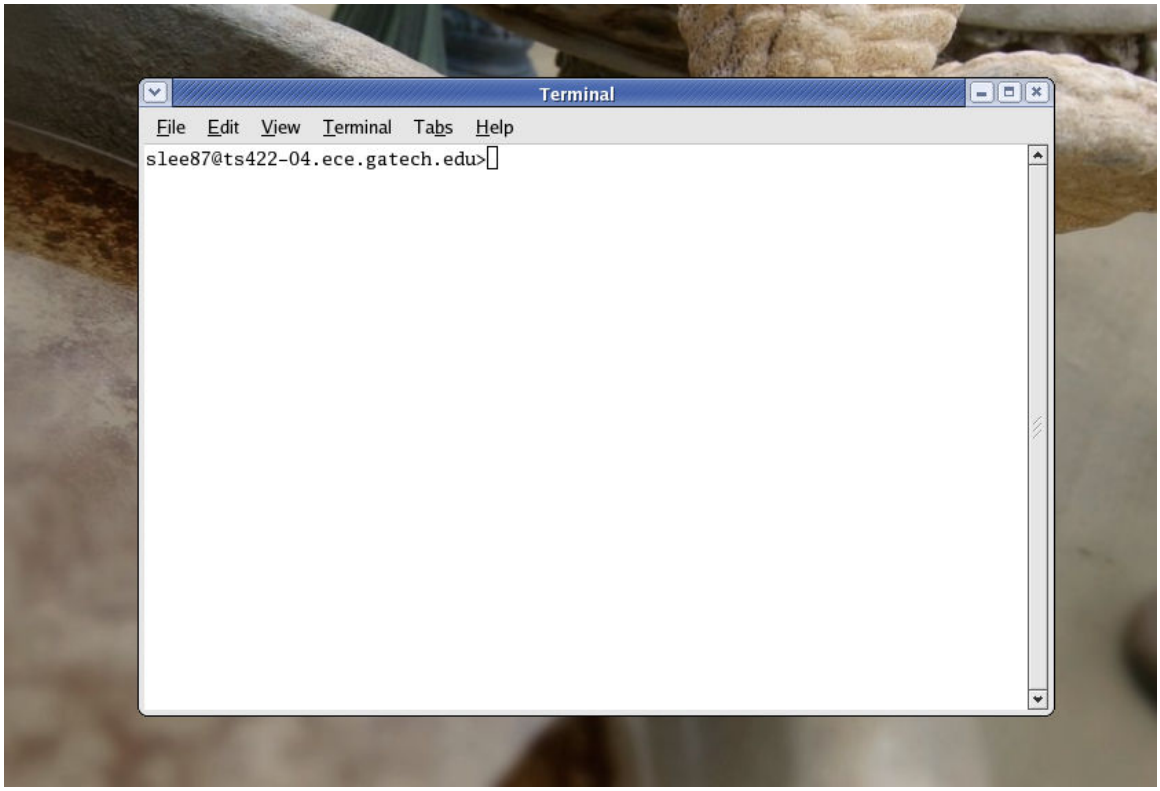
CADENCE SETUP

This short tutorial shows how to configure Cadence to use the NCSU Cadence Design Kit (CDK) with access to the ON Semiconductor C5 0.5-µm and the TSMC 0.35-µm CMOS processes libraries.

In LINUX
◊ Right button of mouse -> Open Terminal



◊ Make cadence directory
ece.gatech.edu> mkdir cadence
◊ Move to cadence directory
ece.gatech.edu> cd cadence
◊ Copy setup files (This should be done only once)
cp /tools/linsoft2/cadence/ic5141usr5/ncsu/cdssestup/cdsinit .cdsinit
cp /tools/linsoft2/cadence/ic5141usr5/ncsu/cdssestup/cdsenv .cdsenv

◊ Edit cds.lib file
ece.gatech.edu> gedit cds.lib&
◊ Add lines to cds.lib file

```
SOFTINCLUDE $SYSTEM_CDS_LIB_DIR/cds.lib
DEFINE          basic                   $CDK_DIR/lib/basic
DEFINE          NCSU_Analog_Parts       $CDK_DIR/lib/NCSU_Analog_Parts
DEFINE          NCSU_Digital_Parts      $CDK_DIR/lib/NCSU_Digital_Parts
DEFINE          NCSU_TechLib_ami06      $CDK_DIR/lib/NCSU_TechLib_ami06
DEFINE          NCSU_TechLib_tsmc04_4M2P
$CDK_DIR/lib/NCSU_TechLib_tsmc04_4M2P
DEFINE analogLib $CDS/tools/dfII/etc/cdslib/artist/analogLib
```

◊ **save and quit**
◊ **Source**
> source /tools/linsoft2/cadence/ic612hf517/cshrc.meta
> source /tools/linsoft2/cadence/ic612hf517/cshrc.ncsu61.lin
> source /tools/linsoft2/cadence/mmsim71/cshrc.mmsim

If you edit .my-cshrc file, then you do not have to type this line always.

- How to edit my-cshrc file?
  - Go to root directory. ("cd").
  - gedit .my-cshrc&
  - Add line:
    > source /tools/linsoft2/cadence/ic612hf517/cshrc.meta
    > source /tools/linsoft2/cadence/ic612hf517/cshrc.ncsu61.lin
    > source /tools/linsoft2/cadence/mmsim71/cshrc.mmsim
  - Save and quit
  - Go to cadence directory

◊ **Execution cadence**
ece.gatech.edu> virtuoso &

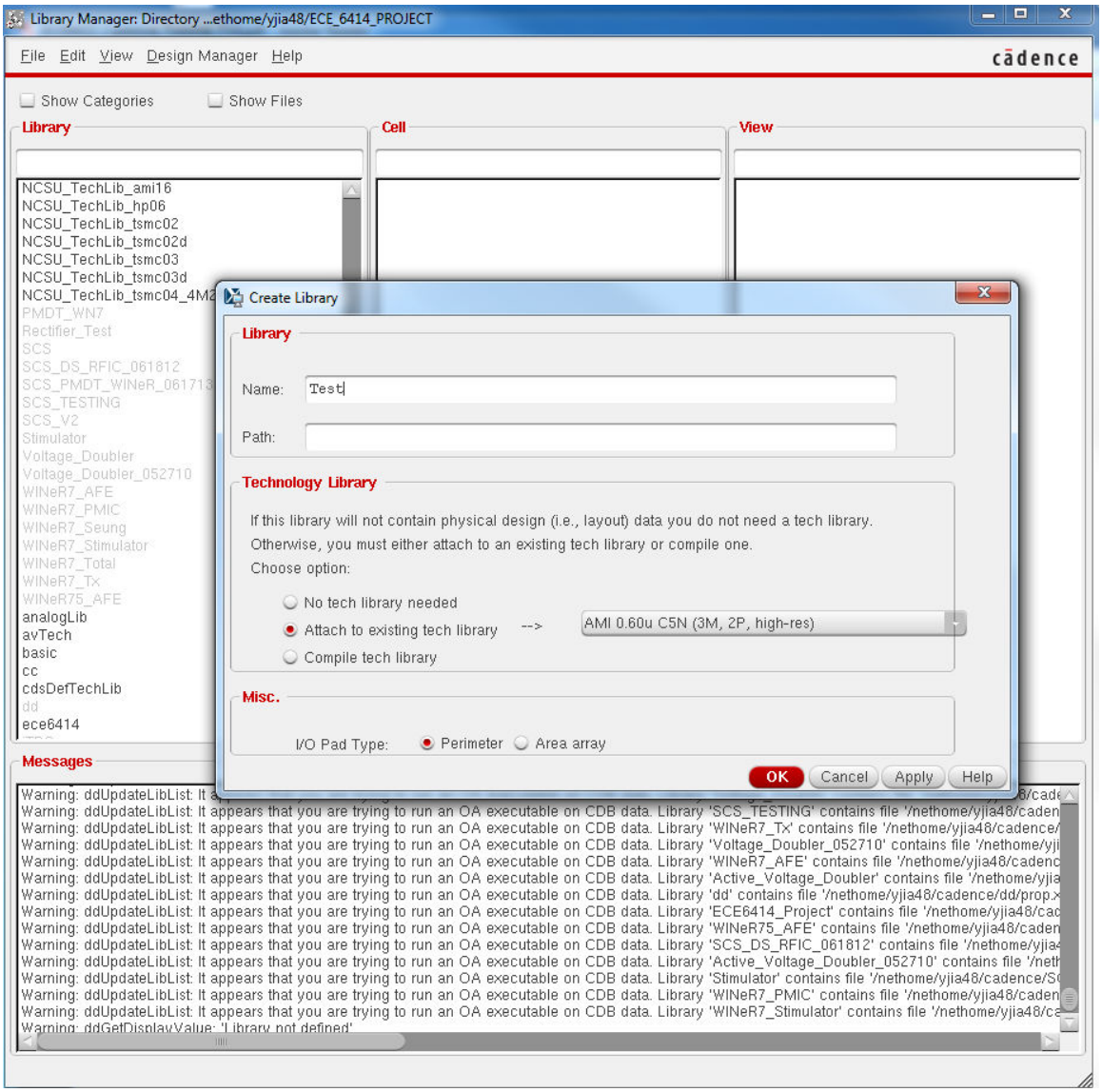
◊ **Tutorials**
Below tutorials are useful materials.
http://www.eda.ncsu.edu/wiki/Tutorial:Contents
➔ **Old Tutorials**
- **Analog Artist Tutorial**
  http://www.cadence.ncsu.edu/NCSUCadenceTutorial/ArtistTutorial.pdf

