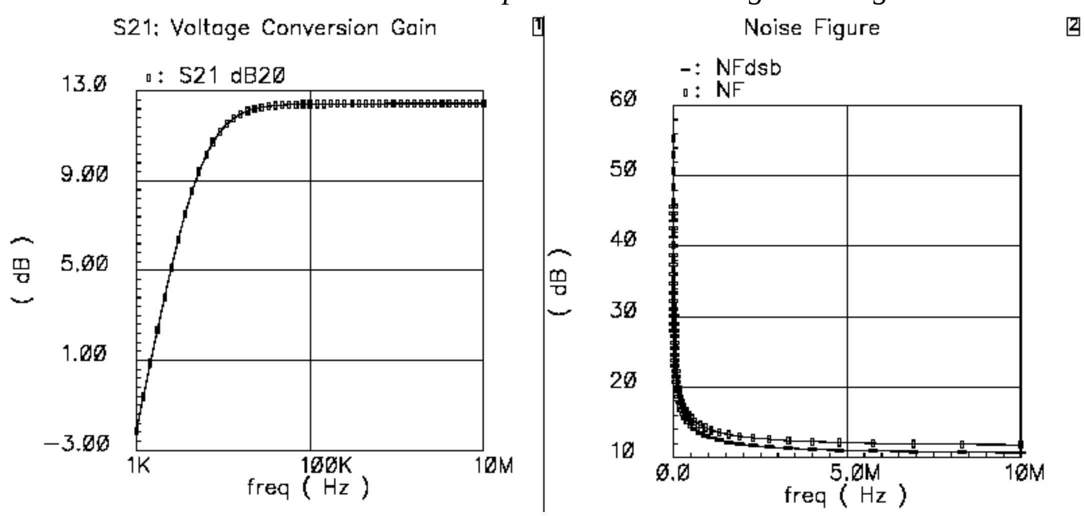


Fig 8: NF and S-Parameter Plots

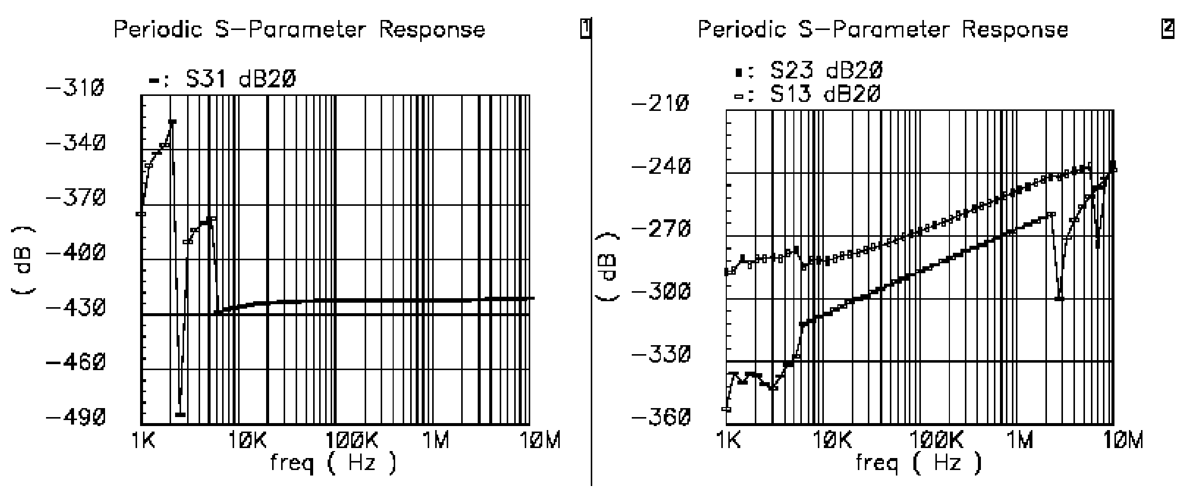


Fig 9: S-Parameters Isolation Plots

2.7. 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 ie. 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 meaning less)*
  - *LO port → sine (flo,flo,plo)*
- Verify the variable values in the **affirma** window
  - *flo = 2.4 GHz , 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 2.4GHz (press update from schematic button)*
  - *Beat Frequency → 2.4G*
  - *Output Harmonics → 10*
  - *Accuracy Default → Moderate*
  - Sweep → variable
  - Variable name→ plo
  - Sweep Range → -10dBm to 20dBm
  - Sweep Type → Linear
  - No of steps → 10 , Enable and apply
- The *Choose Analysis* window shows up
  - *Select Pnoise for Analysis*
  - *Sweep type → absolute*
  - *Start-stop → 10M* (noise is calculated at this frequency, the1/f noise effect will not present, to see that make this frequency 10K or 1K)
  - *Maximum side band →10*
  - Output→ voltage→ select mixout and gnd
  - Input source → port → select RF port

