SCL 0.18µm (Analog) PDK

User Guide

Advanced VLSI Design Lab

IIT Kharagpur

Revision History

Rev.	Author	Last updated on
1.0	Samiran Dam	6th Nov., 2015

1. Basic Setup

1.1. Setup your working directory

Step 1: Copy the working directory from the following path into your home directory -

Command:

cp ~design_installer/scl/scl180_analog.zip .

Step 2: Unzip the **scl180_analog.zip** and enter into **scl180_analog** directory (it is the working directory <**WORKING_DIR**>).

1.2. Working directory structure

<**WORKING_DIR**>
- **cdl_netlists**
  - *scale.cdl*
- **DEMO**
- **Docs**
- **physical_verification**
  - **drc**
    - **rsf**
      - *drc.rsf*
      - *DRC.header*
    - **temp**
  - **antenna**
    - **rsf**
      - *antenna.rsf*
      - *ANTENNA.header*
    - **temp**
  - **lvs**
    - **rsf**
      - *lvs.rsf*
      - *LVS.header*
    - **temp**
  - **pex**
    - **rsf**
      - *pex.rsf*
      - *RCX_4LM.header*
    - **temp**
- **simulation**
  - **spice_files**
    - *inverter_testcase.sp*
  - **results**
- *cds.lib*
- *.cdsinit*
- *.bashrc*
- *user_guide.pdf*

Description of the above items are as follows.

- **cdl_netlist** directory contains the cdl netlists which will be required for LVS. It also contains *scale.cdl* file which is used during LVS and PEX run.
- **DEMO** is a a user design library. It is used to demonstrate various steps throughout this document.
- **Docs** contains two documents which are -
  DR2_0018SL_SCL_Rev1.pdf >> Design Rules for TS18 Standard Logic Process for SCL
  DRS2_0018SL_SCL_manual.pdf >> SPICE Models for TS18 Standard Logic Process
- **physical_verification** contains four separate directories for DRC, antenna, LVS and PEX run using **Calibre**.
- **simulation** contains two directories – **spice_files** and **results**. **spice_files** contains one sample spice netlist for a CMOS inverter circuit (which is inside the DEMO library). This spice netlist (*inverter_testcase.sp*) is used to run the post-layout simulation using Synopsis's **HSPICE** simulator. **results** directory is used to store the simulation output files.

1.3. Change the paths

Replace the string <WORKING_DIR> with the actual path in the following files *antenna.rsf*, *drc.rsf*, *lvs.rsf* and *pex.rsf.*

1.4. About “cds.lib”

cds.lib file contains

- SCL 0.18um PDK library (**ts018_scl_prim**)
- 1.8V I/O library (**CIO150**)
- 3.3V I/O library (**CIO250**)
- HSPICE compatible analogLib (**analogLib_hspice**)

2. Schematic simulation

2.1. Launch cadence

Step-1: Open a terminal window in <WORKING_DIR>.

Step-2: run *.bashrc* [command >> **source .bashrc**]

Step-3: run virtuoso [command >> **virtuoso**]

2.2. Simulate in ADE-L

Step-1: Open **Tools > Library Manager**. Go to the **DEMO** library and open **testinv_testbench** schematic.

Step-2: Launch **ADE-L**.