<CAUTION> You should use the tech library and transistor model files of AMI 0.60 um technology. This is different from NCSTATE tutorial.

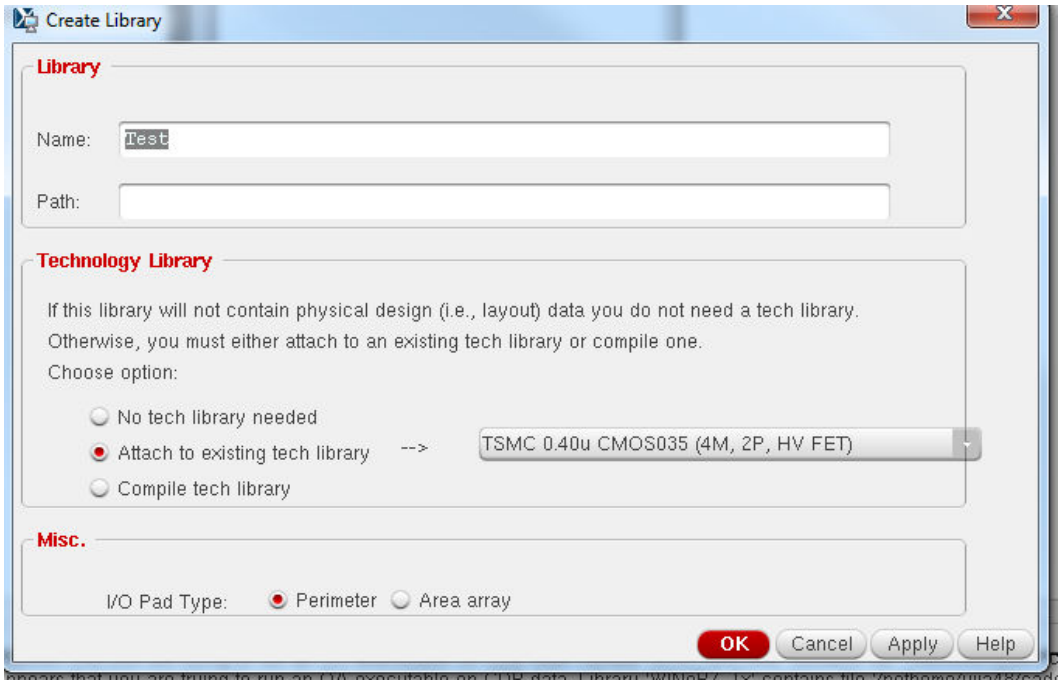
◊ **Generation of a Cadence Library with attachment to an existing technology file**

➔ **Generate Library with ON Semiconductor C5 0.5-µm**
In **Library Manager**, Click **File -> New -> Library**
In the **Name** field, enter "**Test**"
In the **Technology Library** box, select

