# 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

- Go to the Lab directory in the class webpage, download tutorial-hspice.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

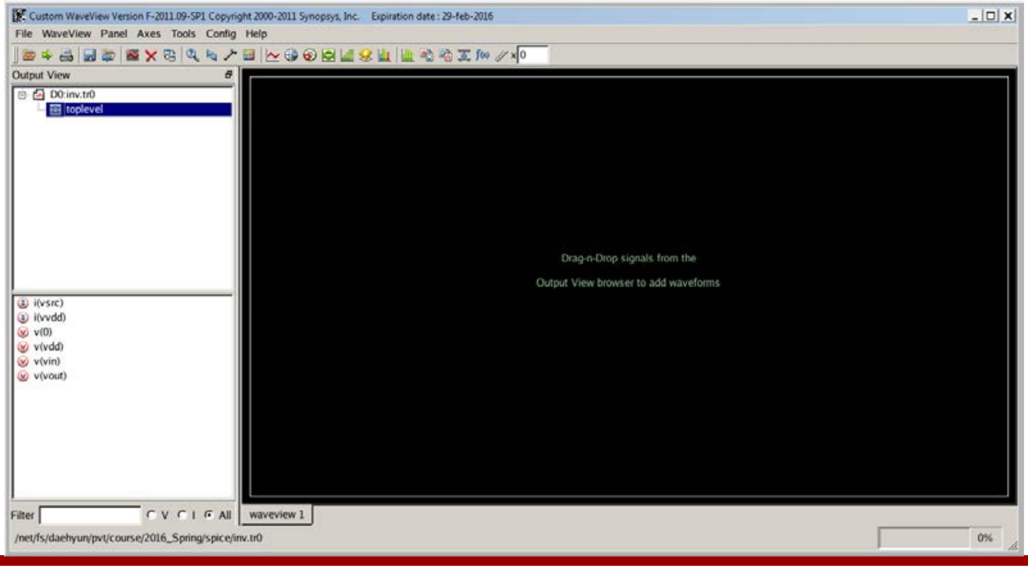
- If something is wrong, you should debug it.

