Circuit Simulation using Spectre

In order to do circuit simulation in Cadence, we need to setup *Analog Artist* first. 1. Go to **Tools** in Composer-Schematic menu and choose *Analog Environment* to open Analog Environment.

- Affirma	Analog Circuit Design Environment (1)	- 1
Status: Ready	T=25 C Simulator: hspice	s 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	4
Library abc	Iype Arguments Enable	HRC TTRAN HDC
New schematic		±₽‡
Design Variables	Outputs	Dŧ″,
I Name Value	# Name/Signal/Expr Value Plot Save March.	*
		1
		1
•		2

2. Go to **Setup** option in **Analog Environment** and choose the option **Model Libraries** and enter the following lines and click **Add**:

/home/cad/kits/ IBM_CMRF8SF-LM013/IBM_PDK/cmrf8sf/V1.2.1.2LM/Spectre/models/allModels.scs

A. Transient Analysis

1. In the *Analog Environment* Window go to *Setup* and select *Design* and the Design window opens.

OK	Cancel		Help
Library	Name	abc	
Cell Na	ne	inverter	
Aew N	artua	schematic	

3. Once this is done choose the type of analysis you wish to perform. In this case now we are going to do a Transient Analysis i.e. plotting a time-voltage curve. Click on *Analyses* in the *Analog Environment*.

		Construction of the local distribution of th			
Analysis	• tran sens pac	dc sp pnotse	ac pdisto pxf	noise pss envip	⊖xf
	Tr	ansient An	afysis		
Stop Time	19-7				
Accuracy De	faults (em	preset)			
Conserv	ative i	noderate	liberal		

4. In the Choosing Analyses Window, select **tran** and enter the stop time as 1e -7. Choose **conservative** under *Accuracy Defaults*. Make sure **Enabled** is ON. Then click on options.

-		Tra	nsient Option	\$
ОК	Cancel	Defaults	Apply	Hel
SIMULAT	TON INT	ERVAL PAR	AMETERS	-1
start	Q			
outputsta	n I			
TIME STI	ep para	METERS		
step	10	⊢ ¶		
maxstep	L			
INITIAL C	ONDITIC	DN PARAMI	TERS	
ic	厚	dć 🔲 node	n ⊡ dev ⊡ all	
skiptic	m	yes 🗔 no	waveless	rampup 🕞 autodo
readic	1			

5. Under this input the start time as "0" and the step time equal to 1e-9. This means that the transient analysis will start from time t=0 and go until t=100ns having a step time of 1ns.Click **OK** in the Transient Options window and finally click **OK** for the Choosing Analyses window. These values can be fixed according to your requirements.

6. In the Analog Environment Window go to *Outputs* and select *Save All.* To plot *select signals to output*, and select allpub and click OK. This will save the voltage values of all the nets. However this is not advisable for a big design since the simulation would consume a lot of time just deselect allpub and choose *selected*. Then select only those node voltages and branch currents which you require.

7. In order to observe the voltage at particular node, go to select **Output, to be saved Select on schematic,** and then click on the particular wire. The wire will be highlighted to indicate that it is selected. In order to plot the current, select the drain node.