Attach to existing tech library -> AMI 0.60u
Press **OK**

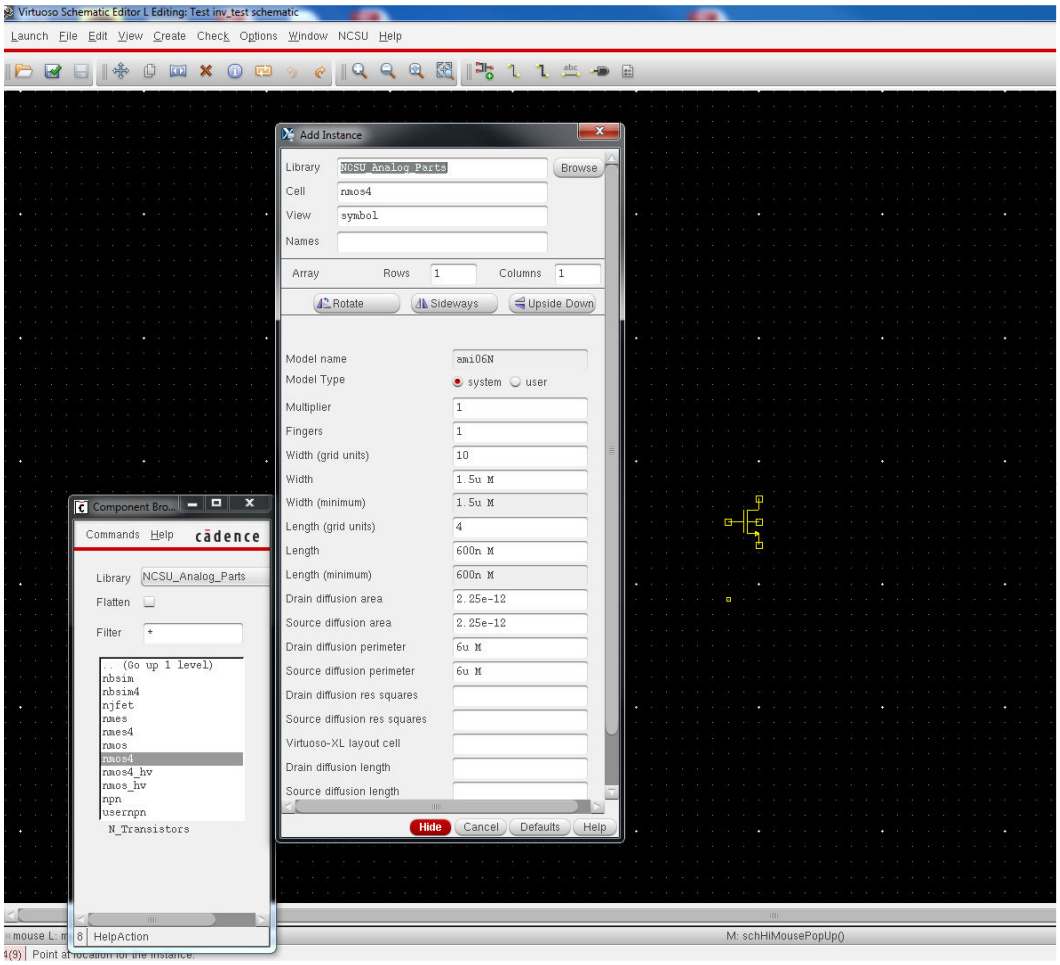


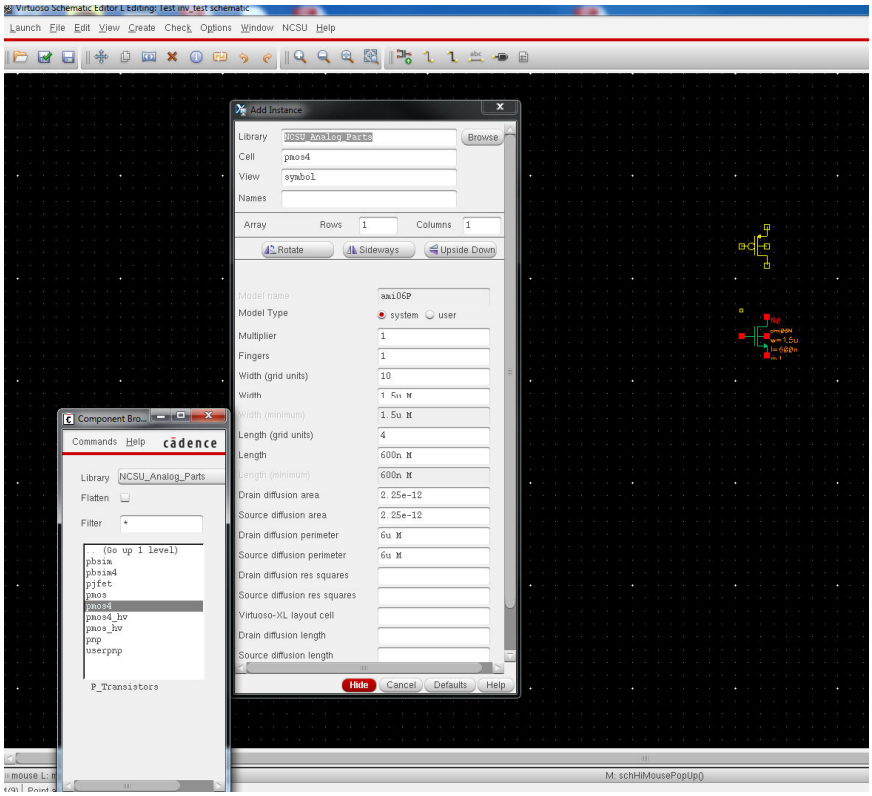
➔ **Generate Library with TSMC 0.35 um process**
Do the same as shown above for generating a Library with the AMI 0.5 um tech file, **but** In the **Technology Library** box, select
Attach to existing tech library -> TSMC 0.40u CMOS035 (4M, 2P, HV FET)
Press **OK**



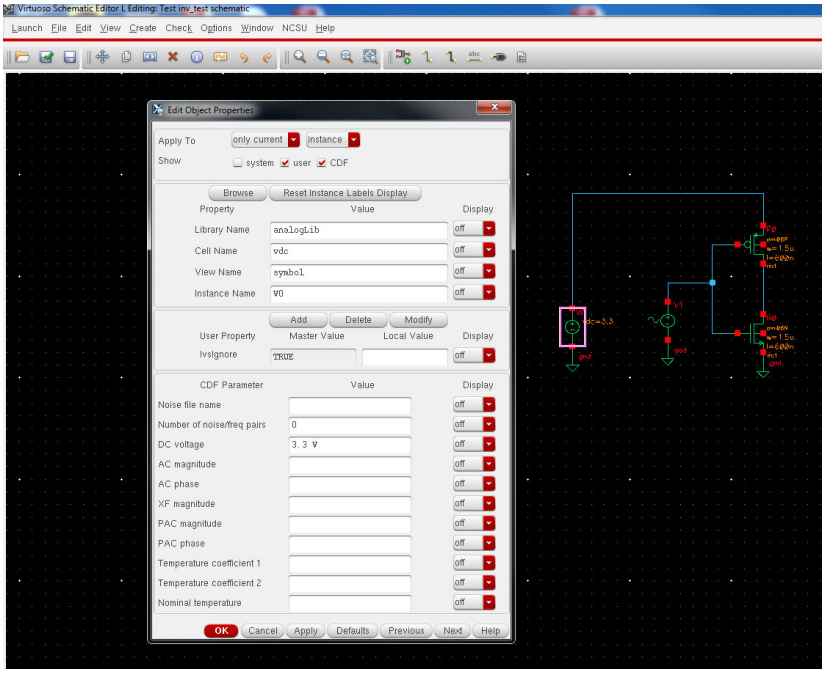
➔ **Making a cell for designing schematic**
In **Library Manager**, Click **File -> New -> Cell View**
Select Library as "Test"
In the Cell name field, enter "**inv_test**"
In the **type** box, select "Schematic"
Press **OK**

➔ **Design a circuit (example: make a simple inverter)**
In schematic, press "I" for the component browser, and select library "NCSU_Analog_Parts", and "nmos4" in "N_Transistors".
Find "pmos4" in "P_Transistors".

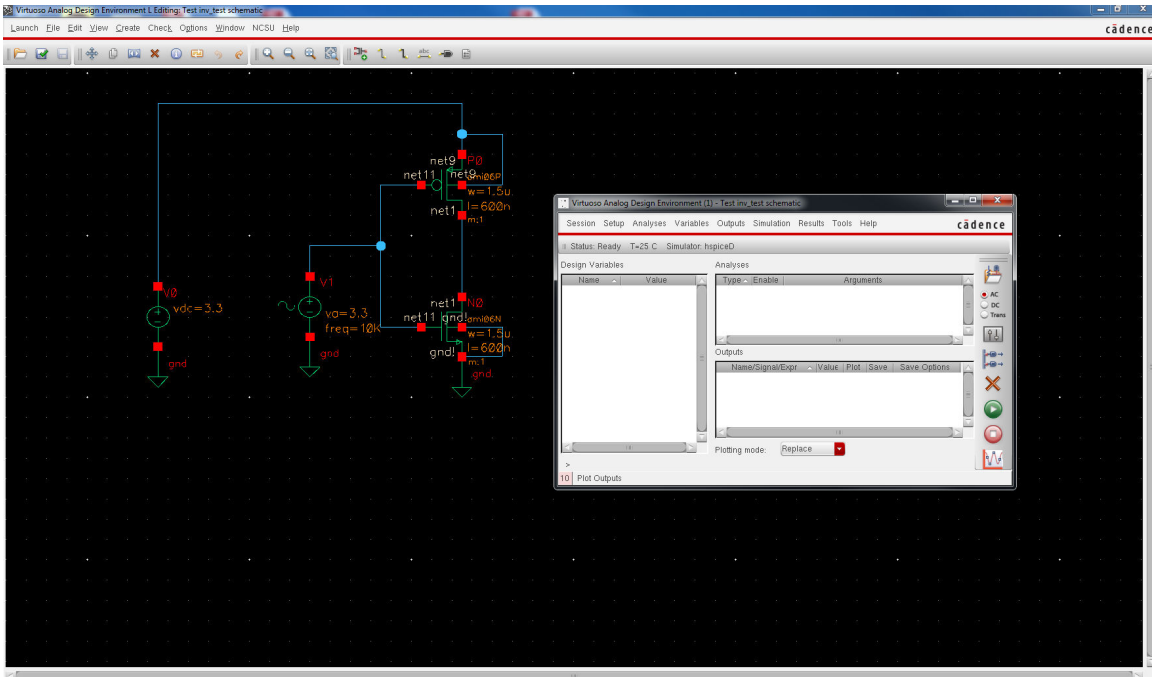




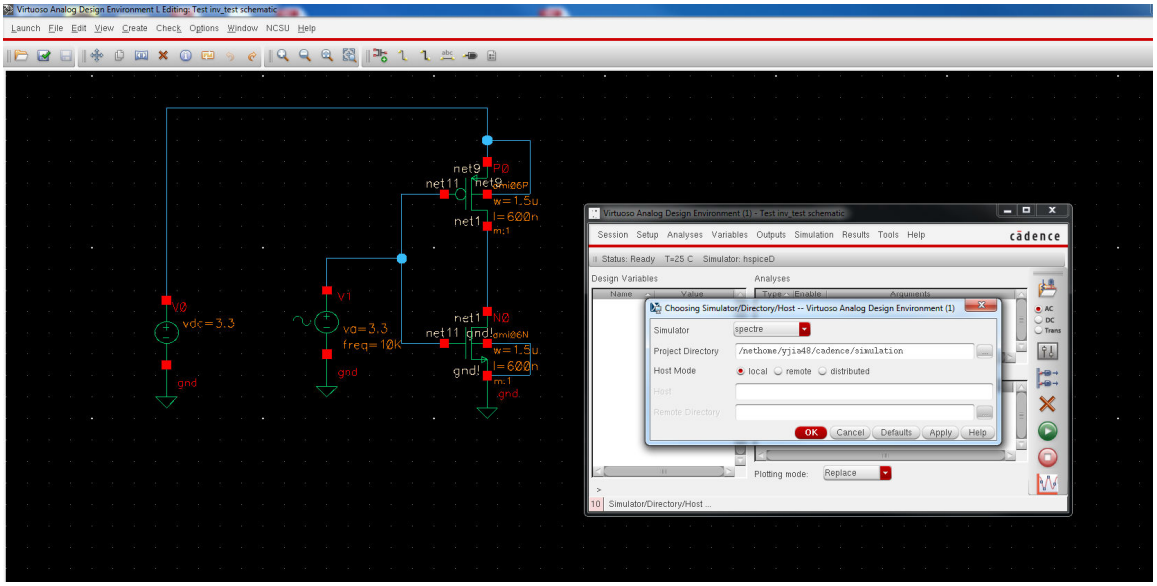
➔ **Find proper sources (vdc, vsin) and ground (gnd) in Component Browser "AnalogLib" Library, and complete the circuit. (Wiring : press "w"). Also input the proper voltage for testing the circuit. (Select component and press "q") Save the schematic (Ctr + s).**



