

Cheat Sheet on using Electric for Design and Simulations

By

Sai Kashyap Nutulapati

Revised - 04 October 2010

Instructions before Starting

- Wherever you see the word <username>, replace it with your CAT account Username.
- All the commands will be in italics. eg:
mkdir models
- Log on to Asynchronous Research Center's Website @ <http://arc.cecs.pdx.edu/>

Downloading Electric on your machine

- Electric can be used to build designs on Windows, Linux and Mac OS which are later converted into a netlist file by Electric.
- The netlist can be used for simulating designs with the Hspice Simulator. Hspice is licensed to run on server “nemo”.
- This tutorial will explain the procedure to simulate designs using Hspice on nemo.
- First of all download Electric by logging on to: <http://arc.cecs.pdx.edu/teaching> and by clicking on the download link: [electricBinary8.11.jar](#) given on the webpage.
- The next 3 slides guide you in setting up the nemo from Windows, Linux and Mac machines respectively.

Downloading software packages for Windows

- Before you can start building designs on Electric, you have to download and install the following software packages:
 1. Java Runtime Environment 6
 2. Putty
 3. WinScp
- Download Java Runtime Environment : select the online java version at <http://www.java.com/en/download/manual.jsp#win>
- Download WinScp Client at <http://sourceforge.net/projects/winscp/files/WinSCP/4.2.8/winscp428setup.exe/download>
- Download Putty on to your windows machine at <http://www.chiark.greenend.org.uk/~sgtatham/putty/download.html>
- Install all these packages.

Setting up nemo from Windows

- After downloading and installing the software packages, you need to setup Hspice on nemo.
- Putty is used for remote login to nemo.
- Open Putty and type in <username>@nemo.ece.pdx.edu in the Host Name slot and click open.
- Type in your nemo server password.
- You will be logged on to the nemo server from your windows machine.

Using WinScp software

- WinScp can be used to transfer files from your Windows machine to nemo.
- Open WinScp and type the following in the slots:
 - Hostname: nemo.ece.pdx.edu
 - Username: your CAT account username
 - Password: your CAT account password
- Save and login. You will see two windows; files on the left are on your windows machine, the files on the right are on nemo.
- In order to transfer files from windows to nemo or vice versa, you can just drag and drop the files into the directory you want.

Setting up nemo for Linux

- If you use Electric on a linux machine, you need java on linux.
- Download JRE 6 package at <http://www.java.com/en/download/manual.jsp#win> and select the suitable version.
- After you are done with JRE download and install, open a terminal and type:
- `ssh -X nemo.ece.pdx.edu` and then type in your nemo server password.
- You will be logged on to the nemo server from your Linux machine.

Setting up nemo for Mac

- If you use Electric on a Mac machine, you need to download the Java version supplied by Apple and available at: http://support.apple.com/kb/HT1338?viewlocale=en_US
- After you are done with downloading and installing Java, open a terminal and type:
- `ssh -X <username>@nemo.ece.pdx.edu` and then type in your nemo server password.
- You will be logged on to the nemo server from your Mac machine.

Adding Hspice and Java Packages on nemo

- Java version 6 is required to run Electric on nemo, and so is hspice.
- Add hspice and Java_6 packages
 - I. use the command: *addpkg* and hit return.
- Select hspice and Java_6 from the list that *addpkg* offers. Log out and Log back in for these changes to take effect.
- After logging on to nemo , create a work directory “YOU” for your Electric and hspice simulations below your username:
 - “YOU” = /u/<username>/(wherever you want to create a folder)
 - e.g. “YOU” can be: /u/<username>/electricsim/
- Use the following commands to create the following directories under your work directory “YOU”:
 - I. *mkdir models*
 - II. *mkdir simulations*

Download necessary files for Electric

- All the files that you need for running the simulations are on the website at <http://arc.cecs.pdx.edu/fall10>
- Here are the list of files you need:
 1. MOSIS_180nm_header.hsp
 2. MOSIS_180nm_trailer.hsp
 3. models.sp
- Download the files mentioned above and save them under “YOU” on your nemo machine.
- Download the library “a180Library-20sep10.zip” on your machine, unzip it and transfer the resulting directory to nemo directory “YOU”.
- Windows users can use WinScp to transfer the file. Linux and Mac users can use the scp protocol to transfer the file.

