## **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.

Status: Ready	T=25 C Simulator: hspice	S 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	×,
Library abc	I Type Arguments	URC TTRAN VEC
Cell inverter Aew schematic		ţţį
Design Variables	Outputs	
/ Name Value	# Name/Signal/Expr Value Plot Save March	y
		8
		10
		in

2. Go to **Setup** option in **Analog Environment** and choose the option

Model Libraries and enter the following lines and click Add:

/home/cad/ee6326/ncsul1.3/local/models/spectre/nom/tsmc35P.m.n88y

/home/cad/ee6326/ncsul1.3/local/models/spectre/nom/tsmc35N.m.n88y

			spectre	): Model Library Setup	
ок	Cancel	Defaults	Apply		Hel
Model	Library 1	File		Section	
e/c	ad/ee632	6/ncsu1.3)	(local/mod	dels/spectre/nam/tsmc35P.m.n88y	
				dels/spectre/nom/tsmc35P.m.n88y dels/spectre/nom/tsmc35N.m.n88y	
e/(		6/ncsul.3/			pt.)
e/(	ad/ee632	6/ncsul.3/		dels/spectre/nom/tsmc35N.m.n88y	pt.)

## 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	me	inverter	
View N	lame	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*.

OK Cance	al Defaults	Apply	He
Analysis	• tron sens pac	dc ac noise sp pdisto pss pnoise pxf envlp	C xf
	Tn	ansient Analysis	
Stop Time	10-7		
Accuracy D	a second second second	preset) 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.

OK (	ancel	Defaults	Consta	Help
UN C	ancei	Deraults	wheek	nes
				- 1
SIMULATIC	ON INT	ERVAL PAR	AMETERS	
start	Ő			
outputstarl	. 1			
ouquisian	· +			
TIME STEP	PARA	METERS		
step	le	⊢¶		
maxstep	L			
INITIAL CO	NDITIC	on Parami	ETERS	
ic	厚	dc 🥅 nodi	a ⊡ dev ⊡ all	
skipdc		yes 🥅 no	🗔 waveless 📋 ran	npup 🗌 autodic
	I			

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	:				Sel	: 0		S	Stat	us:	Se	lectin	g ol	stput	s t	to be	e pl	oti -	F=2	7 C	S	imul	ato	r: s	pec	tre	
ools	Des	sign	W	indo	w	Ed	it	Add	a	neck	5	Sheet	Op	otions	s	NCS	U										Help
Y	11 次 11 以	1 1 1	変更						「山谷						18 11		2011年	1910 - Z	変動		学生	211	16 St	1910 - E	を長		微調
\$ <u>}</u>																											
2																											
2								vdd														мd					
2																						11 M					
$\gtrsim$					×	/dd!	Ē	V1 ) <sup>vdc</sup>	:=3	.3									ne		Id! Vc	Р0 dlam	035F				
- /* //* \					ç	Ind!														ne	et7	₩=  =4 m:1	2u 100	n			
2							$\downarrow$	gnd				net1		0													
												J.T.L. gnd!	÷	v2=3	5.3						Ċ	2					
Time																			ne	ne t1	st7 gine	tom w=	:60(	i. Øn			
×																		1997 - 1994 1997 - 1997 - 1994 1997 - 1994 1997 - 1994 1997 - 1994 1997 - 1997 - 1997 1997 - 1997 - 1997 1997 - 19		ġr	nd!	<u>ا:</u> تر) ا:تر)	100	n			
·· x																						( 					
	》 (新 2、 <del>第</del>	21. 24	青 法		42 回	12.12	t.		<b>林</b>	読み		蒲 精 河 著		部 40	清朝		<b>*</b>	101 - Se	济法	100 - 200 100	4	7	清	- 10 - 10	許に注	2	が、金
$\rightarrow$	mous	se L	: s	shou	wCl	ick	In	fo()			M	schi	HiM	ouse	Poj	pUp	()		R:	sev	Ch	ange	eOut	:50	nSc	hen	uat

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.

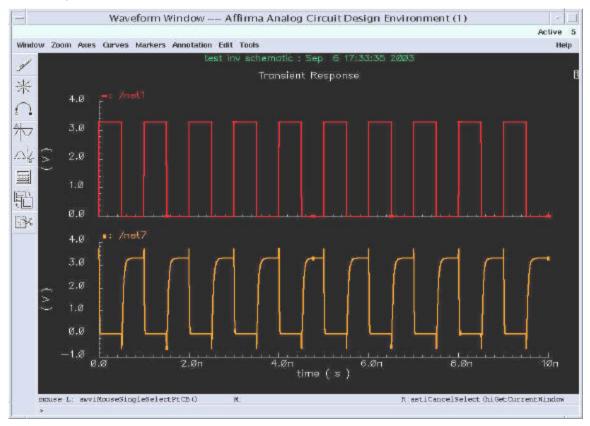
9. Once this is done, go to **Simulation** on the **Analog Environment** Window and select **Netlist and Run.** 

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**.

