LAB4

VIRTUOSO

Outline

1. Virtuoso setup
2. Create Library
3. Schematic
4. Run Spectre simulation

1. Virtuoso setup

Step 1: Create and change directory to "*layout_env*" folder. In this Lab 4, we will work at this place:

```
%> cd /home/cc??group??/vlsi/${Student_ID}/work/layout_env
```

Step 2: Copy Sample Environment Kit from Cadence to each of your synthesis working folder

```
%> cp -rf /home/share_file/cadence/pdk/ pdk
```

1. Virtuoso setup

Step 3: Get license and start virtuoso with a executable file:

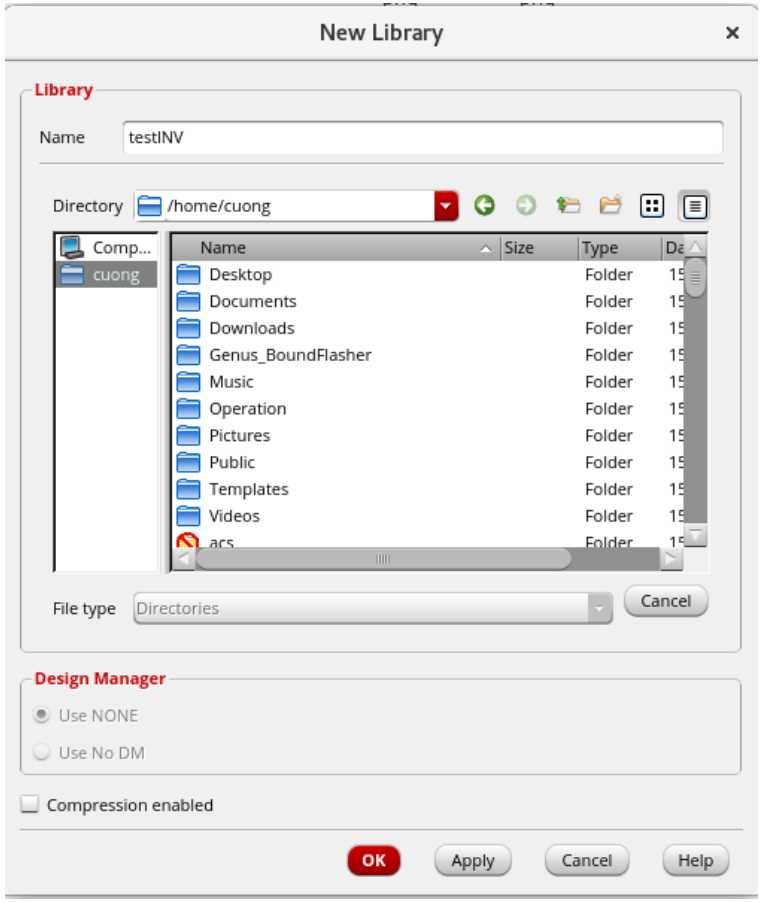
Create a file called “go_vir” with the content as follows:

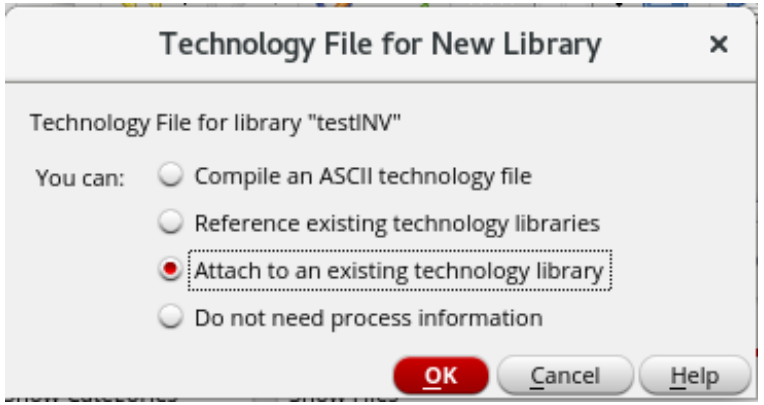
```
#!/bin/bash -f
cd pdk/gpdk045_v_6_0/
cd /home/share_file/cadence/
source add_path
source add_license
cd -
virtuoso &
```

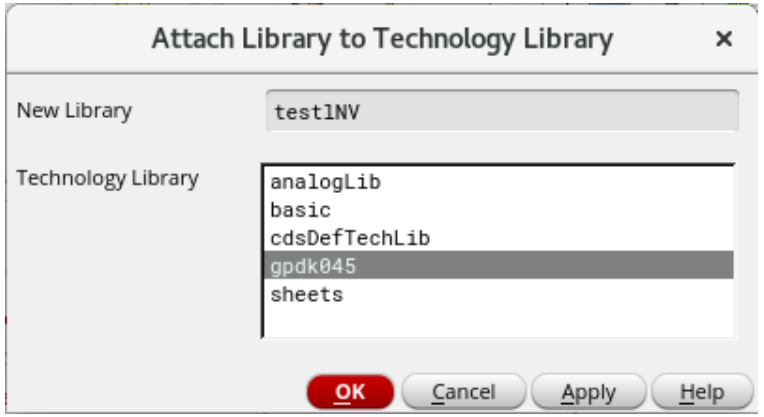
2. Create Library

<u>Step 1:</u> Open ***Tools → Library Manager***

<u>Step 2:</u> Select ***File → New → Library*** and enter a name for the library



