HSPICE and CosmosScope Tutorial

Phyllis R. Nelson
Electrical and Computer Engineering Department
California State Polytechnic University, Pomona

Winter 2013

This tutorial provides a basic introduction to the use of HSPICE and CosmosScope for students in the Electrical and Computer Engineering Department at Cal Poly Pomona.

1 Linux server logon

Log onto a computer in one of the College of Engineering computer labs. Click

Start → All Programs → Xming → Xming

There will now be an `Xming` icon  in the lower right corner of the screen. (If you don't see it, click on the "Show hidden icons" icon.) **Right click** on this icon and select

EGR Linux hosts → `es-linux-02`

A new window will open. Log on using your Bronco Name (all lower case) and College of Engineering password.

2 Basic linux commands

The following table lists useful commands.

Command	Result
`pwd`	print working directory
`clear`	clear the terminal window
`ls`	list current directory contents
`ls -l`	`ls`, long format
`ls -S`	`ls`, sort by file size
`ls -t`	`ls`, sort by modification time
`cd <dir>`	change directory to dir
`cp <src> <dest>`	copy src to dest
`mv <src> <dest>`	move src to dest
`rm <file>`	remove a file
`man <cmd>`	view the system's manual page for cmd
`info <cmd>`	read system's info document for cmd

Check the man page for these commands to make sure that you know how to use them! Also, note that you don't have to type an entire directory or file name. Once you have started typing, the `tab` key causes the shell to auto-complete as much of the name is unique. You can continue to type one or more characters followed by `tab` until the name is complete. Note that in linux subdirectories are separated with "/".

Create a directory called "hspice" for your spice simulations, and make that directory your durrent directory.

```
mkdir hspice
cd hspice
```

3 Emacs editor

Open the emacs editor in a new window by typing `emacs &` (The "&" opens emacs in a new window.)

Copy the following file into the editor window and save it using the filename `nmos-ex.sp`.

```
nmos cs amp with diode-connected load

vdd   3  0  dc  5v
m1    3  3  2  2  n-fet
m2    2  1  0  0  n-fet
vi    1  0  dc  2v
```

```
.model n-fet nmos (level=1 kp=1m vto=1v)

.op
.dc vi 0v 5v 50mv
.option post

.end
```

The command `.option post` tells `hspice` to save the analysis results in appropriate output files (`*.sw0` for the DC sweep, `*.tr0` for a transient analysis, etc.).

Now click on the linux server window (black background) and type `ls` to list the files in your current working directory. You should see `nmos-ex.sp` in the list.

4 Running HSPICE

Run the simulation using the following command.

```
hspice nmos-ex.sp > nmos-ex.log
```

Once you receive the message that the hspice job has concluded, you will find four new files. They are

- nmos-ex.ic0: the node voltages - result of the `.op` command
- nmos-ex.log: a file that contains information on device models, power consumed, and much more. You can page through this file by typing

  ```
  cat nmos-ex.log | more
  ```

  Use the space bar to page through the file and "q" to quit.
- nmos-ex.st0: a log of the simulation process
- nmos-ex.sw0: the data from the dc sweep

5 Displaying results with CosmosScope

Start CosmosScope with the command

```
cscope &
```

Click

File → Open → Plotfiles

and choose `nmos-ex.sw0`. Two more windows will open, the signal manager and the plot file window. Plot the out voltage (the voltage at node 2) by double-clicking $v(2)$ in the plot file window. The default x axis variable is the input voltage because that is the signal that was swept by the `.dc` command. To demonstrate that the horizontal axis is the input voltage, click on $v(1)$ in the plot file window, right-clicking on the graph, and choosing "Plot."

6 Ending your session

Close cscope and emacs. Then type `exit` on the linux command line.