Cmd	2				Sel	: 0		5	stat	tus: :	Se	lect	ting	ou	tput	s t	o bi	e pl	oti -	F=2	7 C	Si	mul	lato	r: s	pec	:tre	
ools	Des	sign	W	indo	w	Ed	it	Add	a	neck	S	shee	et	Op	tion	s I	NCS	SU										Help
x			た山			1	*	-*	-	-+ t .	•			1		ų.	10 10	U.		1	-					900 11.		
3																												
1																												
2																												
2						-															-	- 1						
4								ved																				
N.																												
202																												
2 2343					. '	/dd!	I	۲۹.													VO	d!	PØ					
0- N							¢) vac	;	1.59										пе	t1	Ve	dam	c35P	3			
1					1	and!															Ň	Ŀ	W=	:2u 100	n			
1							I														ne	t/	mit	100	4.5			
1													+ 1							-								
-							V	7				nie Co	1	5.1	1:0													
												III	S	27	2=;	5.3						0	>					
1												gn	d!								пе	7	N/z					
Tur																				пе	et1	pń	Kam	o35)	U_			
h I																						L.	W=	-600 400	an : n			
×																					gr		m:1					
20														+		÷	. 11	t	÷	i.	8 1 2							
.,																							ni.					
-																						1	7					
	÷	124	÷	14		116		-+		+				-	+:	-	- 45		+:	$\left \cdot \right $.+:-	1	ł,	÷	ę	÷	1.4	itte
T	noue	e L		sho		ick	Int	foll	1		M	50	chF	iMo	nise	Po	allp	0		R:	sev	Cha	inge	eOut	sO	nSc	the	nat

8. Then to Analog Environment Simulation Window, select Output, to be plotted,

Select on Schematic, and then select the output node in your schematic. The voltage node will be highlighted and the current node will be denoted by a circle as in the earlier case above.

9. Once this is done, go to **Simulation** on the **Analog Environment** Window and select **Netlist and Run.**

10. Wait for a few seconds and after the simulation is finished, a waveform window will automatically appear.

NOTE: In case if the designed circuit is huge and if the no. of outputs to be plotted is large, the simulation might take some time. Also sometimes it might appear that there is no output waveform appears .This means that there is a problem in your circuit you designed or you have made a fundamental mistake or short circuited some node. So always check twice before running your simulation.

11. Another way of opening the Waveform Window is going to the *Results,* and select *Plot Outputs* and choose *Transient.*



B. DC Analysis

1. In order carry out a DC Analysis, go to *Choose* in the *Analog Environment* Window and in the *Choosing Analyses* Window, select dc. DC analysis can be carried out for a number of parameters. Under the **Sweep Variable** select *Component Parameter*. Then click on *Select Component* and go to the schematic and click the component whose parameter you want to sweep. Select what sort of parameter you want to sweep. Then under the **Sweep Range** enter the initial and final values of the sweeping range. Then in **Sweep Type** select *Linear* and under step size enter a suitable step size or simulation. Then click OK.

Tene	Josing A	manyses	- Annina Analo	g CI	reun Desig	in E
ок	Cancel	Defaults	Apply			He
Analy	rsis	tran sens pac	dc ac sp pdisto pnoise pxf	C ni C p: C ei	aise 🔿 xf ss avlp	
Save	DC Ope	rating Poir	DC Analysis			
Supp	an Mariah	la	The second s			
- Uwca	Temperal	nc. hume	Component Na	me	<i>rva</i>	
Ē	Design Vi	ariable	Selec	t Cor	nponent	
	Compone Model Pa	nt Parame rameter	ter Parameter Na	me	dđ	
Swee	ep Range	r.				
8	Start-Sti Center-S	op Ipan St	urt 🦉	Stop	3.3	
Swee	ер Туре		a Ohan Dina			
U	inear		Number of Ste	ps	0.1	
Add s	Specific I	Points 🗌				
Frankl					Ontions.	

2. Once this is done, go to the *Analog Environment window* and follow the same procedure for plotting as you did for Transient Analysis.