Edit the MOSIS_180nm_header.hsp

- In the .INCLUDE statement replace the sub-path /u/kashyap/models by sub-path YOU/models/.

```
*****
* Header File for Simulation with 180nm MOSIS Transistor Models
*
* Header Written by William Koven, 26 June 2009
*****

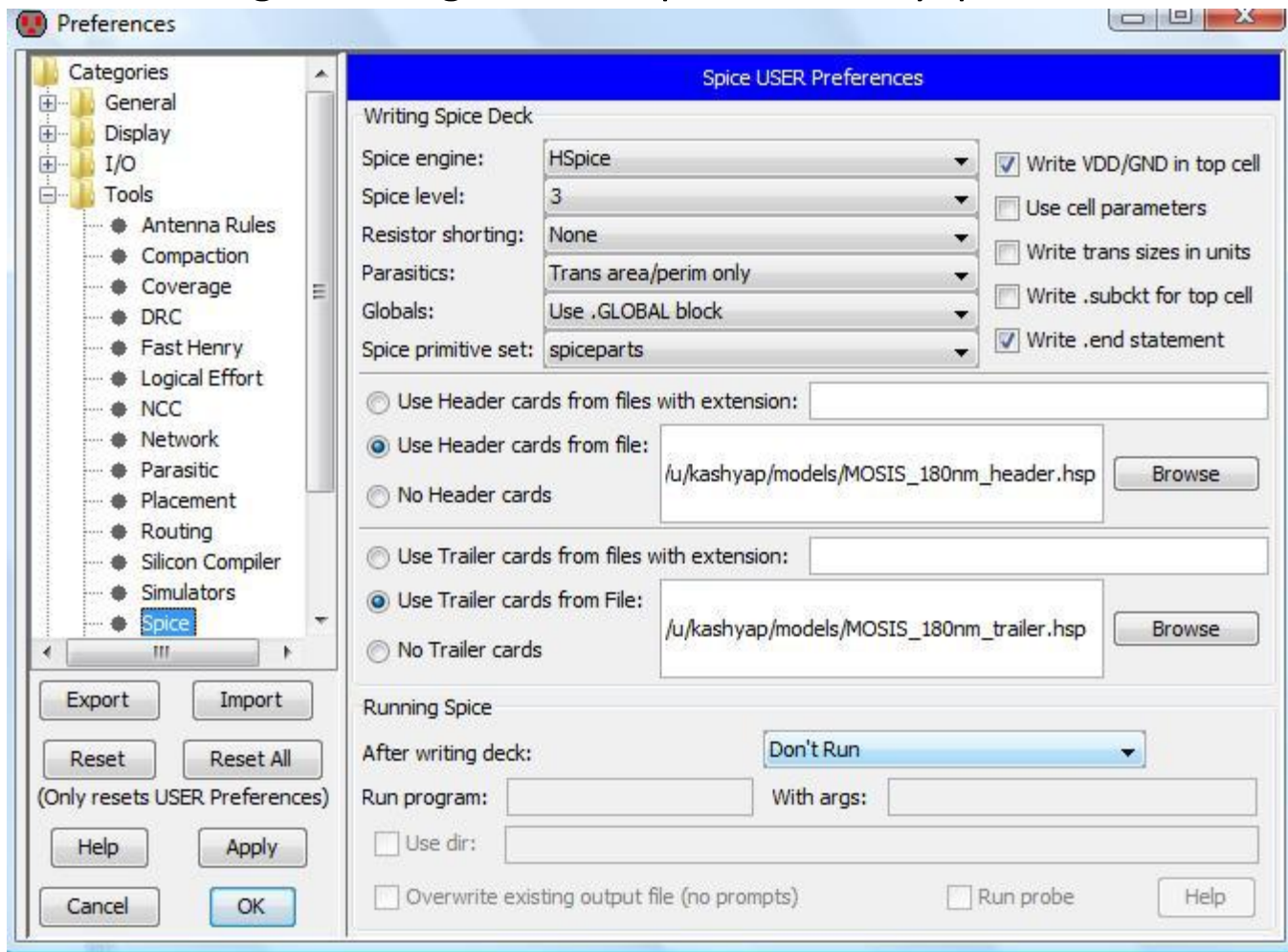
*****
* Transistor models
* Currently location of models is assumed to be /u/ivans/models.sp
* on the PSU ECE nemo server.
* EDIT THE PATH IN THE .INCLUDE STATEMENT
*****
.INCLUDE /u/kashyap/models/models.sp

*****
* Options and Parameters
*****
.OPTION POST
.OPTION SCALE=90n
.PARAM ABN=0
.PARAM ABP=0
.PARAM AVTOP=0
.PARAM AVTON=0
.PARAM hdifn=0
.PARAM hdifp=0
.PARAM SUPPLY=1.8V

*****
* Netlist
*****
```