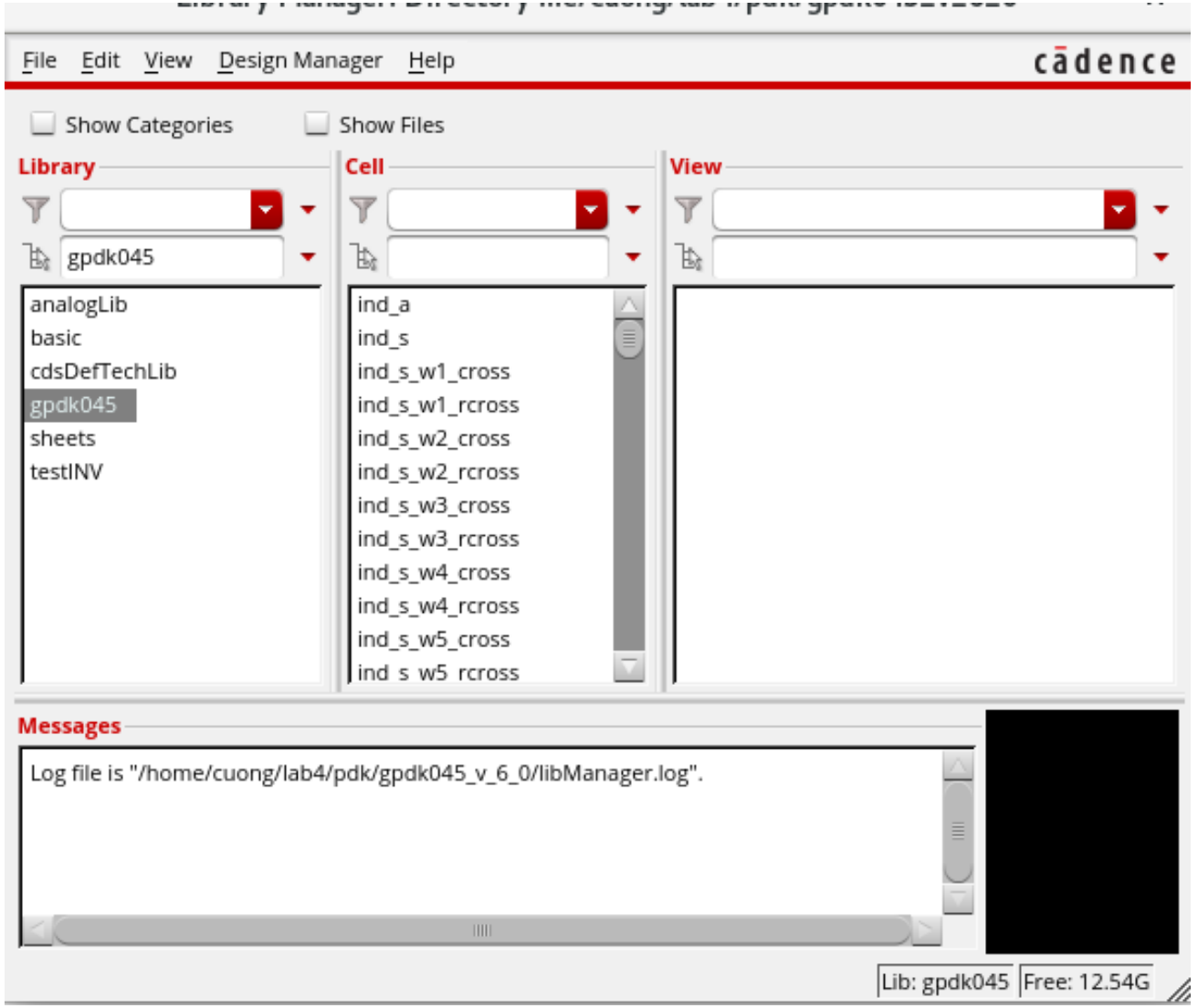




2. Create Library

Step 3: *Tools → Library Manager…*

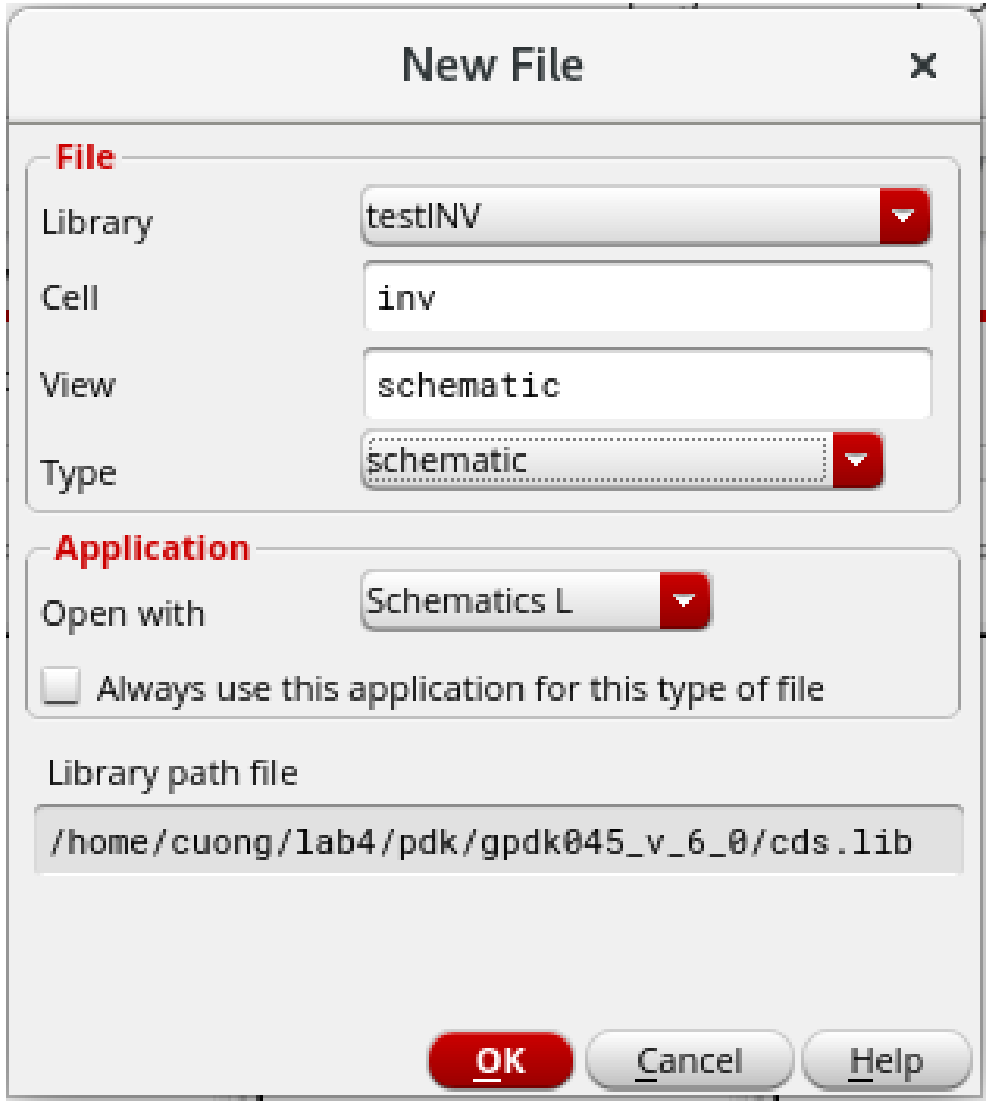
Here you can find all your libraries as well as the built-in libraries and their contents.



3. Schematic

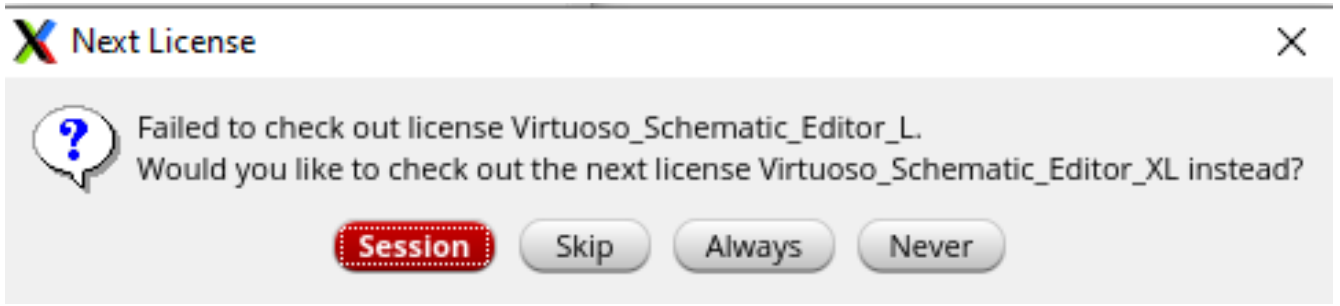
Step 1: Create a cell view: Select ***File → New → Cell View***

Name the cell “inv” and click OK.



3. Schematic

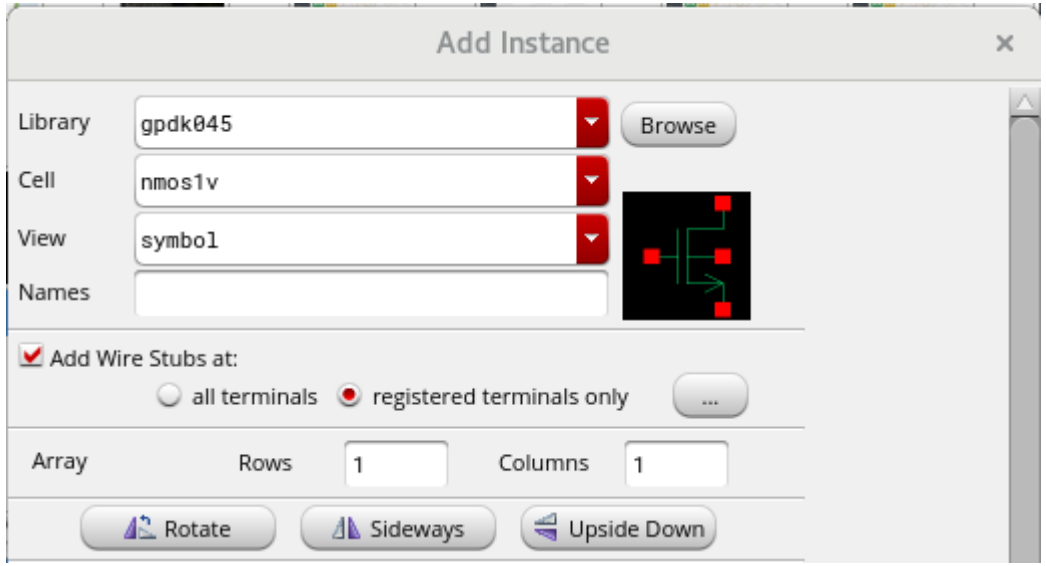
Step 1: If this pop up appears, choose **Always**



3. Schematic

Step 2: Draw a schematic:

