# Massachusetts Institute of Technology Department of Electrical Engineering and Computer Science

6.776 High Speed Communications Circuits Spring 2005

## Cadence and SpectreRF Tutorial By Albert Jerng 02/13/05

#### Introduction

This tutorial will introduce the use of Cadence and SpectreRF for performing circuit simulation in 6.776. Cadence contains an entire design framework for IC design, including schematic capture, layout, circuit simulation, and verification tools. We will be running Cadence Version 4.4.6 on MIT server SUN machines. The Spectre circuit simulator is run in the Affirma Analog Design Environment within the Cadence design framework. Spectre is an advanced SPICE simulator that simulates analog and digital circuits at the differential equation level. SpectreRF includes additional simulation capabilities such as periodic steady state (PSS), s-parameter analysis, and nonlinear noise analysis that make simulating RF circuits easier. This tutorial will first explain how to get the 6.776 Cadence environment running on MIT server. Then, two examples will be presented that will help you get familiarized with the SpectreRF circuit simulator.

#### Setting Up Cadence

- 1. Login to an MIT Server SUN machine
- 2. Type the following lines : *add* 6.776

source /mit/6.776/setup cadence

You can add these lines to your .cshrc.mine file so that you do not have to repeat this step each time. You must type source .cshrc.mine for the changes to take place.

 For the first time running Cadence, remove or move your ~/cds directory, then type :

cadence

Cadence version 4.4.6 should start. A  $\sim$ /cds directory will be created with the files needed for 6.776.

At this point, you should see two windows titled icfb and Library Manager. In the Library Manager, you will see the following pre-loaded libraries :

6776\_Examples, 6776\_Primitives, analogLib, basic

6776\_Primitives contains symbols for the NMOS and PMOS transistors we will be using in this class. They have a minimum channel length of 0.18 µm. 6776\_Examples contains the two example circuits that will be presented in this tutorial. Example 1 contains a narrowband RF amplifier while Example 2 contains a high frequency oscillator circuit. analogLib and basic contain many useful components for circuit simulation, including ideal voltage and current sources, and ideal resistors, capacitors, and inductors.

#### Creating a Schematic, Symbol, and Test-Bench

The first step is to create a new library that will contain the new schematics and symbols to be built.

- 1. In the Library Manager window, left-click on *File -> New -> Library*
- 2. Type in a new library name, i.e. exampleLib, and left-click OK

X	New Libra	агу	×							
	— Library —									
	Name	exampleLib								
	Directory									
	/afs/mit	.edu/user/a/j/ajerng/cdšį́								
	— Design M	anager								
	🔵 Use IN	ONE								
	🔿 Use No	DM								
_										
	ОК	Apply Cancel He	lp							

3. In the pop-up window, left-click on Don't need a techfile and then left-click OK



You should now see your new library name appear in the list of libraries of the Library Manager window. To create a new schematic :

- 1. Left-click on your library name
- 2. Go to the Cell heading and type in a schematic name, i.e. example1
- 3. Go to the *View* heading and type *schematic* and press enter
- 4. A window called *Create New File* will appear. Verify the information and leftclick *OK*

🗶 Create New File										
ОК	Cance	I	Defaults		Help					
Library Name			exampleLib 🖃							
Cell Name	• [	example1[								
View Nan	ne	schematič								
Tool	ļ	Co	mposer-S	Schematic						
Library path file										
mit.edu/user/a/j/ajerng/cds/cds.lib										

A blank schematic capture window will now appear. Play around with the pull-down menus to get familiar with the schematic capture environment. Note that many of the commands have bindkeys associated with them. The main commands you will be using . to build schematics are  $Add \rightarrow Instance$ , and  $Add \rightarrow Wire$  (Narrow). Using the bindkeys, you can invoke these commands by simply typing *i*, and *w*, respectively. When adding a wire, left-click once to start the wire router and also to stop and change directions. A double left-click will end the wire, and right-clicking will modify the snap type for the routing (orthogonal, diagonal, etc.).