# Run WV

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

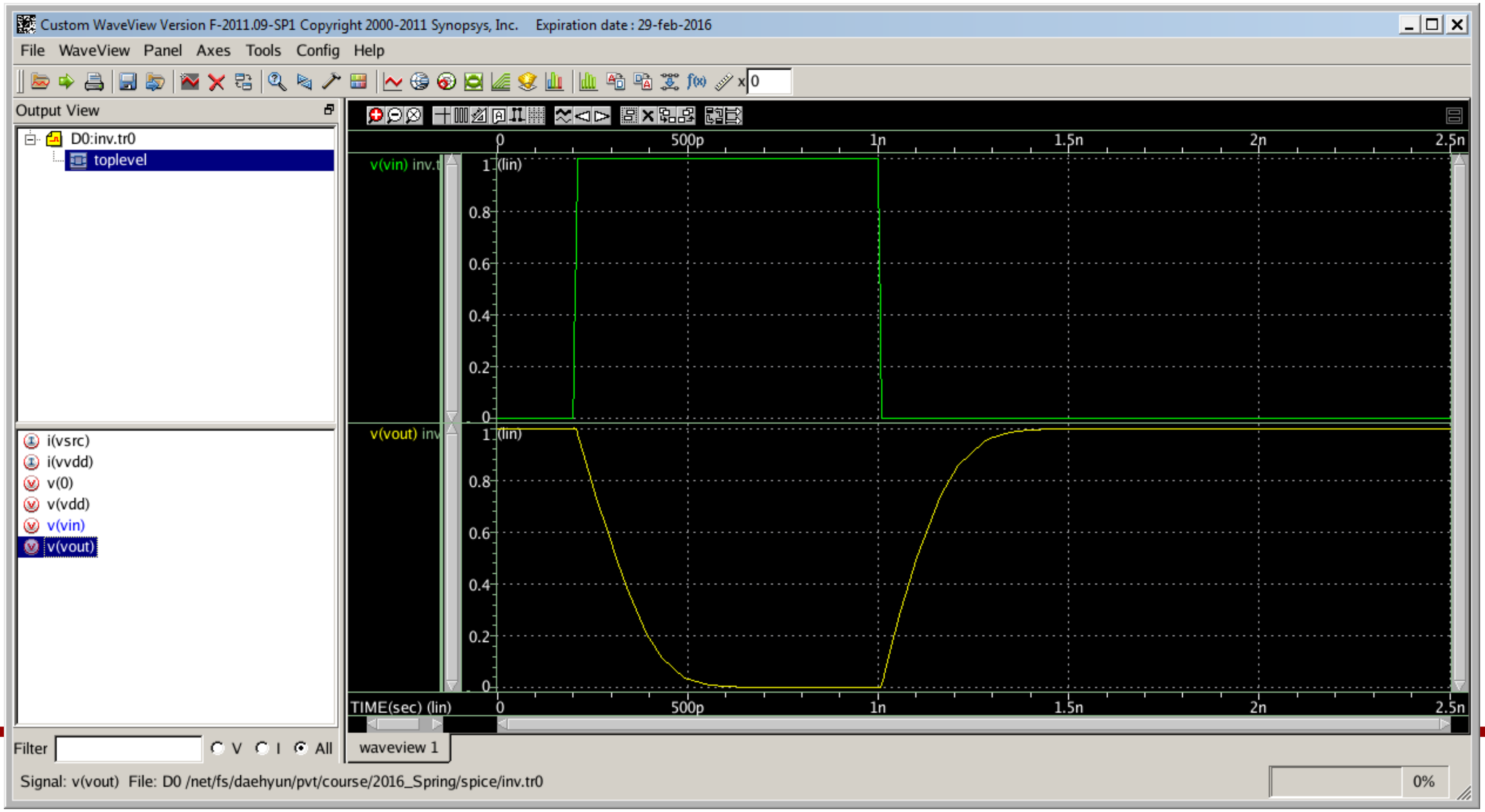
  > wv  inv.tr0

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



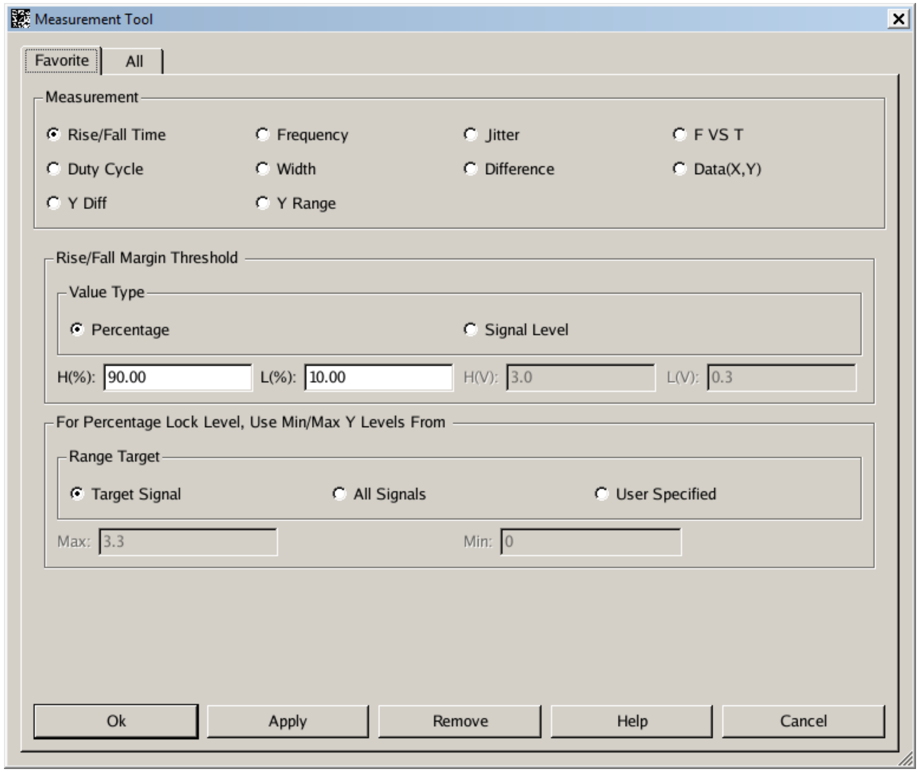
# Run WV

- Double-click
  - v(nIn)
  - v(nOut)



