TSMC 180nm Process Setup

For students who haven't set up Cadence environment, please follow the website below to set up your Cadence environment before adding TSMC 180nm process.

https://wikis.ece.iastate.edu/vlsi/index.php/Cadence_6.1_Setup

For students who have set up Cadence environment.

1. Create a folder in your home directory, e.g. "TSMC180".

2. Open terminal in your created folder in step 1 and create a file named "cds.lib" in the folder with the command "gedit cds.lib". Then add the following lines in the "cds.lib" file:

PDK Libraries

DEFINE tsmc18rf /remote/cadencelib/tsmc018/tsmc18rf

System Libraries

DEFINE analogLib$CDS_INST_DIR/tools/dfII/etc/cdslib/artist/analogLib

DEFINE basic $CDS_INST_DIR/tools/dfII/etc/cdslib/basic

DEFINE cdsDefTechLib$CDS_INST_DIR/tools/dfII/etc/cdsDefTechLib

DEFINE avTech/usr/local/cadence/assura/tools.lnx86/assura/etc/avtech/avTech

SOFTINCLUDE /usr/local/cadence/ic/share/cdssetup/cds.lib

DEFINE avTech /usr/local/cadence/assura/tools.lnx86/assura/etc/avtech/avTech

3. Run "virtuoso&" in the terminal and the Command Interpreter Window (CIW) will be open,
Then click "File-New-Library" to create your own library for your project, as shown in Fig.3.1.

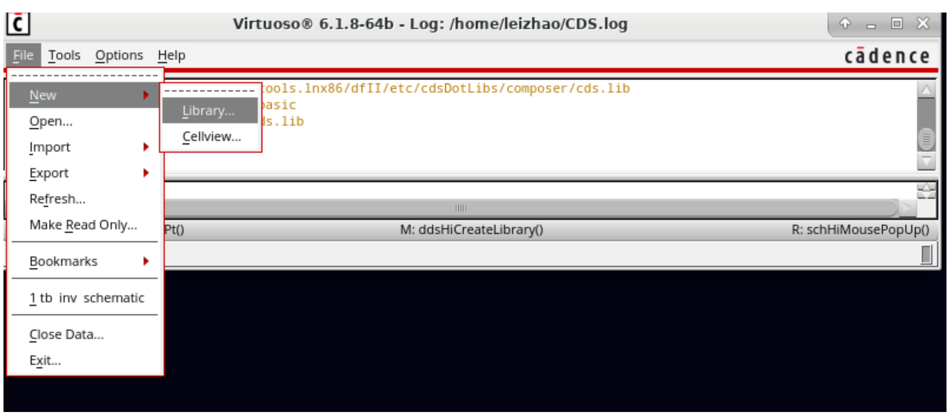

Fig 3.1

When you create your own library, you need to attach your library to an existing technology library, as shown in Fig 3.2. Click OK and choose “tsmc18rf” as the technology library.

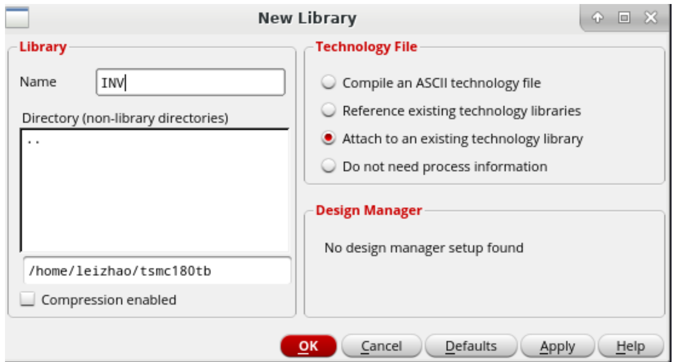

Fig. 3.2

4. Open the Library Manager from “Tools” in CIW as shown in Fig. 4.1.

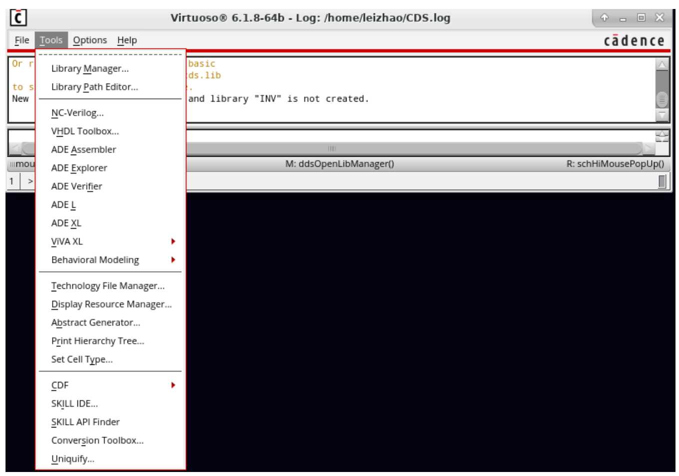

Fig. 4.1

Now you can create new cell view (schematic) for your circuits, as shown in Fig.4.2. and Fig.4.3.