- *Reference sideband → -1*
- *Noise Type → sources*, , Enable and apply

- 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)→ Pnoise*
- The *PSS* results window appears.
  - *Analysis Type → Pnoise*
  - *Function → NF or NFdsb or Output Noise*
  - *You will see the plot as shown in Fig10*

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.

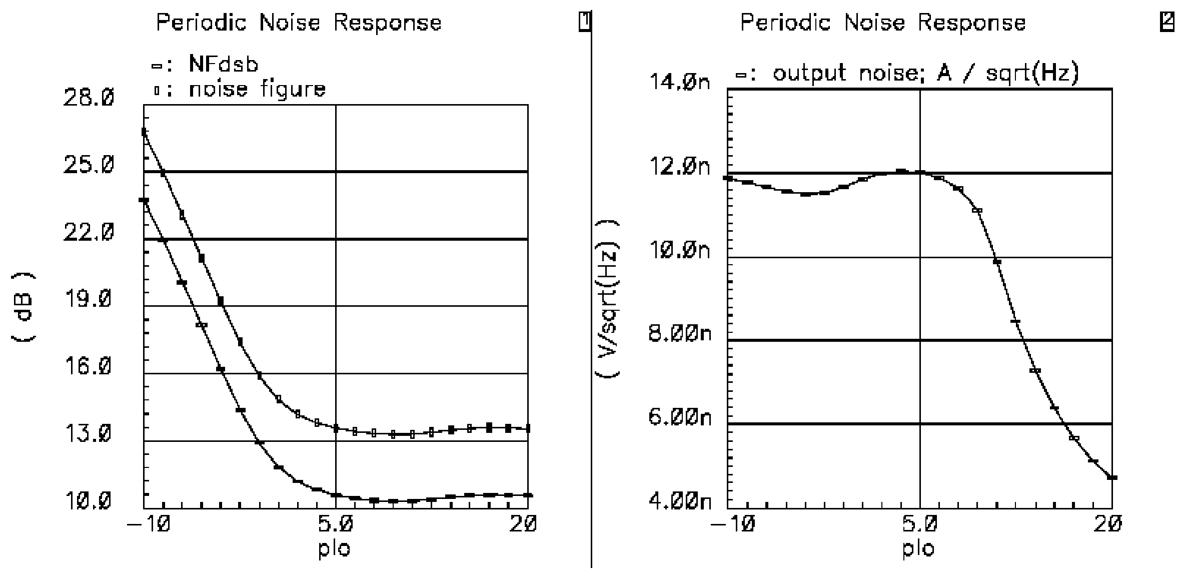


Fig 10: Noise Figure SSB , DSB and Output Noise

2.8. 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 1dB 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 1dB.

The recommended approach to calculate the 1dB CP 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 2^{nd} small signal RF tone has to small enough that IM1 and IM3 are in there asymptotic ranges.

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

- *frf* in *Frequency 1 field*
- *prf* in *Amplitude1(dBm)* field
- Click on the Box ***Display Signal Parameters***
- *pacmagdb in PAC Magnitude field (dB field)*

- Verify the variable values in the **affirma** window
  - *flo = 2.4 GHz, frf =2.401 GHz, prf = -10 ,plo= 5, pacmagdb=prf*
- In the **affirma** 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	PORT2	5
frf	frf	2.401G	modrate	PORT1	4

  - *Accuracy Default* → Moderate
  - High light the *Sweep* Button
  - Select Design Variable, small window appears, choose *prf* in it
  - *Sweep Range* → Choose the start : -70dBm and Stop: 10dBm
  - *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* →*2.4011GHz*
  - *Max Clock Order* →*2, Enable and apply*
- Click OK in the **affirma** window click on *Simulation*→ Netlist and Run to start the simulation.
- In the **affirma** window, select *Results*→*Direct plot (main form)*→*Main Form*
  - *Analysis* →*QPSS*
  - Select *Function* → *Compression Point*
  - *Gain Compression* → *1dB*
  - *Extrapolation Point* → *-70dB*
  - *1st Order Harmonic* → -1 1 (1M)
  - *Select Port (Fixed R (Port))*→ *click* IF PORT
  - *The resulting plot is shown in Fig11*
- In the **affirma** 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* → *-60dB*
  - 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 Fig12*

