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)

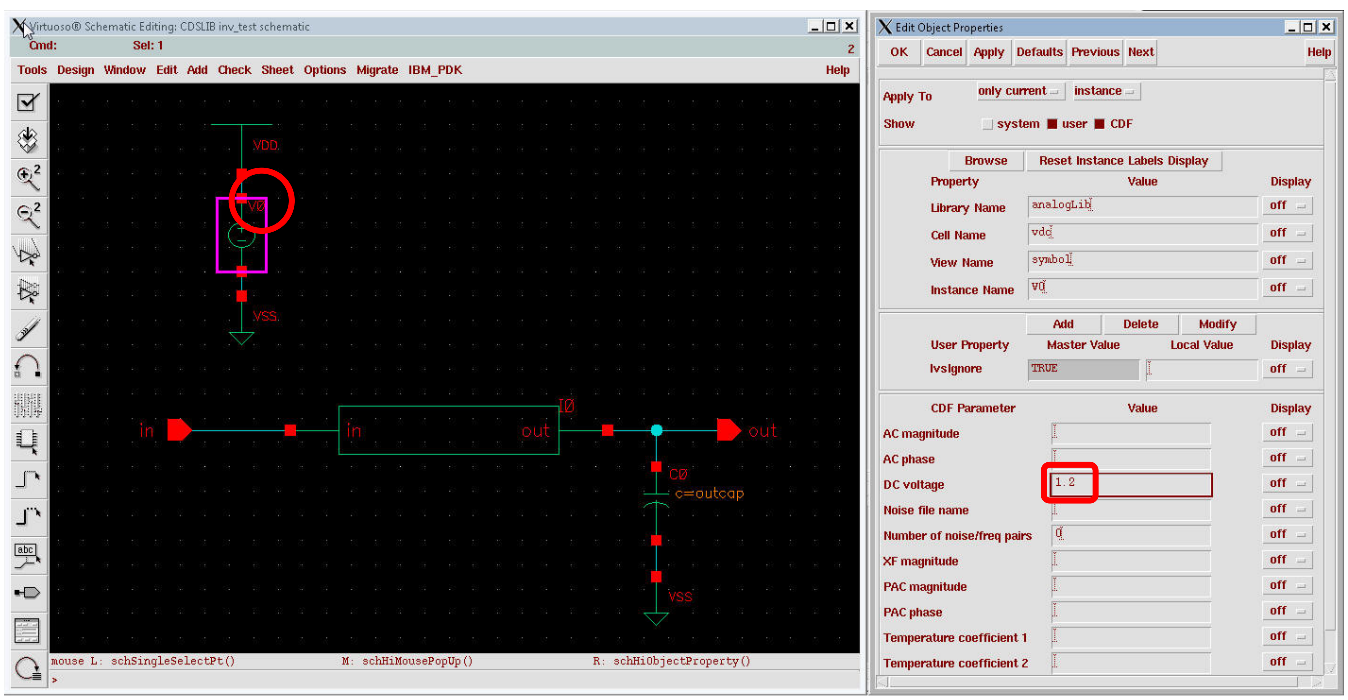

Fig. 1

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.

