

How to Start with Cadence ®  
ECE721 Project

Instructor: Prof. Steve Bibyk  
Prepared by: Golsa Ghiaasi Hafezi  
Spring 2005  
Ohio State University

Follow the instruction for installing AMI05 process posted on the class web site under CAD Design Project section. After you finished that procedure and while you are inside `~/cadence/NCSU`, run `icfb`. The window in Figure 1 pops up. Go to the “Tools” and click on “Library Manager”. You will see the window shown in Figure 2, which is called “Library Manager”. Note that you may not see some of the libraries that I have and are shown in this figure. Go to the File/New/Library and you will see the window in Figure 3. Inside Library Name, type “EE721Project1” and click OK.

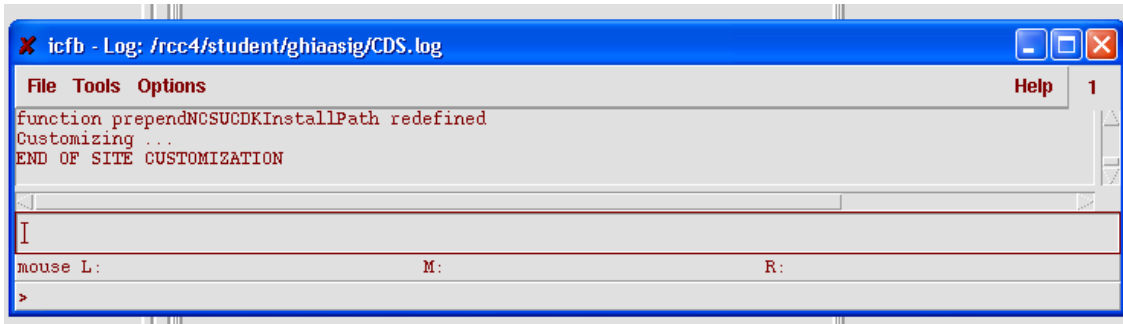


Figure 1

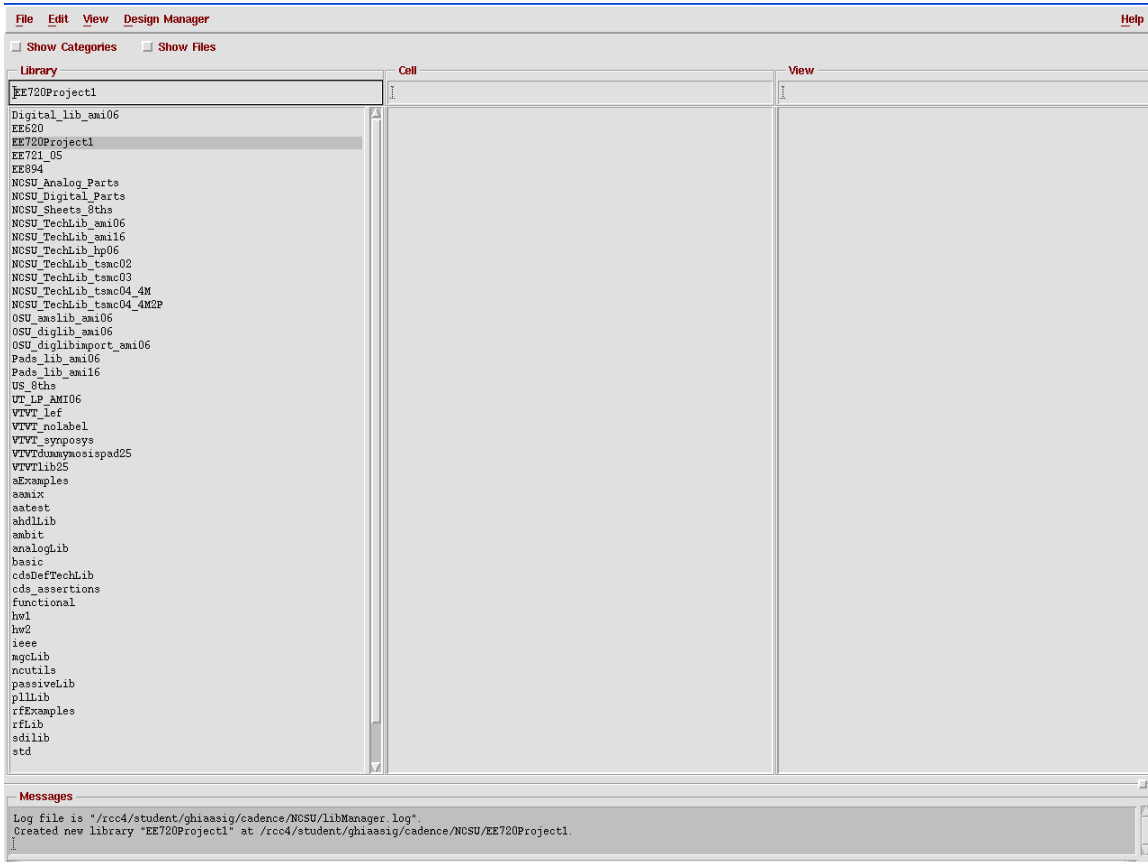


Figure 2

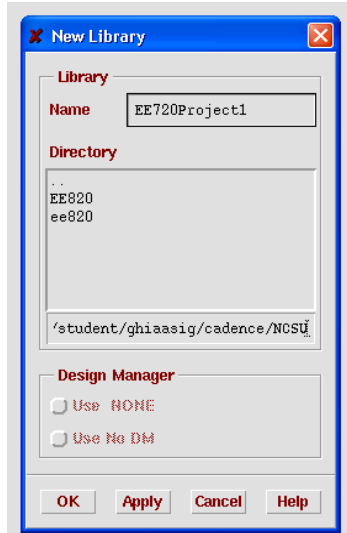


Figure 3