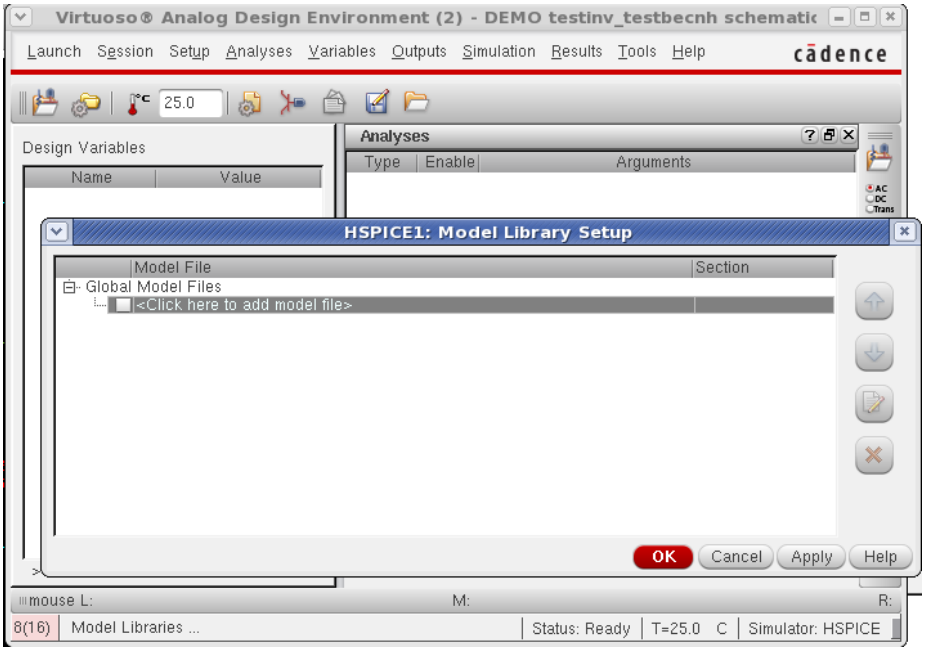
2.2.1. Add model file path

Step-1: Open **Setup > Model Libraries**

Step-2: Enter the following path of the model files.

/designPackages/design_installer/scl/scl_pdk/design_kit/models/hspice/ts18sl_scl.lib

Step-3: Choose appropriate *Section* from the drop-down list. For this demo example, choose **tt_hv**.

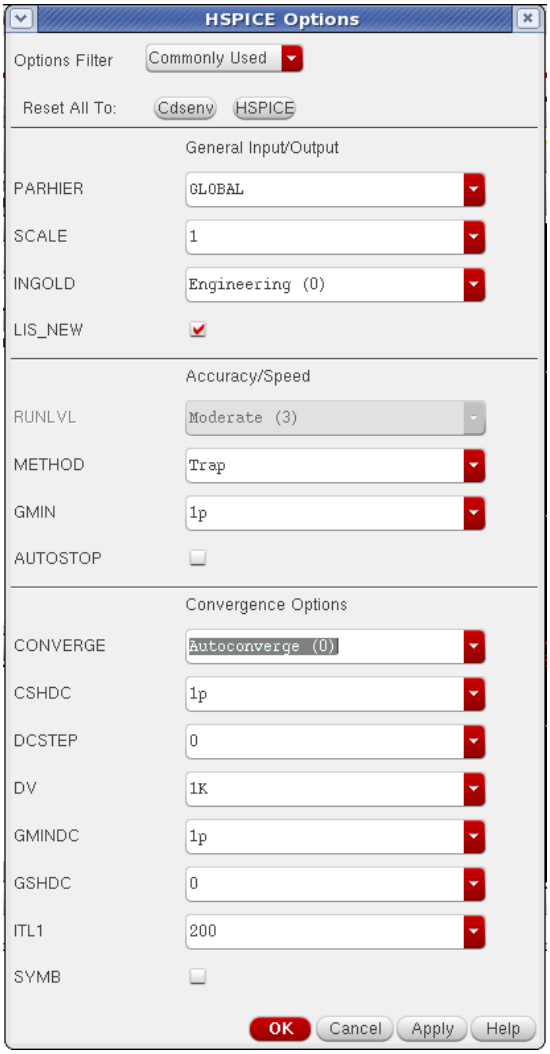


2.2.2. Change scale factor

In SCL process, dimensions of the devices are assumed to be in *micrometer* scale. However, in by default the HSPICE simulator assumes all the dimension to be in *meter*. Therefore, it is necessary to change the default setting.

Step-1: Open **Simulation > Options > Analog > Commonly Used... I**t will open a window (HSPICE Options)

Step-2: In **HPICE Options** window, change the default value of the parameter, **Scale** from **1** to to **1u**.



3. Physical verification

Open the **testinv** schematic from the **DEMO** library. Launch **Layout-XL**.

3.1. DRC

Step-1: Launch **Calibre > DRC**

Step-2: Go to **File > Load Runset** and browse to the ***drc.rsf*** file.