Invoking *Add -> Instance* brings up a window that allows one to select a pre-built symbol using a library browser. In order to add an NMOS transistor from the 6776\_Primitives library :

- 1. Type i
- 2. Click on *Browse* and then select 6776\_*Primitives*, *nmos*, *symbol* under Library, Cell, and View, respectively

🗶 Add In	istance						
Hide	Cancel	Default	s		Help		
Library Cell	6776_Pr nmoš	6776_Primitives] nmoš					
View	symboli				.		
Names							
Array	I	Rows	1.	Columns	1		
Rotate	e	S	lideways		Upside Down		
Model næ	ne		cmosn				
Multiplier			1				
Width			10u M <u>ě</u>				
Length			180.00n	ı M <u>İ</u>			
Drain diff	usion area	a	Y				

- 3. Type in the desired dimensions for the transistor Width and Length. The units are in meters so remember to type in *u* for microns and *n* for nanometers when necessary.
- 4. Place the transistor symbol in the schematic window. You can continue adding additional transistor instances or different instances using the same Add Instance window, or stop adding instances by hitting ESC.
- 5. To save any work, left-click Design -> Check and Save. This command will check your schematic for errors or warnings, in addition to saving the schematic.

Create the schematic shown in the screenshot below using nmos transistors from the 6776\_Primitives library and ideal resistors, capacitors, and inductors from analogLib (res, cap, ind). This schematic can also be found in the 6776\_Examples library. It is named example1\_amp. Circuit nodes can be labeled by left-clicking *Add -> Wire Name*, or by typing *l*. Pins define inputs and outputs for the schematic. They can be added using *Add -> Pin*, or by typing *p*.



Now we can create a symbol for this schematic that can then be placed as an instance in a separate test bench schematic. In order to create a symbol from the schematic cell view :

1. Click Design -> Create Cellview -> From Cellview

🗶 Cellvie	ew From (	Cellview			×
ОК	Cancel	Defaults Apply			Help
Library N	lame	6776_Examples			Browse
Cell Name		example1_ampį́			
From Vie	w Name	schematic 🖃	To View Name	symboli	
			Tool / Data Type	Compose	er-Symbol 😑
Display (	Cellview	<b>I</b>			
Edit Opti	ons	×			

2. Make sure *symbol* is entered in the To View name field

3. The pins can be placed on the left, right, bottom, and top of the generated symbol. Arrange the pins in a sensible manner for use in a test bench schematic and click *OK* 

X Symbol Gen	eration Options				
OK Cancel	Apply				Help
Library Name		Cell Name		View Name	
6776_Example	50 <sup>1</sup>	example1_a	npi	symbol	
Pin Specificatio	ns				Attributes
Left Pins	VIN IB_500u				List
Right Pins	VOUT <u>ě</u>				List
Top Pins	ADDĮ				List
Bottom Pins	GND SOURCE				List
Load/Save 📃	Edit Attr	ibutes 🔟	Edit Labels	Edit	Properties 📃

4. The generated symbol should now appear. You can edit the symbol graphics as desired. Click *Design -> Check and Save* when you are done and then click *Window -> Close* 

We will now create a test bench schematic for the purpose of simulating the narrowband amplifier found in example1\_amp. Create a new schematic in the same library. In this schematic, add an instance of the symbol that you just created. Now wire up the test schematic as shown in the screenshot on the next page. A useful command that will allow you to descend into the symbol view or schematic view of the example1\_amp instance for editing is *Design -> Hierarchy -> Descend Edit*. The bindkey for this command is *E*. Typing CTRL - E will pop you back up in the hierarchy. The input and output voltage sources have 50 ohm source resistances associated with them. They are specialized sources called ports, and can be found in the analogLib library.



In this schematic, we use ideal voltage and current sources to provide the DC bias to our amplifier. VDC is an ideal voltage source found in analogLib. IDC is an ideal current source found in analogLib. We tie the GND pin of the amplifier symbol to an ideal ground component, also found in analogLib. In order to accurately model the gain of the RF amplifier, we connect the SOURCE pin of the amplifier to a 1 nH ideal inductor that goes to ground. This inductance models the bondwire that is typically present in an integrated RF amplifier. The input pin, VIN, connects to the input source through a 2.5 nH inductor. This inductance is used to match the input of the amplifier to the source resistance of 50 ohms. It is physically realized using a combination of the bondwire inductance.

Fill out the input port properties as follows.