Then it asks you for technology file, choose the second option which is “attach to an existing tech file” and click OK( Figure 4.a). The next window asks you for the Technology Library choose “NCSU\_TechLib\_ami06”among the options and then OK (Figure 4.b).

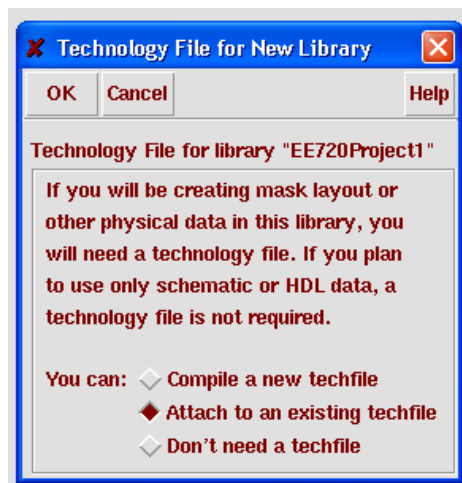


Figure 4.a



Figure 4.b

Now you should be able to see a Library named “EE721project1” inside your Library Manager window. Select the library EE721Project1 in the Library Manger window. Go to /File/New/Cellview. The window in Figure 5 pops up. Call the cell name “Not” and choose the View Name to be “schematic”. Then click OK.

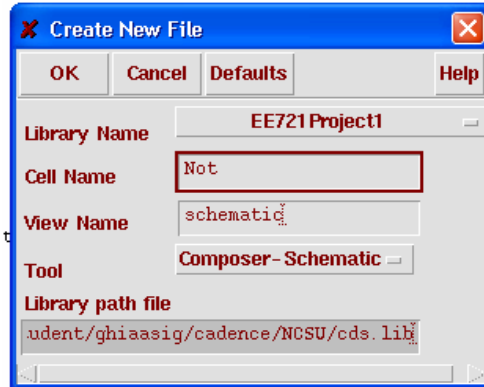


Figure 5

In Library Manger window, double click on Not. The Virtuoso Schematic Editing window will open. You can start designing your circuit now. Go to ADD/Instance. For the Library, choose “NCSU\_Analog\_Parts”. In this project you are only using the components from Library. As it is shown in Figure 6 you can change the parameters of the components, such as size of transistors in the ADD Instance window.

