cādence

SpectreRF Workshop

LNA Design Using SpectreRF

MMSIM6.0USR2

November 2005

Contents

Lower Noise Amplifier Design Measurements

The procedures described in this workshop are deliberately broad and generic. Your specific design might require procedures that are slightly different from those described here.

Purpose

This workshop describes how to use SpectreRF in the Analog Design Environment to measure parameters which are important in design verification of Low Noise Amplifier, or LNA. New features of MMSIM6.0USR2 are included.

Audience

Users of SpectreRF in the Analog Design Environment.

Overview

This application note describes a basic set of the most useful measurements for LNAs.

Introduction to LNAs

The first stage of a receiver is typically a low-noise amplifier (LNA), whose main function is to set the noise boundary as well as to provide enough gain to overcome the noise of subsequent stages (for example, in the mixer or IF amplifier). Aside from providing enough gain while adding as little noise as possible, an LNA should accommodate large signals without distortion, offer a large dynamic range, and present good matching to its input and output, which is extremely important if a passive bandselect filter and image-reject filter precedes and succeeds the LNA, since the transfer characteristics of many filters are quite sensitive to the quality of the termination.

The Design Example: A Differential LNA

The LNA measurements described in this workshop are calculated using SpectreRF in the Analog Design Environment. The design investigated is the differential Low Noise Amplifier shown in below:



The following table lists typically acceptable values for the performance metrics of LNAs used in heterodyne architectures.

Measurement	Acceptable Value
NF	2 dB
IIP3	-10 dBm
Gain	15 dB
Input and Output Impedance	50 Ω
Input and Output Return Los	-15 dB
Reverse Isolation	20 dB
Stability Factor	>1

Testbench

The following figure shows the testbench for a differential LNA. The baluns used in the testbench are three-port devices. The baluns convert the input single-ended signals to the differential signals. Sometimes, they also perform the resistance transformation.



LNA design is a compromise among power, noise, linearity, gain, stability, input and output matching, and dynamic range. They are characterized by the design specifications in Table in page 4.

Example Measurements Using SpectreRF

The LNA measurements described in the following labs are calculated using SpectreRF in the Analog Design Environment.

We'll begin our examination of the flow by bringing up the Cadence Design Framework II environment and look at a full view of our reference design:

Change directory to...

Action: cd to **./Ina** directory

Action: Invoke tool **icfb**&

Action: In the CIW window, select **Tools->Library Manager...**

Lab1: Small Signal Gain (SP)

The S Parameter (SP) analysis is the most useful linear small signal analysis for LNAs. Set up an SP analysis by specifying the input and output ports and the range of sweep frequencies.

- Action1-1: In the Library Manager window, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*
- Action1-2: Select the **PORTrf** source by placing the mouse cursor over it and clicking the left mouse button. Then in the Virtuoso Schematic Editor select **Edit—Properties—Objects...** The Edit Object Properties window for the port cell should come up. Set up the port properties as follows:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	dc

- Action1-3: Set the source type of **PORT load** to DC.
- Action1-4: check and save the schematic.
- Action1-5: In the Virtuoso Schematic Editing window, select **Tools->Analog** Environment
- Action1-6: You can choose **Session—Load State** in Virtuoso Analog Design Environment load state "**Lab1_sp**", then skip to Action1-10 or ...
- Action1-7: In Analog Design Environment window, select Analyses->Choose...
- Action1-8: In the Choosing Analyses window, select the **sp** button in the **Analysis** field of the window.
- Action1-9: In S-Parameter Analysis window, in the **Ports** field, click the **Select** button. Then, in the Virtuoso Schematic Editing window, in order, select the port cells, **rf** (input) and **load** (output). Then, while the cursor is in the schematic window, hit the **ESC** key.

In the Sweep Variable field, select Frequency.

In the Sweep Range field, select Start-Stop, set Start to 1.0 G and Stop to 4.0G, set Sweep Type to Linear, select Number of Steps and set that to 50. In the Do Noise field, select yes, set Output port to /load and Input port to /rf.

The form should look like this:

OK Cance	Deraults	, data à			He
Analysis	<pre> tran xf pz pac qpss qpsp </pre>	 √ dc √ sens ♦ sp √ pnoise √ qpac 	 ↓ ac ↓ dcmatch ↓ envlp ↓ pxf ↓ qpnoise 	◇ noise ◇ stb ◇ pss ◇ psp ◇ qpxf	
	S-Pa	arameter Ana	alysis		
Ports			Select	Clea	r
rf load					
 Design Temper Comport Model I 	Variable rature 1ent Parame Parameter	iter			
 Design Temper Comport Model I Sweep Ran 	icy Variable rature hent Parame Parameter ge	iter			
 Design Temper Compor Model I Sweep Ran Start-S Center- 	Variable rature nent Parame Parameter ge itop St Span	eter art 1.0g	Stop	4.0 ġ	
 Design Temper Comport Model I Sweep Ran Start-S Center- Sweep Typ Linear 	Variable rature nent Parame Parameter ge itop St Span	art 1.0g Step S Numbe	Stop ize r of Steps	4.0ġ	
 Design Temper Compor Model I Sweep Ran Start-S Center- Sweep Typ Linear Add Specific 	Variable rature nent Parame Parameter ge itop St Span e c Points	eter art 1.0g Step S Numbe	Stop ize r of Steps	4.0ğ	
 Design Temper Compor Model I Sweep Ran Start-S Center- Sweep Typ Linear Add Specific Do Noise 	Variable rature nent Parame Parameter ge Stop St Span e C Points Out	eter art 1.0g Step S Numbe	Stop ize r of Steps /load	4.0ġ	

Status: Ready	T=27 C Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	۲.
Library RFworkshop	# Type Arguments Enable	J AC
Cell Diff_LNA_test View schematic	1 sp 16 46 50 Lineyes	- DC
Design Variables	Outputs	Œ
# Name Value	# Name/Signal/Expr Value Plot Save March	4
1 frf 2.46 2 prf -50		
	Plotting mode: Replace -	t.

Now your Virtuoso Analog Design Environment looks like this:

- Action1-10: Choose **Simulation—Netlist and Run** to start the simulation or click on the **netlist and Run** icon in the Virtuoso Analog Design Environment window.
- Action1-11: In Analog Design Environment window, select **Results->Direct Plot->Main Form...** A Waveform window and a Direct Plot Form window should open.
- Action1-12: In Direct Plot Form window, set Plotting Mode to Append. In the Analysis field, select sp. In the Function field, select GT (for Transducer Gain). In the Modifier field, select dB10. Select Add to Outputs. The form should look like

	CEC PION FOI	m		
OK	Cancel			Help
Plotting	g Mode	Append		
Analys	is			
🔶 sp	8			
Functio) <mark>n</mark>			
\diamondsuit SI	P 🔷 ZP	\diamondsuit YP	⇔ HP	
\diamondsuit GI	D \Diamond VSI	WB 🔷 NFmii	n 🔷 Gmin	
🔷 Br	n ⊘mr	I NF	🔷 Kf	
🔷 B1	lf 🛛 🔶 GT	i GA	🔷 GP	
🔷 Gi	nax 🔷 Gm	isg 🔷 Guma	: 💸 ZM	
-\$ NO	C 🔷 GA	C 🔷 GPC	🔷 LSB	
⇔s:	SB			
Descrij Modifie	ption: Trans er	ducer Gain		
\bigcirc Ma	gnitude 🔶	dB10		
Add To	o Outputs		Plot	
Pres	s plot hutto	n on this for	n	

- Action1-13: Hit the **Plot** button. In the **Function** field, select **GA** (for Available Power Gain). Hit the **Plot** button again. In the **Function** field, select **GP** (for Operating Power Gain). Hit the **Plot** button once more. We plotted GT, GA and GP in one waveform window.
- Action1-14: In waveform window, click on New Subwindow. Go back to Direct Plot Form. Select **Gmax** (for maximum Transducer Power Gain), Hit the **Plot** button. In the **Function** field, select **Gmsg** (for Maximum Stability Gain). Hit the **Plot** button. Select **Gumx** (for maximum Unilateral Transducer Power Gain), Hit the **Plot** button again.

