

How to Use Cadence and HSpice on PSU CSE

1. Add the following lines to your ~/.cshrc

```
#####  
## Cadence  
setenv CADENCE_HOME      /import/software/cadence-2008  
setenv CDS                $CADENCE_HOME  
setenv CLS_CSDS_COMPATIBILITY_LOCKING NO  
setenv DD_DONT_DO_OS_LOCKS set  
setenv LM_LICENSE_FILE 1700@neuromancer  
setenv CDS_LIC_FILE 1700@neuromancer  
setenv CDS_SITE /home/chip/chip/cadence/local  
setenv USE_NCSU_CDK /home/mdl/euk139/cadence_2008/  
setenv CDS_Netlisting_Mode Analog  
setenv SKIP_CDS_DIALOG  
  
set path = (      $path  
                $CDS/rc/bin  
                $CDS/ldv/tools/bin  
                $CDS/soc71/tools/bin  
                $CDS/ldv/tools/nclaunch/bin  
                $CDS/dsmse/tools/bin  
                $CDS/dsmse/tools/dsm/bin  
                $CDS/ic/tools/bin  
                $CDS/ic/tools/dfII/bin  
                $CDS/icc/tools/bin  
                $CDS/psd/tools/bin  
                )  
  
#####  
## Synopsys  
setenv PATH /home/software/sparc/SunOS/software/synopsys-  
2008/hspice/hspice/bin:${PATH}  
  
#####  
## FreePDK  
#setenv CDSHOME /home/mdl/euk139/newcadence/  
setenv PDK_DIR /home/mdl/euk139/FreePDK45/
```

2. Go to http://www.eda.ncsu.edu/wiki/NCSU_EDA_Wiki and download FreePDK45 (ex: NCSU-FreePDK45-1.2.tar.gz) You need to enter your name and institution, etc, and a link will be e-mailed to you for download. Put that file into your CSE account (use sftp for instance).

3. Extract NCSU-FreePDK45-1.2.tar.gz at /home/mdl/euk139/
It should create a folder named FreePDK45.

4. Create a folder `/home/mdl/euk139/cadence_2008/`
5. `goto /home/mdl/euk139/FreePDK45/ncsu-basekit/cdssetup/`
copy the `setup.csh` to `/home/mdl/euk139/cadence_2008`
6. `goto /home/mdl/euk139/cadence_2008`
7. edit `setup.csh`, change these 2 lines:
 `setenv PDK_DIR /home/mdl/euk139/FreePDK45`
 `setenv CDSHOME /import/software/cadence/ic`
8. run
 `chmod 700 setup.csh`
 `./setup.csh`
9. `goto /home/mdl/euk139/cadence_2008/`
10. type “virtuoso” to run Cadence
 if you encounter some warning saying something like “should I get a higher tiered license?”,
 click on “Always”.
11. When you are inserting a transistor, insert it from the `NCSU_Devices_FreePDK45` library
12. To simulate, do “Launch->ADE L” (analog design environment)
13. Click “Setup->Simulator/Directory/Host”, and select `HSpiceD` as the simulator.
14. Click “Setup->Model Libraries” and insert the following library:
 “`/home/mdl/euk139/FreePDK45/ncsu_basekit/models/hspice/hspice_nom.include`”
15. Select simulation type, duration, outputs to be plotted, etc. and click on “Netlist and Run”
 output should be plotted.

Good Luck!
Emre Kultursay