Step-3: Run **DRC**

3.2. Antenna rule check

Step-1: Launch **Calibre > DRC**

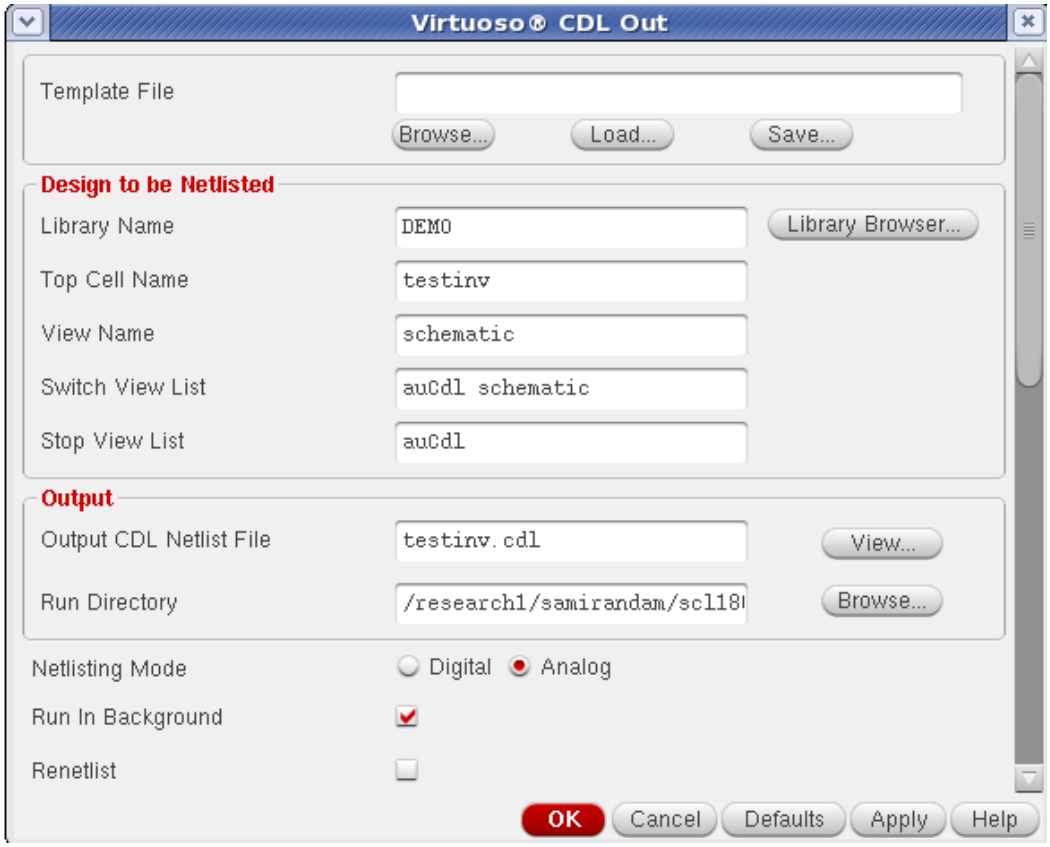
Step-2: Go to **File > Load Runset** and browse to the ***antenna.rsf*** file.

Step-3: Run **DRC**

3.3. LVS

Step-1: Create the **CDL netlist** of the schematic.

- Go to the **CIW** window.
- Go to **File > Export > CDL**
- In the window that appears, click on the button **"Library Browser"** and browse to the schematic of the **testinv** cell.