Now you should get the following waveforms:



Action1-15: Close waveform window, go back to Direct Plot form. In the Function field, select GAC (Available Gain Circles). In Plot Type, choose Z-Smith, Sweep Gain Level (dB) at Frequency 2.4GHz from 0 to 18dB with step as 2 dB.

riot Ty	pe		
♦ Z-S	mith 🔷 Y-:	Smith	
Sweep	🔷 fre	equency 🔶 G	ain Level (dB)
Freque	ncy (Hz)	2.46	Ĩ
	ango (dB)		
rever P	ande (an)		
Start	Q	Stop	18

- Action1-16: Press **plot** button.
- Action1-17: In waveform window, click on **New Subwindow**.
- Action1-18: Go back to Direct Plot Form, in Function field, select **GPC** (Power Gain Circles). And press Plot button.

The waveforms should look like this:



Action1-19: Close the waveform window, go back to **Direct Plot Window**. In **Function** field, choose **Kf**. Hit **Plot** button.

Action1-20: In **Function** field, choose **B1f**. Hit **Plot** button.

You will get the Stability Curves:



Action1-21: Close the waveform window; go back to **Direct Plot Window**. In **Function** field, choose **LSB** (Load Stability Circles). In Plot Type, choose Z-Smith. Specify Frequency Rage from 2G to 3G with step as 0.2G. Hit **Plot** button.

♦ Z-S	mith 🔷 Y-Si	nith	
reque	ncy Range (H	łz)	
Start	2 4	Stop	3ġ
Step	0.2 <u>Ğ</u>		A:
Add To	Outputs	J	Plot

Action1-22: In waveform window, click on **New Subwindow**.

Action1-23: Go back to **Direct Plot Form**, in **Function** field, select **SSB** (Source Stability Circles). And press **Plot** button.

You will get the Load Stability Circles and Source Stability Circle:



- Action1-27: In the **Direct Plot Form** window, select **NF** (**Noise Figure**) in the **Function** field. In the **Modifier** filed, select **dB10**. Hit the **Plot** button.
- Action1-26: In the waveform window, click on **New Subwindow**.
- Action1-24: Close the waveform window, go back to Direct Plot Form window, in the **function** field, and choose **NC** (Noise Circles). In the **Plot type** field, choose **Z-Smith**. Sweep Noise Level at Frequency 2.4G Hz staring from 0 to 10 dB with step of 1 1dB.

🔶 Z-S	mith 🔷 Y-Smith		
Sweep	🔷 frequency	🔶 N	oise Level (dB)
	Part of Water		2
reque	ncy (Hz) 2.46		
irequel .evel F	ncy (Hz) 2.46 lange (dB)		
irequel .evel F Start	ncy (Hz) 2.46 lange (dB)	Stop	2.5

Action1-25: Hit the **Plot** button.

You should get the following plot:



Action1-28: Close the waveform window, go back to the Direct Plot Form window, in the **function** field, and choose **VSWR** (Voltage standing-wave ratio). In the **Modifier** field, select **dB20**. Press on **VSWR1**, then **VSWR2**.

You should get the following waveforms:



Action1-29: Close the waveform window and click Cancel on the Direct Plot form

Lab2: Large Signal Noise Simulation (PSS and Pnoise)

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. For most cases, LNAs work with very low input power level, so SP or noise analysis is enough.

- Action2-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*
- Action2-2: Select the **PORTrf** source. Use the **Edit**—**Properties**—**Objects** command to ensure that the port properties are set as described below:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	sine
Frequency name 1	RF
Frequency 1	frf
Amplitude 1 (dBm)	prf
Frequency name 2	(blank)
Frequency 2	(blank)
Amplitude 2 (dBm)	(blank)

- Action2-3: Check and save the schematic.
- Action2-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.
- Action2-5: You can choose **Session—Load State**, load state "**Lab2_Pnoise**" and skip to Action2-15 or ...
- Action2-6: In Vituoso Analog Design Environment, choose Analyses—Choose...

Action2-7:	In the Choosing Analyses window, select the pss button in the Analysis field of the window.
Action2-8:	In pss analyses window, select Auto Calculate button. This automatically calculates either the Beat Frequency or Beat Period of the circuit. If the circuit contains frequency dividers or the input sources do not come from analogLib , it might be necessary to manually calculate the Beat Frequency (or Beat Period).
Action2-9:	In the Output Harmonics field, set the cyclic to Number of Harmonics and set the number of harmonics to 3. This will allow us to look at, in the frequency domain results, 3 harmonics of the Beat Frequency.
Action2-10:	In the Accuracy Defaults (errpreset) field, select the moderate button.

Your Choosing Analyses->PSS window should look like...

Periodic Steady State Analysis ngine Shooting Periodic Steady State Analysis ngine Shooting Fundamental Tones Name Expr Value Signal Strond Signal Sweep Periodic Steady State Analysis Periodic Steady State Analysis Periodic State Analysis Fundamental Tones Name Expr Value Signal Sweep Periodic State Analysis Periodic State Analysis Periodic State Analysis Periodic State Analysis Period Period Period </th <th></th> <th></th>		
Ingine Shooting Flexible Balance Fundamental Tones Name Expr Value Signal SrcId Image:	Periodic Steady State Analysis	
Fundamental Tones Name Expr Value Signal SrcId IRF frf 2.46 Large Clear/Add Delete Update From Schematic • Beat Frequency 2.46 Auto Calculate Output harmonics Number of harmonics Number of harmonics Number of harmonics Sweep Sweep	Shooting Elexible Balance	
Name Expr Value Signal SrcId RF frf 2.46 Large rf Clear/Add Delete Update From Schematic Beat Frequency 2.46 Auto Calculate Beat Period 2.46 Dutput harmonics Number of harmonics Image: Conservative image: moderate image: liberal Viditional Time for Stabilization (tstab) Sweep Sweep Sweep	nental Tones	
RF frf 2.46 Large Clear/Add Delete Update From Schematic • Beat Frequency 2.46 Auto Calculate • Beat Period 2.46 Auto Calculate Output harmonics Number of harmonics 14 • Conservative I moderate liberal • Additional Time for Stabilization (tstab) Save Initial Transient Results (saveinit) no • Sweep	: Expr Value Signal SrcId	1.5
Clear/Add Delete Update From Schematic	frf 2.4G Large rf	
Clear/Add Delete Update From Schematic	Large -	
Beat Frequency Beat Period 2.4g Auto Calculate Auto Calculate Dutput harmonics Number of harmonics IQ Conservative moderate _ liberal Additional Time for Stabilization (tstab) Save Initial Transient Results (saveinit) _ no _ yes Dscillator _ Sweep _	ar/Add Delate Undate Emm Schematic	<u>1</u>
Beat Frequency 2.46 Auto Calculate Beat Period Output harmonics Number of harmonics1 Accuracy Defaults (errpreset) Conservative I moderateIiberal Additional Time for Stabilization (tstab) Save Initial Transient Results (saveinit) noyes Dscillator Sweep	spuare from Schemauc	
Beat Period Purce Calculate Output harmonics Number of harmonics Index Accuracy Defaults (errpreset) Conservative Image: moderate Iberal Actio Calculate Auto Calculate Output harmonics Index Auto Calculate Output harmonics Index Auto Calculate Auto Calculate Output harmonics Index Auto Calculate Auto Calculate Auto Calculate Auto Calculate Auto Calculate Auto Calculate Index Auto Calculate Index Auto Calculate Index Auto Calculate Index	at Frequency	
Output harmonics Number of harmonics IQ Accuracy Defaults (empreset) conservative moderate liberal Additional Time for Stabilization (tstab) Save Initial Transient Results (saveinit) Dscillator	at Period	
Output harmonics Number of harmonics III Accuracy Defaults (errpreset) Conservative moderate Iberal Additional Time for Stabilization (tstab) Save Initial Transient Results (saveinit) Dscillator		
Number of harmonics1[Accuracy Defaults (errpreset) conservative III moderate liberal Additional Time for Stabilization (tstab) [Save Initial Transient Results (saveinit) no yes Dscillator Sweep	harmonics	
Accuracy Defaults (empreset) Conservative moderate liberal Additional Time for Stabilization (tstab)	r of harmonics = 11	
Accuracy Defaults (empreset) Conservative I moderate liberal Additional Time for Stabilization (tstab) Save Initial Transient Results (saveinit) no yes Dscillator Sweep		
_ conservative ■ moderate _ liberal Additional Time for Stabilization (tstab) Save Initial Transient Results (saveinit) _ no _ yes Oscillator _ Sweep _	y Defaults (errpreset)	
Additional Time for Stabilization (tstab)	nservative 🔳 moderate 🔄 liberal	
Save Initial Transient Results (saveinit) no yes Oscillator Sweep	al Time for Stabilization (tstab)	
Oscillator	itial Transient Results (saveinit) 🔄 no 🔄 yes	
Oscillator		
3weep _	or 🔄	
Sweep		
Sweep 🔄		
JWCCP		