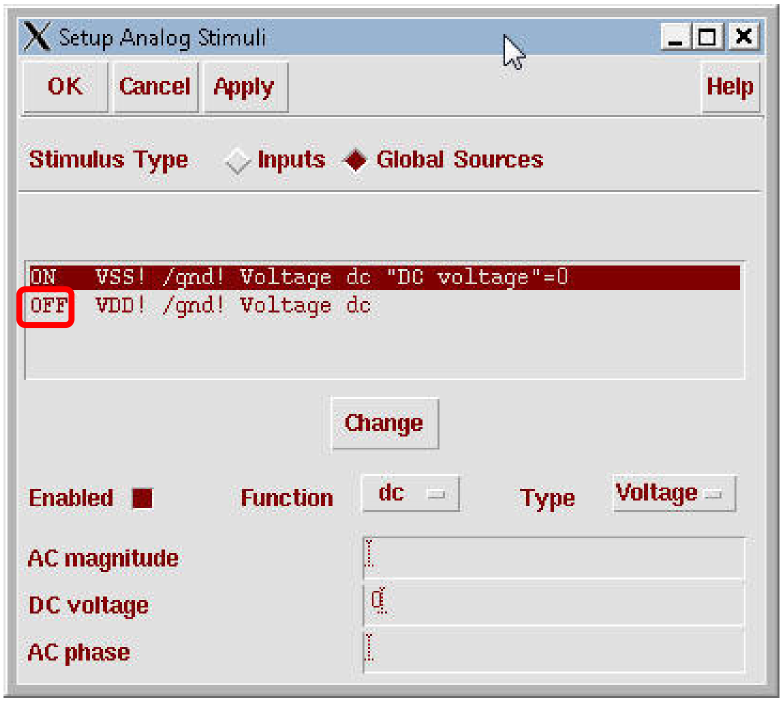

Fig. 2

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

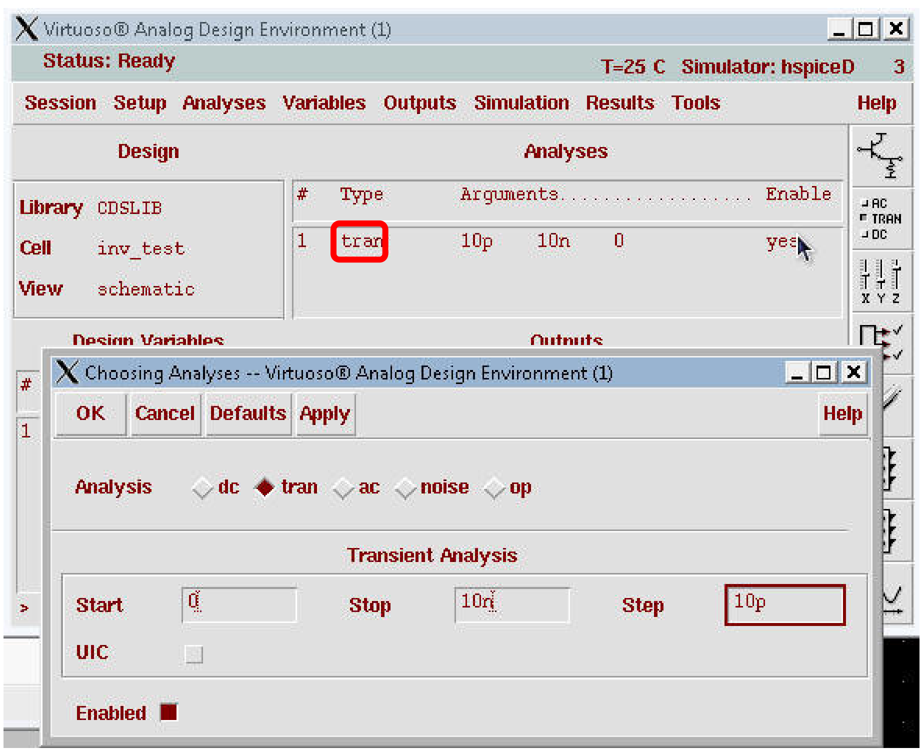

Fig. 3

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.