🗶 Edit	t Object Properti	es	
ок	Cancel Apply	Defaults Previous Next	Hel
	Property	Value	Display
	Library Name	analogLib]	off 🖃
	Cell Name	porti	off 🖃
	View Name	symboli	off 🖃
	Instance Name	PORTI	off 🖃
		Add Delete Modify	
	User Property	Master Value Local Value	Display
	lvsignore	TRUE	off 🖃
	CDF Parameter	r Value	Display
Resist	tance	50 Ohnš	off 🖃
Port n	umber	1	off 🖃
DC vo	Itage	Y	off 🖃
Source	e type	sine 🖃	off 🖃
Fre	equency name 1	First	off 🖃
Fre	equency 1	50 Hž	off 🖃
An	iplifiide 1 (Vpk)		off 🖃
Am	plitude 1 (dBm)	Piri	off 🖃
Pha	ase for Sinusoid		off 🖃
Sin	ie DC level	Y	off 🖃
De	lay time	Y	off 🖃
Displa	y second sinusoi	d 🔳	off 🖃
Fre	equency name 2	Second	off 🖃
Fre	equency 2	5.16 Hz	off 🖃
Am	iplifiide 2 (VpK)		off 🖃
Am	plitude 2 (dBm)	Pinj	off 🖃
Ph	ase for Sinusoid	2	off 😑
Displa	y modulation par	ams 🔟	off 🖃
Displa	y small signal pa	rams 🔳	off 🖃
PA	C Magnitude	¥	off 🖃
PA	C Magnitude (dB	n) 👔	off 🖃
PA	C phase	Ĭ.	off 🖃
AC	Magnitude	1 Vi	off 🖃
	nhoon	Y	-66

The input port resistance is 50 ohms. It is being used as a sine wave source with two tones. Frequency 1 is called "First" and is 5 GHz with an amplitude of Pin dBm. Pin is a variable representing power in dBm that will be defined later. Frequency 2 is called "Second" and has the same amplitude, but at 5.1 GHz. The port is also being used as a small signal source with AC amplitude of 1.

Fill out the output port properties as follows -

Edit Object Properties		
OK Cancel Apply De	faults Previous Next	Hel
Apply To only curr Show syste	ent = instance = i m III user III CDF	
Browse	Reset Instance Labels D	isplay
Property	Value	Display
Library Name	analogLib	off 🖃
Cell Name	portį	off 🖃
View Name	symbolį́	off 🖃
Instance Name	PORTŐ	off 🖃
	Add Delete	Modify
User Property	Master Value L	ocal Value Display
lvsignore	TRUE	off 💻
CDF Parameter	Value	Display
Resistance	50 Ohmsį	off 🖃
Port number	Ž	off 💷
DC voltage	Ĭ.	off 🖃
Source type	sine 💷	off 🚍
Frequency name 1	,	off 😑
Frequency 1	¥	off 🗖
Amplitude 1 (Vpk)	Ĭ.	off 🗖
Amplitude 1 (dBm)		off 🖃
Phase for Sinusoid 1	j.	off 🖃
Sine DC level	Ĭ.	off 🖃
Delay time	).	off 🖃
Display second sinusoid		off 🖃
Display modulation param	•	off 🖃
Display small signal paran	s 🔟	off 🖃
Display temperature para	ns 🔟	off 🖃
Display noise parameters		off 🖃
Multiplier	1.	off 🚍

The output port is numbered "2" and also has a resistance of 50 ohms. This port is used as an output load. No signals will be applied from this port.

We are now ready to use Spectre to simulate our test schematic.

## **Simulating using Affirma**

Left-click on *Tools -> Analog Environment* of the schematic window to launch the Affirma Analog Design Environment. This will bring up the Affirma simulation window which is used to define analyses, variables, and outputs for simulations. In order to link the transistors in the 6776\_Primitives library to their appropriate models, we must first add the model files to the Affirma Setup menu.

- 1. Left-click on Setup -> Model Libraries
- 2. In the Model Library File box, type in /mit/6.776/Models/0.18u/cmos018.scs
- 3. Click Add and Click OK.

🗶 spectr	e0: Mode	l Library S	etup			X
ок	Cancel	Defaults	Apply			Help
Model	Library 1	File				Section
/mit/6	.776/Mod	els/0.18u	/cmosO18	8.scs		
Model Li	ibrary File					Section (opt.)
/mit/6	. 776/Mod	els/0.18u	/cmos018	8.scs		Ĭ.
Add	D	elete	Change	Edit File		Browse

The next step will be to define the variables we are using in the schematic, VDD and Pin.