➔ **Select “Launch” -> “ADE_L” for the simulation.**

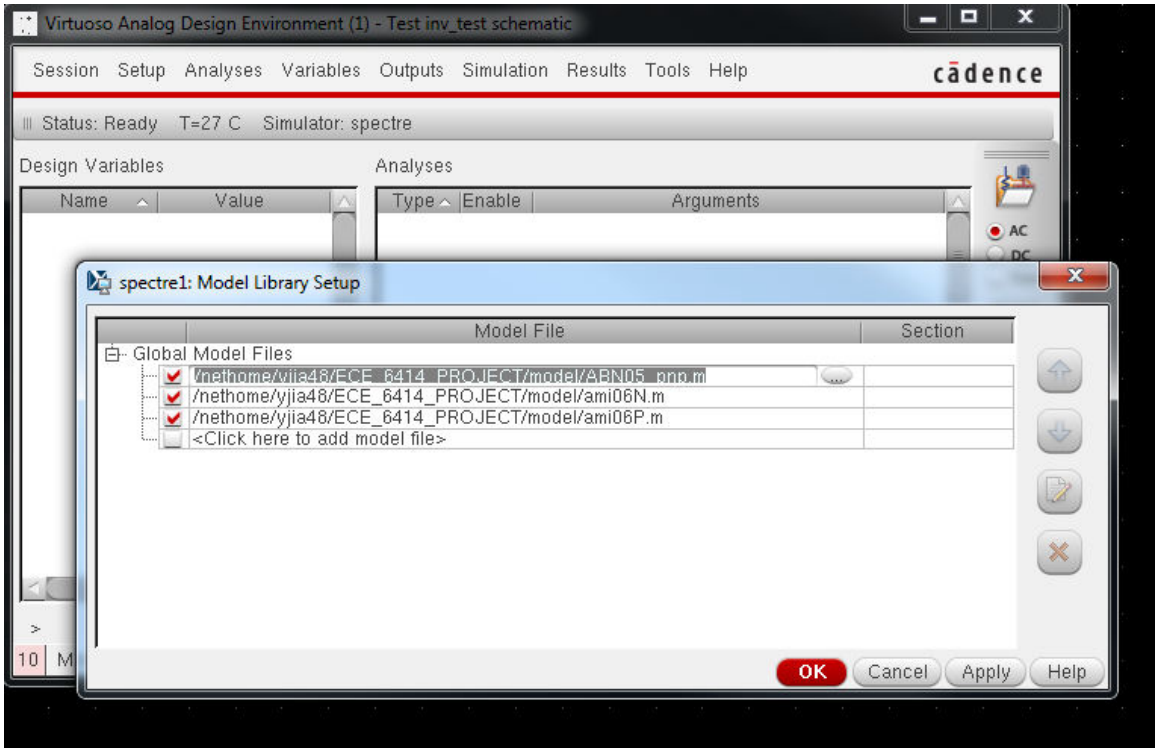


➔ **Setup the simulator**
In Virtuoso Analog Design Environment, select “setup” -> “Simulator/Directory/Host”, and select the “spectre” for the simulator. Press “OK”



➔ **Including transistor model file**
In Virtuoso Analog Design Environment, select "setup" -> "Model Libraries", and add the model for AMI05.

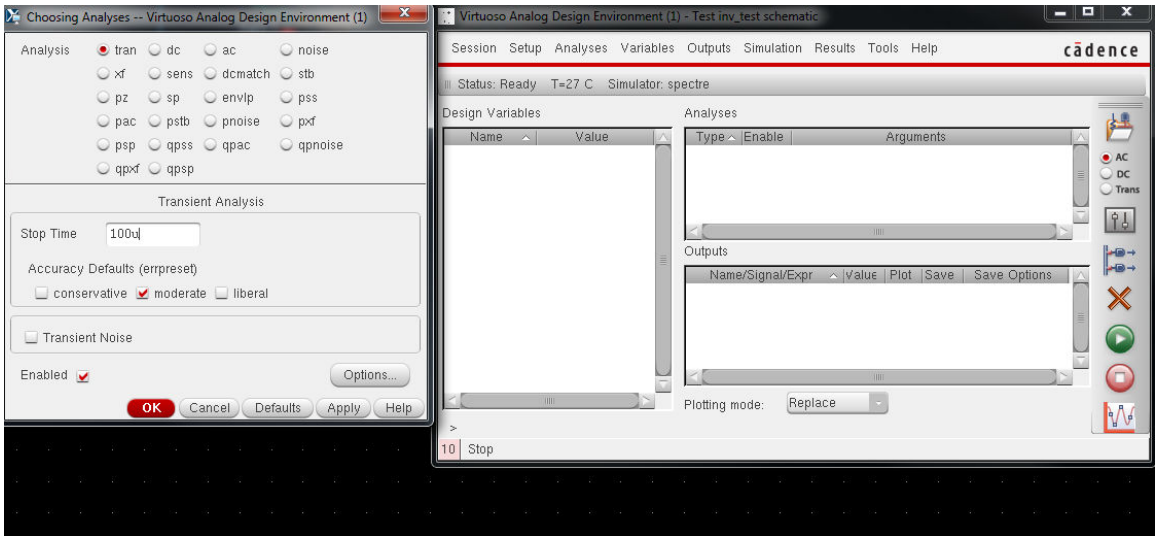
- **To use the ON Semiconductor C5 0.5-µm, browse and add 'ami06N.m' and 'ami06P.m' and 'ABN05'.**
- **For the TSMC 0.35-um process, browse and add the following transistor model files: 'tsmc35N.m' and 'tsmc35P.m'.**

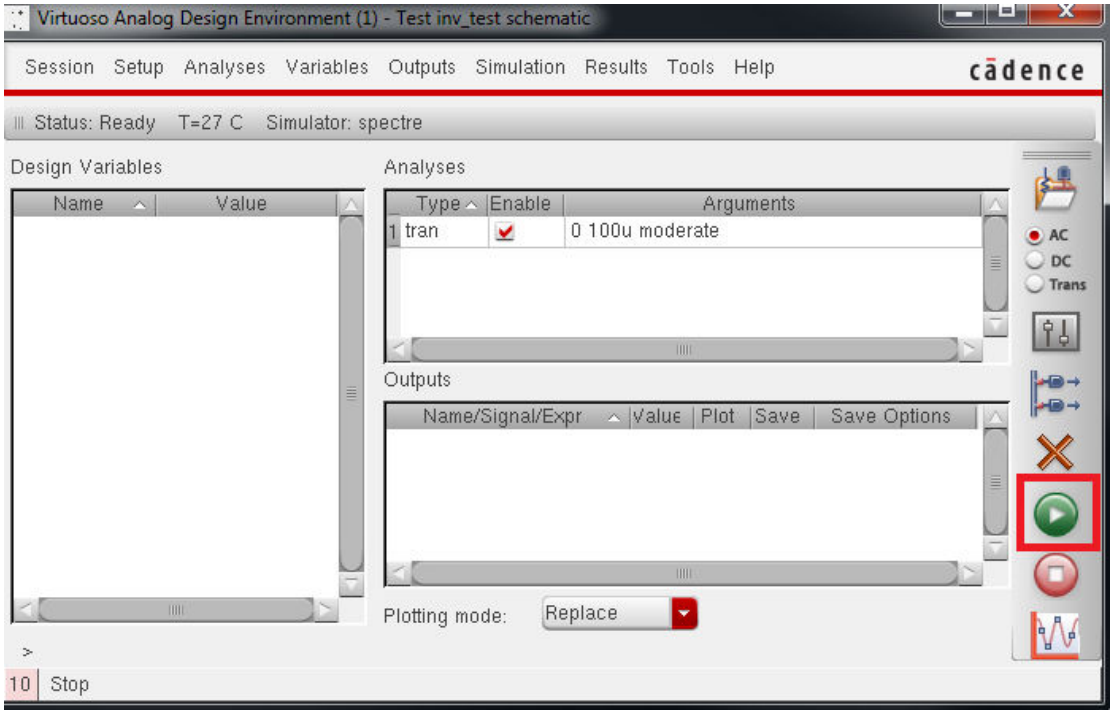


The required transistor model files are located in /tools/linsoft/lincad/ic5141usr5/ncsu/models/spectre/standalone/.