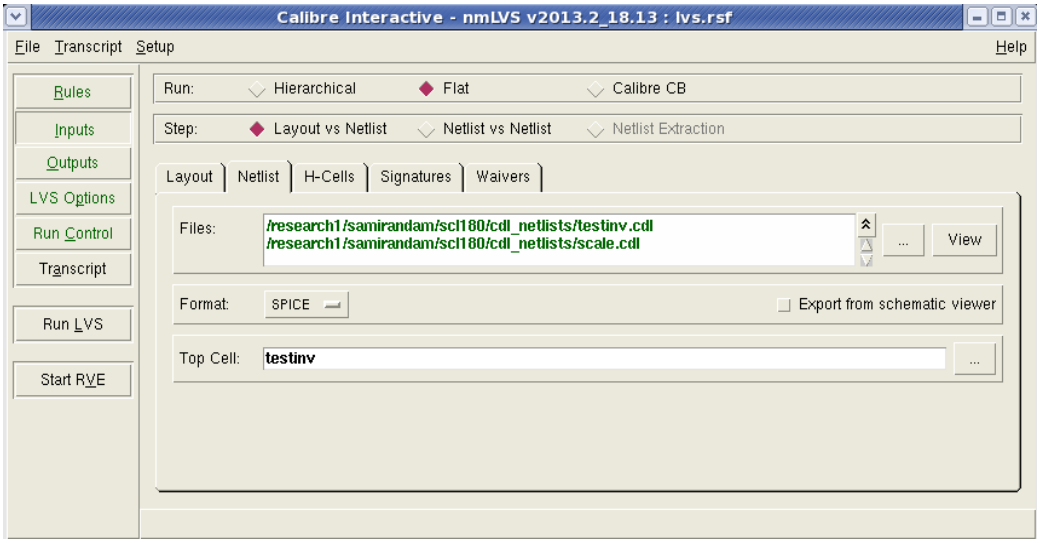
- In the Output section of the window, specify the **"Run Directory"** as the **cdl_netlist** directory in the <WORKING_DIR>. And specify the **"Output CDL Netlist File"** as the name of the cell, which for this example is **testinv.cdl**.

Step-2: Launch **Calibre > LVS**

Step-3: Go to **File > Load Runset** and browse to the ***lvs.rsf*** file.

Step-4: In the **"Inputs"** section of the LVS window, select the **Netlist** tab. And in the Files section browse to the **cdl_netlist** directory and select the **testinv.cdl** and **scale.cdl** files and add them in the list of files.