- 1. Left-click Variables -> Edit
- 2. In the Name box, type VDD.
- 3. In the Value box, type 1.8.
- 4. Left-click Add

🗶 Editing Design Variables Affirma Analog Circuit Design 🗙										
OK Cancel Apply Apply & Run Simulatio	n	Help								
Selected Variable	Table of	Design Variables								
Name	# Name	Value								
Value (Expr) 1.8										
Add Delete Change Next Clear Find										
Cellview Variables Copy From Copy To										

5. Repeat to add Pin with value -20 We will now define DC and AC analyses.

- 1. Left-click *Analyses -> Choose* ...
- 2. Left-click *dc* and left-click on the box *Save DC Operating Point*
- 3. Left-click *ac* and fill out the box as follows

🗶 Cho	osing /	Analyses -	- Affirma An	alog Circuit	Design Envi	r 🔀
ОК	Cance	el Default	ts Apply			Help
Analy	sis	<ul> <li>↓ tran</li> <li>↓ xf</li> <li>↓ pss</li> <li>↓ psp</li> </ul>	<ul> <li>↓ dc</li> <li>↓ sens</li> <li>↓ pac</li> <li>↓ pdisto</li> </ul>	<ul> <li>◆ ac</li> <li>◇ sp</li> <li>◇ pnoise</li> <li>◇ qpnoise</li> </ul>	◆ noise ◆ envlp ◆ pxf	
			AC Analys	is		
◆ F	p van requer esign emper Compor lodel F	aoie Variable rature nent Paran Parameter	neter			
Swee Swee Li	p Ranı Start-S Center- p Typo near	ge Stop • Span e =	Start <u>1</u> िष् ्रिर्घि ् Numb	Size Size Der of Steps	op 9eğ 1eğ	
Add 5	specific	: Points )				
Enabl	ed 🔳				Option	IS

4. Left-click OK

The AC analysis will sweep frequency from 1 GHz to 9 GHz with a step size of 1 MHz. The Affirma simulation window should now look like this.

🗶 Affi	rma Analo	g Circuit De	esign Env	rironment (1	)				
Stat	us: Ready					T=27.0 C	Simulato	or: spectr	e 4
Sessi	on Setup	Analyses	Variable	s Outputs	Simulation	Results	Tools		Help
	Design				Analys	ies			Ļ
Library	6776_Exa	amples	# Ту	pe A	rguments			Enable	J AC ■ TRAN
Cell	example1	l_amp_test	1 dc	t 1	e 0e	1 M	Line	yes wee	112
View	schemati	ic	2 00	1		IM	HINC.	yco	T + I X Y Z
C	esign Varia	ables	,		Outpu	its			[]‡′
# N	ame Va	alue	# Na	me/Signal/	Expr V	alue Pl	lot Save	March	M
1 VI 2 P:	DD 1. in -:	. 8 20							হ
									<b>₩</b>
> Resu	lts ina/j/	'ajerng/simu	lation/ex	ample1_amp	_test/spect	tre/schema	atic		$\sim$

Before we simulate, we can save the variables and analyses defined in a state by leftclicking *Session -> Save State* ... and filling in a state name. Later on, we can re-open the schematic, launch Affirma, and recall the settings by left-clicking Session -> Load State ...

To run a simulation and plot results -

- 1. Make sure that you have left-clicked *Design -> Check and Save* on the schematic window to save any changes made to the schematic.
- 2. Then, go to the Affirma window and left-click Simulation -> Netlist and Run.
- You can observe node voltages or device operating points by left-clicking *Results* -> Annotate -> DC Node Voltages or Results -> Annotate -> DC Operating Points, respectively.
- 4. Left-clicking on *Results -> Direct Plot* will enable you to plot various AC parameters such as AC Magnitude, AC Phase, and AC dB20.
- 5. Alternatively, you can also left-click *Tools -> Calculator* ... This will bring up a calculator window that can be used to plot various mathematical quantities from your circuit schematic nodes.

Plot the following expressions

- 1. dB20(VF("/out"))
- 2. dB20((VF("/in")-1))



Based on what we have learned in class about S-parameters, you should see that these two quantities represent S21 and S11, respectively, for the amplifier. We will now run through several other simulations using this same test schematic.