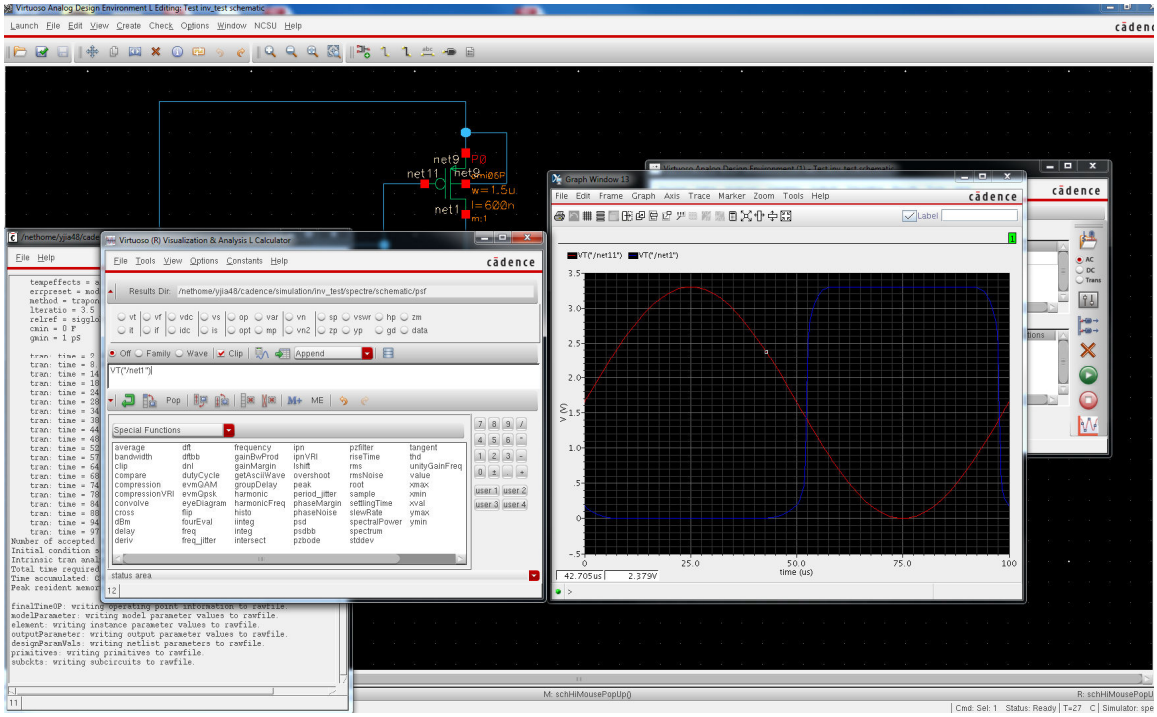
***If you save the simulation state, then you don't need to setup model library and simulator again when you load your saved state. (Session -> Save state & Load state). Also you can load your saved sate for other schematic.**

➔ **Choosing Analysis for the simulation**
In this example, transient simulation is selected.
Click "OK" and run "Netlist and run" button.