Change Electric preferences

- Open Electric and go to File → Preferences. Then Click on Tools → Spice and make sure the simulator preferences look like this and then click apply and OK to save the changes. Change the sub-path /u/kashyap/models to YOU/models.



Example –Ring Oscillator

- Download Class1_ringOsc-01oct10.jelib file from <http://arc.cecs.pdx.edu/fall10/class1> and save it on your machine.
- Now open Electric and open the library a180Library-20sep10 you have downloaded earlier. In the a180Library-20sep10 library select the sub-library “ anOpener”. A schematic window opens.
- Now open Class1_ringOsc-01oct10.jelib by clicking File → Open Library → Class1_ringOsc-01oct10. If you open Class1_ringOsc-01oct10 library without opening a180Library-20sep10 , you will have errors in Electric.
- Continued on next slide.....

Example –Ring Oscillator Cont. (1)

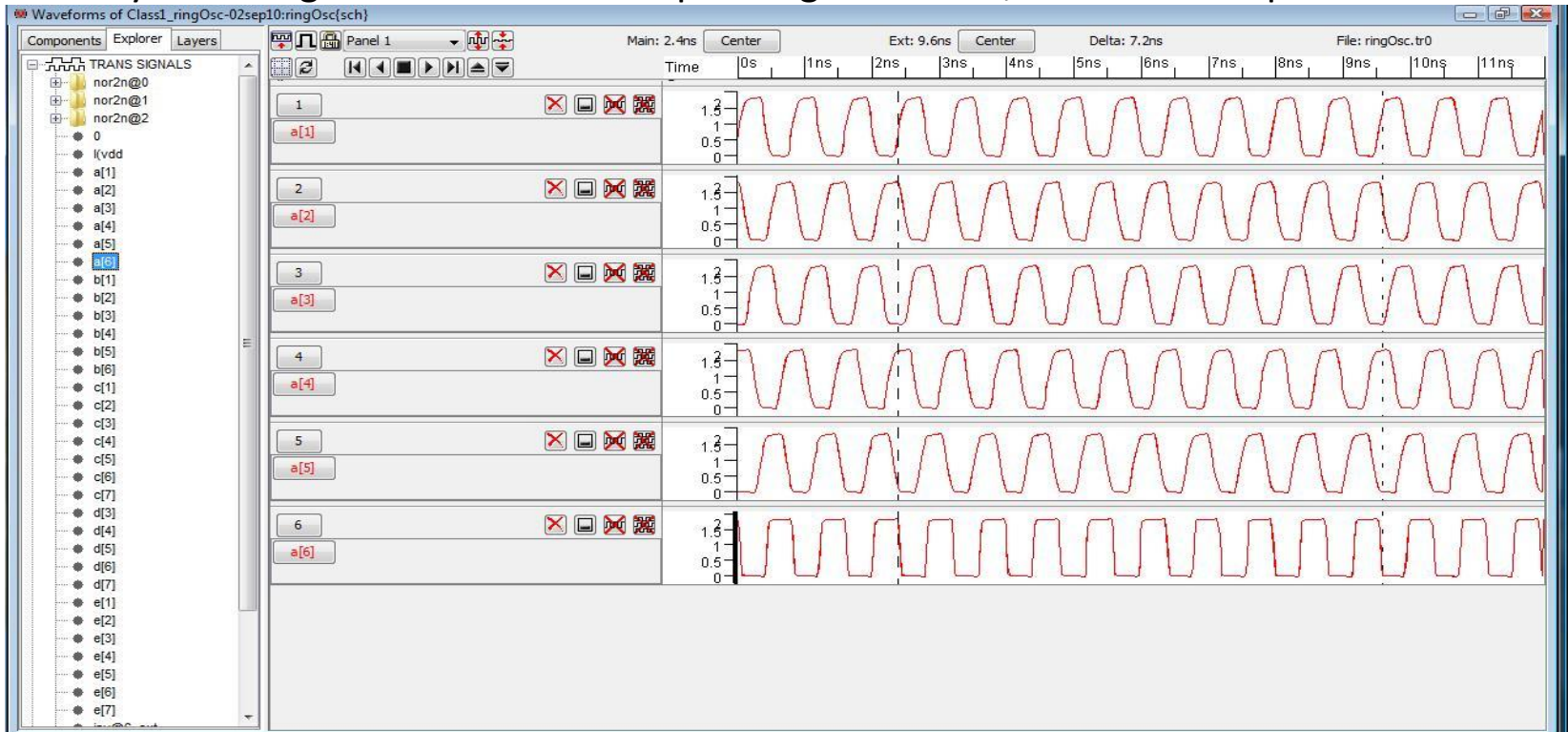
- The left side now shows the name Class1_ringOsc-01oct10 with three tabs underneath it: Components, Explorer, Layers. In tab Explorer, select ringOsc{sch} under libraries/class1_ringOsc_01Oct10
- Generate the spice netlist for this schematic, by clicking Tools → Simulation (Spice) → Write Spice Deck and write the result file ringOsc.spi on your machine.
- Transfer ringOsc.spi from your machine to your simulation directory “YOU”/simulations on nemo by using WinScp (windows) or the scp protocol (linux or Mac users).
