EE434 ASIC & Digital Systems

HSPICE

Dae Hyun Kim daehyun@eecs.wsu.edu



• HSPICE is a SPICE software for transistor-level circuit analysis.

How to Run HSPICE

- Go to the Lab directory in the class webpage, download tutorialhspice.zip, and unzip it.
 - unzip tutorial-hspice.zip
- Run the following command:
 - > source ictools_generic.sh
 - > source synopsys.sh

(If you are using cshell, run "bash" first and then source the above files.)

How to Run HSPICE

 When you source synopsys.sh, you should be able to see the following messages:

synopsys.sh: adding component "hspice" (ver K-2015.06-SP1) synopsys.sh: adding component "syn" (ver K-2015.06-SP2) synopsys.sh: adding component "vcsmx" (ver K-2015.09)

- If you don't see these messages, run the following command:
 - source synopsys.sh hspice
- Now you are ready to run HSpice.

How to Run HSPICE

- Run HSPICE:
 > hspice <netlist_file_name>
- Run WaveView:
 > wv <wave_file_name>

Library Files

- You will see the following files:
 - 45nm_PTM_HP_v2.1.pm
 - 45nm transistor models for SPICE
 - inv.sp
 - An HSPICE netlist for an inverter

SPICE Netlist

- Open inv.sp in a text editor and see the contents.
- There are comments, so it won't be too hard to understand the netlist.

Run HSPICE

- Perform HSPICE simulation for the inverter as follows:
 > hspice inv.sp
- If the simulation is successful, you will see the following message:

***** hspice job concluded

• If something is wrong, you should debug it.

Run WV

- Once the simulation is done, HSPICE generates some output files.
- Let's open the waveform.

> wv inv.tr0

• Then, click "D0:inv.tr0" and click "toplevel". You will see some signals in the bottom.



Run WV

- Double-click
 - v(nln)
 - v(nOut)



How to Measure

- Click the "ruler" icon (Measurement Tool) in the icon bar.
- Choose "Rise/Fall Time" and set H(%) to 90.00 and L(%) to 10.00.

easurement Rise/Fall Time C Frequency C Jitter C Duty Cycle C Width C Difference C Y Diff C Y Range Rise/Fall Margin Threshold - Value Type C Percentage C Signal Level H(%): 90.00 L(%): 10.00 H(V): 3.0 L For Percentage Lock Level, Use Min/Max Y Levels From - Range Target C Target Signal C All Signals C User Spe Max: 3.3 Min: 0	rite All				
Rise/Fall Time C Frequency C Jitter C Juter C Jitter C Juter C Juter C Vidth C Difference C Y Diff C Y Range Several Margin Threshold Value Type C Signal Level C Signal C All Signal C Level C Signal C Level C Signal C Sign	asurement				
Duty Cycle C Width C Difference C Y Diff C Y Range Image: C Signal Level Image: C Signal Level Value Type Image: C Signal Level Image: C Signal Level Image: C Signal Level (%): 90.00 L(%): 10.00 H(V): 3.0 Image: C	Rise/Fall Time	C Frequency	C Jitter	C F VS T	
Y Diff C Y Range	Duty Cycle	C Width	C Difference	C Data(X,Y)	
ise/Fall Margin Threshold Value Type Percentage (%): 90.00 L(%): 10.00 H(V): 3.0 L(%): 3.0 L	Y Diff	C Y Range			
Value Type © Percentage C Signal Level (%): 90.00 L(%): 10.00 H(V): 3.0 L(or Percentage Lock Level, Use Min/Max Y Levels From Range Target C User Spe C User Spe ax: 3.3 Min: 0 0	se/Fall Margin Thresh	old			
Percentage C Signal Level (%): 90.00 L(%): 10.00 H(V): 3.0 L or Percentage Lock Level, Use Min/Max Y Levels From Range Target C Target Signal C All Signals C User Spe ax: 3.3 Min: 0	Value Type				
(%): 90.00 L(%): 10.00 H(V): 3.0 L or Percentage Lock Level, Use Min/Max Y Levels From	Percentage		C Signal Level		
cross posse cross posse r(v); posse posse or Percentage Lock Level, Use Min/Max Y Levels From Range Target © Target Signal C All Signals C User Spe ax: 3.3 Min: 0	(%): 90.00	1 (%): 10 00	H00: 3.0	100:03	
Range Target C All Signals C User Spe ax: 3.3 Min: 0	r Percentage Lock L	wel Lise Min/May X Levels From			
© Target Signal C All Signals C User Spe ax: 3.3 Min: 0	Range Target	vel, Ose MillyMax T Levels Floh			
ax: 3.3 Min: 0	 Target Signal 	C All Signals		C User Specified	
iax: J3.5 Mint. Ju			Mine 0		
	ax. 5.5		MIR: 0		
Ok Anniv Bemove Heln		Annhy D		Holm C	ancel

How to Measure

• Click OK. Drag and drop the measurement icon to measure the fall time. You can measure the rise time in the same way.



HSPICE Syntax (Important)

- Transistor names should begin with "m" and unique.
- Node names should begin with "n" and unique.
 - However, the ground node is always "0".
 - You can use positive integers for node names, but I do not recommend it.
- Resistor names should begin with "r" and unique.
- Capacitor names should begin with "c" and unique.
- Voltage source names should begin with "v" and unique.

HSPICE Syntax

- HSPICE is case-insensitive.
- Sub-circuit
 - Sub-circuits are used for hierarchical designs.
 - .subckt <module_name> <I/O pin 1> <I/O pin 2> ...
 statements
 .ends <module_name>

 - Sub-circuit instantiation

<instance_name> node_mapping <module_name>
*** instance_name should begin with X.