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.

Č	Virtuoso® 6.1.8-64b - Log: /home/leizhao/CDS.log	
File Tools Options	Help	cādence
New Open Import Export	ools.lnx86/dfII/etc/cdsDotLibs/composer/cds.lib Library Scientific State	
Refresh Make <u>R</u> ead Only Bookmarks	Pt() M: ddsHiCreateLibrary()	R: schHiMousePopUp()
<u>1</u> tb inv schematic 		

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.

	New Library 🔹 🔍
Library Name INV Directory (non-library directories) //home/leizhao/tsmc180tb Compression enabled	Technology File Compile an ASCII technology file Reference existing technology libraries Attach to an existing technology library Do not need process information Design Manager No design manager setup found
	OK Cancel Defaults Apply Help

Fig. 3.2

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

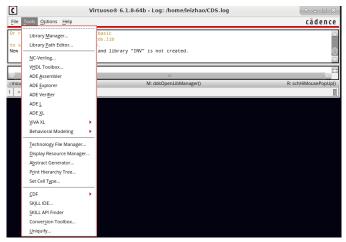


Fig. 4.1

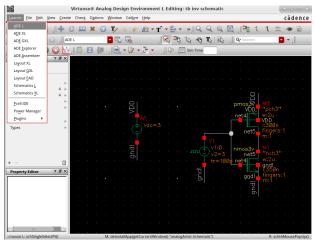
Now you can create new cell view (schematic) for your circuits, as shown in Fig.4.2. and Fig.4.3.

Elle Edit View Design Manager	Help		c a d e n c e
<u>N</u> ew >	네 Library Ctrl+N		
📂 Open Ctrl+O	<u>Cell View</u> Ctrl+N	Cell	View
Open (<u>R</u> ead-Only) Ctrl+R	Category Ctrl+N		▼▼
😂 Open With	-	B _t inv ▼	Bt schematic
Load Defaults	_	irv	View A Lock
Save Defaults			
Open Shell Window Ctrl+P			
E <u>x</u> it Ctrl+X			
rfExamples			
rfLib rfTlineLib			
tb			
tsmc18rf			
L		10	

	New File	Ŷ		\times	
File					
Library	tb				
Cell	inv				
View	schematic				
Туре	schematic		•		
Application					
Open with	Schematics L				
Always use this application for this type of file					
Library path file					
/home/leizhao/tsmc180tb/cds.lib					
			<u>H</u> el	p)	



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





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.

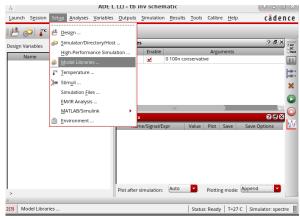


Fig.6.1

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

spectre0: Model Library Setup		
Model File	Section	
🖻 Global Model Files		
/remote/cadencelib/tsmc018/models/spectre/cor_3v.scs	tt_3v	
/remote/cadencelib/tsmc018/models/spectre/cor_std_mos.scs	tt	
Click here to add model file>		4
	OK <u>C</u> ancel <u>App</u> l	y <u>H</u> elp

Fig.6.2