EE434
ASIC & Digital Systems

HSPICE

Dae Hyun Kim
daehyun@eecs.wsu.edu

Overview

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

How to Run HSPICE

- Run the following command:

  > source the synopsys.sh file you downloaded from tutorial-dc.zip.

  (If you are using cshell, run “bash” first and then source the above file or just source “/net/ictools/csh/synopsys.csh”)

- Run HSPICE:

  > hspice <file_name>

- Run WaveView:

  > wv <file_name>

Library Files

- Download the following file into your working directory:
  - http://eecs.wsu.edu/~ee434/Labs/tutorial-hspice.zip

- Unzip it
  - unzip tutorial-hspice.zip

- 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:

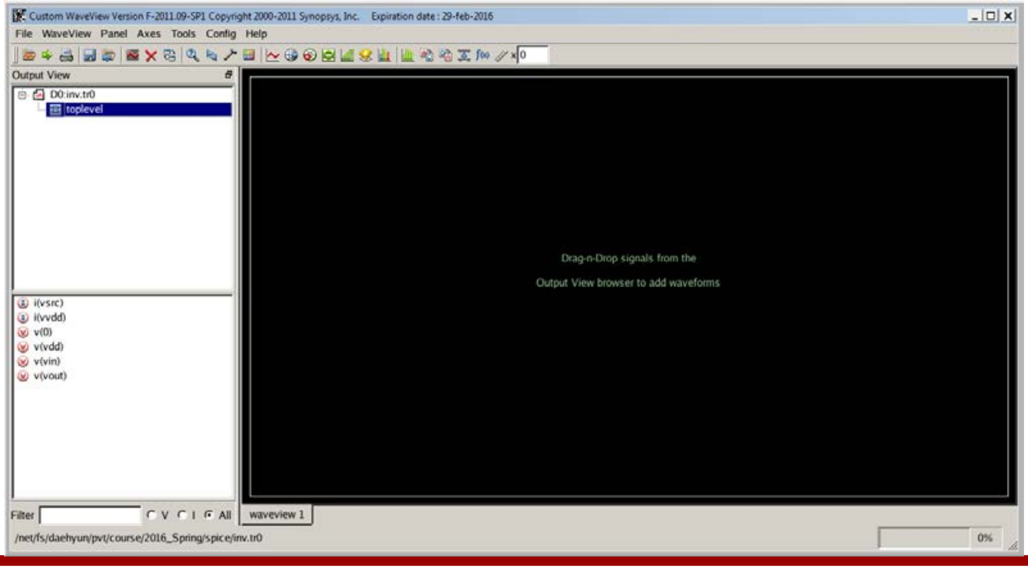
  ***** hspice job concluded

- If something is wrong, you should debug it.