Action2-11: Now that you have set up the PSS analysis. Click **pnoise** in the Choosing Analyses form. You must specify the noise source and the number of sidebands for inclusion in the summation of the final results. The larger the number, the more accurate the results will be, until the point where the higher order harmonics are negligible. Spectre will give you warning message regarding accuracy for any maxsideband number lower than 7. You specify the reference sideband as 0 for an LNA because an LNA has no frequency conversion form input to output. The form should look like this:

	GLATCOT	Defaults	Apply		He
		Perio	dic Noise Ana	lysis	
88.8	leat Frequ	uency (Hz	2.46		
	11-				
Swe	eptype _	default	Swee	p is Currently	7 Absolute
Outp	out Freque	ency Swe	ep Range (Hz)	
Sta	art-Stop	- St	art <u>10</u>	Stop	4G
Swe	ер Туре				
1	inear .		Step Si	ze of Stons	8
1			- Humber	or oups	
Add	Specific	Points 🔄			
casal.	S - 2				
Side	bands	27.011114.02	10		
Max	kimum sid	leband 😑			
Outp	ut				
Outp pro	out obe	Output	Probe Instanc	e /load	Select
Outp pro	out obe	Output	Probe Instanc	e /load	Select
Outp pro	ut be t Source	Output	Probe Instanc	e /load	Select
Outp pro Input	ut be = t Source rt =	Output Input P	Probe Instanc Port Source	e /load /rf	Select
Outp pro Input po Refe	out obe	Output Input P le-band	Probe Instanc Port Source	e /load	Select
Outp pro Input po Refe Ent	out obe t Source ort rence sid	Output Input P le-band	Probe Instanc Port Source	e <mark>/load</mark> /rf	Select Select
Outp pro Input po Refe Ent	out obe t Source ort rence sid	Output Input P le-band	Probe Instanc Port Source	e /load /rf	Select Select
Outp pro Input po Refe Ent	out obe t Source ort rence sid ter in field se Type	Output Input P Ie-band I sources	Probe Instanc Port Source	e /load /rf	Select
Outp pro Input Po Refe Ent Noi: Sou	out obe	Output Input P le-band I sources jle sidebai	Probe Instanc Port Source	e /load /rf	Select

Action2-12: Make sure the **Enabled** button is active, and click **OK** in the Choosing Analyses form.

Action2-13: In the Virtuoso Analog Design Environment window, double click on prf in the field of Design Variables. Change the input power to -20.

ок	Cancel	Apply	Apply & Run Simulat	ion		Help
	5	elected	Variable	Т	able of De	sign Variables
Hame		prf		#	Name	Value
Value ((Expr)	-20		1	frf	2.4G
1 1 1	· · · [r r 7	12	prf	-50
Add	Delete	Change	Next Clear Find			
			1 - V	2		
Cellvie	w Variab	les Co	py From Copy To			

Action2-14: Click **Change**. Click on OK to close the Editing Design Variables window.

Your Virtuoso Analog Environment will look like this:

Status: Ready	T=27 Č Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	⊦₹ _₹
ibrary RFworkshop	# Type Arguments Enable	JAC F TRAN
Cell Diff_LNA_test Aew schematic	1 pnoise 3 16 46 8 yes 2 pss 2.46 2 yes	コ DC - - - - -
Design Variables	Outputs	Œ,
# Name Value	# Name/Signal/Expr Value Plot Save March	Ű
l frf 2.46 2 prf -20		<i>≫</i> ₩
		8
	Plotting mode: Replace -	to,

Action2-15: In your Analog Design Environment, Choose Simulation—Netlist and Run or click the Netlist and Run icon to start the simulation.

- Action2-16: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.
- Action2-17: In the Direct Plot Form, select the **pnoise** button, and configure the form as follows:

ок	Cancel		Help
Plotting	j Mode	Append	
Analys	is		
💸 ps	s 🔶 pnoise		
Functio	in		
🔷 Ou	itput Noise	😞 Input Noise	
🔶 No	ise Figure	🔷 Noise Factor	
\odot NF	dsb	😞 Fdsb	
<> NF	ieee	😞 Fieee	
10000	ase Noise	Transfer Function	

Action2-18: Click the **Plot** button.



The Waveform window displays the Noise Figure. The NF from Phoise is slightly larger than the NF from SP because at Pin = -20 dBm, the LNA demonstrates very weak nonlinearity and noise as other high harmonics are convoluted.

Action2-19: Close the waveform window and click **Cancel** on the Direct Plot form. Close the Virtuoso Analog Design Environment window.

Lab3: Gain Compression and Total harmonic Distortion (Swept PSS)

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. After the PSS analysis with swept input power level, plot the output power against the input power level. Determine the 1 dB compression point from the curve.

- Action3-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*
- Action3-2: Select the **PORTrf** source. Use the **Edit**—**Properties**—**Objects** command to ensure that the port properties are set as described below:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	sine
Frequency name 1	RF
Frequency 1	frf
Amplitude 1 (dBm)	prf

Action3-3: Check and save the schematic.

Action3-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.

- Action3-5: You can choose **Session—Load State**, load state "**Lab3_P1dB**" and skip to Action3-9 or ...
- Action3-6: In Vituoso Analog Design Environment, choose Analyses—Choose...

Action3-7: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window. Set up the form as follows:

ок	Cancel	Default	s Appl	У					H
Funda	nental	Tones							
t Nam	e	Expr	v.	alue	Sig	nal	Src	Id	
2 RF		frf	2	. 46	Lar	ge	rf		
		×.			Large		8		
Cle	ar/Add	Del	ete	Upd	ate Fro	m Sch	ematic	:	
Be Be	eat Free eat Peri	quency iod	2.46)		Auto	Calcu	late 🔳	
Numbe	r of ha	nics rmonics		14]			
Contraction	onserv nal Tim	ative ∎ ne for St	moder abilizat	ate 🔄 tion (ts	liberal tab) [
⊥ c Additio Save I Oscilla	nal Tim nitial Ti tor 🗌	ative 🔳 ne for St ransient	moder abilizat Result	rate ⊔ ion (ts s (savi	liberal tab) [einit) _) no _) yes	<u> </u>	
c Additio Save I Oscilla Sweep	nal Tim nitial Ti tor Variable	e =	rrprese moder abilizat Result	ion (ts ion (ts s (save Freque Variab	liberal tab) [einit) ency Va le Nam	on sideire e pri) yes ? ◆ no	i ⊘ye	S
c Additio Save I Oscilla Sweep	nal Tim nitial Ti tor Variable	e =	rrprese moder abilizat Result	rate ion (ts s (savu Freque Variab	liberal tab) [einit) ency Va le Nam Select	no ariable e pri Desigi) yes ? ◆ no [n Varia	i ⊘ye ble	5
c Additio Save I Oscilla Sweep Sweep ♦ St	nal Tim nitial Ti tor variable Range art-Sto	e	rrprese moder abilizat Result	rate ion (ts: s (savi Frequi Variab	liberal tab) [einit) ency Va le Nam Select] no ariable e [pr:] Design) yes ? ◆ no [] -S	i ⊘ye: ble	s
C Additio Save I Oscilla Sweep Sweep ◆ St ~ Ce	variable Range art-Sto	e op ; pan	rrprese moder abilizat Result	rate ion (ts: s (savi Frequi Variab	liberal tab) [einit) ency Va le Nam Select	no ariable e pri Desigi Stop	yes ? ◆ no (n Varia	ble	s
Additio Save I Oscilla Sweep	variable Range art-Sta Type	e for St	rrprese moder abilizat Result	rate ion (ts s (savi Freque Variab	liberal tab) [einit) ency Va le Nam Select	no ariable e pri Design Stop	yes ? ◆ no (1 Varia	ble	5
☐ Cr Additio Save I Oscilla Sweep \$Sweep \$St \$Sweep \$Sweep \$Sweep \$Li \$Li	nal Tim nitial Ti tor wariable Range art-Sto nter-S Type near garithm	e op span	rrprese moder abilizat Result	rate ion (ts: s (savu Freque Variab -30 Step S Numbe	liberal tab) [einit) ency Va le Nam Select Select	no ariable e pri Design Stop) yes ? ◆ no [10]	i ⊘ye: ble	5
☐ Cl Additio Save I Oscilla Sweep Sweep ♦ SI ◇ Ce Sweep ♦ Li ◇ Lo	nal Tim nitial Ti tor variable Range art-Sto nter-S Type near garithm secific	e	rrprese moder abilizat Result	rate ion (ts' s (save Freque Variab	liberal tab) [einit) ency Va le Nam Select Select	no ariable e pr: Design Stop) yes ? ◆ no (10 10	ı ⊘ye: ble	s

Action3-8: Make sure the **Enabled** button is on. Click on **OK** on the choosing analyses form.