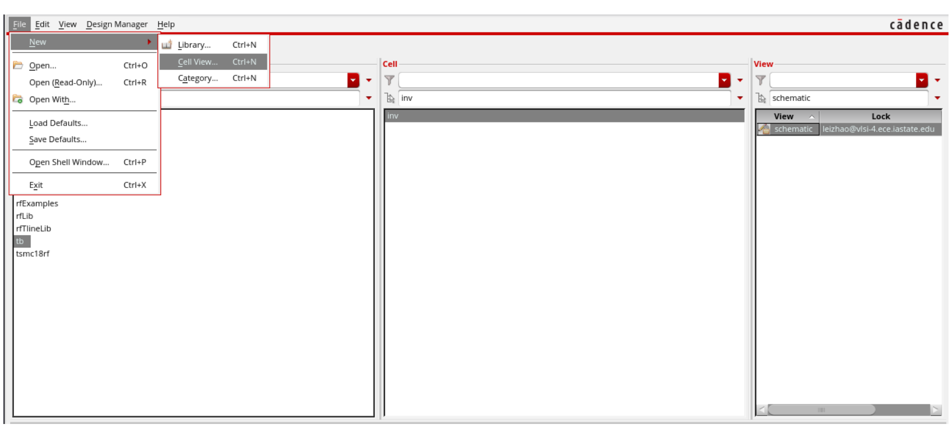

Fig. 4.2

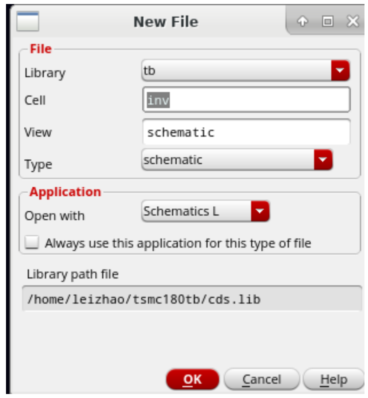

Fig. 4.3

5. To do simulation with your circuit, you can click “Launch-ADE L”.

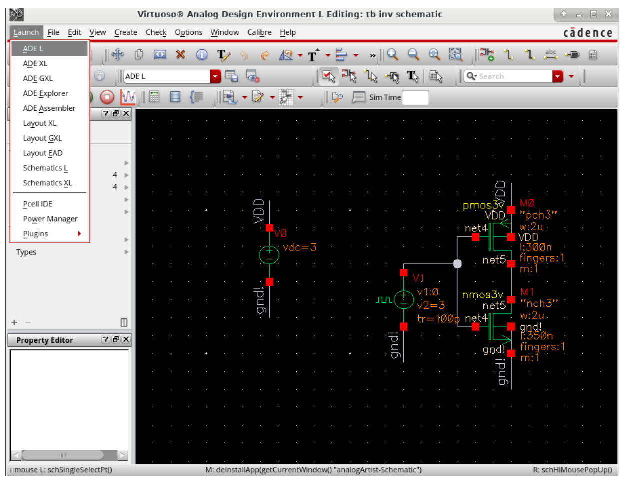

Fig.5.1

6. In the ADE window, before running simulation, you need to add the models for the device. Click “Setup-Model Libraries” as shown Fig.6.1.

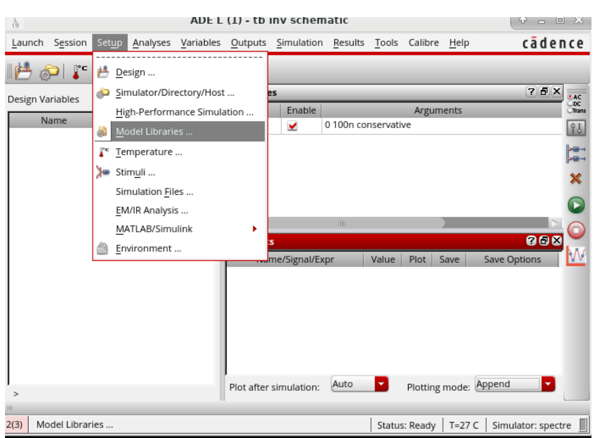

Fig.6.1

And then setup the model library as shown in figure 6.2.

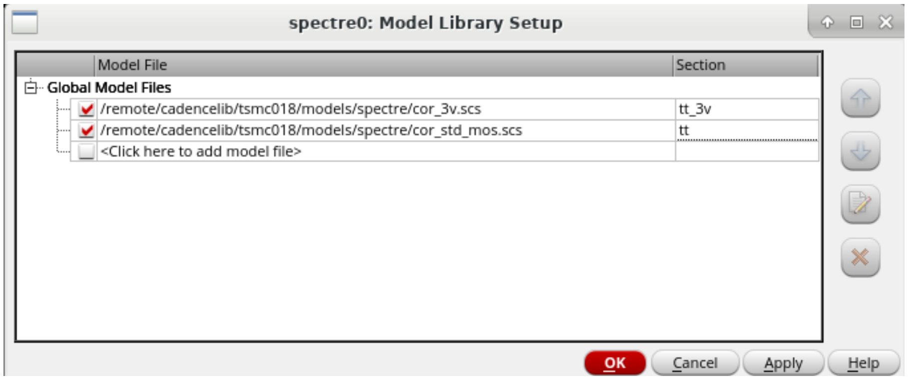

Fig.6.2