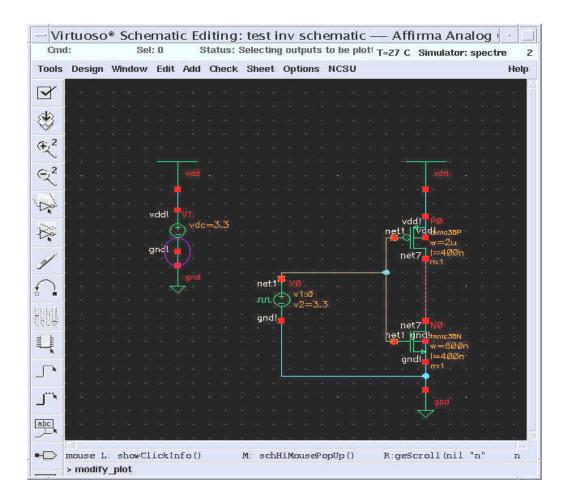


## **B. DC Analysis**

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.

OK	Cancel	Defaults Ap	ply		Help
Analys	is (		p C pdisto C p	oise () xf ss ivlp	
Save I	DC Ope	DC rating Point	: Analysis		
	Variab Imperat	ture	Component Name	₽VĂ	
	esign Vi	ariable nt Parameter	Select Cor	nponent	
	Contraction of the second second	rameter	Parameter Name	dď	
. 5	) Range art-Sti inter-S	op our	q stop	3.3	
and the second second	i Type ear	E (	<ul> <li>Step Size</li> <li>Number of Steps</li> </ul>	0 I	
Add Sj	ecific I	Points 🗌			

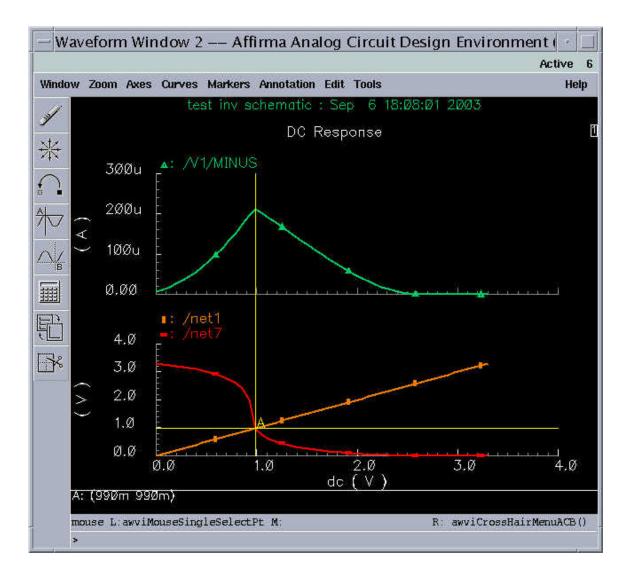
2. Once this is done, go to the *Analog Environment window* and follow the same procedure for plotting as you did for Transient Analysis.



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.

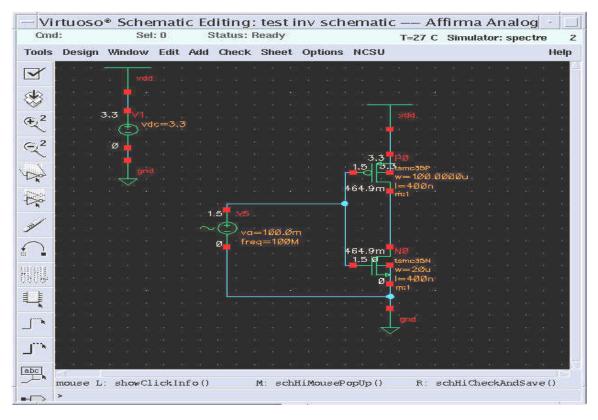
5. A plot of the DC characteristic curve of the inverter is shown and the drain current **I**<sub>d</sub> is shown. On the Waveform window go to Markers and click on Marker "A". The marker can be useful to observe values at a particular point on the graph.



## 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**.

- 4. Click OK, choose the outputs you want to see and run the simulation.
- 5. The output is as shown in the waveform in the fig below.
- 6. Also perform a transient analysis and observe the differential inputs.

The University of Texas at Dallas

	Cancel	Defaults	Apply				He
Amathy	ysis	tren sens pac	dc sp pnoise	e ac pdisto perf	pss env		
			AC Anal	lysis			
Swe	ep Variab	le					
	Frequenc	y.					
	Design Vi						
	Temperal Compone	ture nt Parame	ter				
1.1	Model Pa	rameter					
5.7	Model Pa	rameter					
( ))	Model Pa	rameter					
	Model Pa ep Range						
Swe	ep Range Start-St				Star	16	
Swe	ep Range		art [][		Stop	Iġ	
Swe	ep Range Start-St Center-S		art 👔			Iđ	
Swe	ep Range Start-St Center-S ep Type		art 👔	ints Per De	cade	10	
Swe	ep Range Start-St Center-S		art 👔	ints Per De mber of Str	cade		
Swe Swe Log	ep Range Start-Sti Center-S ep Type artthmic	i op span St 21	art 👔 • Poi Nu		cade		
Swei Swei Log	ep Range Start-Sti Center-S ep Type artthmic		art 👔 • Poi Nu		cade		

dow 2	Zoen Aves C	urves Mar	kers Annotation	a Edit. Tools			Ho
			test inv sci	hematic : Sep	6 19:08:56 2	003	
				AC Respo	nse		
	630m 🚅						
1	610m						
8							
-	. 590m E						
	57øm E						
	189.9	: phage	DegUnwropp	ved(VF(**/net)	?")/VF("/net1	<u>))</u>	
~	177.0						
-Fep							
č	174.0					1	
	171.0		100	1øk	ÍM	120M	1øg
			1040		(Hz)	1000001	

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**.