Status: Ready	T=27 C Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	×.
Library RFworkshop	# Type Arguments Enable	JAC F TRAN
Cell Diff_LNA_test	1 pss 2.46 10 -30 -5 yes	- DC
View schematic		I I X Y Z
Design Variables	Outputs	Œ,
# Name Value	# Name/Signal/Expr Value Plot Save March	4
1 frf 2.46		<u> </u>
2 prf -50		000
		8
	Plotting mode: Replace	+

Now your Virtuoso Analog Design Environment will look like:

- Action3-9: In your Analog Design Environment, Choose Simulation—Netlist and Run or click the Netlist and Run icon to start the simulation.
- Action3-10: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.
- Action3-11: In the Direct Plot Form, select the **pss** button, and configure the form as follows:

	Cancel	Hel
Pottin Malvs	g Mode Ap	pend 💴
A ne		
• p:	55	
uncti	on	
\sim V	oltage	😞 Current
<\$₽	ower	Voltage Gain
$\diamond 0$	urrent Gain	Power Gain
<□	ransconductanc	e VTransimpedance
• 0	ompression Poil	nt VIPN Curves
~ P	ower Contours	Reflection Contours
∨ H	armonic Freque	ncy Ver Added Eff.
Y P	ower Gain vs P	out V Comp. As Pout
√n	oue complex in	iþ. √1πυ
Select	t Port	(fixed R(port)) 👘 🛁
orma	t Output Powe	1 1 ,
Sain (Compression (d)	
Jouri V	winpression (u	uy [**
" nnut	prf" ranges from Power Extranol:	n -30 to -5 ation Point (dBm)
	(Defaults to	-30)
	it Referred 1dB	Compression -
Inpu		
Inpu 1st O	rder Harmonic	
Inpu 1st O 0	rder Harmonic 0	
Inpu 1st O 0	rder Harmonic 0 2.46	
Inpu 1st O 0 1 2 3	rder Harmonic 0 2.46 4.86 7.26	
Inpu 1st 0 0 1 2 3 4	rder Harmonic 0 2.46 4.86 7.26 9.66	
Inpu 1st 0 0 1 2 3 4 5	rder Harmonic 0 2.46 4.86 7.26 9.66 126	
Inpu 1st 0 1 2 3 4 5	rder Harmonic 0 2.46 4.86 7.26 9.66 126 0 Outputs	

Action3-12: Select output port **load** on schematic.

The P1dB plot appears in the Waveform window.



The gain at -30 dBm input power level is -14.7 - (-30) = 15.3 dBm which is a good match for the small signal gain.

After the PSS analysis, you 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.

- Action3-13: Close the waveform window.
- Action3-14: In the Direct Plot Form, select the **pss** button, and configure the form as follows:

100 C	ancel	H
lotting M nalysis	lode Appe	nd 💷
🔶 pss		
Function		
 Volta; Powe 	ge ir	✓ Current ✓ Voltage Gain
🔷 Curre	nt Gain	🔷 Power Gain
Trans	conductance	Transimpedance
Comp	ression Point	Poflection Contours
Ham	nnic Frequency	Power Added Fff.
Powe	er Gain Vs Pout	Comp. Vs Pout
Node	Complex Imp.	THD
	194610	
Select	Net	
Select Sweep	Net	
Select Sweep ♦ spectr	Net um _v time	
Select Sweep ♦ spectr Signal Lev	ven ⇒ time	~ ms
Select Sweep ♦ spectr Signal Lev Modifier	vım √time ∕el ∳peak	~ ms
Select Sweep spectr Signal Lev Modifier Magni Real	vım ç>time /el ∳peak tude Ç Phase Ç Imagin	√ ms ♦ dB20 ary
Select Sweep spectr Signal Lev Modifier Magni Real Vidd To Ou	vel ∳peak tude ◇ Phase ◇ Imagin	√ ms ♦ dB20 ary

Action3-15: Select output **net RFout** on schematic.



it is obvious that the DC and all the even modes at the output are suppressed because the LNA we investigated is a differential LNA.

Action3-16: After viewing the waveforms, close waveform window.

Action3-17: In the Direct Plot Form, select the **pss** button, and choose **THD** function.

VIX VII	ncel	He	elp
Plotting Mo Analysis	de Appe	nd	
🔶 pss			
Function			
Voltage			
Power		🗸 Voltage Gain	
↓ Curren	t Gain		
Transc	onductance	Transimpedance	
Compr	ession Point	VIPN Curves	
		Mar.	
Power	Contours nic Frequency	 Reflection Contours Power Added Eff. 	
Power Harmo Power Power Node C Select	Contours nic Frequency Gain Vs Pout Complex Imp. Net	 Reflection Contours Power Added Eff. Comp. Vs Pout THD 	
Power Harmo Power Power Node C Select Fundament:	Contours nic Frequency Gain Vs Pout Complex Imp. Net al	 Reflection Contours Power Added Eff. Comp. Vs Pout THD 	
Verver Verver Verver Node C Select Fundament	Contours nic Frequency Gain Vs Pout Complex Imp. Net al	 Reflection Contours Power Added Eff. Comp. Vs Pout THD 	
 Power Harmo Power Node C Select	Contours nic Frequency Gain Vs Pout Complex Imp. Net al 0 Complex Imp.	 Reflection Contours Power Added Eff. Comp. Vs Pout THD 	
 Power Harmo Power Node C Select Fundamenta 0 1 2 4.8 3 7 	Contours nic Frequency Gain Vs Pout Complex Imp. Net	 Reflection Contours Power Added Eff. Comp. Vs Pout THD 	
Power Harmo Power Power Node C Select Fundamenta 0 1 2 4.8 3 7.2 4 9.6	Contours nic Frequency Gain Vs Pout Complex Imp. Net al 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	 Reflection Contours Power Added Eff. Comp. Vs Pout THD 	
Power Harmo Power Power Node C Select Fundamenta 0 1 2.4 3 7.2 4 9.6 5 12	Contours nic Frequency Gain Vs Pout Complex Imp. Net	 Reflection Contours Power Added Eff. Comp. Vs Pout THD 	
Power Harmo Power Power Power Node C Select Fundamenta 0 1 2.4 2 4.8 3 7.9 4 9.6 5 12 Add To Out	Contours nic Frequency Gain Vs Pout Complex Imp. Net al 0 10 10 10 10 10 10 10 10 10 10 10 10 1	 Reflection Contours Power Added Eff. Comp. Vs Pout THD 	

Action3-18: Select output **net RFout** on schematic.

The THD plot appears in the Waveform window.



Action3-19: Close the waveform window and click **Cancel** on the Direct Plot form.

IP3 measurements

SpectreRF offers Several ways to simulate IP3.

The first method treats one tone as a large signal, for example ω_1 , and performs a PSS analysis on this signal. The other tone, for example ω_2 , is treated as a small signal and a PAC analysis is performed based on the linear time-varying systems obtained after the PSS analysis. The IP3 point is the intercept point between the power for the signal ω_2 and the power for the signal $2\omega_1 - \omega_2$. Since the magnitude of this component is $0.75\alpha_3A_1^2A_2$, it has a linear relationship with the power level of the tone ω_2 . Thus the ω_2 component can be treated as a small signal. It is necessary to set the power level of both tones the same.

The second method treats both tones as large signals and uses a QPSS analysis.