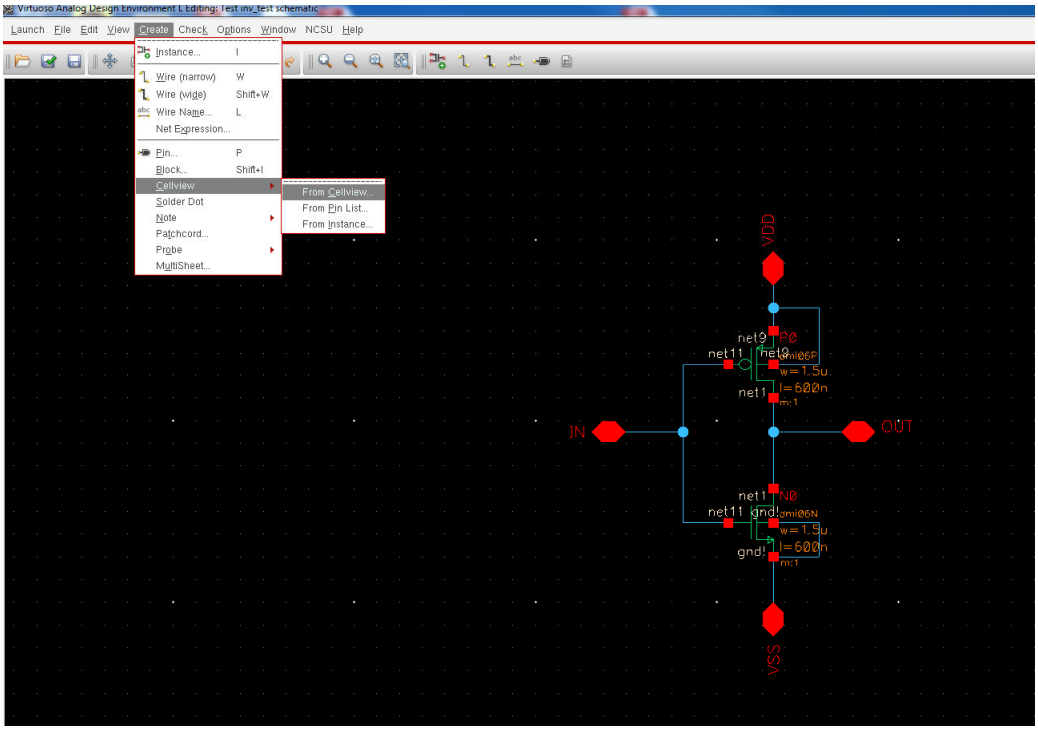


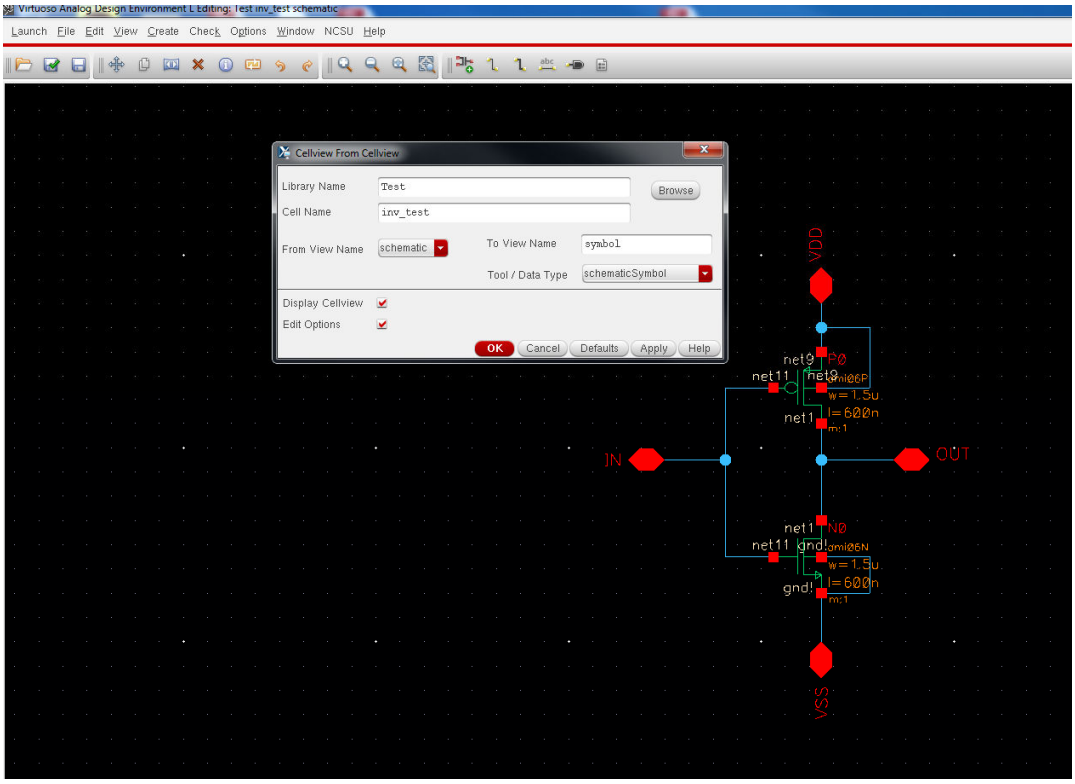


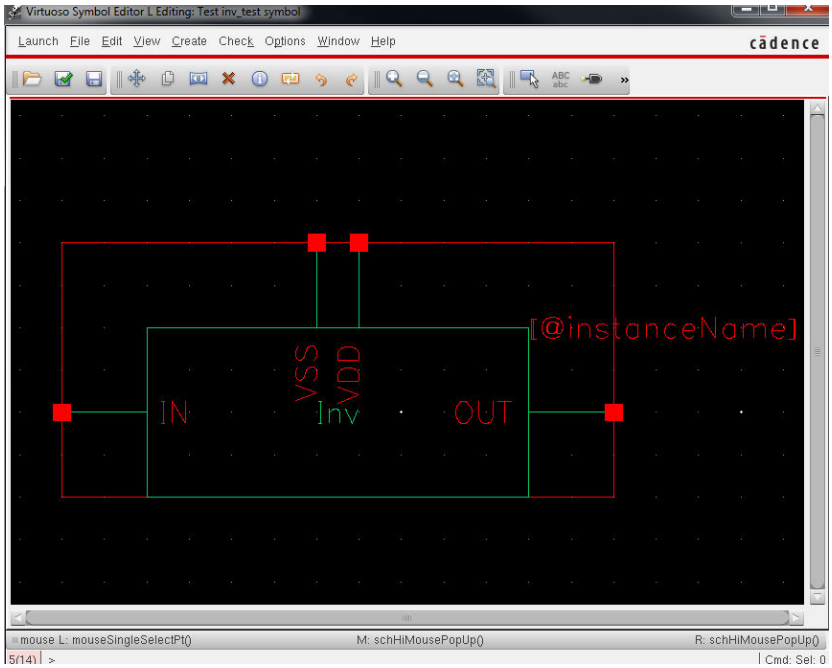
- ➔ **To see the simulated waveforms, you might use calculator in "tools"-> "Calculator" in Virtuoso Analog Design Environment**
- ➔ **For the transient waveforms, click "vt" (Voltage transient) and click the node where you want to see the waveforms.**



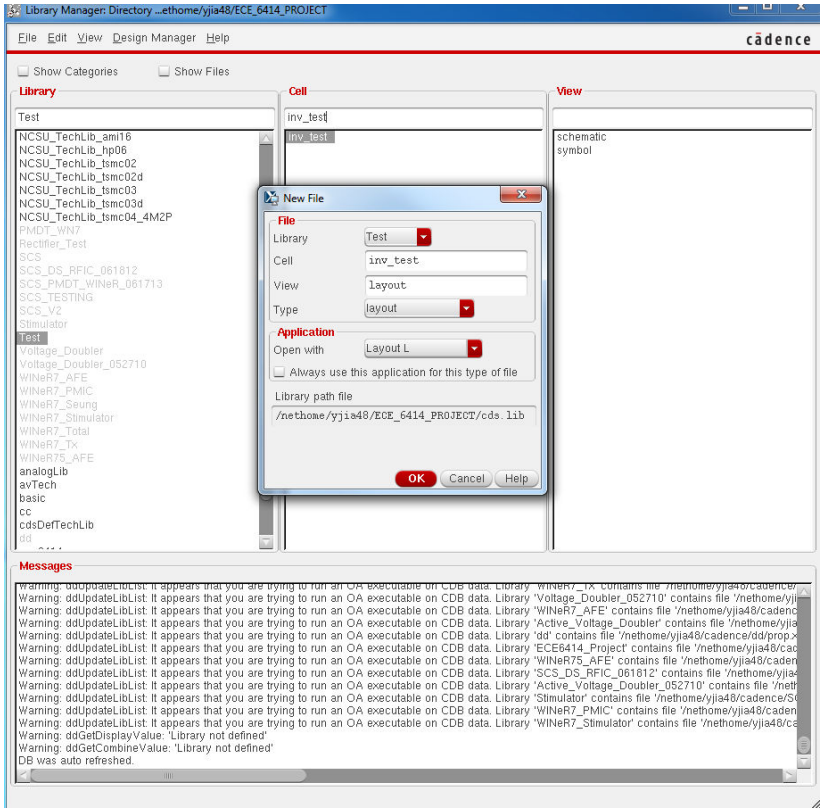
➔ **After simulation is done, you might make a symbol for the circuit. Add the proper pins for inputs and outputs of your circuit. (Hot key "p") Then click "Create"->"Cellview"->"From Cellview" to create symbol.**