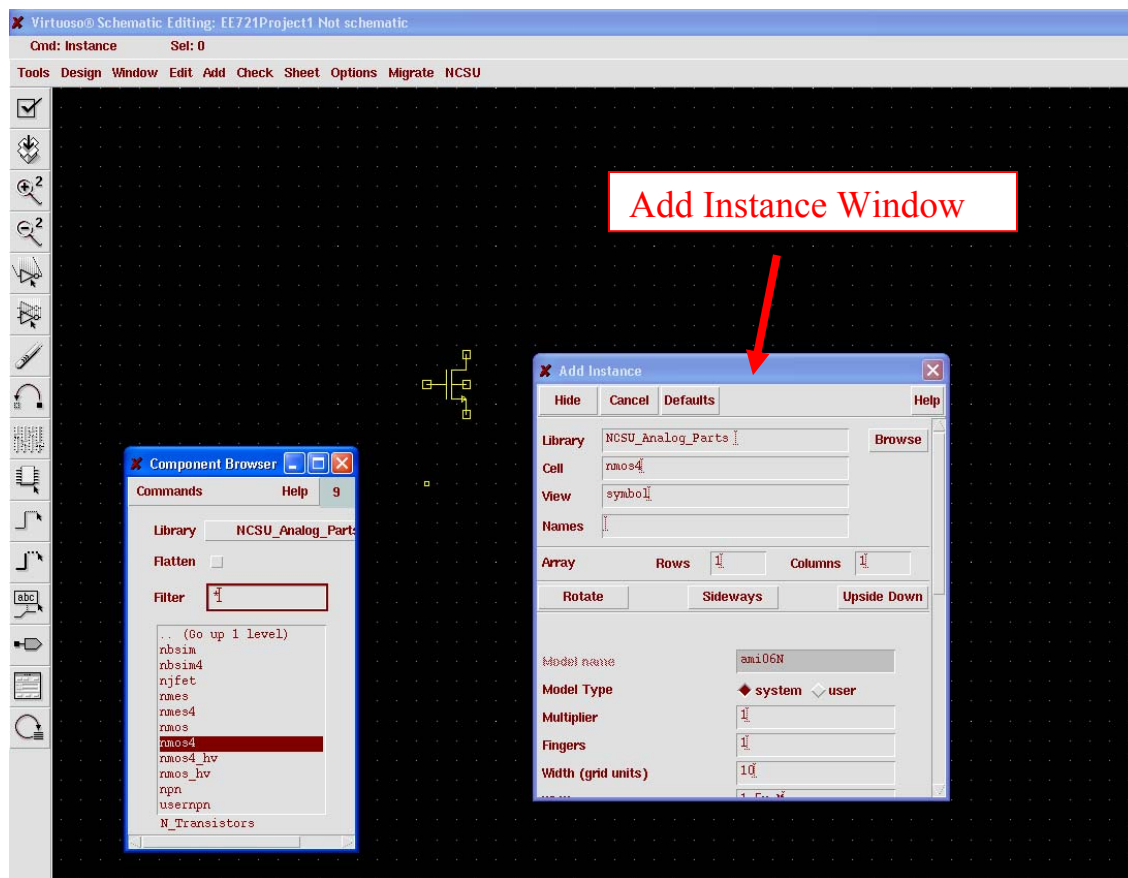


Figure 6

Note: For this project, you should use nmos4 and pmos4 components for NMOS and PMOS type transistors.

You can change the parameters of the components at any time, by selecting the component(s) and choosing /Edit/Properties. Connect the components using the wires from “Add/Wire” menu. After the circuit is done, click on “Save and Check”. Make sure that there is no error and warning before proceeding further.

To simulate the circuit, you need to run the “Analog Artist” tool by clicking on “File/Analog Environment “. The window in Figure 7 pops up.