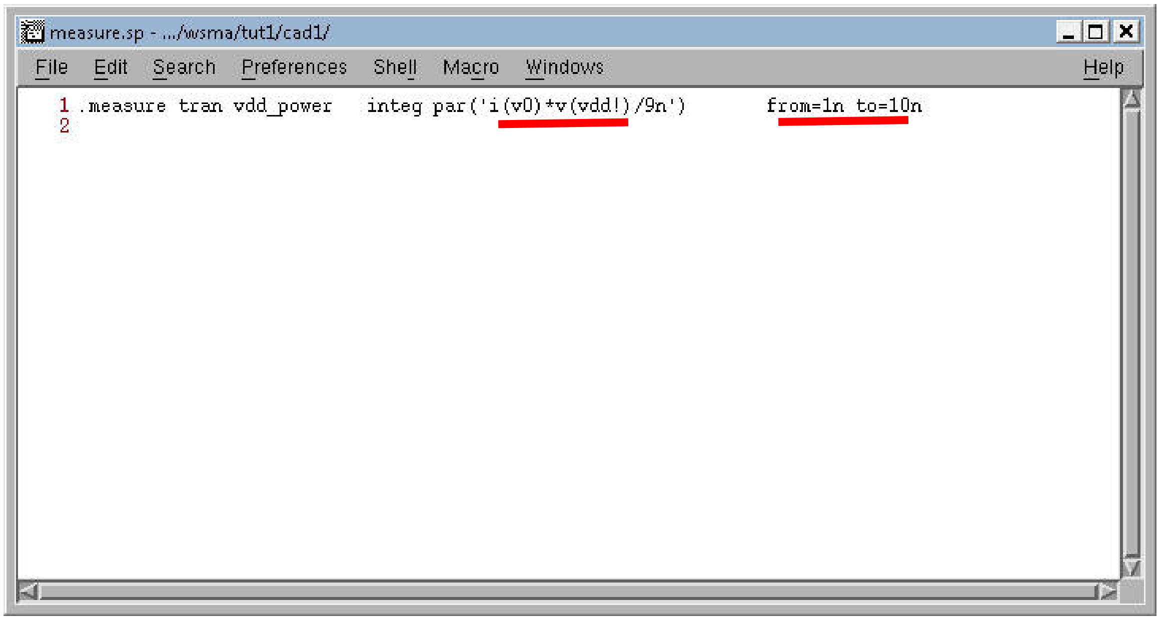

Fig. 4

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

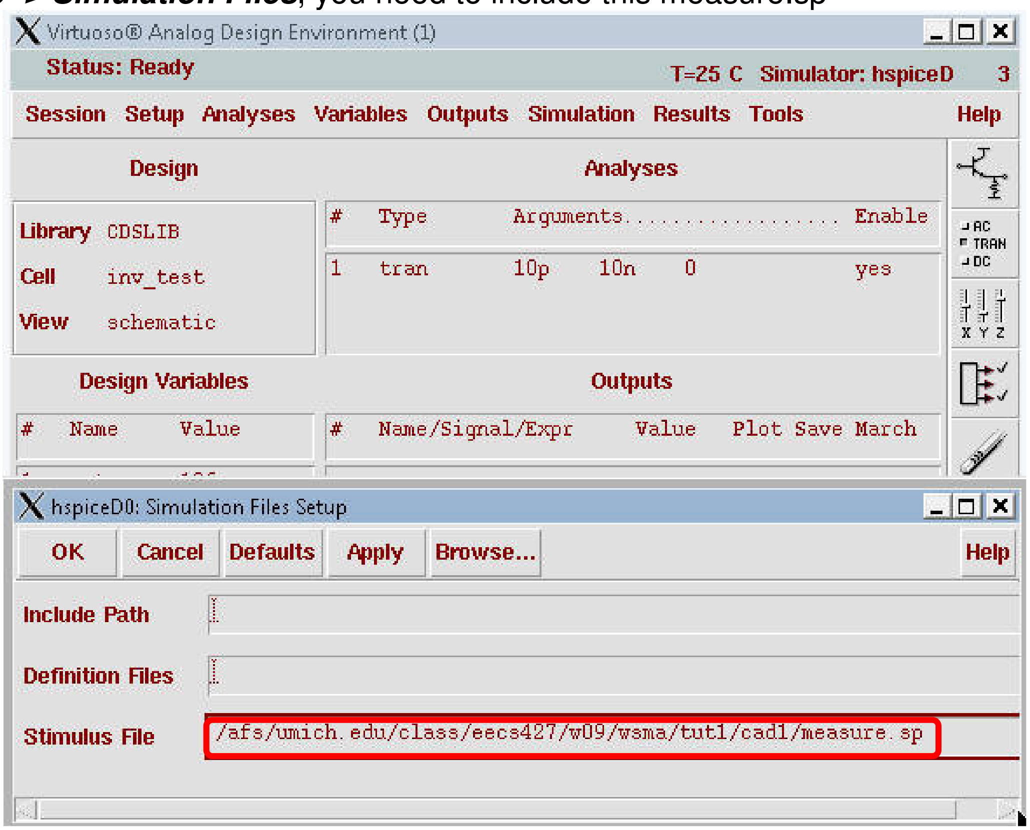

Fig. 5

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