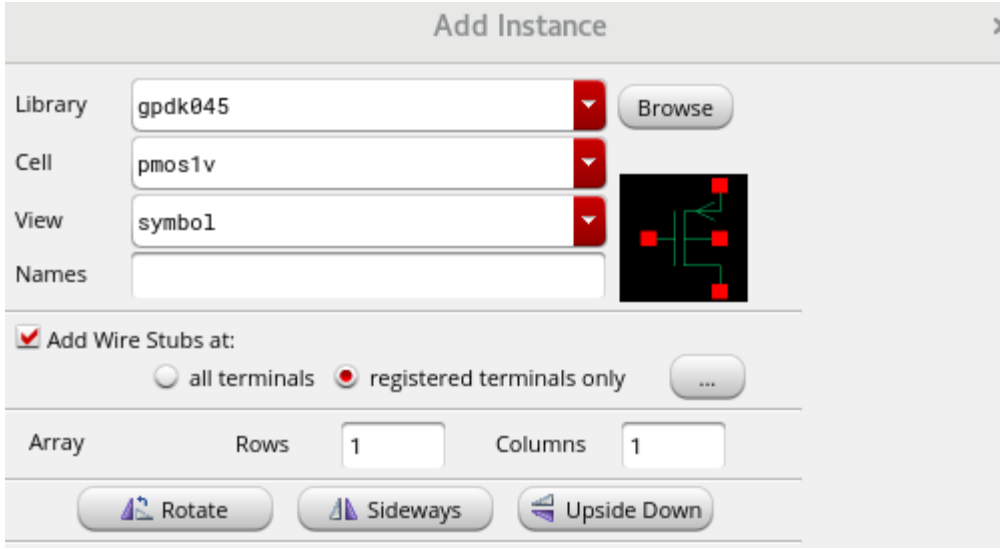
1. select ***Create → Instance (I)*** and choose the NMOS transistor



3. Schematic

Step 2: Draw a schematic:

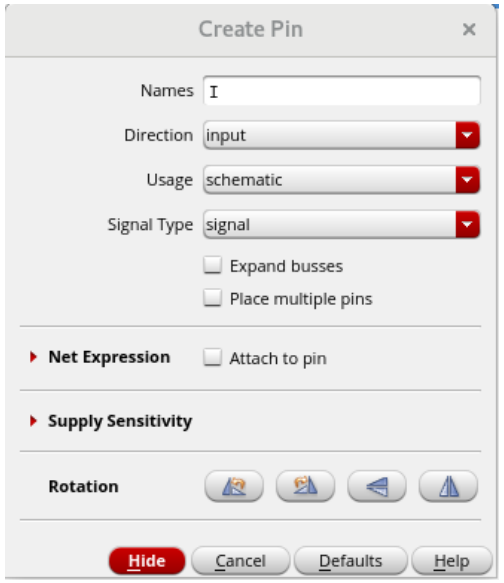
2. select ***Create → Instance (I)*** and choose the PMOS transistor

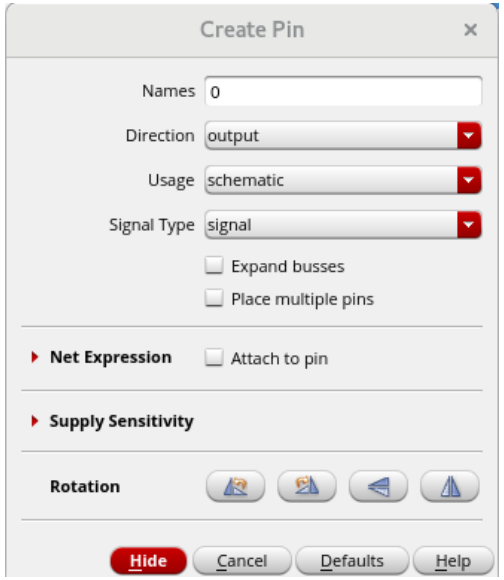


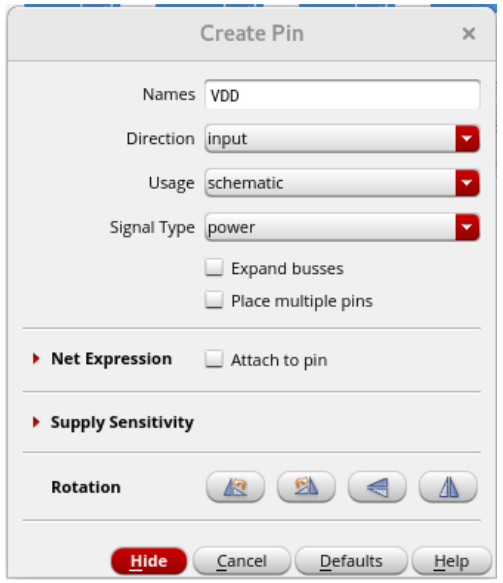
3. Schematic

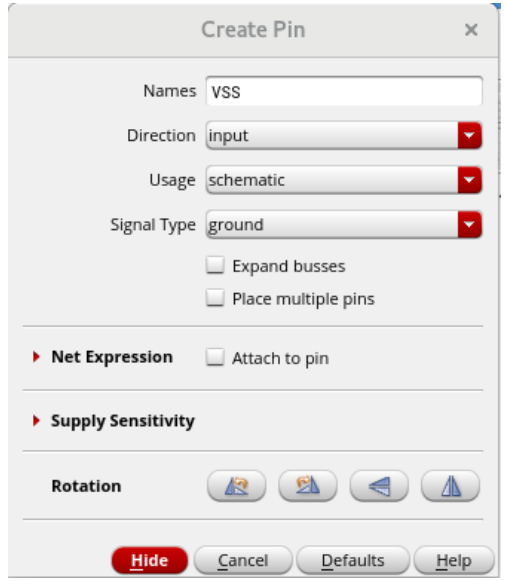
Step 2: Draw a schematic:

3. To add pin, Select ***Create → Pin. (P),*** then add **input, output, vdd** and **gnd**







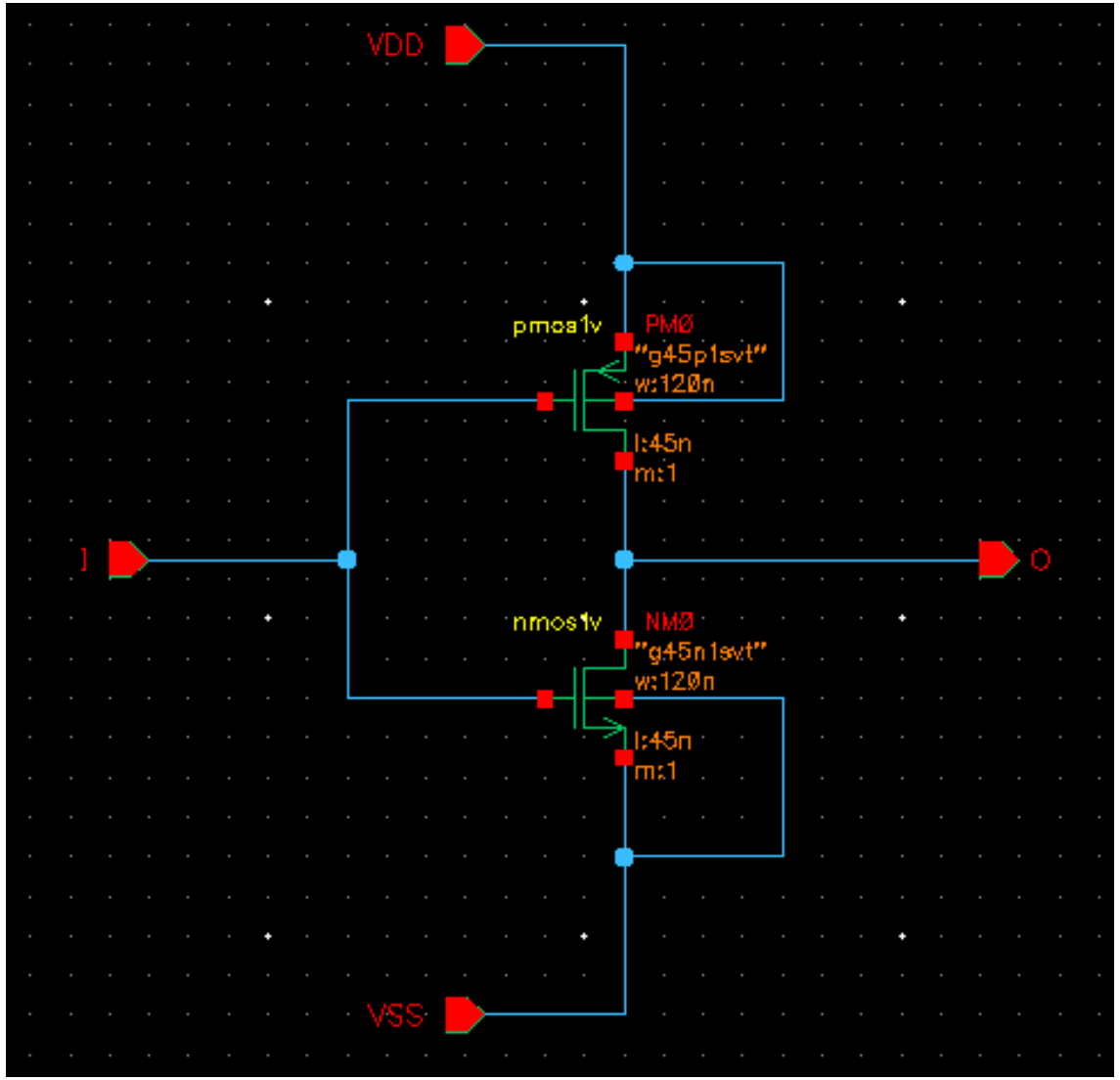


3. Schematic

Step 2: Draw a schematic:

4. Connect the components: Select ***Create → Wire. (W)***

Complete the circuit



3. Schematic

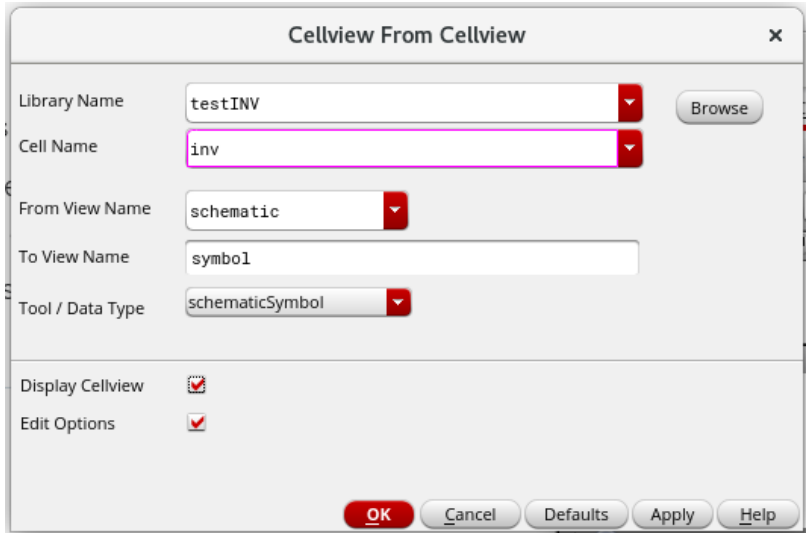
Step 2: Draw a schematic:

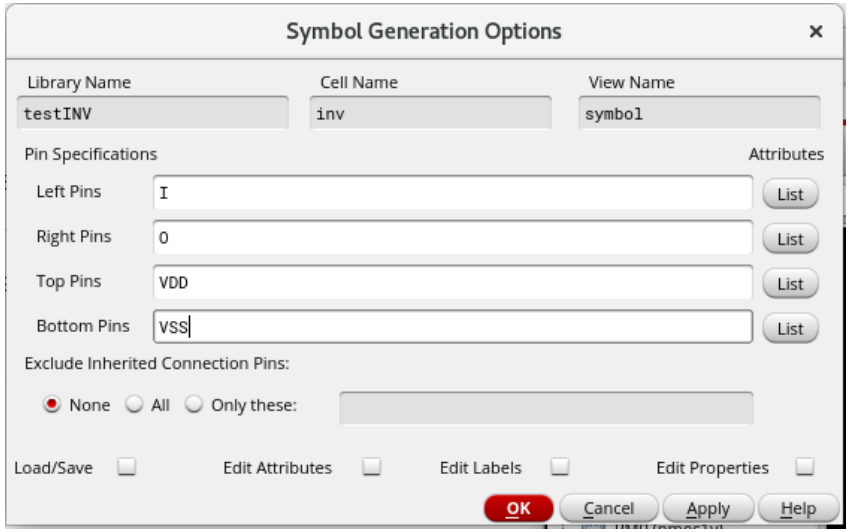
5. Check and save: Select ***File → Check and Save***.

3. Schematic

Step 3: In order to use your cell in a test bench, you first need to create a symbol that represents it.

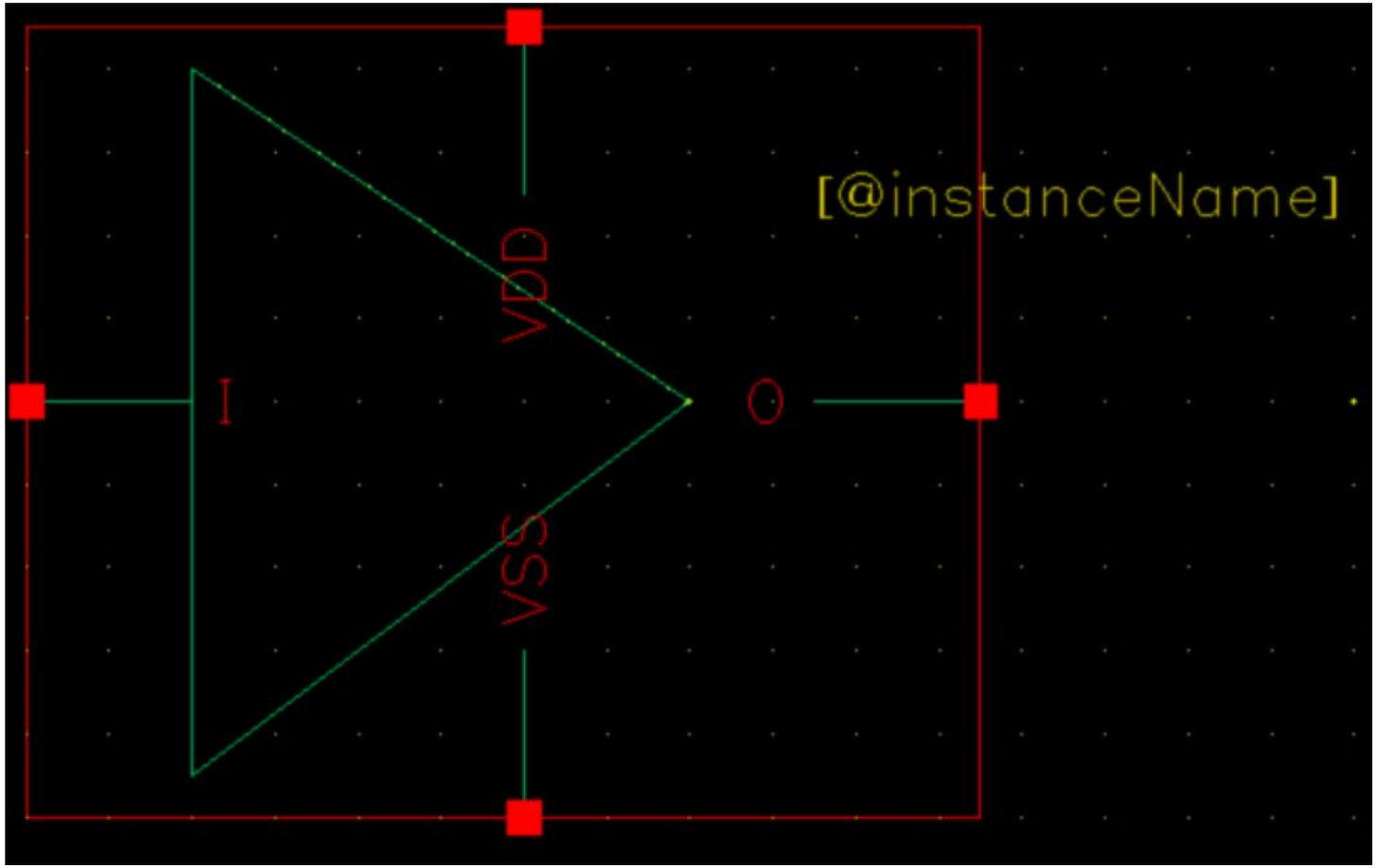
In order to do that, select **Create → Cellview → From Cellview**.