Both the first and second methods are equivalent because of the linear dependence of the output component's magnitude, $2\omega_1 - \omega_2$, on the input component's magnitude, ω_2 . The recommended method is to use the PSS and PAC analyses for IP3 simulation because the PSS with PAC analysis method is more efficient than the QPSS analysis, and because the calculated IP3 is theoretically expected to be the same and is actually very close numerically.

The third method uses the PSS analysis with the beat frequency set to be the commensurate frequency of the two tones. Because the commensurate frequency can be very small, the simulation time for this method can be very long. This method is not recommended.

MMSIM6.0 USR2 provides new solution to calculate Rapid IP3 based on PAC or AC simulation. It is the fastest way for IP3 calculation. Rapid IP3 is a perturbative approach to solve weakly nonlinear circuit based on Born approximation. The method does not require explicit high order derivatives from device model. All equations are formulated in the form of RF harmonics. They can be implemented in both time and frequency domains.

For nonlinear system, the circuit equation can be expressed as:

$L \cdot v + F_{NL}(v) = \varepsilon \cdot s$

Here the first term is the linear part, the second one is the nonlinear part, and s is RF input source. Parameter ε is introduced to keep track of order of perturbation expansion. Under weakly nonlinear condition, nonlinear part is small compared to the linear part, so the above equation can be solved by using Born approximation iteratively:

$$u^{(n)} = v^{(1)} - L^{-1} \cdot F_{NL} \left(u^{(n-1)} \right)$$

where $u^{(n)}$ is the approximation of *v* and it accurate to the order or $O(\varepsilon^n)$.

Since the evaluation of F_{NL} takes full nonlinear device evaluation of F and its first derivative, no higher order derivative is needed. This allows us to carry out higher order perturbations without modifications in current device models. Also, the dynamic range of perturbation calculations covers only RF signals. It gives the perturbative method advantages in terms of accuracy.

Lab4: IP3 Measurement---PSS Plus PAC analysis

- Action4-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*
- Action4-2: Select the **PORTrf** source. Use the **Edit**—**Properties**—**Objects** command to ensure that the port properties are set as described below:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	sine
Frequency name 1	RF
Frequency 1	frf
Amplitude 1 (dBm)	prf
PAC magnitude (dBm)	prf

Action4-3: Check and save the schematic.

- Action4-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.
- Action4-5: You can choose **Session—Load State**, load state "Lab4_IP3_PSSPAC_shooting" and skip to Action4-12 or ...
- Action4-6: In Vituoso Analog Design Environment, choose Analyses—Choose...
- Action4-7: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set up the form as follows:

	Cancel	Defaults	Apply			He
Fund	lamental	Tones				
# N	ame	Expr	Value	Signal	SrcId	
2 R	F	frf	2.46	Large	rf	
Ĭ		. Y		Large 🚽		- 2
	Clear/Add	l Delet	e Upo	late From Sci	ematic	
\diamond	Beat Per	iod	2.4G	Aut	o Calculate 📕	
Outp Num	out harmo ber of ha	onics Armonics -	. 1			
Outp Num Accu Addil	out harmo ber of ha iracy De conserv tional Tin e Initial T	onics armonics faults (erm rative m n ne for Stal ransient R	preset) noderate pilization (ts esults (sav	liberal :tab) [einit) no	yes	
Outp Num Accu Addit Save Osci	ut harmo ber of ha racy De conserv tional Tin e Initial T llator	onics armonics faults (erry rative m n ne for Stal ransient R	preset) noderate bilization (ts lesults (sav	liberal :tab) [einit) no	_ yes	

Action4-8: Enable the Sweep button and set the sweep values as follows:

OK	Cancel	Defaults	Apply	(Help
Swe	ep 🔳			Frequency Vari	able ?	🔶 no 😞 yes	
	variaum	s		Variable Name	prf		
				Select D	esign	Variable	
Swe	ep Range						
$\overline{\diamond}$	Start-Sto Center-S	^{op} St pan	art [-50	Stop	-5	**
Şwe	Start-Sti Center-S ep Type	op St pan	art [-50 :	Stop	-5	
Swe	Start-Stu Center-S ep Type Linear	^{op} St pan	art [~	-50 ; Step Size	Stop	-5	
Swe	Start-Stu Center-S ep Type Linear Logarithn	^{op} St pan nic	art [◆	-50 ; Step Size Number of Step	Stop os	-5 20	
Swe	Start-Sta Center-S ep Type Linear Logarithm Specific I	op St pan nic Points 🗌	art [∳	-50 Step Size Number of Step	Stop os	-5 <u>.</u> 20 <u>.</u>	

- Action4-9: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window.
- Action4-10: Set up the form as shown here:

ок	Cancel	Defaults	Apply			H
Analy	ysis	<pre>tran xf pz pac qpss qpsp</pre>	 ✓ dc ✓ sens ✓ sp ✓ pnoise ✓ qpac 	 ↓ ac ↓ dcmatch ↓ envlp ↓ pxf ↓ qpnoise 	◇ noise ◇ stb ◇ pss ◇ psp ◇ qpxf	
15 8	eat from	Pe upper (11	riodic AC Ana	lysis		
Swe	eptype	-	Swe	ep is Currently	y Absolute	
Swei Input Sing Beca	eptype Frequer (1e-Poin use the a single	t [] f	Swe p Range (Hz) req 2402.! ction of the I	ep is Currently i i SS analysis i is currently st	y Absolute s active,	
Swee Input Sing Beca only Sidel Max	eptype Frequer (le-Poin use the a single bands imum sid	t [] f sweep se point for f	P Range (Hz) req 2402.! ction of the I this analysis	ep is Currently	y Absolute s active, ipported.	
Swei Input Sing Beca only Sidel Max Spec	eptype Frequer (le-Poin use the a single bands imum sid	t [] F sweep se point for f leband	P Range (Hz) req 2402.1 ction of the I this analysis	ep is Currently	y Absolute s active, ipported.	

Action4-11: Click **Ok** in the choosing Analyses form.

Your Simulation Environment should look like this:

Status: Ready	y	T=27 C Simulator: spectr	e 4
Session Setup	Analyses	variables Outputs Simulation Results Tools	Help
Desig	n	Analyses	÷Ę,
Library RFwork:	shop	# Type Arguments Enable	JAC F TRAN
Cell Diff_L! View schemat	WA_test tic	1 pac 2 2.4026 yes 2 pss 2.46 2 -50 -5 yes	
Design Var	riables	Outputs	Œ,
# Name 1	Value	# Name/Signal/Expr Value Plot Save March	j
1 frf 9 2 prf -	2,46 -50		
		Plotting mode: Replace 💷	1.

- Action4-12: In your Analog Design Environment, Choose Simulation—Netlist and Run or click the Netlist and Run icon to start the simulation.
- Action4-13: After the simulation ends, in the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.
- Action4-14: Choose **pac** and Set up the forms as follows:

- SUFERING AND IN	*****
OK Cancel	Help
Plotting Mode Append Analysis	(and
⇔pss ♦ pac	
Function	
🔷 Voltage 🛛 🔷 Voltage	e Gain
🔷 Current 🛛 🔶 IPN Cu	rves
Select Port (fixed	I R(port)) 📃 📃
Circuit Input Power 🛛 🔷 Sin	igle Point
Var prf" ranges from -50 t Input Power Extrapolation P	riable Sweep ("prf") o -5 'oint (dBm) -40
◆ Var "prf" ranges from -50 t Input Power Extrapolation F Input Referred IP3 →	riable Sweep ("prf") o -5 'oint (dBm) -40 Order <u>3rd -</u>
◆ Vai "prf" ranges from -50 t Input Power Extrapolation P Input Referred IP3 → 3rd Order Harmonic 1	riable Sweep ("prf") o -5 'oint (dBm) -40 Order 3rd st Order Harmonic
◆ Vai "prf" ranges from -50 t Input Power Extrapolation P Input Referred IP3 → 3rd Order Harmonic 1 -2 2.39756	riable Sweep ("prf") o -5 oint (dBm) -40 Order <u>3rd -</u> st Order Harmonic -2 2.39756
◆ Var "prf" ranges from -50 t Input Power Extrapolation P Input Referred IP3 = 3rd Order Harmonic 1 -2 2.39756 -1 2.5M 0 2.4025c	riable Sweep ("prf") o -5 Point (dBm) -40 Order 3rd st Order Harmonic -2 2.39756 -1 2.5M 0 2.40955
 ♦ Var "prf" ranges from -50 t Input Power Extrapolation F Input Referred IP3 = 3rd Order Harmonic 1 2.39756 1 2.5M 0 2.40256 1 4.80256 	riable Sweep ("prf") o -5 'oint (dBm) -40 Order 3rd - st Order Harmonic -2 2.39756 -1 2.5M 0 2.40256 1 4.80256
 ♦ Vai "prf" ranges from -50 t Input Power Extrapolation F Input Referred IP3 = 3rd Order Harmonic 1 -2 2.39756 -1 2.5M 0 2.40256 1 4.80256 2 7.20256 	riable Sweep ("prf") o -5 Point (dBm) -40 Order 3rd - St Order Harmonic -2 2.39756 -1 2.5M 0 2.40256 1 4.80256 2 7.20256
 ♦ Var "prf" ranges from -50 t Input Power Extrapolation P Input Referred IP3 - 3rd Order Harmonic 1 2.39756 1 2.40256 1 4.80256 2 7.20256 Add To Outputs 	riable Sweep ("prf") o -5 Point (dBm) -40 Order 3rd St Order Harmonic -2 2.39756 -1 2.5M 0 2.40256 1 4.80256 2 7.20256
 Val "prf" ranges from -50 t Input Power Extrapolation P Input Referred IP3 3rd Order Harmonic 1 2.39756 1 2.5M 0 2.40256 1 4.80256 2 7.20256 Add To Outputs freqaxis = absout 	riable Sweep ("prf") 0 -5 Point (dBm) -40 Order 3rd St Order Harmonic -2 2.39756 -1 2.5M 0 2.40256 1 4.80256 2 7.20256

Action4-15: Select **port load** in the Diff_LNA_test schematic.

The IP3 plot appears in the Waveform window.



Action4-16: Click Cancel in the Direct Plot form and close the waveform window.

Lab5: IP3 Measurement---QPSS Analysis with Shooting or Flexible Balance Engine

- Action5-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*
- Action5-2: Select the **PORT rf** source. Use the Edit—Properties—Objects command to ensure that the port properties are set as described below:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	500 mV
Source type	sine
Frequency name 1	RF
Frequency 1	frf
Amplitude 1 (dBm)	prf
Frequency name 2	RF2
Frequency 2	frf+2.5M
Amplitude 2 (dBm)	prf

- Action5-3: Check and save the schematic.
- Action5-4: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.
- Action5-5: You can choose **Session—Load State**, load state "Lab5_IP3_QPSS_shooting" and skip to Action5-9 or ...
- Action5-6: In Vituoso Analog Design Environment, choose Analyses—Choose...
- Action5-7: In the Choosing Analyses window, select the **qpss** button in the **Analysis** field of the window and set the form as follows:

	Cance	l Defaults /	Apply			He
ngi	(ine	Quasi-Periodi	c Steady St ng 🔄 Flexibl	ate Analysis le Balance		
Fu #	indamenta	l Tones	Value	Simal	Spetd H	2776
	Houne	rohr	Value	Signai	SICIU II	arms
2 3	RF2 RF2	frf frf+2.5M	2.46 2.40256	Large Moderate	rí rf	3
	RF	frf	2.46	Large 🖃] rf	1
	Clear/Ad	d Delete	Upda	te From Sch	ematic	
		Default				
Ha ev	rmonics en/odd/all	(for each m	od. tone) or	blank I		j
Ha ev Ac Sa	rmonics en/odd/all curacy De conser Iditional Ti tve Initial	(for each m efaults (errpr vative I m me for Stabi Transient Re	od. tone) or reset) oderate li lization (tsta sults (savei	blank iberal ab) nit) _ no _	yes	
Ha ev Ac Sa Sv	rmonics en/odd/all curacy De conser ditional Ti ve Initial veep	(for each m efaults (empr vative I mo me for Stabi Transient Re	od. tone) or reset) oderate 🛄 li lization (tsta sults (savei	blank [iberal ab) [nit) _ no	yes	

Action 5-8: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Now your Virtuoso Analog Design Environment will look like:

5	itatus: Re	eady	T=27 C Simulator: spectr	e 4
Se	ssion Se	tup Analyses	Variables Outputs Simulation Results Tools	Help
	De	esign	Analyses	Ŧ
Libr	ary RFw	orkshop	# Type Arguments Enable	J AC
Cell	Dif	E_LNA_test	1 qpss RF RF2 3 3 yes	ा DC च च छ
Viev	w sche	ematic		Ť † Ť x y z
	Design	Variables	Outputs	D.
#	Name	Value	# Name/Signal/Expr Value Plot Save March	4
1 2	frf prf	2.46 -40		<i>≫</i> ₩
				100
			Plotting mode: Replace -	to

- Action5-9: In your Analog Design Environment, Choose Simulation—Netlist and Run or click the Netlist and Run icon to start the simulation.
- Action5-10: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.
- Action5-11: In the Direct Plot Form, select the **qpss** button, and configure the form as follows:

	uncel			Hel
lotting M	ode Appe	nd 💴		
nalysis				
🔶 qpss				
unction				
🔷 Voltaç	le	🔷 Curre	nt	
Powel	r	🔷 Volta	ge Gain	
🔷 Currei	nt Gain	Powe	r Gain	
Trans	conductance	Trans	impedance	
🔷 Comp	ression Point	🔶 IPN C	urves	
Power	r Contours	🔷 Refle	ction Conto	urs
Power	r Added Eff.	Powe	r Gain Vs F	Pout
Comp	. Vs Pout	Node	Complex In	np.
elect	Port (f nt Input Power	ixed R(po r Value (d	rt)) Bm) -40	(m)
ielect ingle Poir Input Re	Port (f nt Input Power eferred IP3	ixed R(po r Value (d	rt)) Bm) <u>-40</u> Order <u>3n</u>]
ielect Single Poir Input Re	Port (f nt Input Power eferred IP3 Freq.(Hz)	ixed R(po r Value (d RF	rt)) Bm) <u>40</u> Order 3n RF2	
ielect ingle Poir Input Re	Port (f nt Input Power eferred IP3 Freq.(Hz) 2.3956	ixed R(po r Value (d 	rt)) Bm) -40 Order 3n RF2 -2	 d
ielect ingle Poir Input Re	Port (f nt Input Power eferred IP3 Freq. (Hz) 2.3956 2.39756	ixed R(po r Value (d RF 3 2	rt)) Bm) -40 Order 3n RF2 -2 -1	
ielect ingle Poir Input Re Input Re	Port (f nt Input Power eferred IP3 Freq. (Hz) 2.3956 2.39756 2.46 2.40256	ixed R(po r Value (d 	rt)) Bm) -40 Order 3n RF2 -2 -1 0 1	
ielect ingle Poir Input Re Input Re rd rder larmonic	Port (f nt Input Power eferred IP3 Freq.(Hz) 2.3956 2.39756 2.46 2.40256 2.40256 2.4056	ixed R(po r Value (d 	rt)) Bm) -40 Order 3n RF2 -2 -1 0 1 2	
ielect ingle Poir Input Re Input Re rd order Iarmonic	Port (f nt Input Power eferred IP3 Freq.(Hz) 2.39756 2.39756 2.40256 2.40256 2.40756	ixed R(po r Value (d RF 3 2 1 0 -1 -1 -2	rt)) Bm) -40 Order 3n RF2 -2 -1 0 1 2 3	
ielect ingle Poir Input Re rd order larmonic	Port (f nt Input Power eferred IP3 Freq. (Hz) 2.3956 2.39756 2.46 2.40256 2.4056 2.4056 2.4056 2.4056	ixed R(po r Value (d 	rt)) Bm) -40 Order 3n RF2 -2 -1 0 1 2 3	
ielect ingle Poir Input Re Input Re Input Re Input Re Input Re	Port (f nt Input Power eferred IP3 Freq. (Hz) 2.3956 2.39756 2.40 2.40256 2.40256 2.40756 2.40756 2.4056 2.4056 2.4056	ixed R(po r Value (d RF 3 2 1 0 -1 -2 0 -1 -2	rt)) Bm) -40 Order 3n RF2 -2 -1 0 1 2 3 3	
ielect ingle Poir Input Re Input Re Input Re Input Re Input Re Input Re Input Re Input Re Input Re Input Re	Port (f nt Input Power eferred IP3 Freq. (Hz) 2.3956 2.39756 2.40 2.40256 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056	ixed R(po r Value (d 	rt)) Bm) -40 Order 3n RF2 -2 -1 0 1 2 3 1 2 3 -1	
ielect	Port (f nt Input Power eferred IP3 Freq. (Hz) 2.3956 2.39756 2.40 2.40256 2.40256 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 4.79756 4.86	ixed R(po r Value (d 	rt)) Bm) -40 Order 3n RF2 -2 -1 0 1 2 3 -1 0	
ielect ingle Poir Input Re Input Re Inder larmonic st Inder	Port (f nt Input Power eferred IP3 Freq. (Hz) 2.3956 2.39756 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 2.4056 4.79756 4.80256	ixed R(po r Value (d 	rt)) Bm) -40 Order 3n RF2 -2 -1 0 1 2 3 -1 0 1 1 2 3 -1 0 1	

Action5-12: Select output port **load** on schematic.