Step-5: Run **LVS**



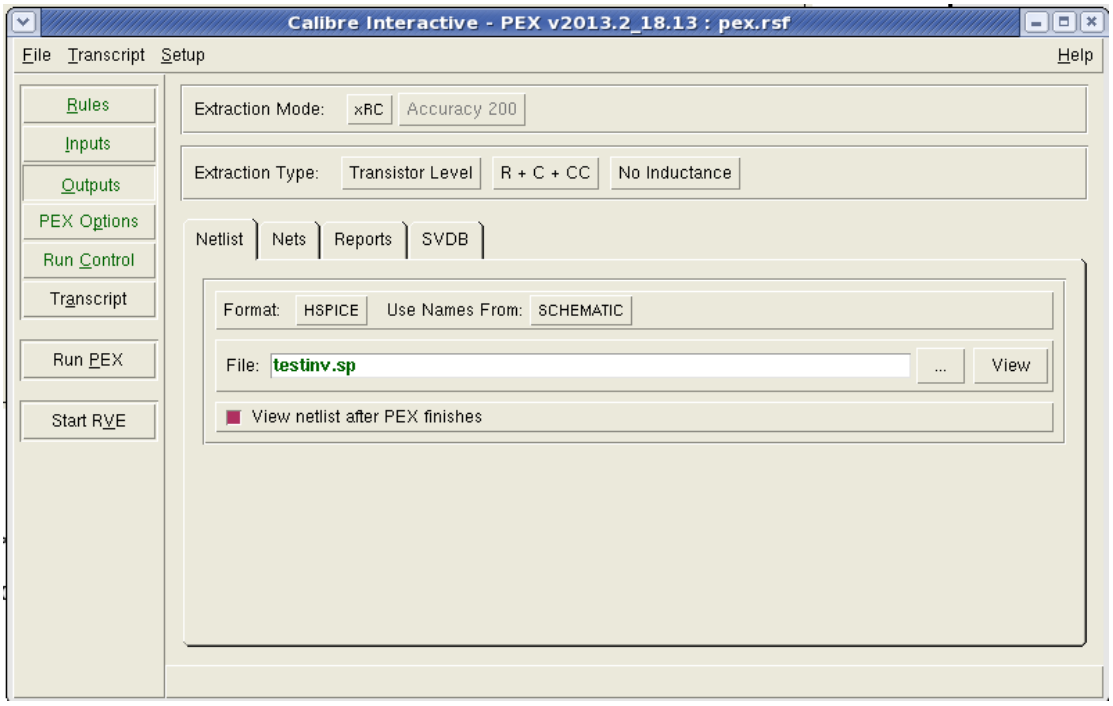
3.4. PEX

Step-1: Launch **Calibre > PEX**

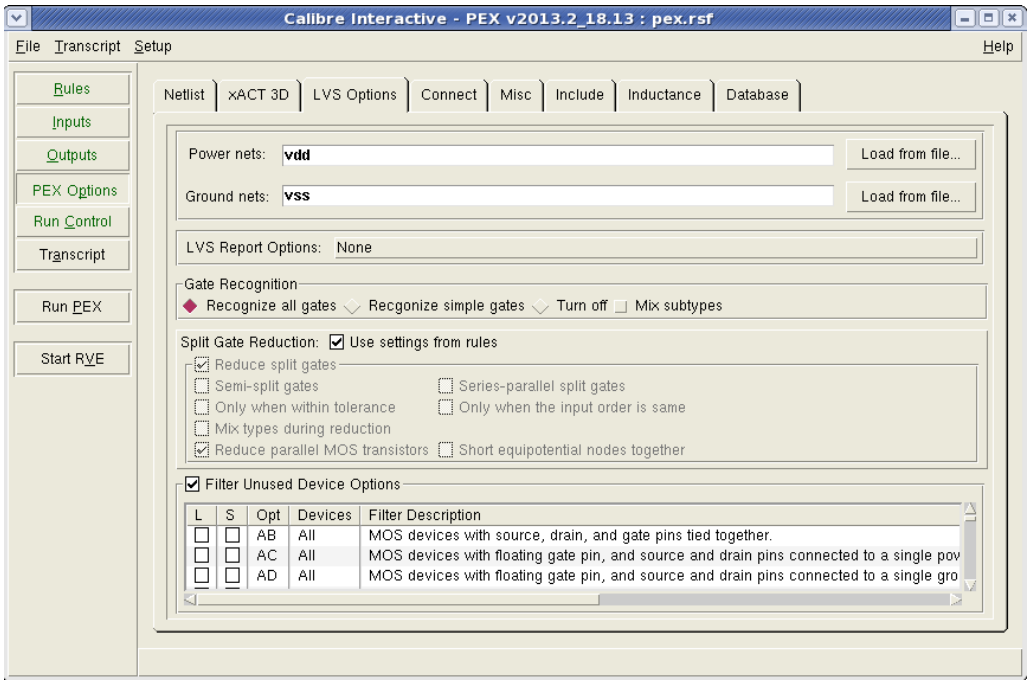
Step-2: Go to **File > Load Runset** and browse to the ***pex.rsf*** file.

Step-3: In the **"Inputs"** section of the PEX window, select the **Netlist** tab. And in the Files section browse to the **cdl_netlist** directory and select the **testinv.cdl** and **scale.cdl** files and add them in the list of files.

Step-4: Go to the **"Outputs"** section of the PEX window and choose **"Format"** as **HSPICE** and change the **"File"** name to ***testinv.sp***. This file will be saved in - *<WORKING_DIR>/physical_verification/pex/temp/testinv.sp* path



Step-5: Go the **"PEX Options"** and then select the tab **"LVS Options"** and enter **vdd** in the field **Power nets** and **vss** in the field **Ground nets**.



Step-6: Run **PEX**

4. Post-layout simulation

Post-layout simulation is done using HSPICE in command-mode. A sample spice netlist for transient simulation of the our example inverter is saved in -

<WORKING_DIR>/simulation/spice_files/inverter _testcase.sp

Before starting the simulation, modify the *inverter_testcase.sp* to include the extracted netlist

.include <WORKING_DIR>/physical_verification/pex/temp/testinv.sp

4.1. Simulation using HSPICE

Execute the following command to run HSPICE on the spice netlist.

hspice64 -mt 2 -i <WORKING_DIR>/simulation/spice_file/inverter_testcase.sp -o <WORKING_DIR>/simulation/results/testinv_sim &

Note: *<WORKING_DIR>/simulation/spice_file/inverter_testcase.sp* is the path of the spice netlist of the testbench

<WORKING_DIR>/simulation/results/testinv_sim is the path of the simulation result directory

4.2. Viewing simulated waveforms

Run the following command to open the waveform window.

wv &

Browse to the simulation result directory (**File > Import Waveform File**) and select ***inverter_testcase.tr0*** file.

5. Monte-Carlo simulation

...to be updated in the next revision.

6. I/O Library

...to be updated in the next revision.