Cmd	:			;	Sel	: 0		5	Sta	tus:	Se	elec	ting	l on	tpu	ts t	o be	e pl	oti -	F=2	7 C	S	imu	late	or: s	peo	ctre	3
ools	Desi	gn	W	ndo	w	Ed	it	Add	C	heck		She	et	Op	tion	s I	NCS	su										Hel
X																												
3																												
-																												
4																												
2						-	Т																					
<								veld															Ŵ					
Ň							4															1						
202																												
					4	vaa!		vda Vda		3.3											vd	Id!	Pe					
0							Ę	2		900 B.C										ne	±l -C	VC	dan	nc35	P			
14					ş	and!	4													T	na	+7	w- Li≘	-2u 4Ø0	n.			
							4														110		m:					
												me	+1	14	a.					1								
-							Ý						. 7-	5	1:0					1								
199												31	5	2	(2=)	3.3												
												gr	1d!								ne	t7	i Ni					
Ł																				he	t1	gno	lltsn	nc39	N .			
The last																				1 1		Ļ	W=	=80 4 04	0n			
*																					gn	rai	ΰı:	1	1			
18.																					1000							
· .																												
CANE -																						~	Ļ	ia.				
C																												
-*	- 161	Æ		ai.	4	- 1	4							- 14		14		- 242	1.#31	dir:	TES.	-	14	- 46	- 14	+		-
╮.	101120	- T		hee	(C)	ick	Iné	00			M		chH	N MA	100	Day	silles	0		p.	nac	ny	-11	(m i	1 "	 "		

3. Then go to **Simulation** in *Analog Environment window* and under Simulation choose *Netlist and Run.*

4. A waveform will automatically appear showing the DC response of the circuit. We can separate all the graphs by clicking on *Switch Axes mode* on the Waveform window to

the left. Also in order to combine two graphs just select one of them and place them over the other.

5. A plot of the DC characteristic curve of the inverter is shown and the drain current **Id** is shown. On the Waveform window go to Markers and click on Marker "**A**". The marker can be useful to observe values at a particula r point on the graph.



C. AC Analysis

In order to conduct an AC Analyses, we will make use of an inverter circuit. Draw it in the schematic window with the specifications given below. Some of the components we will be using will be a sinusoidal source (vsin). In the *Composer* window go to *Instance* and choose vsin.



1. Follow the same steps as above and invoke the Analog Environment window.

2. Then go to *Choose* in the Analog Environment window and select AC.

3. In Choosing Analyses window, under Sweep Variable select Frequency. In the Sweep Range, enter in Start as 1 and in Stop as 1G. Then in Sweep Type, choose Logarithmic and select Points per Decade enter 10.
Click OK, choose the outputs you want to see and run the simulation.
The output is as shown in the waveform in the fig below.
Also perform a transient analysis and observe the differential inputs.

OK	Cancel	Defaults	Apply				He
Analy	ysis E	tran sens pac	dc sp pnotse	e ac polisto per	p c noi pss env	se ∩xf tp	
			AC Anal	lysis			
Swe	ep Variab	te					
000	Frequenc Design Va Temperal	y ariable lure					
C	Compone Model Pa	nt Parame rameter	iter				
Swe	Compone Model Pa ep Range	nt Parame rameter	iter				
Swe	Compone Model Pa ep Bange Start-Str Center-St	nt Parame rameter ap pan St	iter art 👔		Stop	14	
Swe Swe Log	Compone Model Pa ep Bange Start-Str Center-S ep Type arithmic	nt Parame rameter pp pan St	art 1	ints Por De mber of Si	Stop scade teps	10	
Swe Swe Log Add	Compone Model Pa ep Bange Start-Str Center-S ep Type arithmic Specific I	nt Parame rameter pp pan St Points [art 1	ints Por De mber of Si	Stop scade teps	Iď	



In order to learn Cadence much more, try out your own circuits and simulate them and observe their response. Also to learn about Cadence and Analog Environment, go to **Help** and view **Openbook Main Menu**.

D. Differential circuit DC & AC Analysis

Above DC & AC Analysis is for single-ended circuit. If the circuit is differential, such as differential opamp, the schematic of analysis will be changed. Draw the differential opamp in the schematic window with the specification given below. Some of the components we will be using will be two voltage control voltage source (vcvs) and two voltage source (vdc). In the Composer window go to instance and choose vcvs and vdc.



Then follow the same steps as above single-ended circuit DC & AC Analysis.