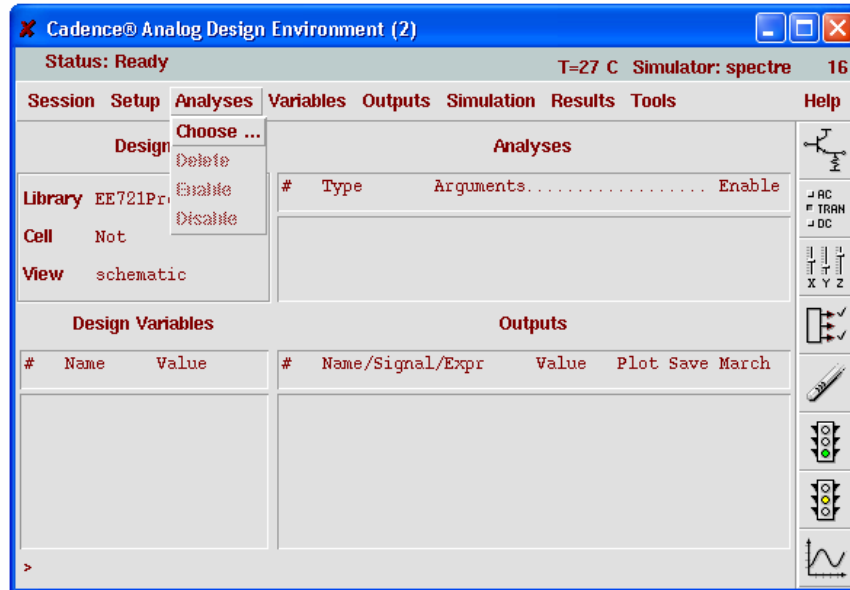


Figure 7

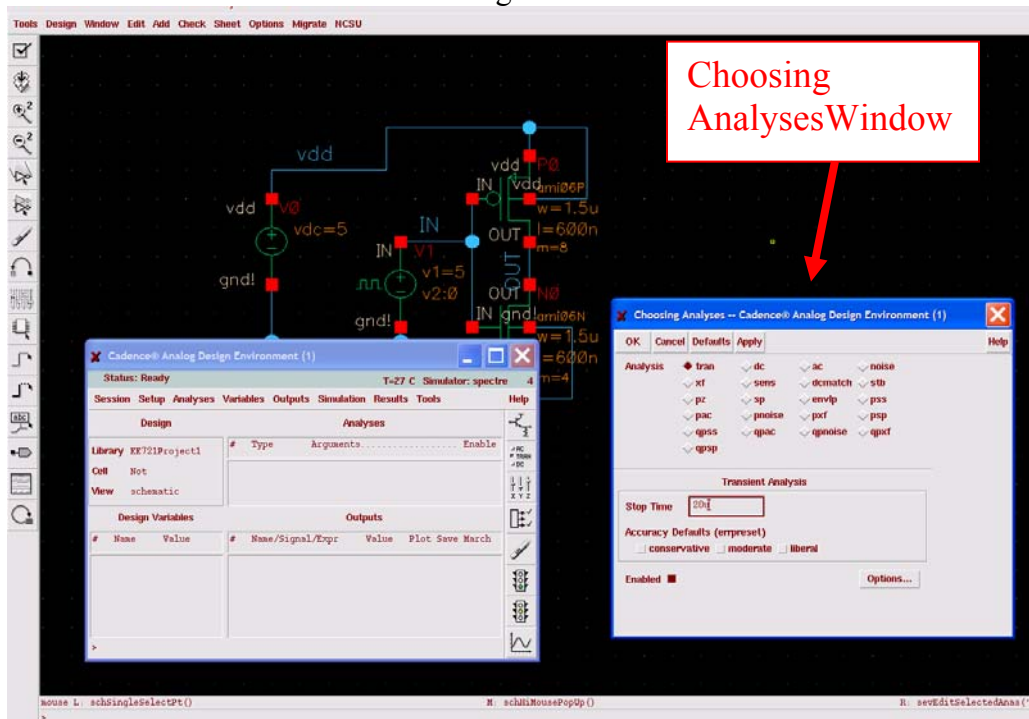


Figure 8

To set up analysis type, Click on Analyses/Choose. The Choosing Analyses window will pop up (Figure 8). Select the analysis to “tran” and specify the simulation time in the “stop time” field. Click OK.