3. Schematic

Step 4: You can modify the symbol by simply delete things that you don't want, and add the shape by: (from the Symbol Editor) Create → Shape → Line/Rectangle/Polygon/Circle/ …

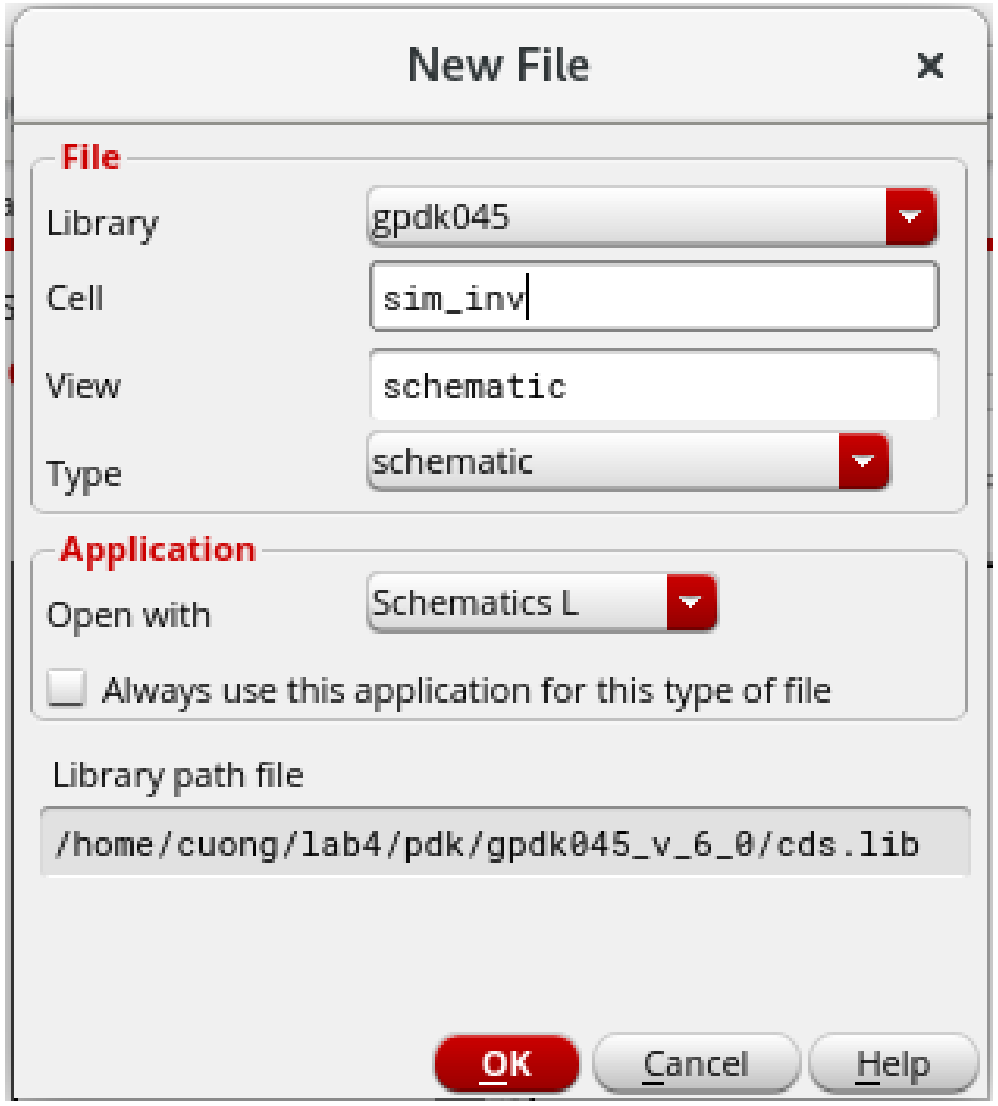


4. Run Spectre simulation

Step 1: To simulate the circuit, we create a new schematic: (from the Library Manager)

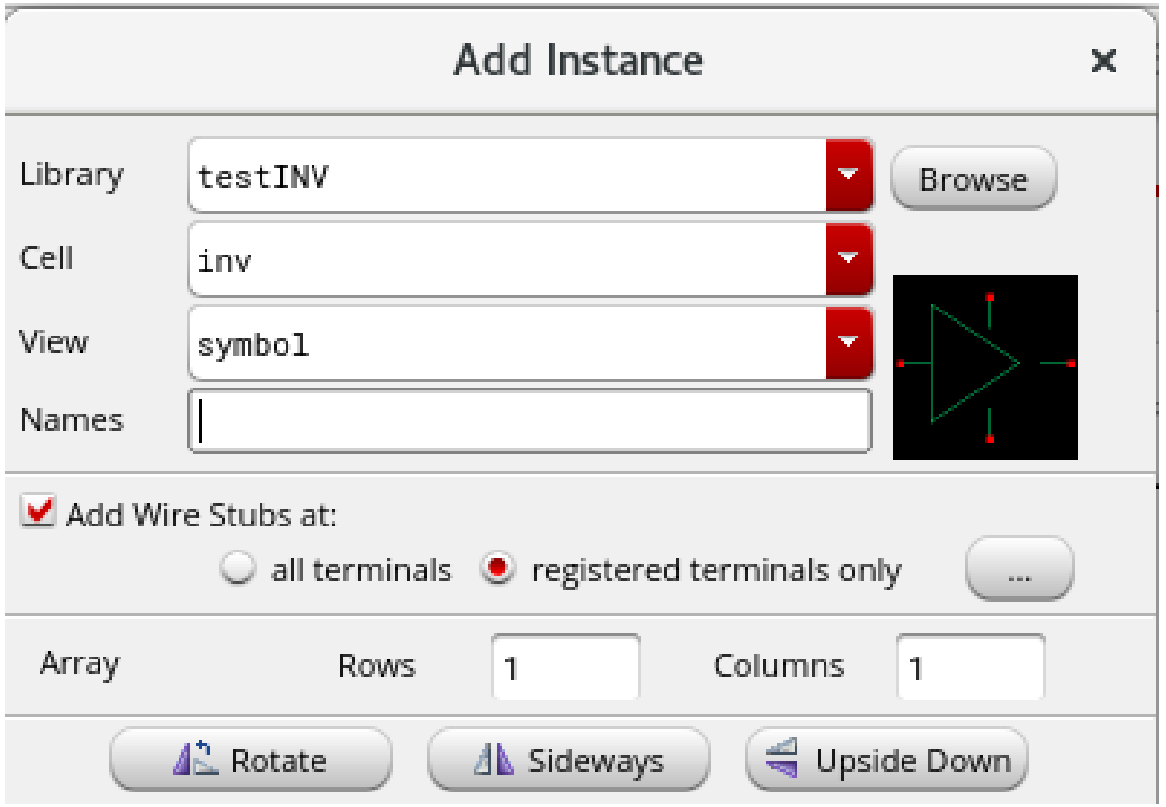
File →New → Cellview.