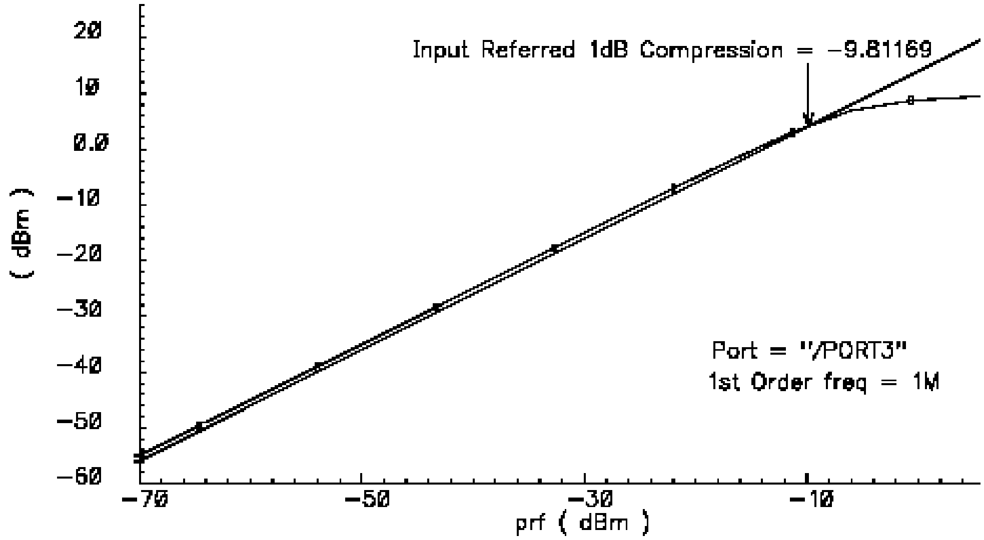


Fig 11: 1db Compression point and IIP3

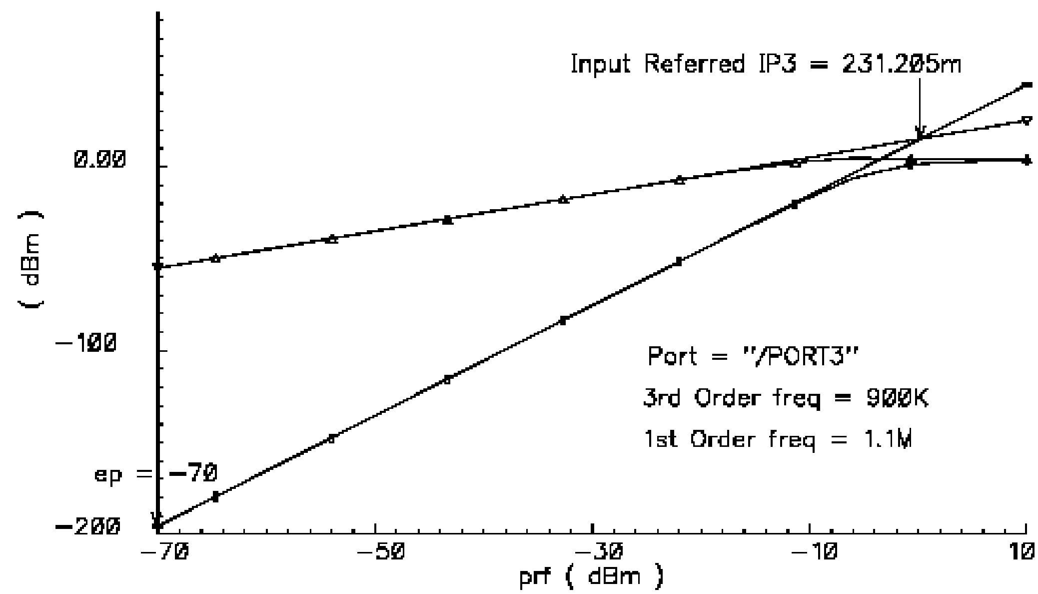


Fig 12: IIP3 using QPSS and QPAC

2.9. 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 blocker desensitize the receiver i.e. the gain and NF of the receiver for 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 Schematic Window
    - *LO port → sine (flo,flo,plo)*
    - *IF port → DC and 50 Ohms*