The IP3 plot shows in the waveform window.



Action5-13: Close the waveform window. Click on Cancel on the Direct Plot form.

We are going to simulate the IP3 with Flexible Balance engine and compare its results with Shooting engine.

Action5-14:	You can choose Session—Load State , load state " Lab5_IP3_QPSS_FB " and skip to Action5-21 or
Action5-15:	In Vituoso Analog Design Environment, choose Analyses—Choose
Action5-16:	In the Choosing Analyses window, select the qpss button in the Analysis field of the window.
Action5-17:	In Engine filed, choose Flexible Balance.
Action5-18:	In field of Function Tones, choose RF. Change the Maxharms to 10, because Flexible balance need more harmonics to calculate. Click Update.
Action5-19:	Put 10n in the field of Additional Time for Stabilization (stab) .

The form should look like this:

	Cancel	Defaults	Apply			He
Analy	sis 🔍	tran	🔷 dc	4 ac	🔷 noise	
	4	∕ xf	🔷 sens	land the second	🔷 stb	
	3	∕ pz	🔷 sp	🔷 envlp	🔷 pss	
	<	/ pac	🔷 pnoise	🔷 pxf	🗇 psp	
	1	• qpss	🔷 qpac	🔷 qpnoise	🔷 qpxf	
	K	>qpsp				
	Q	lasi-Period	lic Steady S	tate Analysis		
ingine		_ Shoot	ing 📕 Flexil	ole Balance		
Tone	s					
Name	Expr	Value	SrcId Ma	axharms Over	sample Ts	tab
RF	frf	2.4G	rf	10	l ye	5
RF2	frf+2	.5M 2.40	25G rf	3	1 n	0
RF	frf	2.46	rf	10 1	yes	
	1.16	1			Update	
				1		

Action5-20: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Note: Harmonic balance engine shares the same PSS/QPSS statement with time-domain engine. A toggle button is available for user to switch between time domain shooting and HB in ADE PSS and QPSS set up form. When setting up HB QPSS/PSS analyses, the following parameter should pay attention to:

1. Maximum harmonic: Maximum harmonic ("harms" in PSS and "maxharms" in QPSS) has the most impact on HB accuracy. When inadequate harmonics are used, spectrum outside of maxharm will be folded back into harmonics inside by aliasing effect and cause error. To obtain accurate results, maxharm should be big enough to cover the signal bandwidth.

Parameter "**reltol**" or "**errpreset**" also determines the simulation accuracy. HB uses the same convergence criteria as the shooting method.

2. **tstab**: Similar to time domain shooting method, **tstab** is a valid parameter for initial transient analysis in HB. The default **tstab** for both PSS and QPSS is one cycle of signal period. For QPSS analysis, user can choose the specific tone during tstab period and only one tone is allowed. One additional cycle is run for FFT. If **tstab** is set to 0, dc results will be used as initial condition for HB.

3: **Oversample Factor:** In general oversampling is not needed. For extremely nonlinear circuits driven by sources at very high power level, it might help with convergence.

Status: Ready	T=27 C Simulator: spectr	e t
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	-7
ibrary RFworkshop	# Type Arguments Enable	JAC F TRAN
Cell Diff_LNA_test Aew schematic	1 gpss RF RF2 10 3 yes	JDC
Design Variables	Outputs	Œ
i Name Value	# Name/Signal/Expr Value Plot Save March	4
l frf 2.46 2 prf -40		
	Plotting mode: Replace	1.

Now your Virtuoso Analog Design Environment will look like:

Action5-21: In your Analog Design Environment, Choose Simulation—Netlist and Run or click the Netlist and Run icon to start the simulation.

As the simulation progresses, note messages in the simulation output log window that are different from time domain qpss:

		b
qpss: time = 10.82 ns (97.6 %), step = 4.167 ps	(1 %)	- 11-
Flexible balance		
********* initial residual *********		
Resd Norm=7.03e+03 at node RFin harm=(0 1)		
********* itor = 1 ********		
Delta Norm=1.41e+03 at node 19:bal n bal n int flow harm=	(0 1)	
Resd Norm=4.74e-01 at node RFin harm=(0 1)		
******** iter = 9 ********		
Delta Norm=2.79e+03 at node V2:p harm=(1 1)		
Resd Norm=1.22e-03 at node RFin harm=(0 1)		
********** iter = 3 *********		
Delta Norm=2.07e+00 at node V2:p harm=(1 1)		
Resd Norm=2.52e-05 at node 18.122.n1 harm=(8 0)		
****** iter = 4 ********		
Delta Norm=3.66e-01 at node V2:p harm=(1 1)		1
Resd Norm=2.61e-05 at node 18.122.n1 harm=(8 0)		
CPU time=0 s		
Total time required for gpss analysis `gpss' was 980 ms.		

- Action5-22: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.
- Action5-23: In the Direct Plot Form, select the **qpss** button. The form will the same as the one using shooting engine.

Action5-24: Select output port **load** on schematic.

The results are plotted in the Wavescan window.



Note: Flexible Balance is new feature of MMSIM6.0USR2. Flexible balance complements shooting method. It provides efficient and robust simulation for linear and weakly nonlinear circuits.

HB method is very efficient in simulating weakly nonlinear circuits such as LNA. Only a few harmonics are needed to represent accurately the solution. For highly nonlinear circuits with sharply raising/falling signals, time domain shooting is considered more suitable. However, HB may still has an advantage when exploring design trade-offs using a few harmonics where accuracy is not of primary concern.

Action5-25: After viewing the waveforms, close the waveform window and click **Cancel** in the Direct Plot form.

Lab6: IP3 Measurement---Rapid IP3 using AC analysis

- Action6-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*
- Action6-2: Select the **PORTrf** source. Use the Edit—Properties—Objects command to ensure that the port properties are set as described below:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	dc

Action6-3: Click ok on the Edit Object Properties window to close it.

- Action6-4: Check and save the schematic.
- Action6-5: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the **Tools—Analog** Environment command.
- Action6-6: You can choose **Session—Load State**, load state "**Lab6_RapidIP3_AC**" and skip to Action6-10 or ...
- Action6-7: In Vituoso Analog Design Environment, choose Analyses—Choose...
- Action6-8: In the Choosing Analyses window, select the **ac** button in the **Analysis** field of the window. Choose Rapid IP3 as Specialized Analyses. Set the Input Source 1 to /rf by select PORT rf on the schematic. Push the ESC key on your keyboard to terminate the selection process. Set the Freq of source 1 to 2.4G and Freq of Source 2 to 2.4025G. Set the Frequency of IM Output Signal as 2.3975G and the frequency of Linear Output Signal as 2.0425G. If the Maximum Non-linear Harmonics is not specified, the default value is 4. The form should look like:

	Cancel	Defaults	Apply		Hel
	dire d		AC Analysis	^p	
Swe ◆ ◇ ◇ ◇	ep Variab Frequency Design Va Temperat Compone Model Par	le vriable ure nt Paramo rameter	eter		
Swe	ep Range Start-Sto Center-Sj ep Type	p Si Dan	tart 2.46	Stop	2.40256
Add Spec	Specific I cialized Ar R	Points	1		
Sour	ce Type	isource	e 🔷 vsource	e 🔶 port	10.13
2010.00	t Sources	1 /11		Select	Freq 2.46
Input				Coloct	A REAL PROPERTY AND A REAL
Input Input	t Sources	2 /11		Select	Freq . 40256
Input Input Input	t Sources t Power (2 /r1 dBm) -4	40	Select	Freq . 4025G
Input Input Input Freq	t Sources t Power (uency of	2 /r1 dBm) IM Outpu	40 It Signal	.3975g	Freq . 40256
Input Input Input Freq Freq	t Sources t Power (uency of uency of	2 /rɪ dBm) IM Outpu Linear Ou	40 It Signal Itput Signal	. 39756 . 40256	Freq . 4025 <u>6</u>
Input Input Input Freq Freq Maxi	t Sources t Power (uency of uency of mum Nor	2 /r1 dBm) IM Outpu Linear Ou -linear Ha	40_ It Signal Itput Signal armonics	.39756 .40256	Freq . 4025 <u>6</u>
Input Input Freq Maxi Outp	t Sources t Power (uency of uency of mum Non out \blacklozenge Vol	2 /r1 dBm) IM Outpu Linear Ou -linear Ha tage Ou rent Ou	40 It Signal Itput Signal armonics 5 It+ /RFout It- /gnd!	.3975¢ .4025¢ Select Select	Freq .4025 <u>6</u>

- 5

Action6-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Status: Ready	T=27 C Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	-Ę
Library RFworkshop	# Type Arguments Enable	JAC TRAN
Cell Diff_LNA_test View schematic	1 ac 2.46 2.4026 Auto Star yes	
Design Variables	Outputs	Œ
# Name Value	# Name/Signal/Expr Value Plot Save March	1
1 prf -50		-
2 iri 2.46		
		8
	Plotting mode: Replace 🔤	to

Now your Virtuoso Analog Design Environment will look like:

Action6-10: In your Analog Design Environment, Choose Simulation—Netlist and Run or click the Netlist and Run icon to start the simulation.

As the simulation progresses, note messages in the simulation output log window that are similar to the following:



Action6-11: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**. Action6-12: In the Direct Plot Form, select the **ac** button. Choose Rapid IP3 in the function field.

Dire	ect Plot For	rm)
ок	Cancel			Help
Plottin Analys	g Mode _	Append		
🔶 ac	n.			
Functio	on			
✓ Vo♦ Ra	iltage 😞 xpid IP3	Current		
Resist	ance (Defa	ult is 50.)	Ι	
Add T	o Outputs		Plot	
> Pres	s plot butto	n on this fo	m	

Action6-13: Click Plot button to get IP3 calculation results:



Action6-14: After viewing the waveforms, click **Cancel** in the Direct Plot form.

Lab7: IP3 Measurement --- Rapid IP3 using PSS Plus PAC Analysis

- Action7-1: If not already open, open the *schematic* view of the *Diff_LNA_test* in the library *RFworkshop*
- Action7-2: Select the **PORT rf** source. Use the Edit—Properties—Objects command to ensure that the port properties are set as described below:

Parameter	Value
Resistance	50 ohm
Port Number	1
DC voltage	(blank)
Source type	dc

- Action7-3: Click ok on the Edit Object Properties window to close it.
- Action7-4: Check and save the schematic.
- Action7-5: From the Diff_LNA_test schematic, start the Virtuoso Analog Design Environment with the Tools—Analog Environment command.
- Action7-6: You can choose **Session—Load State**, load state "**Lab7_RapidIP3_PAC**" and skip to Action7-12 or ...
- Action7-7: In Vituoso Analog Design Environment, choose Analyses—Choose...
- Action7-8: In the Choosing Analyses window, select the **pss** button in the **Analysis** field of the window and set the form as follows:

Cillo	osing /	nalyses –	Virtuoso	® Analog D	esign Environm	s.
ок	Cancel	Defaults	Apply			He
		Periodic	Steady St	ate Analysis	5	
ngine		E Shoot	ing 🛄 Fle	xible Balanc	e	
Fund # Na	amental me	Tones	Value	Simal	SrcId	
_				_	s	
Ι		ļ.		Large -		
С	lear/Add	l Delet	e U	pdate From	Schematic	
-	Doot Exo	allaney				
	seat Fre Beat Per	ind	2.416	1	Auto Calculate 🔤	Ê
× .						
Outpi	ut harm	onics				
Numl	per of ha	armonics -	Ś	ĺ		
-			-10 -12			
Accu	racy De	faults (em	oreset)			
	conserv	ative 📕 n	noderate	_ liberal		
Additi	ional Tin	ne for Stab	oilization (tstab) 📗		
Save	Initial T	ransient R	esults (sa	veinit) 🗌 n	o 🗌 yes	
Oscil	lator					
Swee	p □					_
1.00.00.00.00.	and the second second					_
Enabl	led 📕				Options	

- Action7-9: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **apply** button.
- Action7-10: In the Choosing Analyses window, select the **pac** button in the **Analysis** field of the window. Choose Rapid IP3 as Specialized Analyses. Set the Input Source 1 to /rf by select PORT rf on the schematic. Push the ESC key on your keyboard to terminate the selection process. Set the Freq of

source 1 to 2.4G and Freq of Source 2 to 2.4025G. Set the Frequency of IM Output Signal as 2.3975G and the frequency of Linear Output Signal as 2.0425G. . If the Maximum Non-linear Harmonics is not specified, the default value is 4. The form should look like:

OK Cancel Defaults Apply Periodic AC Analysis SS Beat Frequency (Hz) 2.416	Help
Periodic AC Analysis SS Beat Frequency (Hz) 2.416	-
Sweeptype default - Sweep is Currently Absolute	1
Start-Stop - Start 2.46 Stop 2.40256	
Sweep Type Automatic =	
Add Specific Points 🛄	
Sidebands Maximum sideband —	
Specialized Analyses Rapid IP3	
Source Type 😞 isource 🔷 vsource 🔶 port	
Input Sources 1 /rf Select Freq 2.4g	
Input Sources 2 /rf Select Freq .4025	G
Input Power (dBm) -40	
Frequency of IM Output Signal .39756	
Frequency of Linear Output Signal .40256	
Maximum Non-linear Harmonics 🛐	
Output Voltage Out+ /RFout Select	
Enabled	

Note: Incommensurate frequencies should be used for all tones. If multiple combinations of tone frequencies match *Frequency of Linear Output Signal* or *Frequency of IM output Signal*, Spectre will complain because it can't figure out which one user wants to use as IM1 or IM3.

If output is current in a port, user must use the 'save' statement to indicate that the port current needs to be computed. Otherwise, Spectre won't calculate it.

Action7-11: Make sure the **Enabled** button is on. In the Choosing Analyses window, hit the **OK** button.

Status: Ready	T=27 C Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	٠Ę
ibrary RFworkshop	# Type Arguments Enable	JAC F TRAN
Cell Diff_LNA_test Aew schematic	1 pac 2.46 2.4026 yes 2 pss 2.416 5 yes	- DC
Design Variables	Outputs	E:
# Name Value	# Name/Signal/Expr Value Plot Save March	-
		1000
	Plotting mode: Replace 🛶	to .

Now your Virtuoso Analog Design Environment will look like:

Action7-12: In your Analog Design Environment, Choose Simulation—Netlist and Run or click the Netlist and Run icon to start the simulation.

As the simulation progresses, note messages in the simulation output log window that are similar to the following:

- Action7-13: In the Virtuoso Analog Design Environment, Choose **Results—Direct Plot—Main Form**.
- Action7-14: In the Direct Plot Form, select the **pss** button, and configure the form as follows:



Action7-15: Select output port load on schematic.

Action7-16: After viewing the waveforms, close the waveform window. Click **Cancel** in the Direct Plot form

Conclusion

This application note discusses LNA testbench setup, LNA design parameters, and how to use SpectreRF to simulate an LNA and extract design parameters. Some useful SpectreRF analysis tools for LNA design, such as SP, PSS, Pnoise, PAC and QPSS analyses are addressed. The results from the analyses are interpreted.

Reference

- [1] "The Designer's Guide to Spice & Spectre", Kenneth S. Kundert, Kluwer Academic Publishers, 1995.
- [2] "Microwave Transistor Amplifiers", Guillermo Gonzalez, Prentice Hall, 1984.
- [3] "RF Microelectronics", Behzad Razavi. Prentice Hall, NJ, 1998.
- [4] "The Design of CMOS Radio Frequency Integrated Circuits", Thomas H. Lee. Cambridge University Press, 1998.