Right now, you are ready to run the simulation. On the Analog Artist window, click on Simulation/Netlist and Run. The Specter log file will show up. Wait till the simulation is complete.

```

File                               Help 5
ic = all
skipdc = no
reltol = 1e-03
abstol(I) = 1 pA
abstol(V) = 1 uV
temp = 27 C
trno = 27 C
tempeffects = all
expreset = moderate
method = trapzonly
lteratio = 3.5
relref = sigglobal
cmin = 0 F
gmin = 1 pS

tran: time = 716.1 ns (3.58 %), step = 306.9 ns (1.53 %)
tran: time = 1.516 us (7.58 %), step = 400 ns (2 %)
tran: time = 2.716 us (13.6 %), step = 400 ns (2 %)
tran: time = 3.516 us (17.6 %), step = 400 ns (2 %)
tran: time = 4.716 us (23.6 %), step = 400 ns (2 %)
tran: time = 5.507 us (27.5 %), step = 202 ns (1.01 %)
tran: time = 6.707 us (33.5 %), step = 400 ns (2 %)
tran: time = 7.507 us (37.5 %), step = 400 ns (2 %)
tran: time = 8.707 us (43.5 %), step = 400 ns (2 %)
tran: time = 9.507 us (47.5 %), step = 400 ns (2 %)
tran: time = 10.73 us (53.7 %), step = 315.5 ns (1.58 %)
tran: time = 11.53 us (57.7 %), step = 400 ns (2 %)
tran: time = 12.73 us (63.7 %), step = 400 ns (2 %)
tran: time = 13.53 us (67.7 %), step = 400 ns (2 %)
tran: time = 14.67 us (73.3 %), step = 333.3 ns (1.67 %)
tran: time = 15.58 us (77.9 %), step = 237.9 ns (1.19 %)
tran: time = 16.78 us (83.9 %), step = 400 ns (2 %)
tran: time = 17.58 us (87.9 %), step = 400 ns (2 %)
tran: time = 18.78 us (93.9 %), step = 400 ns (2 %)
tran: time = 19.58 us (97.9 %), step = 400 ns (2 %)
Number of accepted tran steps = 191.
Initial condition solution time = 10 ms.
Intrinsic tran analysis time = 20 ms.
Total time required for tran analysis 'tran' was 30 ms.

finalTimeOP: writing operating point information to rawfile.
modelParameter: writing model parameter values to rawfile.
element: writing instance parameter values to rawfile.
outputParameter: writing output parameter values to rawfile.
designParamVals: writing netlist parameters to rawfile.

