Rajagopal Vijayaraghavan
01/20/2005
Email: rvijay@utk.edu

HSPICE Tutorial

I. The purpose of this tutorial is to gain experience using the Hspice circuit simulator for Digital circuit simulation from the UNIX environment. The students are expected to have a UNIX login before they begin this tutorial. After completing this assignment, the student should be able to:

A. Use Hspice to simulate transient response of Digital circuits
B. Use Hspice to estimate the dynamic power dissipation of a circuit
C. Use DC analysis to find out the switching points of Digital Inverters
D. Use the .MEASURE statement in Hspice efficiently

The hspice manual is available in my directory ~rvijay/ ECE491/Hspice_tutorial. Copy it to you directory for reference.

II. Preliminary Setup:

A. Add Hspice to your path (the unix prompt will be assumed to be % from now). From the prompt
% source ~cad/.cshrc
% synopsys_tools
B. Next, create a simulation directory and copy all the files
% mkdir sims
% cd sims
% mkdir hspice_tut
% cd hspice_tut
You will copy the model files and the spice netlist from my directory in to your directory by using the following commands
% cp ~rvijay/ECE491/Hspice_tutorial/Inverter.cir
% cp ~rvijay/ ECE491/Hspice_tutorial /Inverter_DC.cir
% cp ~rvijay/ ECE491/Hspice_tutorial /Inverter_power.cir
% cp ~rvijay/ ECE491/Hspice_tutorial /ami_c5n_typical.txt
The file Inverter.cir is the transient response code, Inverter_DC.cir is the DC analysis spice code and Inverter_power.cir is the spice deck for estimating dynamic power dissipation.
You are ready to start the tutotal now….

III. Rise-time, Fall-time, Propagation Delay (high to low, low to high) Measurement:

A. Refer to the spice code Inverter.cir. The spice code is used to perform transient analysis. The process used here is the AMI-0.5u CMOS process (5V process). The model file is included using the .inc statement. The power supplies are defined and a pulse input is given to the inverter. The .op statement is used to calculate the operating point of the circuit.
B. To simulate the circuit using Hspice run the command
% hspice Inverter.cir > ! Inverter.lis &
C. A message appears indicating the status of the simulation.
D. The file Inverter.lis contains the details of the simulation. The file also contains the results of the .MEASURE statements that are used for computing the rise-time, fall-time and the propagation delay tphl and tplh.
E. To view the results of the simulation you have to open the Inverter.ic0 file using the Avanti Waves. Type the following command in your unix prompt

% awaves Inverter.ic0&

F. This launches the graphical tool that can be used to view the waveforms. Click on the appropriate analysis and view the waveforms.

G. The rise time, fall time etc can also be computed from the waveforms. The .MEASURE statement is an easy way of computing of the above.

H. The total propagation delay of the gate is the average of tplh and tphl.

I. The .alter statement is used to run the same set of simulations for different capacitance load, transistor sizes etc..

IV DC Analysis:

A. Refer to the spice code Inverter_DC.cir. The procedure is almost similar to the transient analysis. Type the follwing commands from your unix prompt
% hspice Inverter_DC.cir > Inverter_DC.lis &

B. The result of the analysis is stored in the Inverter_DC.lis file. The switching point obtained using the .MEASURE statement can also be found in the .lis file.

C. Open Awaves by using the command % awaves Inverter_DC.ic0 and view the waveforms.

V Dynamic Power Disspation:

A. Refer to the Inverter_power.cir . The first step is to add a periodic square wave voltage source to your spice deck. For a meaningful power measurement a periodic signal should be used instead

B. Next you must add a .MEASURE statement to find the RMS current delivered by VDD. The power dissipation can then be calculated as the product of the RMS power supply current and the power supply.

C. Run the file by using the command % hspice Inverter_power.cir > Inverter_power.lis &

Assignment :

1. Measure rise time, fall time, tplh, tphl and the propagation delay tp (tphl+tplh)/2 for load capacitance of 0.2pf, 0.5pf and 1pf. Keep the W/L =3.6u/0.6u for PMOS and 1.8u/0.6u for NMOS. Tabulate the results
2. Measure the above for W/L of 3.6u/0.6u, 7.2u/0.6u, 14.4u/0.6u for PMOS and 1.8u/0.6u , 3.6u/0.6u, 7.2u/0.6u for NMOS. Keep the load capacitance fixed at .5pf
3. Measure the above for W/L of 1.8u/0.6u for both NMOS and PMOS and 3.6u/0.6u for PMOS and 5.4u/0.6u for NMOS (fixed load cap of .5pf).
   Tabulate the results and draw inferences from the results.
4. For the DC analysis calculate the switching points for equal PMOS and NMOS, Width of PMOS larger than NMOS by 2.5 times and vice versa.
5. Tabulate the results and draw inferences.
6. For the dynamic power measurement, vary the load cap (like problem 1 and note the power dissipated.