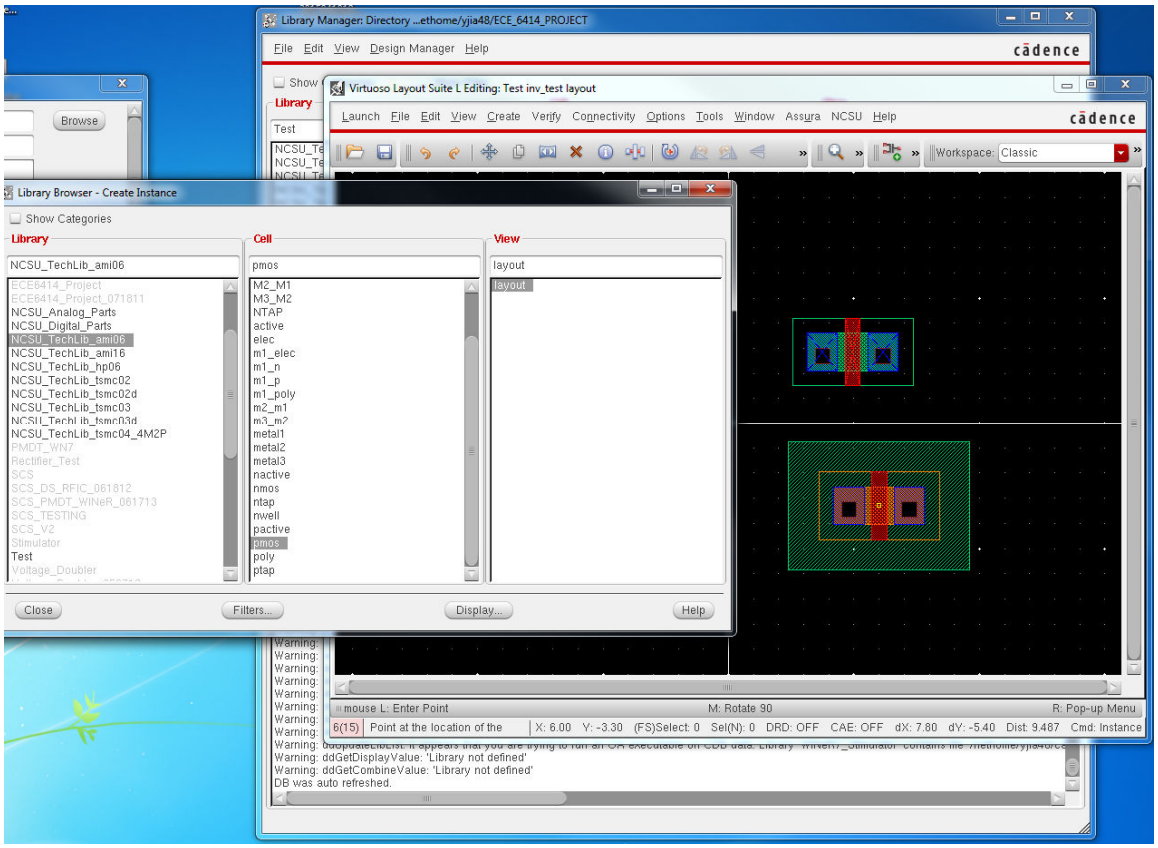




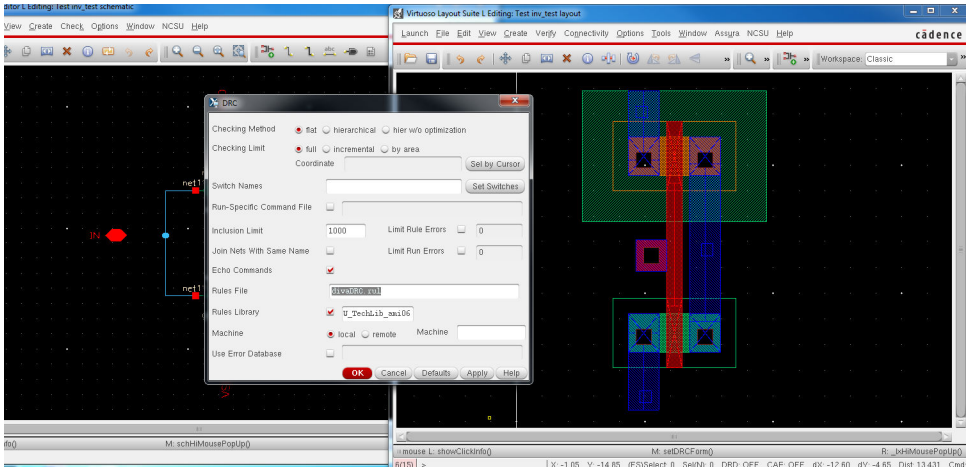
➔ **To create layout, you can create "layout" as same way to create "schematic". Just change the type to "layout".**



- ➔ **Create instance (hotkey: “I”), and find “nmos” and “pmos” in NCSU_TechLib_ami06 for AMI05. (NCSU_TechLib_tsmc03 for tsmc035 process)**
- ➔ **Check the parameters are same in schematic design. (hotkey: “q” on the component)**
- ➔ **Complete the layout with proper input and output pins in your schematic (Pin location hotkey: “Shift+p”)**

