

PROCEDURE FOR CREATING THE SCHEMATIC SIMULATION

I. Commands to get into Cadence

- Right Click and open the terminal window
- Type the following commands as follows and press enter.
 - `cs`
 - `source /home/install/cshrc`
 - `virtuoso &`

II. Procedure for Schematic simulation using Cadence

- Now two windows must open i) virtuoso/command interpreter window ii) "Whats New..."
- Close the 2nd window
- Use 1st window i.e virtuoso window (CIW) for further processing.
 - Create a New Library
 - Create Schematic Cell view.
 - Create the Symbol for schematic Cell view.
 - Create the test Cell view.
 - Analog simulation by spectre
- Procedure for Creating New Library.
 - File –New – Library
 - Name : Give name for ur library Ex: VLSILAB
 - Enable Attach to an existing technology library, Click OK
 - Attach the library to the technology library gpdk180. Click OK
- Create Schematic Cell view.
 - Go to 1st window i.e virtuoso (CIW)
 - File-New-Cell view
 - Setup the new file form
- Library: Select the one you a created.
- Cell : Give the experiment name Ex: Inverter View: Schematic
- Type: Schematic press OK
 - Add the required components from the libraries and make the connections.
 - Go to instance fixed menu or use shortcut key "I" from keypad to go instances
 - Click on browse. This opens the library browser
- Now select the appropriate library for components like
 - gpdk180.....nmos, pmos
 - Analog library.....Vdd, Gnd, Vcc, Vpulse, Vsin
 - Make the connections by using fixed narrow wire key
 - Click Check and Save button

- Creating the Symbol for schematic Cell view
 - In the schematic window, execute
 - Create – Cell view – From Cell view
 - The cell view from cell view window appears
 - Check Lib Name, Cell Name, From View name must be schematic Press ok
 - Now Symbol generation form appears. Click Ok If No changes required
 - A new window with with default symbol is created.
 - Edit the symbol if you want to give actual symbol shape else continue.
 - Execute Create-Cell view-from cell view
 - Library Name and Cell Name must be same which you have used for schematic. Press OK
 - Check for the position of pin side.Prss OK
 - Edit for the shape by Create-Shape-Choose required options to edit.
- Creating the new test cell view
 - Go to CIW window, Execute File-New-Cell view
 - Setup the new file form
 - Library: Select the one you a created.
 - Cell: Cell name must be different from the name used in schematic cell view. Ex: Inverter_test
 - View: Schematic
 - Type: Schematic press OK
 - Make the required connections
- Analog simulation by SPECTRE.
 - In test cell view window Launch – ADE L(Analog Design Environment)
 - Execute Setup—Simulation/directory/Host A new window opens
 - Set the simulation window to spectre and click ok
 - Execute Setup-Model Library. Anew window opens, Check of gpdk.scs as lib and section type as stat then press OK.
 - Execute Analysis – Choose. A window opens.
 - Select the type and set the specifications and press OK
 - Execute Output s—to be plotted – Select on Schematic
 - Then Select the INPUT WIRE(Vin) and OUTPUT WIRE(Vout) from your test Schematic using mouse

Execute Simulation -- Net list and Run