```

Figure 9

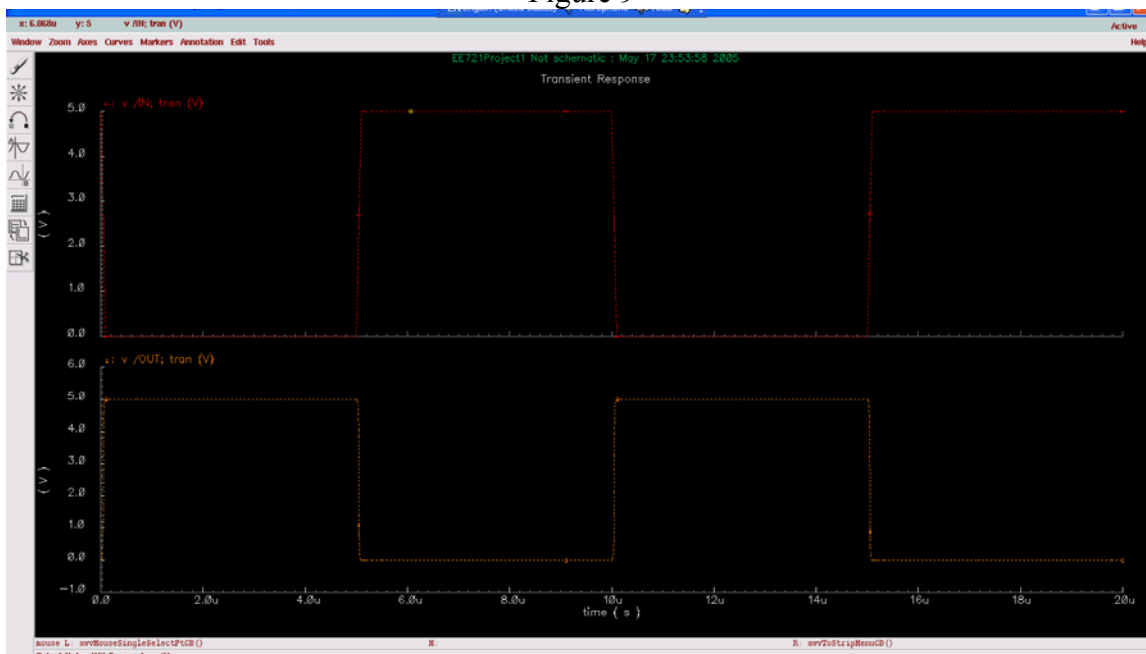


Figure 10

In order to plot the results, choose Results/Direct Plot/Main Form on Analog Artist window. Select the node voltages you wish to plot by clicking on the Virtuoso Schematic window. The Plot window will show up.( Figure 10)

You can use the markers by choosing Markers in the Plot window to analyze the plot.

\*\*\*For the purpose of project, we like to be able to include the circuit views and graphs in the body of a report using MSWord or Latex.

1. To capture the simulation view in cadence, go to Virtuoso Schematic Editing. Click on Design/Plot/Submit. The Submit Plot window (Figure 11.a) will pop up. To change the settings of the plot, click on Plot Options at the bottom right corner of the window. The Plot Option window (Figure 11.b) will open. Change the “Display Type” to “psb”. In order to include it in the text processor, choose the Plotter to be EPS. In this case, choose “Send Plot Only to File” and specify the file name.



Figure 11.a

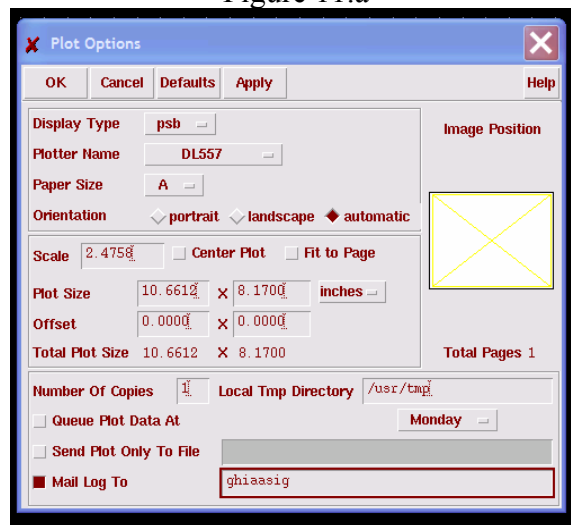


Figure 11.b

1. For the graphs plotted in the Waveform window, click on Window/Hardcopy. The Hardcopy window (Figure 12) shows up. Again, make sure that Display Type is “psb”. To write as a EPS file, select the Plotter to be EPS. Send Plot Only to File” and specify the file name.

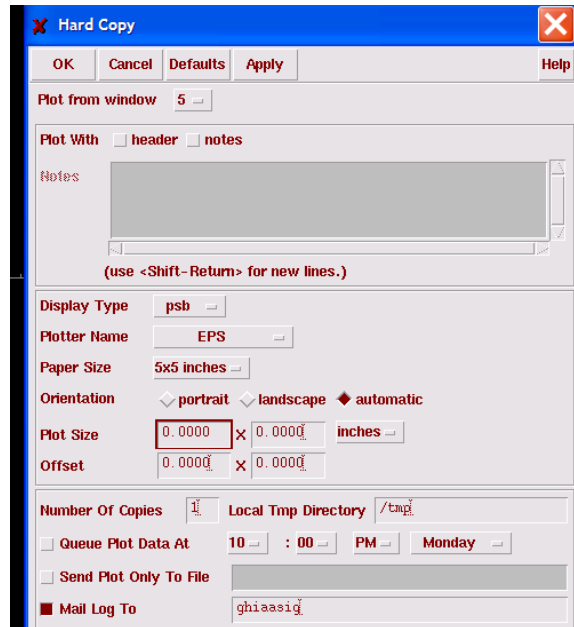


Figure 12

Good Luck!