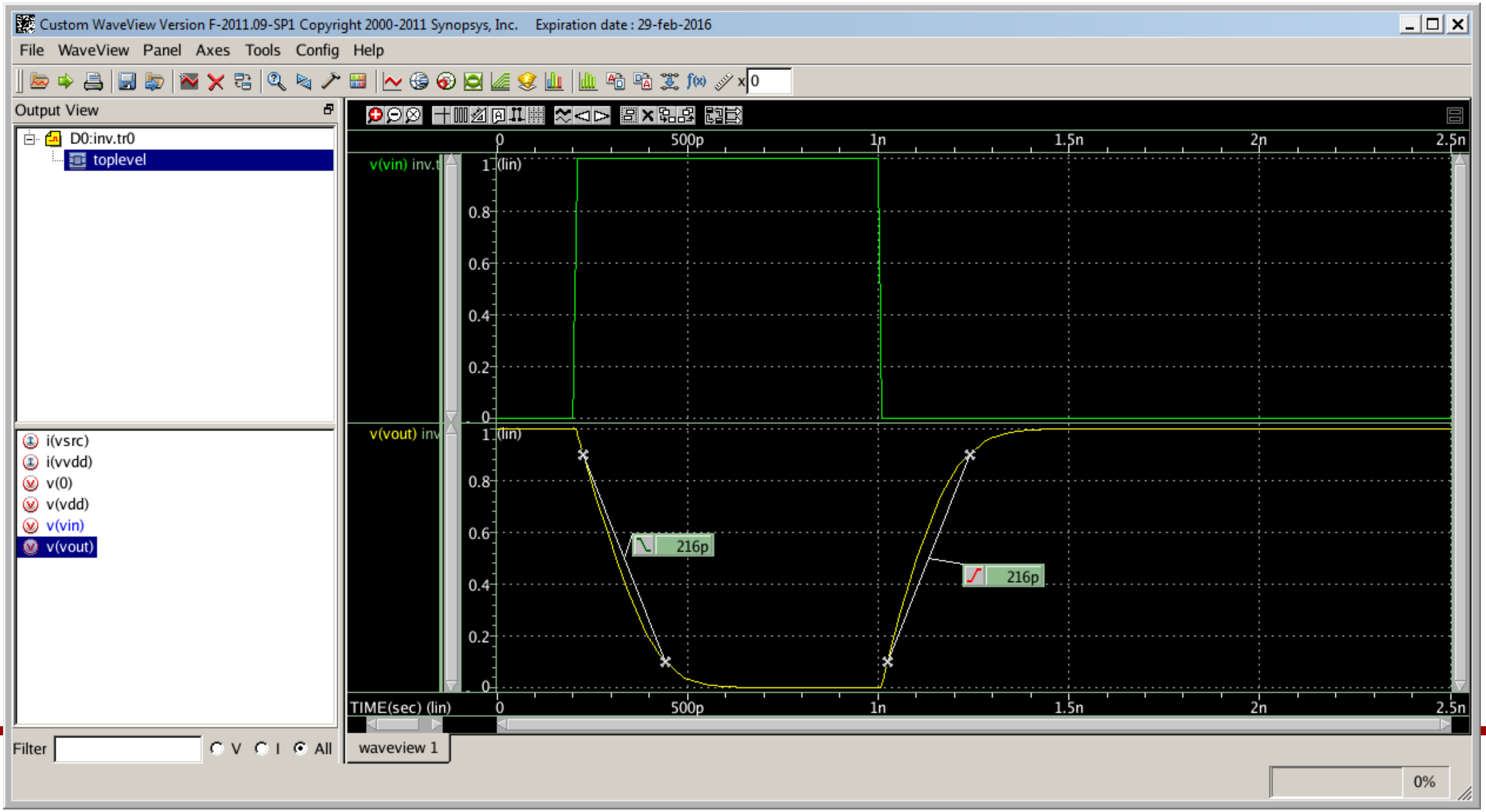
# How to Measure

- 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.



# 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.