# **S-Parameter Simulation**

- 1. Left-click *Analyses -> Choose -> sp*
- 2. Under the Ports box, select the input and output ports on the schematic
- 3. Select Frequency as the Sweep Variable
- 4. Sweep frequency from 1 GHz to 9 GHz with a linear step size of 1 MHz
- 5. Left-click yes under Do Noise and select the output and input ports
- 6. Left-click *OK* to close the Analyses window
- 7. Left-click *Simulation -> Netlist and Run* to run the simulation

🗶 Cho	osing Aı	nalyses -	Affirr	na An	alog Ci	rcuit De	esign	Envir 🔀
ок	Cancel	Defaul	ts App	ly				Help
Analy	sis	tran xf pss psp	⇔da ⇔sa ⇔pa ⇔pa	c ens ac disto	◇ ac ◆ sp ◇ pn ◇ qp	oise noise	noi: env pxf	se /lp
		S-	- Param	eter A	nalysis			
Ports						Selec	t	Clear
PORT	1 PORTO	]						
Swee	p Varial requenc Design V Tempera Compone Nodel Pa	ble Sy ariable ture ent Parai irameter	meter ,					
Swee Swee Swee	p Range Start-Si Center-S p Type near	e op Span	Start ♦	1eg Step Numb	Size er of S	Stop	9	e gr
Add S	specific	Points						
Do No yr i no	oise es o	C	Dutput p nput poi	oort rt	/P0	RTŮ <u></u> RT1 <u>i</u>		Select Select
Enabl	ed 🔳						C	Options

Left-click on Results -> Direct Plot -> S-Parameter ...

Plot S11 (dB20) and S21 (dB20) using the S-parameter results window. How do the results compare with the AC analysis we ran earlier?



You can also plot the NF of the amplifier using the same S-parameter results window. Plot NF dB10 for the amplifier. You should measure a noise figure of < 1.8 dB at 5 GHz.



## **Transient Simulation**

- 1. Left-click Analyses -> Choose ... -> tran
- 2. Set the stop time to 60n and the accuracy defaults to moderate
- 3. Left-click on Options ...
- 4. Under the Time Step Parameters heading, set the maxstep to 5p. The general rule of thumb here is that the time step should be about 1/50 of the period of the frequency of interest. In our example, the period of the 5 GHz waveform is 200 ps.

- 5. Under the Integration Method Parameters heading, set the method to gear2only. This integration method is accepted as being helpful for getting circuits to converge properly.
- 6. Left-click *Simulation -> Netlist and Run* to run the simulation
- 7. To plot transient waveforms, left-click *Results -> Direct Plot -> Transient Signal* and select nodes from the schematic to view their waveforms
- 8. Plot the transient waveform at node out



This waveform contains the addition of two sinusoidal signals at 5 GHz and 5.1 GHz. We will now use the calculator to plot the FFT of the output waveform and decipher the power gain of the amplifier.

- 1. Left-click Tools -> Calculator
- 2. Left-click *vt* and select the output node on the schematic
- 3. Left-click *dft* ... under the Special Functions box
- 4. Fill out the Discrete Fourier Transform box with the following parameters

X Discrete Fourier Transform					
OK Cancel Defaults Apply	Help				
From 10ri To 60ri	Number of Samples 4096				
Window Type 🛛 Rectangular 🖃	Smoothing Factor				
Coherent Gain (none) 💷 1					

5. Left-click OK and then on the Calculator window, take dB20 of the entire quantity by left-clicking dB20

- 6. Left-click erplot to erase the existing plot and plot the FFT
- 7. The FFT should indicate an output voltage power of -22.18 dBm at 5 GHz.

🗶 Calculator 📃 🗖 🔀												
Window Memories Constants Options Help 7												
dB20(dft(VT("/out"), 10n, 60n, 4096, "Rectangular", 1, 1))]												
Evaluate Buffer 🔲 Display Stack 🔲 🔶 standard 💠 RF												
browser	vt	it	lastx	х⇔у	dwn	up	sto	rci	Special Functions 🖃			
wave	vf	if	cle	ear	clst	app	sin	asin	mag	In	exp	abs
family	VS	is	en	ter	undo	eex	COS	acos	phase	log10	10**x	int
erplot	vdc	idc	-	7	8	9	tan	atan	real	dB10	y**x	1/x
plot	ор	opt	+	4	5	6	sinh	asinh	imag	dB20	x**2	sqrt
printvs	vn	var	*	1	2	3	cosh	acosh	fl	f2	f3	f4
print	mp		I	0	•	+/-	tanh	atanh				



Calculate the output power normalized to 50 ohms. Given that the input power is -20 dBm, what is the power gain of the amplifier?