- Change the RF Port Parameters in Schematic Window
  - **Sine** in *Source Type*
  - *frf* in *Frequency name 1* field
  - *frf* in *Frequency 1 field*
  - *prf* in *Amplitude1(dBm)* field
  - Click on the Box ***Display Signal Parameters***
  - *pacmagdb in PAC Magnitude field*
- Verify the variable values in the **affirma** window
  - *flo = 2.4 GHz, frf = 2.403 GHz*
  - *prf = -50 ,plo= 5, pacmagdb =-30db*
- In the **affirma** 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	PORT2	5
frf	frf	2.403G	modrate	PORT1	4

  - *Accuracy Default* → Moderate
  - High light the *Sweep* Button
  - Select Design Variable, small window appears, choose *prf* in it
  - *Sweep Range* → Choose the start : -50dBm and Stop: 10dBm
  - *Sweep Type* → *Liner* 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* →*2.401GHz*
  - *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 Refrence Side Band* →*(1 0), Enable and apply*
- Click OK in the **affirma** window click on *Simulation*→ Netlist and Run to start the simulation.
- In the **affirma** window, select *Results*→*Direct plot (main form)*→*Main Form*
  - *Analysis* →*QPAC , Function* →*voltage*
  - *Select instance with two terminals*
  - *Sweep* →*variable, Modifier* → *dB20*
  - *Output Harmonic* →*1M (-1 0)*

  - *Click in Schematic on IF port*
  - *View the results as shown in Fig14*
- In the **affirma** window, select *Results*→ *Direct plot (main form)*→ *Main Form*
  - *Analysis* → *QPNoise* , *Function* → *Noise Figure*
  - *View the results as shown in Fig13*

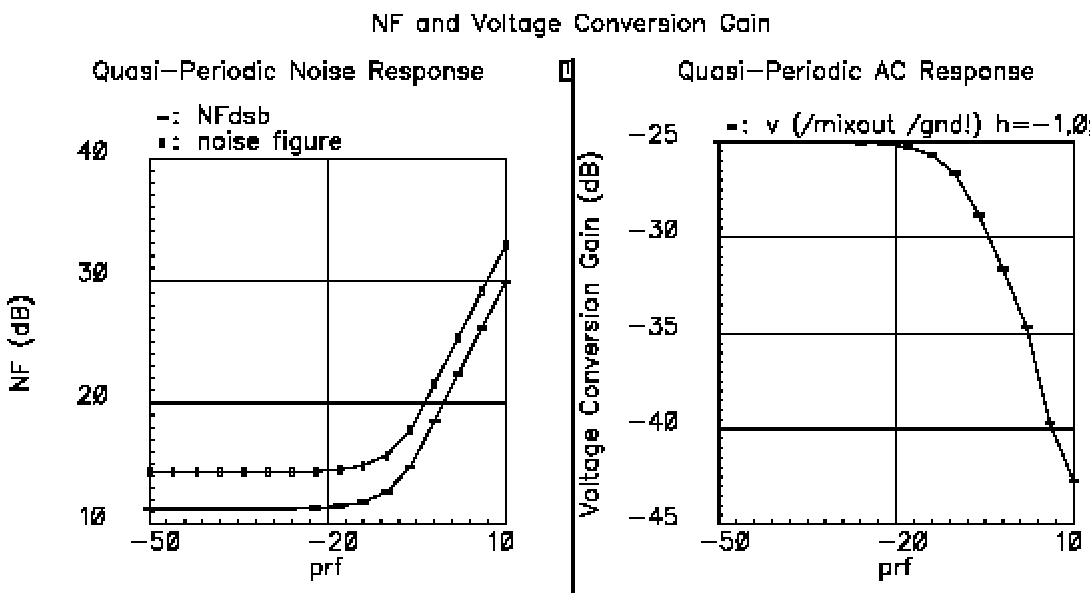


Fig 13: Voltage Conversion Gain & NF in presence of Blocking Signal

Assignment: Modified Gilbert Mixer with Improved Performance

Improve the NF, Gain by 2dB and IP3 by 3dB. To achieve this goal you can use the same circuit used in this tutorial and burn more power. Alternately to save the power you can apply any of the advance technique for gain and linearity enhancement discussed in the class lecture.

Please, report your result in a form of the table as shown below. Please, describe very briefly how you modified the circuit to meet the specifications and problems encountered during simulations.

Parameter	Present Design	Your Design
Gain		
NF		
1dB Comp Point		
IIP3		
…..		