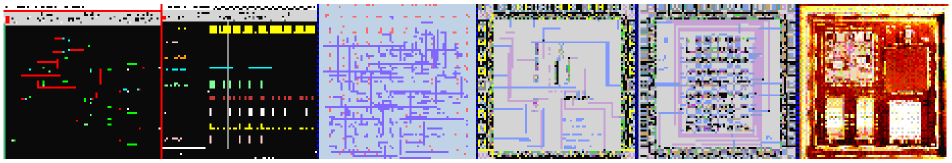
Homework_10 -- Using NanoSim

Revised 11/04/08 by D. Bouldin



Part A -- Delay Measurements

NanoSim is a fast Spice simulator which uses the same format as Hspice.

Login to ada3.eecs.utk.edu

Open your schematic from hw4-case-1:

Click on Tools --> Analog Environment

Select the Hspice simulator:

Click on Setup --> Simulator/Directory/Host and select hspiceS

Generate the netlist:

Click Simulation --> Netlist --> Create Final

Change directory to "simulation/adder4-noload/hspiceS/schematic/netlist"

Copy "hspiceFinal" to "adder4-noload.spi"

Edit "adder4-noload.spi" to include:

```
./usr/local/ncsu-cdk-1.5.1/models/hspice/standalone/ami06N.m
./usr/local/ncsu-cdk-1.5.1/models/hspice/standalone/ami06P.m
```

```
vCIN CIN 0 pwl(0n 0 3n 0 3.5n 2.5 4n 5 10n 5)
.tran 0 10n 0.05n
```

I copied them exactly, looks like .spi file are not case-sensitive!

```
Vdd  vdd! 0 dc=5
vgnd gnd! 0 dc=0
```

(My edited adder4-noload.spi is included in the tar file in Part B below.)

Create a NanoSim configuration file named "nanosim.cfg" with the following contents:

```
print_node_voltage *
print_node_logic *
```

I also added " print_node_current * "

Then, type:

```
nanosim -n adder4-noload.spi -c nanosim.cfg -out fsdb
```

must at the same dir of *.spi & *.cfg files

Synopsys CosmosScope can be used to view the waveforms produced by NanoSim.

Type:

```
cscope &
```

The following window will open:

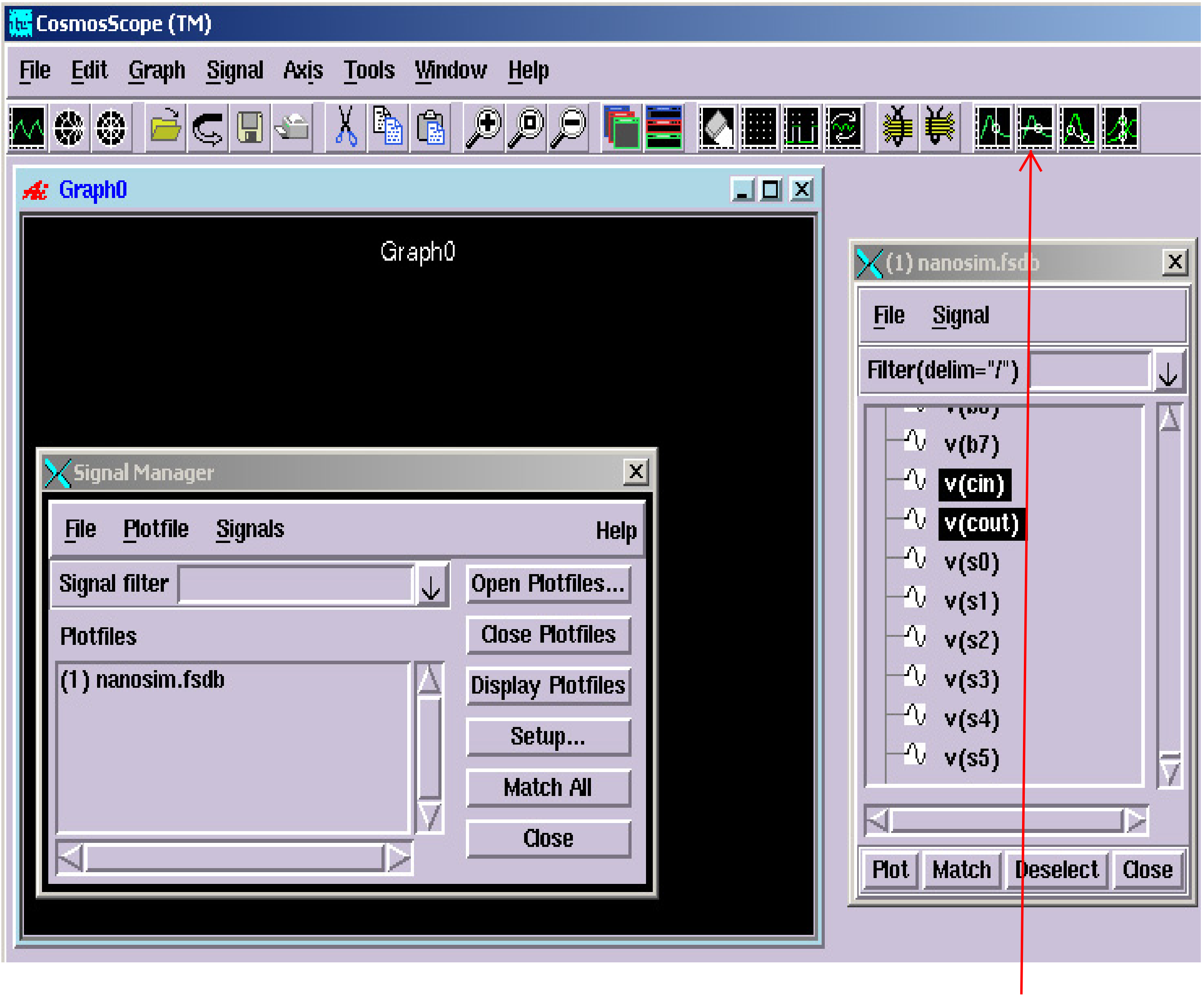
Next, select File --> Open --> Plotfiles