## **Periodic Steady State Analysis**

- 1. Left-click Analyses -> Choose ... -> pss
- 2. Under the list of Fundamental Tones, click on Update From Schematic
- 3. You should see the frequency tones First and Second from the schematic in the list of tones
- 4. Select Beat Frequency and left-click Auto Calculate

- 5. A beat frequency of 100 MHz should appear in the box
- 6. Type in 60 for number of harmonics. This means that the PSS analysis will collect information on 60 harmonics of the 100 MHz beat frequency. In other words, we will have information out to 6 GHz. Since our main tones are at 5 GHz, this should be enough harmonics. If we wanted information on the 2<sup>nd</sup> or 3<sup>rd</sup> harmonics of the 5 GHz input frequency, we would need to pick at least 150 for the number of harmonics.
- 7. Under Accuracy Defaults, click on moderate. Type in 10n for the Additional Time for Stabilization.
- 8. Left-click on Sweep and sweep Pin from -30 to 0 with a stepsize of 5.

X Choosing Analyses Affirma Analog Circuit Design Environ	. 🗙
OK Cancel Defaults Apply	Help
Fundamental Tones	
# Name Expr Value Signal SrcId	
2 First 5G 5G Moderate PORT1	-
3 Second 5.16 5.16 Moderate PORT1	
Second 5.16 5.16 Moderate 2 PORT	
Clear/Add Delete Update From Schematic	
◆ Beat Frequency	
♦ Beat Period	
Output harmonice	
Number of harmonics = 60	
Accuracy Defaults (empreset)	וור
🔟 conservative 🔳 moderate 🛄 liberal	
Additional Time for Stabilization (tstab) $10r_{\underline{k}}$	
Save Initial Transient Results (saveinit) 🔲 no 🛄 yes	
Oscillator T	۱۱ -
Sweep	
Frequency Variable? The vest	
Variable Name Pirg	
Select Design Variable	
Sween Banne	
Start-Ston	
✓ Center-Span     Start   -30 grid   Stop   0 grid	
Sween Tyne	
🔹 Linear 🔹 Sten Size ————	
✓ Logarithmic ✓ Number of Steps	

- 9. Left-click OK to close the form
- 10. Left-click Simulation -> Options -> Analog
- 11. Change reltol to 1e-4. Tightening reltol will improve the simulator's accuracy and push down the noise floor allowing us to resolve individual harmonic tones. The tradeoff is that it will increase the simulation time.
- 12. Left-click *Simulation -> Netlist and Run* to start the simulation.

In order to plot results from the PSS simulation, left-click Results -> Direct Plot -> PSS

First, find the power gain with an input power of -20 dBm by clicking on the following settings :

- 1. Plot Mode -> Replace
- 2. Analysis -> pss
- 3. Function -> power
- 4. Select -> Port (fixed R(port))
- 5. Sweep -> spectrum
- 6. Modifier -> dBm
- 7. Variable Value (Pin) -> -20
- 8. Select the Output port on the schematic and plot



You should measure an output power of -12.15 dBm at 5 GHz. This corresponds to a power gain of 7.85 dB. How does this compare with the results computed from the transient analysis in the previous section?

Next, find the input 1-dB compression point by clicking on the following settings in the PSS Results form :

- 1. Plot Mode -> Replace
- 2. Analysis -> pss
- 3. Function -> Compression Point
- 4. Select -> Port (fixed R(port))
- 5. Gain Compression (dB)  $\rightarrow 1$
- 6. Extrapolation Point -> Default (-30)
- 7. Input Referred 1dB Compression
- 8.  $1^{st}$  order Harmonic -> 50 (5G)
- 9. Select the output port on the schematic
- 10. Click on replot

The plot should indicate an input 1-dB compression point of  $\sim$  -7 dBm.



Next, find the input referred IP3 by clicking on the following settings in the PSS Results form :

- 1. Plot Mode -> Replace
- 2. Analysis -> pss
- 3. Function  $\rightarrow$  IPN curves
- 4. Select -> Port (fixed R(port))
- 5. Circuit Input Power -> Single Point
- 6. Input Power Value (dBm) -> -20

- 7. Plot -> Points
- 8. Input referred IP3
- 9. Order  $-> 3^{rd}$
- 10. 3<sup>rd</sup> order Harmonic -> 49 (4.9 GHz)
- 11. 1<sup>st</sup> order Harmonic -> 50 (5 GHz)
- 12. Select the output port on the schematic
- 13. Click replot



At a Pin of -20 dBm, the plot indicates that the input referred IP3 is 3.5 dBm. Notice that the curve is fairly flat up until a Pin of -15 dBm. In general, IP3 is measured at an input power level around 10 dB less than the input 1-dB compression point. Since the input compression point was measured to be -7 dBm, a Pin of -20 dBm is an appropriate input level to measure IIP3.