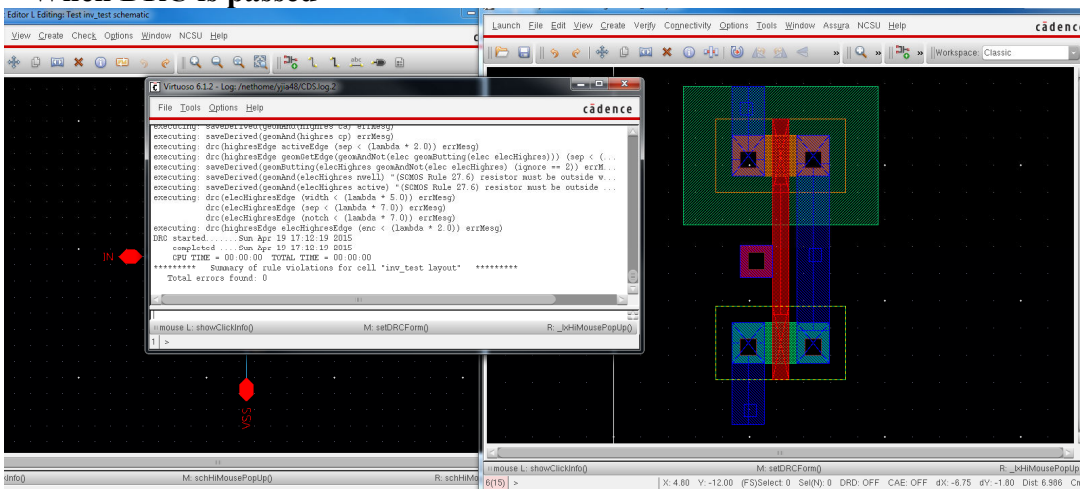


➔ **For DRC, select “Verify”->”DRC” in the layout window.**

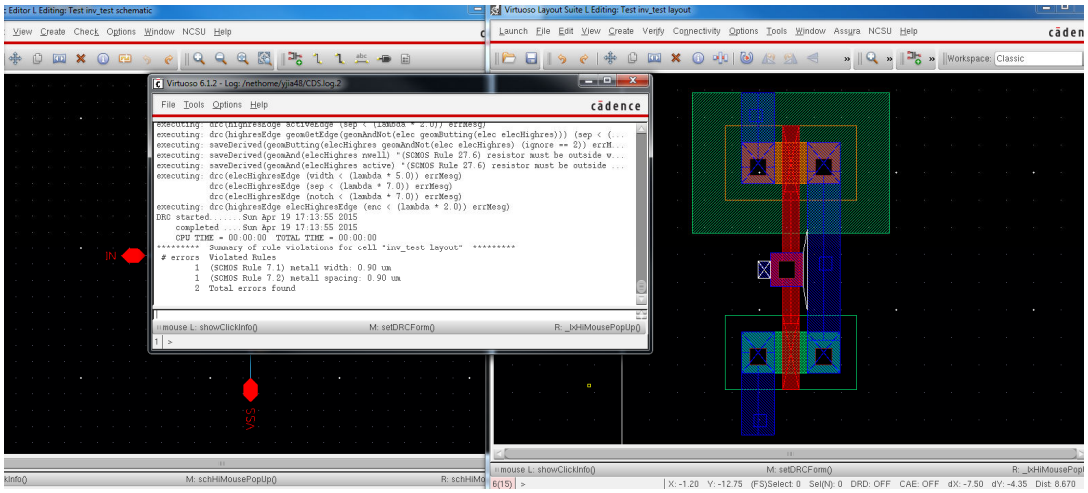


➔ **After DRC, you can see the results in the log file.**

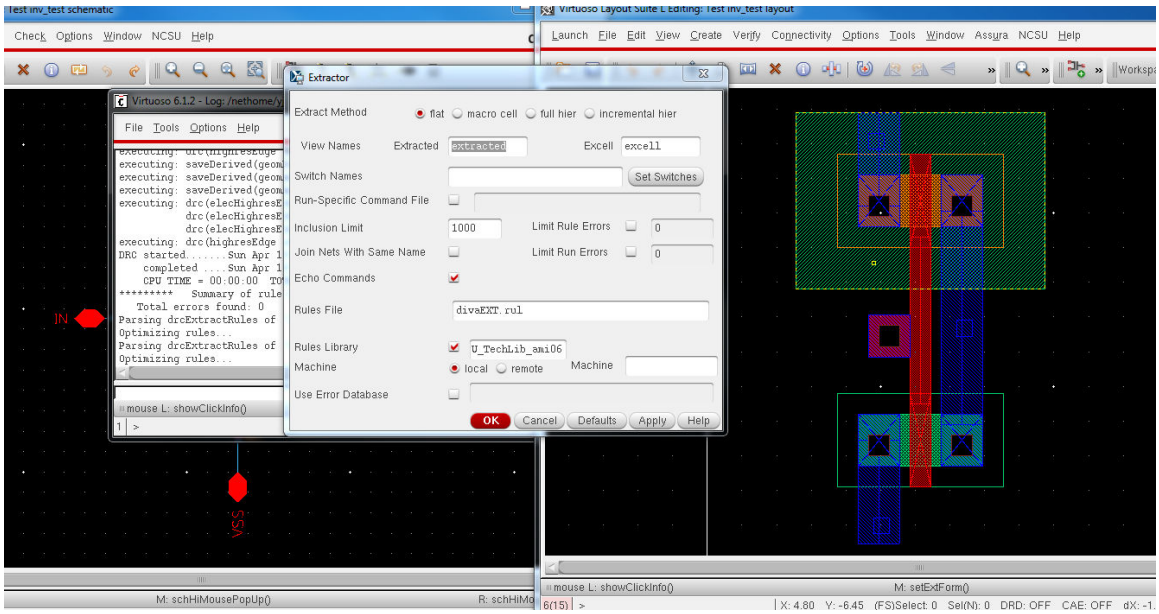
< When DRC is passed >



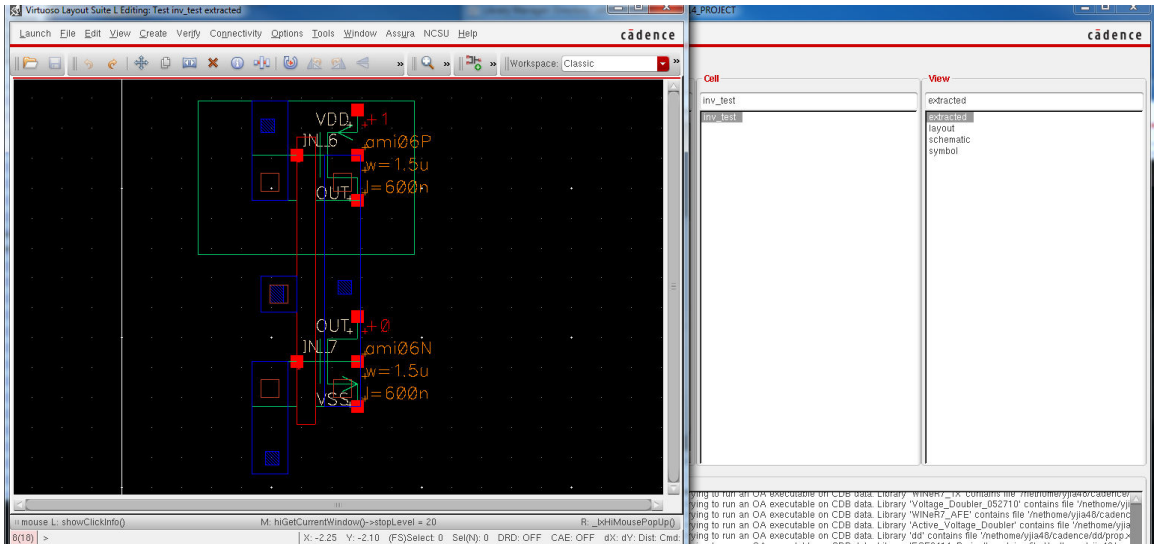
< When DRC has errors >



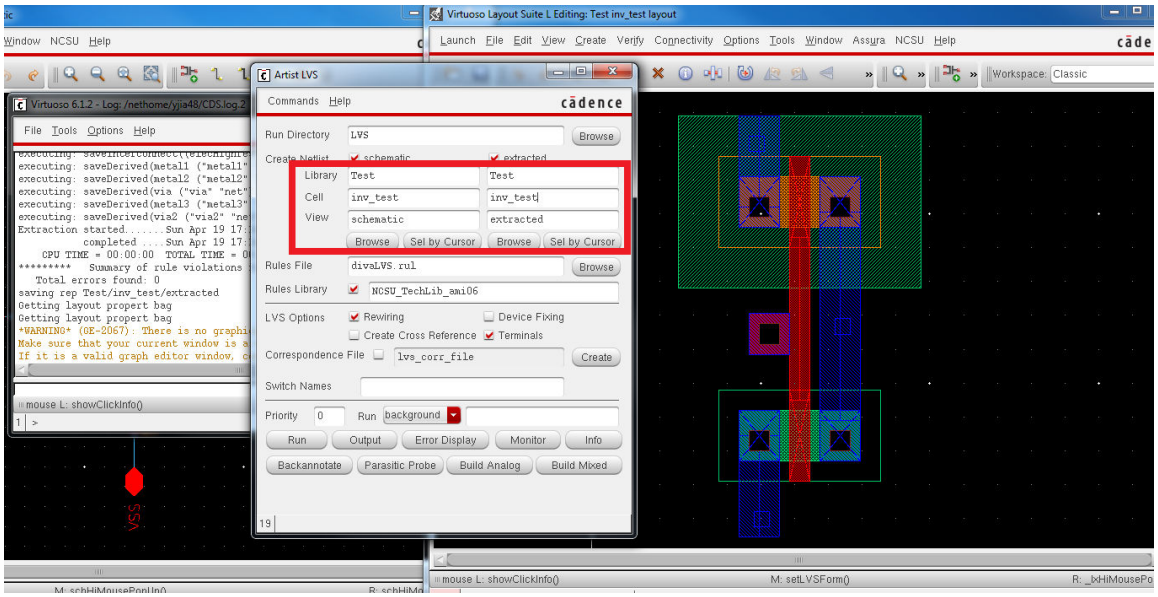
➔ **After DRC has passed, create extracted circuit using "Verify"-> "Extract"**



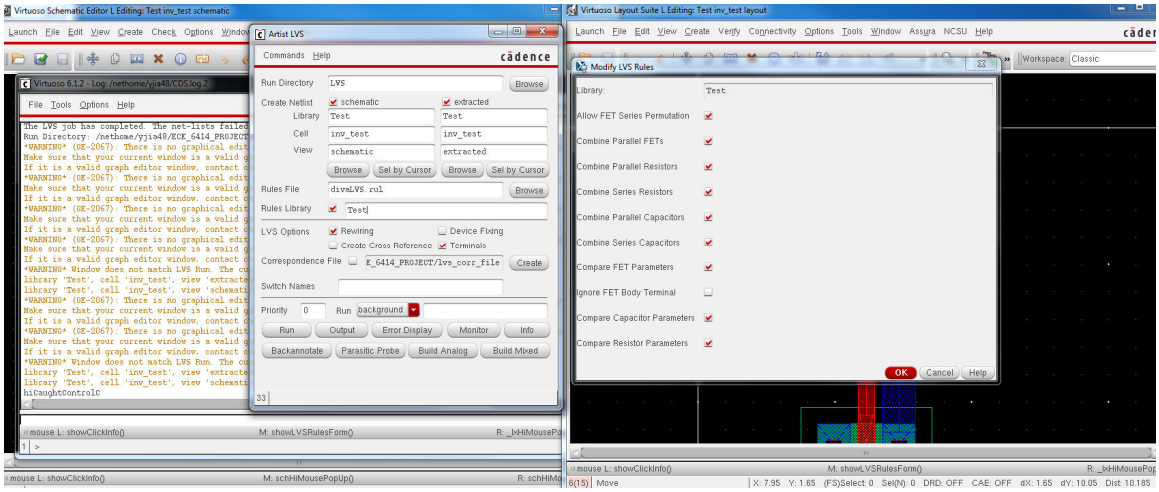
➔ **Then, you can see the extracted circuit in the cell view.**



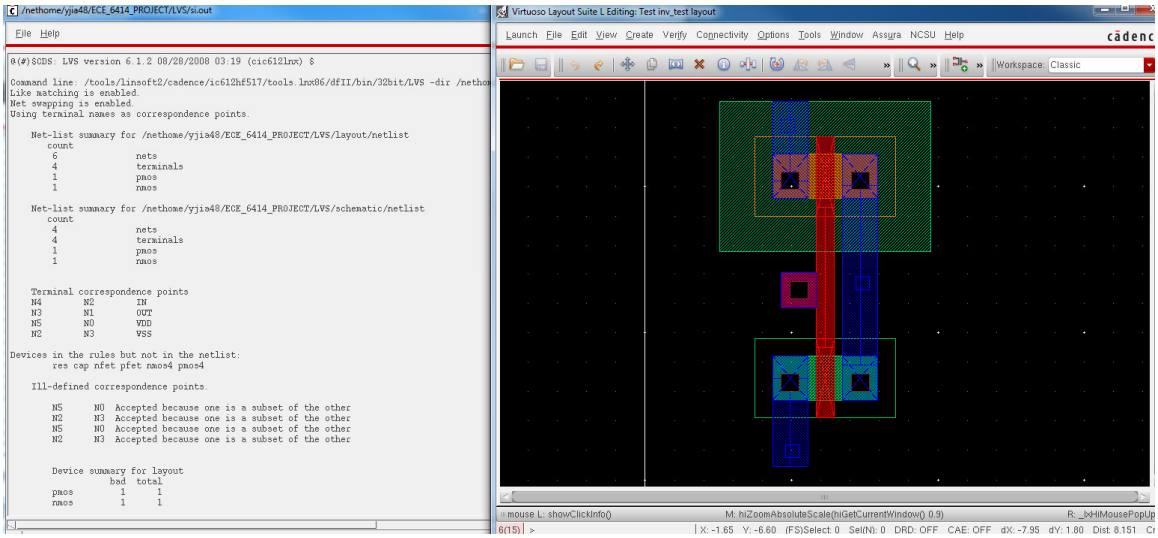
➔ **For LVS, click “Verify”->”LVS”, and select the extracted view.**
Left side should be your schematic, and right side should be its extracted circuit.
Rule file is located on “/tools/solsoft/solcad/ic610/ncsu-cdk-1.6.0.beta/techfile/divaLVS.rul”



➔ **Check LVS rule is correct. You can see the modify LVS Rules in “NCSU”-> “Modify LVS Rules” to change the options.**
➔ **Run “LVS”, and click “Output” to see the error reports.**



< When LVS has errors : ptap and ntap for body terminals are missing >



< When LVS has completed : Let-lists are matching in schematic and layout >