and choose nanosim.fsdb

need to "display all files" then nasosim.fsdb appears.

```
Next, open your file and select the relevant signals to display:
```

looks like NanoSim has both logical and analog waveforms for display!




```
Measure the delay from carry_in to carry_out by clicking the "At Y Measurement" icon
    and then select "Signal --> Measure Results" and capture the image.

In the graph below, v(1) is the carry_in of the adder while v(cin)
is the input to the first inverter.
```

Compare your hspice results with your spectre results.

Part B -- Power Measurements

Using the same methodology as in Part A, we will now explore the power measurement capabilities of NanoSim.

This tutorial was adapted from one provided by Synopsys in two formats: pdf and swf.

A similar tutorial is available at chiptalk.org.

```
cp ~bouldin/webhome/protected/651-hw10/651-hw10.tar .
```

```
tar -xvf 651-hw10.tar
```

```
cd 651-hw10
```

The file "setup.spi" contains an adder circuit and "adder.vec"
contains a series of digital input vectors to stimulate the circuit.
This is called the "switching activity".

notice uppercase "-C" not "-c" in Part A

The "run" file contains:

```
  nanosim -n setup.spi -nvec adder.vec -C cfg
```

Now, type:

```
./run
```

Note that two errors have been captured in the "nanosim.err" file which can be viewed by typing:

```
viewerror -i nanosim.err -o errors
```

"viewerror -i" is enough

and then:

```
vi errors
```

The power report is given in "nanosim.log".

(Also, see power.swf.)

To view the power report in a graphical format, type:

(Skip this since we do not have the license for nWave. Perhaps we can do this with cscope like the figure below.)

```
nWave &
```

indeed license problem!

and select File --> Open --> "adder.out" and from the "Signal" menu, select the signals of interest:

Part C -- Power Measurements on Your Circuit

Adapt the files in Part B to determine the power of your homework_4 circuits for adder4-noload and adder4-4loads.

Match the labels in your spi files to those in adder.vec

The homework_4 circuits had their a[0-3] and b[0-3] inputs tied to Vdd and Gnd. Change the node labels in the spi file to make these variable.

notice also for a[2], b[2], a[1], b[1], a[0],b[0]

Replace: XI0 VDD! 0 NET52 COUT SUM FAX1_G2

with: XI0 a[3] b[3] NET52 COUT SUM FAX1_G2

I also removed .tran command

Also, remove: vCIN CIN 0 pwl(0n 0 3n 0 3.5n 2.5 4n 5 10n 5)

Make other changes in your spi files as needed to match the labels in adder.vec

Post your results on your protected webpage.

===========

Note that power estimates can be made at multiple levels as described in the thesis (PDF and PPT) by Ashwin Balakrishnan.

NanoSim can also be used to simulate a mixture of analog and digital circuits for full-chip mixed-signal simulation: NS-VCS-MX.pdf

---

dbouldin@tennessee.edu

Since we can not use "nWave &" command due to license problem, we will adapt the input vector file and then use "cscope &" to display the variation of voltage and current.
modified *.spi file with a[3:0] and b[3:0] inputs,
predefined *.vec input file,
remember run nanosim at the same dir of these two files
I also copied the nanosim.cfg to the same dir.
I use the command for my 4bit_adder.spi

```
nanosim -n 4bit_adder.spi -nvec adder.vec -c nanosim.cfg -out fsdb
```

Part D -- do the same for the 4bit_adder_case3