This time, we name it something like sim_inv:



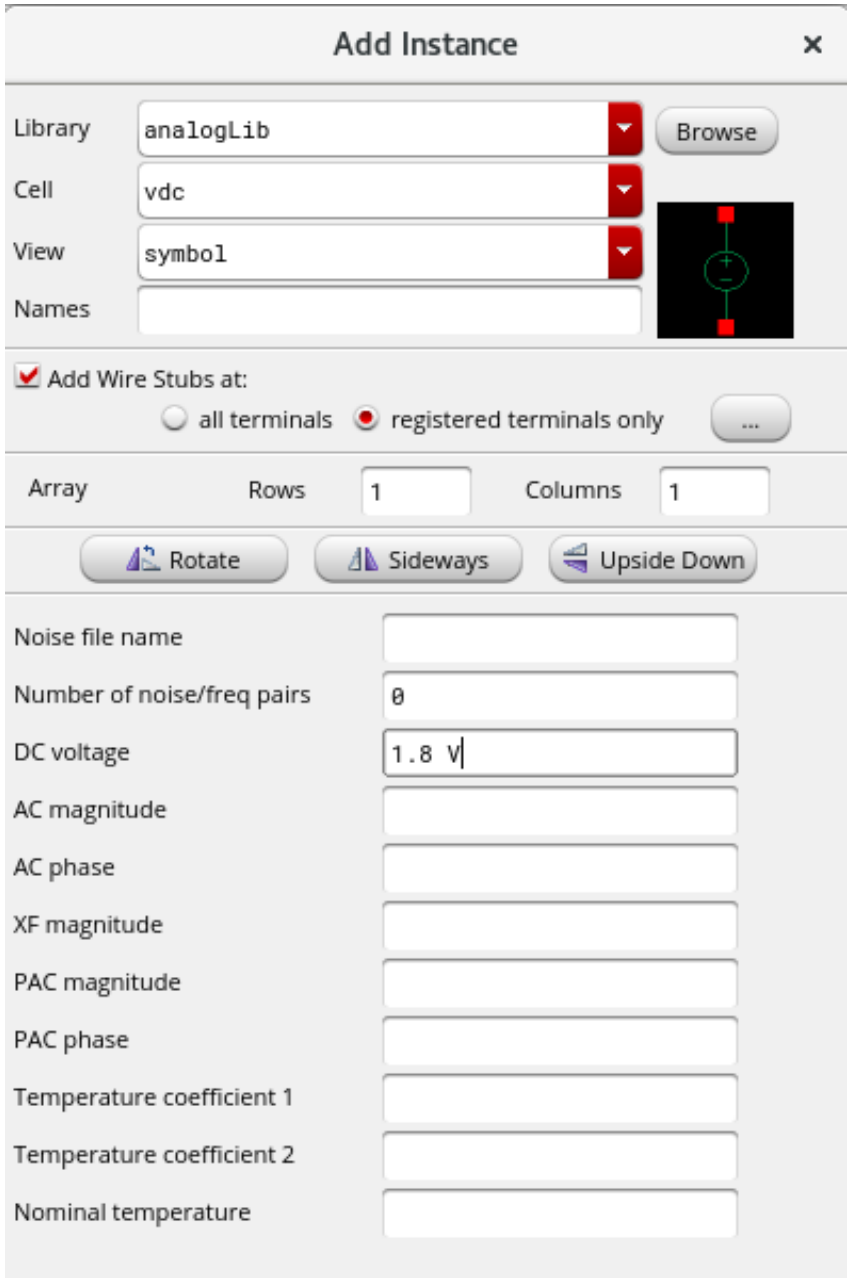
4. Run Spectre simulation

To add the cell inv: press I



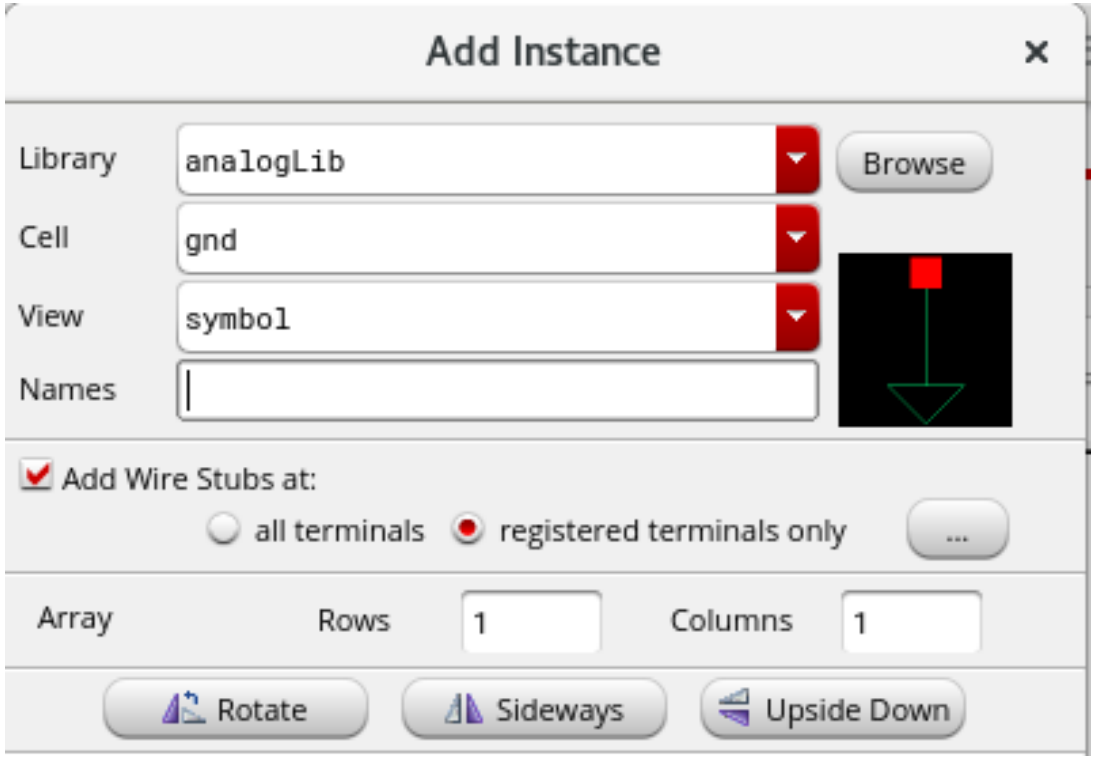
4. Run Spectre simulation

To add **VDD** and **In** voltages: press **I** then browse to **vdc** in the **analogLib** library, set **DC** voltage as **1.8**:



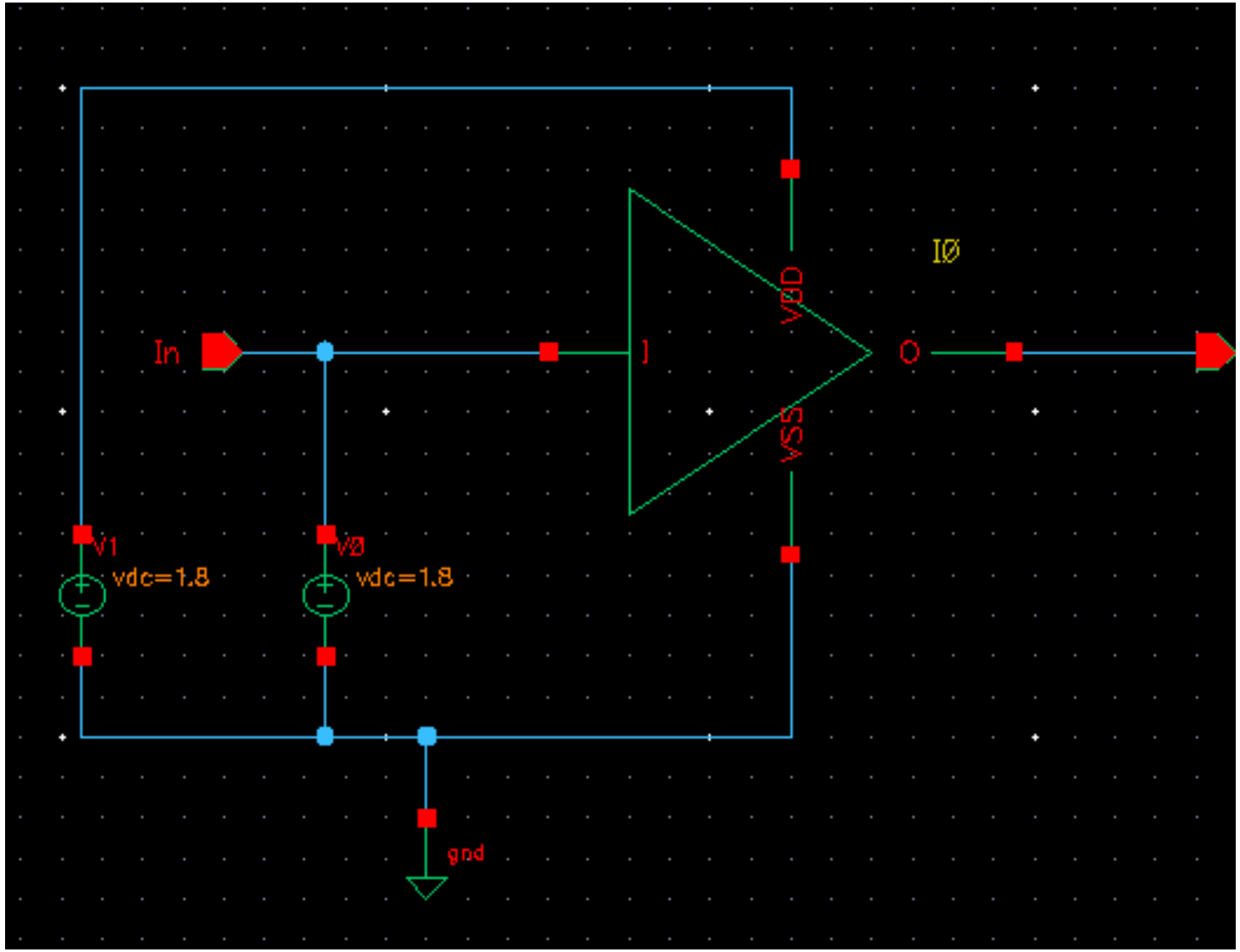
4. Run Spectre simulation

To add **VSS**: press **I** then browse to **gnd** in the **analogLib** library:



4. Run Spectre simulation

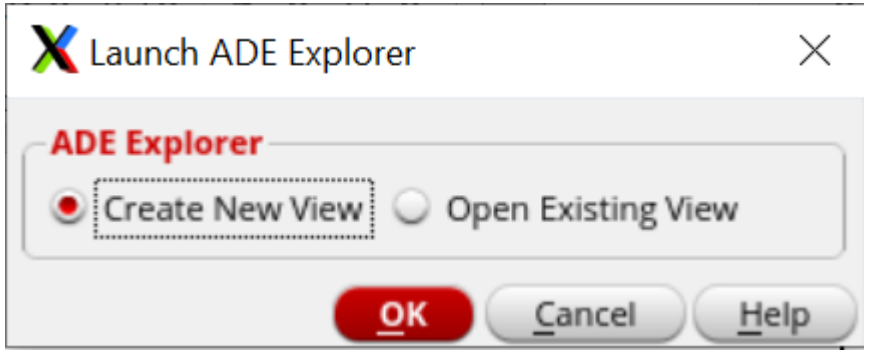
Complete the test circuit.

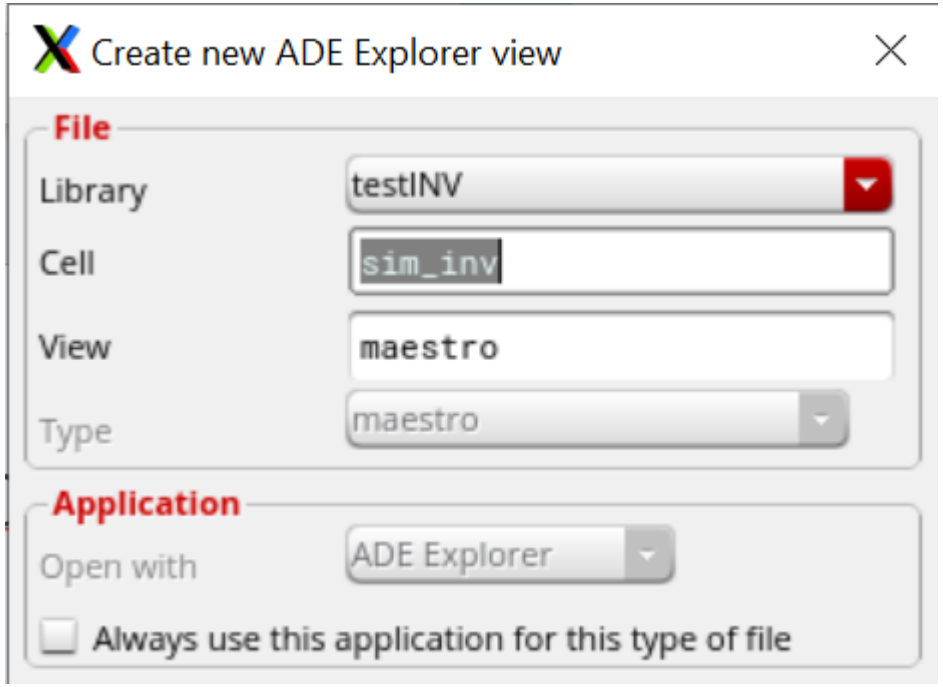


4. Run Spectre simulation

Step 2: Now we will use the tool **ADE Explorer** to run the analyses.

1. In Schematic Editor, select **Launch → ADE Explorer**.



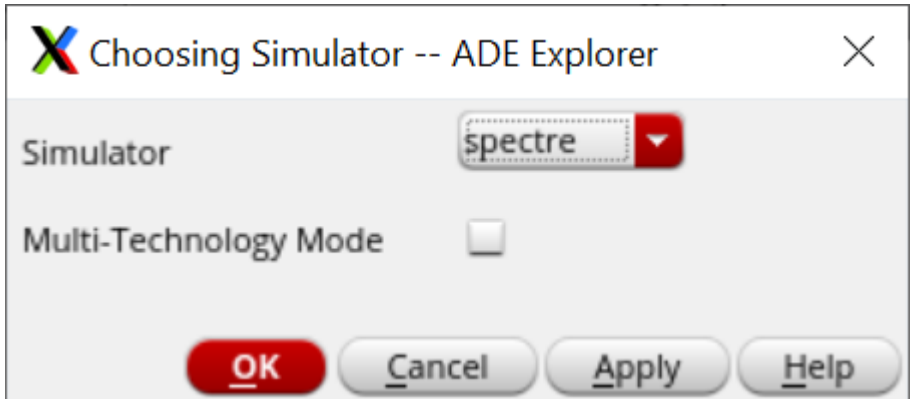


4. Run Spectre simulation

2. From the ADE Explorer,

Setup → Simulator,

then select *spectre*



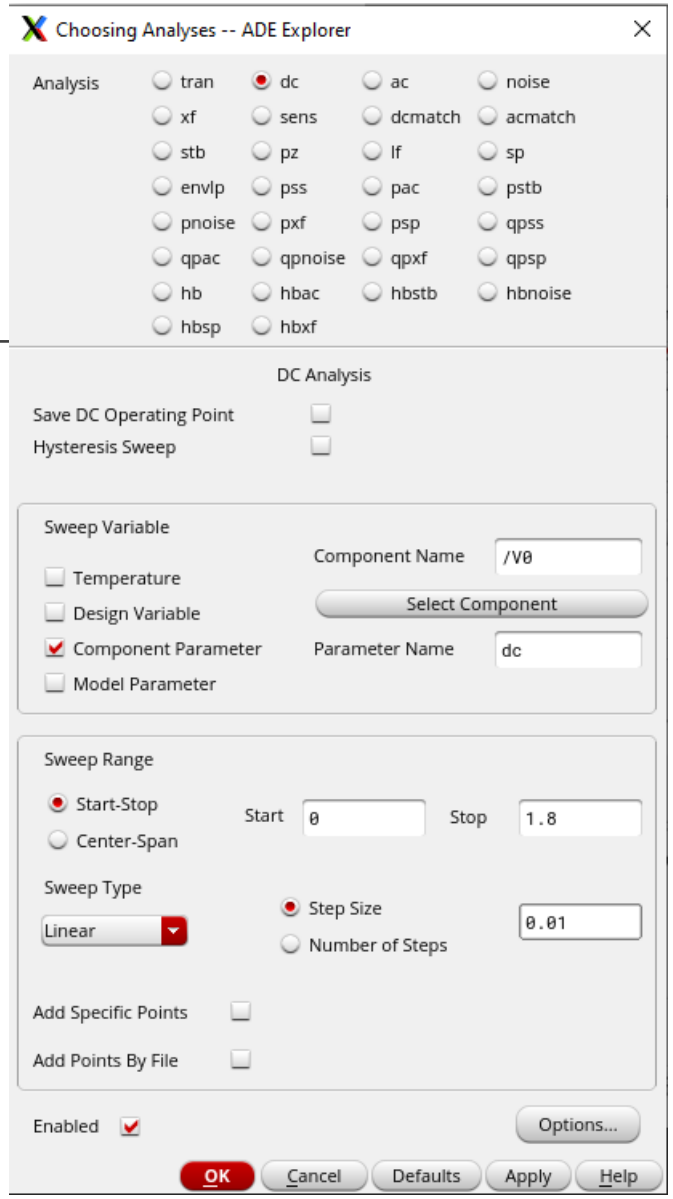
4. Run Spectre simulation

4. To run a simulation, we need to choose the type of analysis. Select **Analyses → Choose...**

Select as below

Select **Select Component** and click on V0 vdc on schematic.