- Continued on next slide

Example –Ring Oscillator Cont. (2)

- You should now have a file called ringOsc.spi in your directory “YOU”/simulations on nemo.
- Run Hspice, as follows:
 - *cd “YOU”/simulations*
 - *hspice64 ringOsc.spi*
- If you see “hspice job aborted”, then fix the errors.
- Once the spice simulation is error-free, you should a message “hspice job concluded”.
- If there are no errors you will see a file with extension .tr0 in “YOU”/simulations. We will use this file to plot waveforms.
- If you don’t run Electric on nemo, then transfer ringOsc.tr0 to the machine where you run Electric using WinScp (windows) or the scp protocol (Linux and Mac users).

Example –Ring Oscillator Cont.(3)

- In Electric, click:
Tools → Simulation (Spice) → Plot Simulation Output (guess file)
- You get a new window with simulation data. From tab Explorer in the new window, open the drop-down menu for TRANS SIGNALS by clicking on its “+” symbol.
- You will now see a set of signal names from the design schematics. You can double click on any of the signals to see its corresponding waveform, similar to the picture below.



Manual on using Electric

- If you want to explore Electric and learn all its features and start using them, refer to the manual available at this link
<http://arc.cecs.pdx.edu/fall10>
- The manual can also be found at
<http://www.staticfreesoft.com/jmanual/ElectricManual-8.11.pdf>