This concludes our simulation of the narrowband RF amplifier example. The analyses covered in this tutorial have been saved as session states in the example1\_amp\_test schematic found in 6776\_Examples. Open the schematic, left-click on *Tools -> Analog Environment*, and then left-click on *Session -> Load State* within the Affirma window to load either the ac, transient, pss, or s-param simulation setups.

# **Oscillator Example**

Next, we will simulate an oscillator circuit using AC, transient, and PSS analyses. Create a new schematic based on the following circuit diagram. This schematic can also be found in the 6776\_Examples library. It is called example2\_osc. No symbol view was created this time. Simulation setups for this schematic are also saved under the state name allsims.



We will find the oscillation frequency of this circuit in three ways. First, we will estimate it by doing a quick AC simulation. Then, we will drive the circuit with an impulse-like current and allow the circuit to reach steady state in a transient simulation. Finally, we will use SpectreRF's PSS analysis to simulate the circuit as well as find its phase noise. The ideal current source between the nodes op and on will be used to provide an AC input as well as the impulse-like current. Fill out the parameters of the current source as follows.

X Edit Object Properties						
OK Cancel Apply D	efaults Previous Next	Help				
Apply To only current instance Show system user CDF						
Browse	Reset Instance Labels Display					
Property	Value	Display				
Library Name	analogLib	off 💷				
Cell Name	ipulse	off 🖃				
View Name	symboli	off 🖃				
Instance Name	II	off 🖃				
	Add Delete Modify					
User Property	Master Value Local Value	Display				
lvsignore	TRUE	off 🖃				
CDF Parameter	Value	Display				
AC magnitude	1 🕺	off 🗖				
AC phase	Ĭ.	off 🖃				
DC current	Ĭ.	off 🖃				
Current 1	0.0 A <u>ř</u>	off 🖃				
Current 2	4m Až	off 🖃				
Delay time	ln š	off 🖃				
Rise time	10p š	off 🖃				
Fall time	10p š	off 🖃				
Pulse width	50p š	off 🖃				
Period	lu š	off 💷				
Frequency name for 1/per	iod impulse	off 🖃				
Noise file name	Ĭ.	off 🖃				
Number of noise/freq pairs	s di	off 🖃				
XF magnitude	<u>.</u>	off 🖃				
PAC magnitude	<u>i</u> .	off 💷				
PAC phase	)	off 🖃				
Multiplier	<u></u>	off 🖃				
Temperature coefficient 1	j	off 🖃				
Temperature coefficient 2	· [	off 🖃				
Nominal temperature	j	off 🖃				
DC source		off 🖃				

Launch the Affirma Analog Design Environment by left-clicking *Tools -> Analog Environment*. You will need to again setup the model library and define variables such as VDD. Or, you can load your saved simulation state from the previous example by choosing *Session -> Load State* ... from the Affirma window. Setup the following simulations :

- 1. AC Simulation
  - a. Linearly Sweep from 1 GHz to 9 GHz
  - b. Use a step size of 1 Mhz
- 2. Transient Simulation
  - a. Use a stop time of 60n
  - b. Use an accuracy default of moderate

Run the simulations and plot the differential output waveform (op-on) for both the AC and transient cases. Calculate the oscillation frequency in the transient case by using the markers to measure the period of the sinusoidal waveform. Try plotting an FFT of the output signal to get a measure of the oscillation frequency. What is the difficulty with this approach?



AC Differential Output







Transient Waveform Period = 257.243 ps



FFT of Transient Output Waveform

# **PSS and Pnoise for Oscillator Analysis**

We will now use PSS to overcome some of the limitations that AC and transient simulations have in simulating oscillator circuits. With PSS, we will be able to accurately find the oscillation frequency of the circuit in the frequency domain and simulate the phase noise.

Setup pss and pnoise analyses according to the following forms.