The **Start** point should be 0, the **Stop** should be 1.8, the **Step Size** should be **0.01**

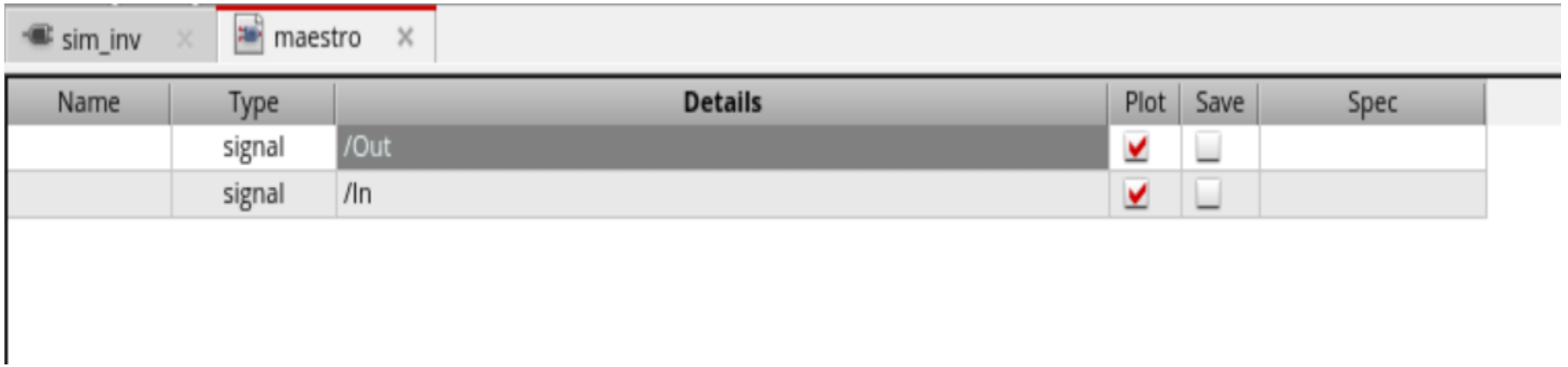


4. Run Spectre simulation

5. Return back to **maestro** view

Select **Outputs → To Be Plotted → Select on Design**.

Point to the **In** and **Out** pins on schematic



4. Run Spectre simulation

6. To run the simulation, select **Simulation → Netlist and Run**

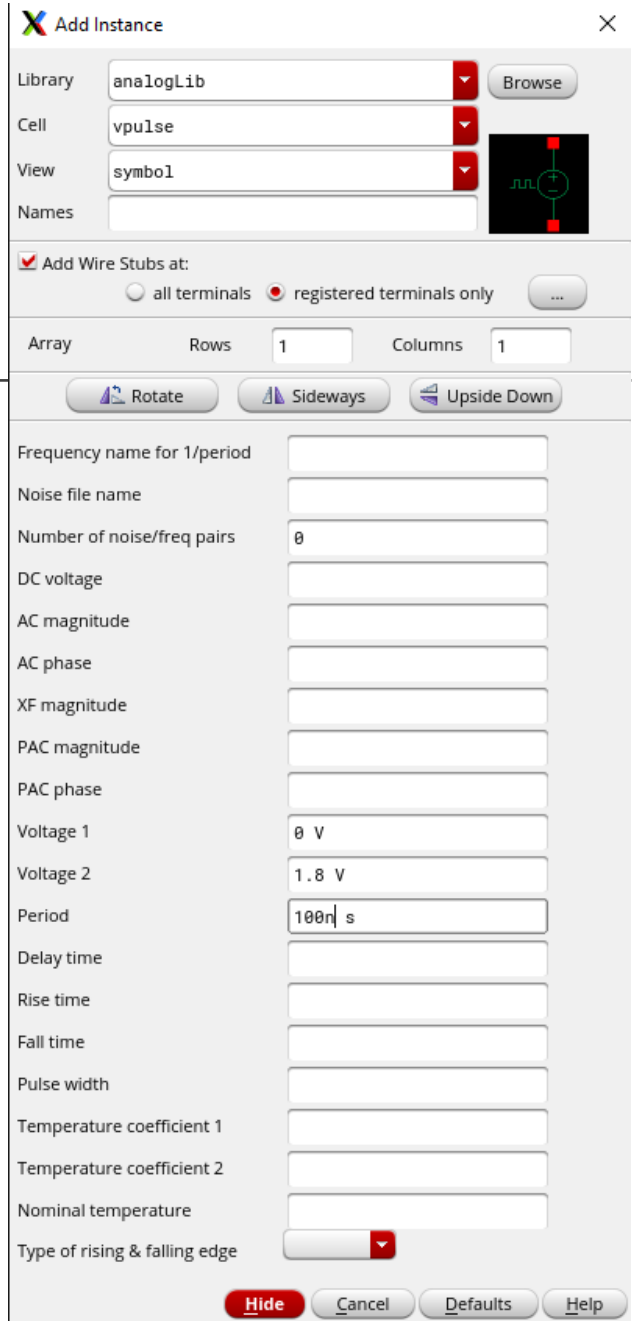
4. Run Spectre simulation

In case you want to simulate multiple inputs:

Step 1: Change the simulation schematic's **vdc** components connecting to the inputs into **vpulse**

Step 2: Set up the **vpulse** component

Example: The image is showing a **vpulse** component moving from 0-1.8V with period 100ns

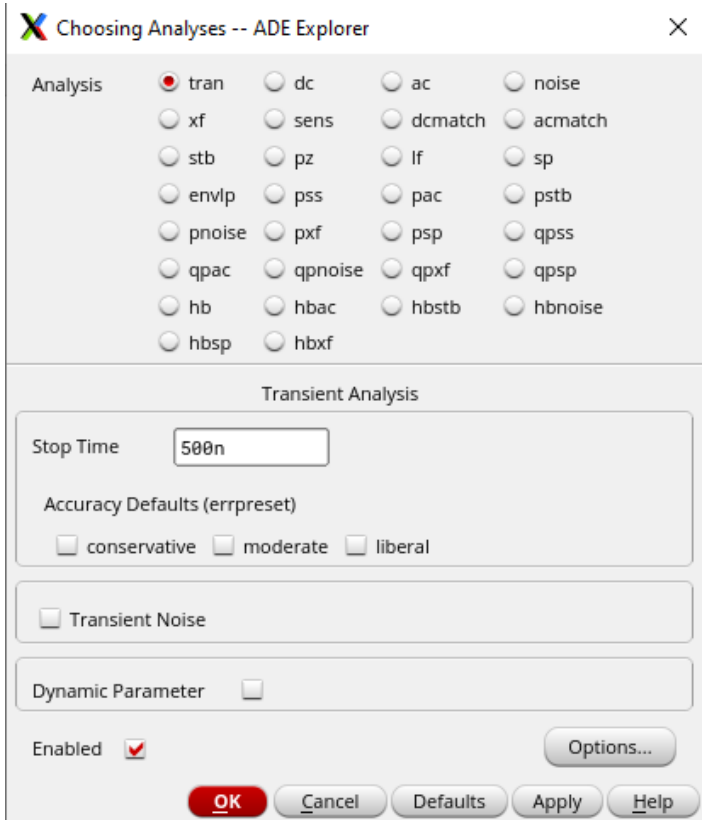


4. Run Spectre simulation

In case you want to simulate multiple inputs:

Step 3: Instead of using the **dc** analysis mode, use **tran** for multiple inputs

Example: the image is showing a **tran** analysis mode running in 500ns



5. Submission

This slide just showed an example of an Inverter

Your exercise is to do the same with And gate and Or gate

Each individual submit a report with schematics and simulation of those gates.

There must be your comments showing your own understanding of the schematics and simulation result you submitted in your report