Run WV

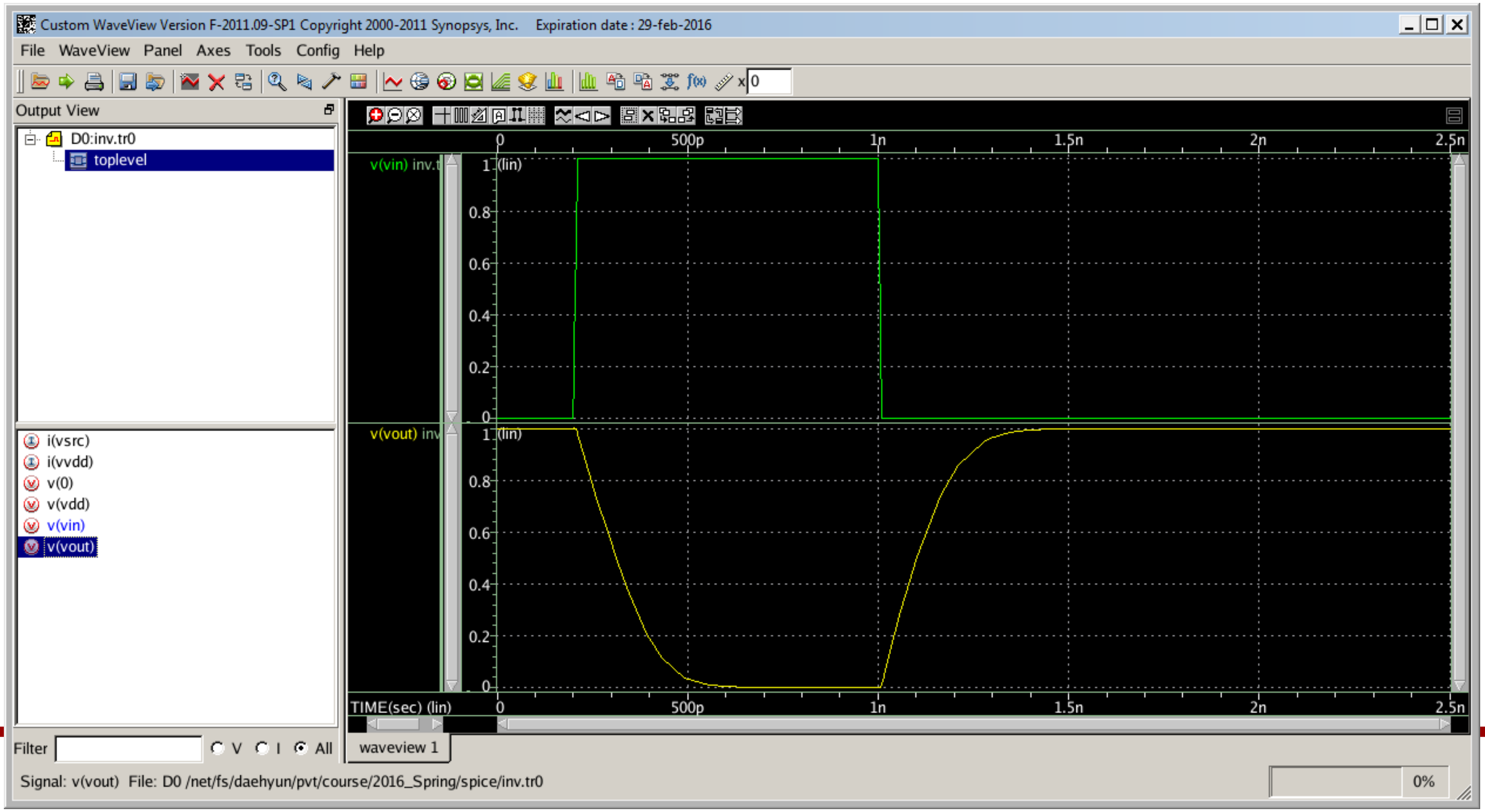
- Once the simulation is done, HSPICE generates some output files.
- Let's open the waveform.

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



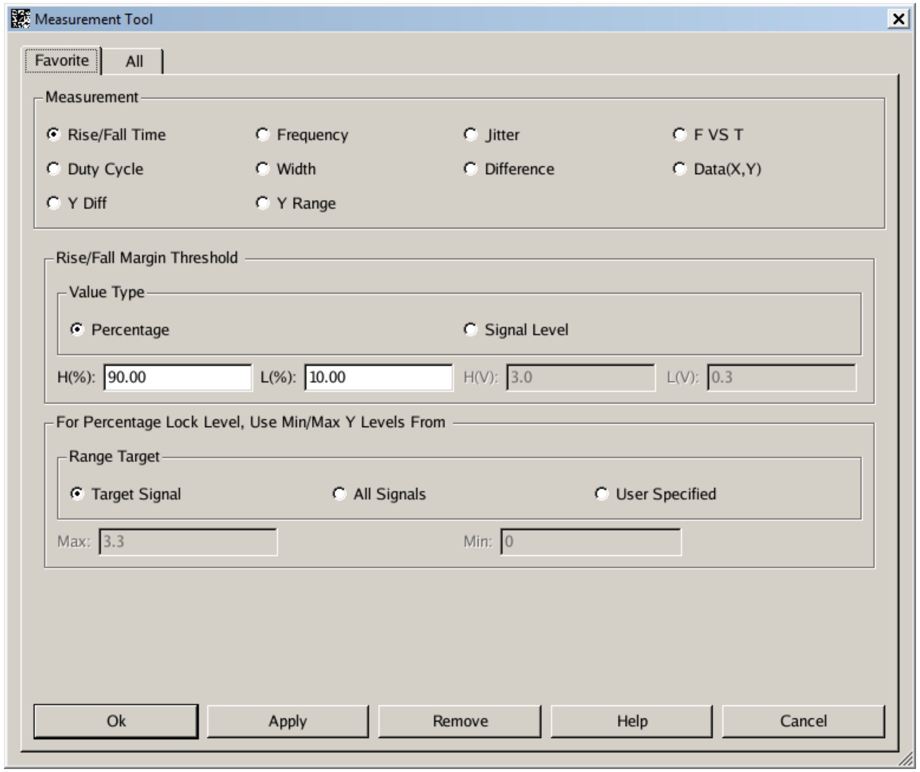
Run WV

- Double-click
  - v(vin)
  - v(vout)



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.



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.