🗶 Choosing Analyses Affirma Analog Circuit Design Envir.	r 🗙 Choosing Analyses Affirma Analog Circuit Design Envir 🗙
OK Cancel Defaults Apply	Help OK Cancel Defaults Apply Help
Analysis       tran       dc       ac       noise         Xf       sens       sp       envlp         pss       pac       pnoise       pxf         psp       pdisto       qpnoise	Analysis ↓ tran ↓ dc ↓ ac ↓ noise ↓ xf ↓ sens ↓ sp ↓ envlp ↓ pss ↓ pac ◆ pnoise ↓ pxf ↓ psp ↓ pdisto ↓ qpnoise
Periodic Steady State Analysis	Periodic Noise Analysis
Fundamental Tones	PSS Beat Frequency (Hz) 4. 2e9
# Name         Expr         Value         Signal         SrcId           1         impulse         1/(1u-0)         1M         Moderate         I1	Sweeptype relative Relative Harmonic
	Frequency Sweep Range (Hz)
Moderate =	Start-Stop Start 114 Stop 1014
Clear/Add Delete Update From Schematic	Sweep Type
◆ Beat Frequency	Logarithmic  Number of Steps
Seat Period	Add Specific Points
Output harmonics	Sidebands
Number of harmonics 🖃	Maximum sideband 🖃
Accuracy Defaults (empreset)	Output
_ conservative ■ moderate _ liberal	voltage  Negative Output Node /og
Save Initial Transient Results (saveinit) _ no _ yes	
Oscillator node /og	
Reference node /gndlig Sel	
Sweep 🔟	Noise Type
Enabled  Options	S
	Enabled F Options

On the pss form, we will provide an estimate of the oscillation frequency under the beat frequency box based on our previous transient simulations. Estimating on the high side of the actual oscillation frequency helps the simulator's convergence. We ask the simulator to gather information from 5 harmonics of the 4.2 GHz beat frequency. As you choose more harmonics, the simulation becomes more accurate and takes longer.

On the pnoise form, we choose a relative sweep of the offset frequencies for which phase noise will be calculated. The offset frequency is swept from 1 kHz to 10 MHz, relative to the 1<sup>st</sup> harmonic of the output waveform.

Run the simulation and plot results using Results -> Direct Plot -> PSS

Find the differential output's oscillation frequency and amplitude by using the following settings on the PSS Results form.

- 1. Plot Mode -> Replace
- 2. Analysis -> pss
- 3. Function -> Voltage
- 4. Select -> Differential Nets
- 5. Sweep -> Spectrum
- 6. Signal Level -> Peak
- 7. Modifier -> Magnitude
- 8. Choose the differential nets op and on by clicking on them in the schematic

What is the oscillation frequency and oscillation amplitude?



Now, plot the phase noise using the PSS Results form as follows.

X PSS Results		×			
OK Cancel		Help			
Plot Mode 🛛 🔷	Append 🔶 Replace				
Analysis					
🔷 pss 🔶 pnoise					
Function					
🔷 Output Noise	🔷 Input Noise				
$\diamond$ Noise Figure	🔷 Noise Factor				
🔶 Phase Noise	$\diamond$ Transfer Function				
Currently, only frequency data is available					



One can also separate out the individual noise contributors to the phase noise by using *Results -> Print -> PSS Noise Summary*.

Experiment with this form to find the top 10 noise contributors to phase noise at a 1 MHz offset frequency.

# **Printing**

Schematics :

- 1) Go to *Design -> Plot -> Submit* on the schematic window
- 2) Under Plot With, uncheck Header
- 3) Click on *Plot Options* at the bottom right of the form
- 4) Click on Send Plot Only To File and put in a filename.eps
- 5) Click *OK* on the Plot Options form
- 6) Click *OK* on the Submit Plot form to print

Waveforms :

- 1) Go to *Window -> Hard Copy* on the waveform window
- 2) Under *Plot With*, uncheck *Header*
- 3) Click on *Send Plot Only To File* and put in a filename.eps
- 4) Click OK on the Hard Copy form to print

# **Exporting Data to Matlab**

You can export data to Matlab indirectly by first printing it to a text file. A general way to do this is to use the printvs function built into the calculator.

- 1) Using the calculator, select the desired output signals
- 2) Click on printvs in the calculator window
- 3) A Printvs Range window will appear. You can define the range of values desired for your output signal, i.e. a frequency range if you are printing an AC signal or a time range if you are printing a transient signal
- 4) A Results Display window will appear. Your data will be displayed in column format. Click *Window -> Print* ... and then select *Print To -> File*. Enter a filename.txt.
- 5) You can then import the column data from the .txt file into Matlab.