***

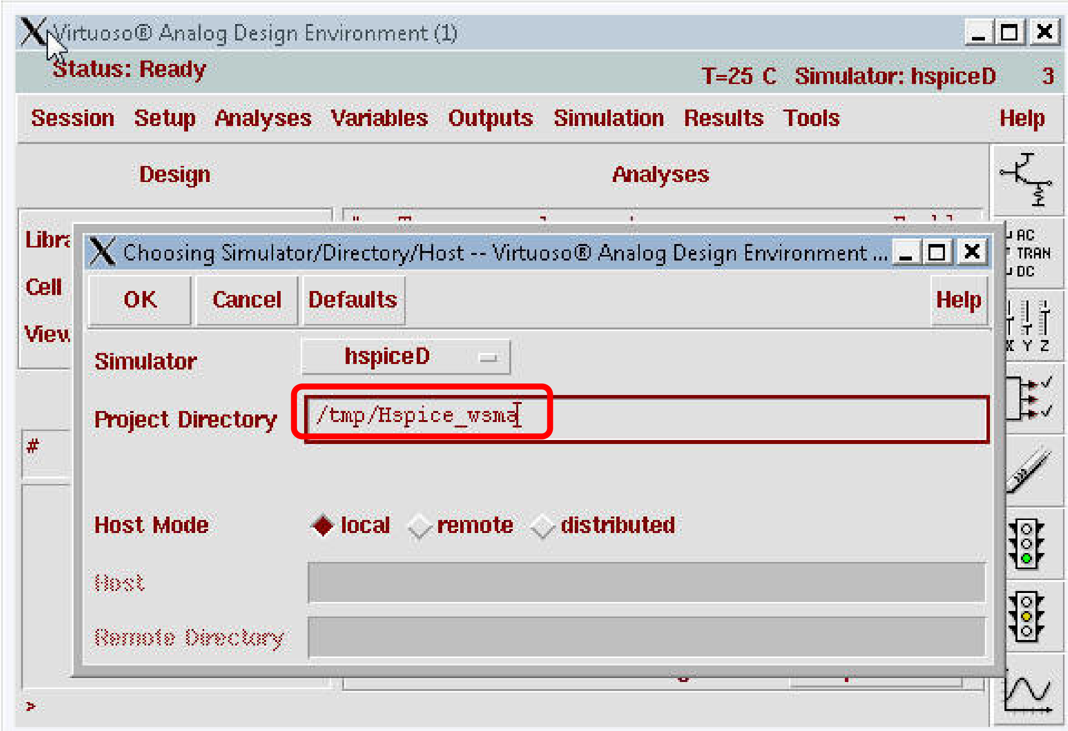

Fig. 6

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

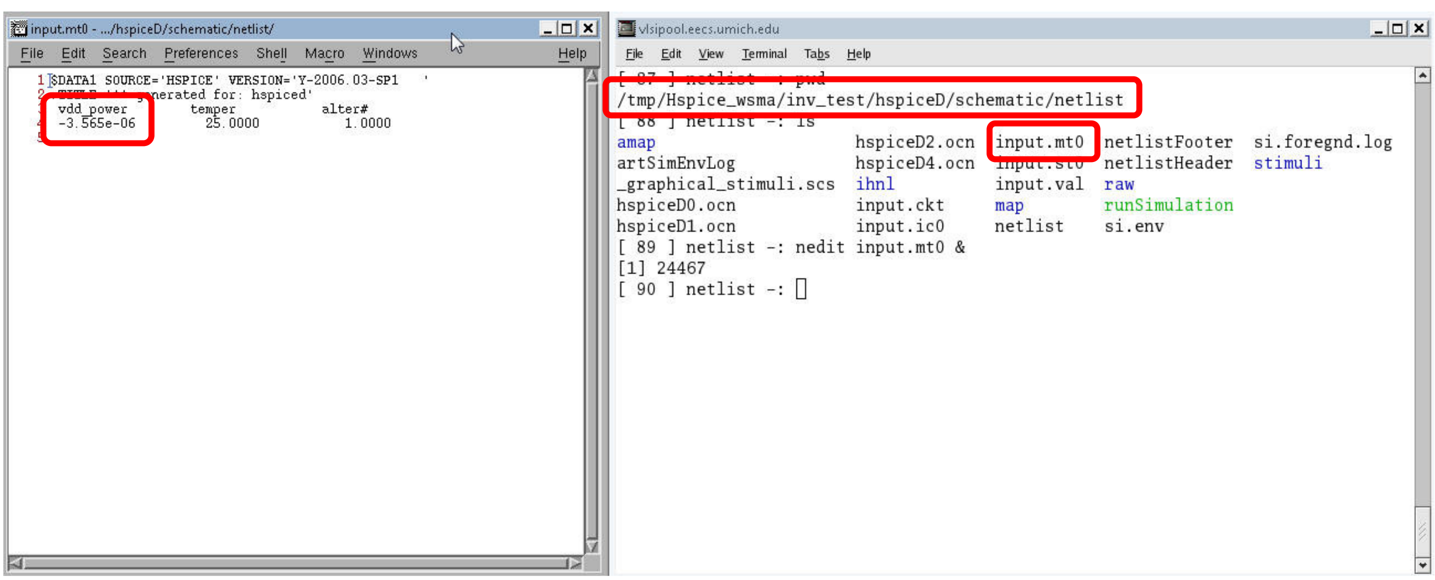

Fig. 7