Power Measurement in Hspice

(Last updated: Nov. 2, 2009)

Before you start

Finish tutorial 1 before you start this tutorial.

1. Make Schematic and Symbol for your circuit

Please refer to tutorial 1 (section C/D)

2. Analog circuit simulation

A. Create a schematic view for testing, and initiate the circuit you want to test (tutorial 1, section F)

B. Add a DC source and set the DC voltage to 1.2V. (Note: the instance name for the voltage source is V0, and you will use this name for power measurement)



Fig. 1

C. Fill the *Stimuli, Design variables, and the Outputs* (tutorial 1, Section F). Make sure you **turn off** the VDD!, since you already have a voltage source in your schematic.

-	1 2 2	08.39.11		13	1000
ок	Cancel	Apply			Hel
Stimul	us Type	\Diamond Inputs	🔶 Global Sou	rces	
10.00	2010-00-00 C				
N I	VSS! /gn VDD! /gn	d! Voltage d! Voltage	de "DC volta de	age"=0	
IN IFF	VSS! /gn VDD! /gn	d! Voltage d! Voltage	dc "DC volta dc	age"=0	
)N)FF	VSS! /gn VDD! /gn	d! Voltage d! Voltage	dc "DC volta dc Change	age"=O	
in able	VSS!∕gn VDDI⁄gn	d! Voltage d! Voltage	dc "DC volta dc Change	age"=0	Voltage
IN I	VSS!/gn VDD!/gn	d! Voltage d! Voltage Function	dc "DC volta dc Change	age"=0 Type	Voltage
inable C ma	VSS!⁄gn VDD!⁄gn d ∎ gnitude	d! Voltage d! Voltage Function	dc "DC volta dc Change dc = [age"=0 Type	Voltage
nable C ma	VSS! /gn VDD! /gn d gnitude Itage	d! Voltage d! Voltage Function	dc "DC volta dc Change dc =	age"=0 Type	Voltage

D. Choose *transition analysis* and give the *start/stop/step time*.

Status: Ready			T=25 C Simula	tor: hspiceD	<u>el – </u>
ession Setup Analyse	es Variables Out	tputs Simulation	Results Tools		Help
Design		Analy	ses	-	۲,
brary CDSLIB	# Type	Arguments	*****	. Enable	
I inv test	1 tran	10p 10n	0	yes	
ew schematic					 † X Y
Design Variables		Outo	ute		Π‡
\mathbf{X} Choosing Analyses -	- Virtuoso® Analog) Design Environmer	nt (1)	,	K
OK Cancel Defa	ults Apply			Hel	p
Analysis 🔷 dc	♦ tran ◇ac ◇ Transie	rnoise 🔷 op ent Analysis			
Start 0	Stop	10n <u>ě</u>	Step 1	.0p	2
		1	-		

E. Make a new file "measure.sp". In this file, you define the voltage source that need to be measured (In this example, the voltage source is **V0**). You can also define the time period for your measurement.

🛅 measure.sp/wsma/tut1/cad1/	_ 🗆 🗙
<u>File Edit Search Preferences Shell Macro Windows</u>	<u>H</u> elp
1 .measure tran vdd_power integ par('i(v0)*v(vdd!)/9n') from=1n to=10n 2	Z

Fig. 4

F. In Setup -> Simulation Files, you need to include this measure.sp

Status: Ready	T=25 C Simulator: hspicel) ;
Session Setup Analyses	/ariables Outputs Simulation Results Tools	Help
Design	Analyses	⊦₹
Library CDSLIB	# Type Arguments Enable	
Cell inv_test View schematic	1 tran 10p 10n 0 yes	
Design Variables	Outputs	
# Name Value	# Name/Signal/Expr Value Plot Save March	A
${f X}$ hspiceD0: Simulation Files Se	up	
OK Cancel Defaults	Apply Browse	Hel
Include Path I. Definition Files I. Stimulus File (/afs/wmin	ch.edu/class/eecs427/w09/wsma/tut1/cad1/measure.sp	

3. Run the simulation and get the result

A. After you finish the simulation, a measurement file will be generated. The measurement file is in your simulation folder. You can find your simulation folder by *Setup -> Simulator Directory*

Sta	aws: Ready	T=25 C Simulator: hspice	D
ess	ion Setup Analyse	s Variables Outputs Simulation Results Tools	Help
	Design	Analyses	Ł
61 F		- I	
Ir a	$\mathbf X$ Choosing Simulate	or/Directory/Host Virtuoso® Analog Design Environment 💶 🗖 🗙	
41	OK Cancel	Defaults Help	
9W	Simulator	hspiceD	T + x y
	Simulator		
	Project Directory	/tmp/Hspice_wsma	1
	Host Mode	▲ local ∧ romate. ∧ distributed	Ľ
	TIDST MOUE		100
	Host		10
	Remote Directory		10

Fig. 6

B. open the *.mt0, and you can see the measured power number.

