Cadence setup

1. Connect to the compute node olympus.ece.tamu.edu
   - Compute nodes on Olympus server https://tamuengr.atlassian.net/wiki/spaces/helpdesk/pages/2164785158/Olympus+Academic+User+Information
2. Go to home directory and create course work folder
3. Go to work folder and copy model, cds.lib, nscu

   cp -rf /mnt/lab_files/ECEN689_605/model .

   cp -rf /mnt/lab_files/ECEN689_605/cds.lib .

   cp -rf /mnt/lab_files/ECEN689_605/ncsu .
4. Run Cadence

   ./ncsu

- Cadence setup on Olympus

```
$ hostname
olympus.ece.tamu.edu
$ load-ecen-620
$ cd ~
$ mkdir ECEN620_example
$ cd ~/ECEN620_example
$ cp -rf /mnt/lab_files/ECEN689_605/model .
$ cp -rf /mnt/lab_files/ECEN689_605/cds.lib .
$ cp -rf /mnt/lab_files/ECEN689_605/ncsu .
$ ls
cds.lib  model  ncsu
$ pwd
/home/grads/l/liutong[redacted]/ECEN620_example
$ ./ncsu
```

Load compute node (620/720)
Go to your home directory
Create your work directory
Go to your work directory

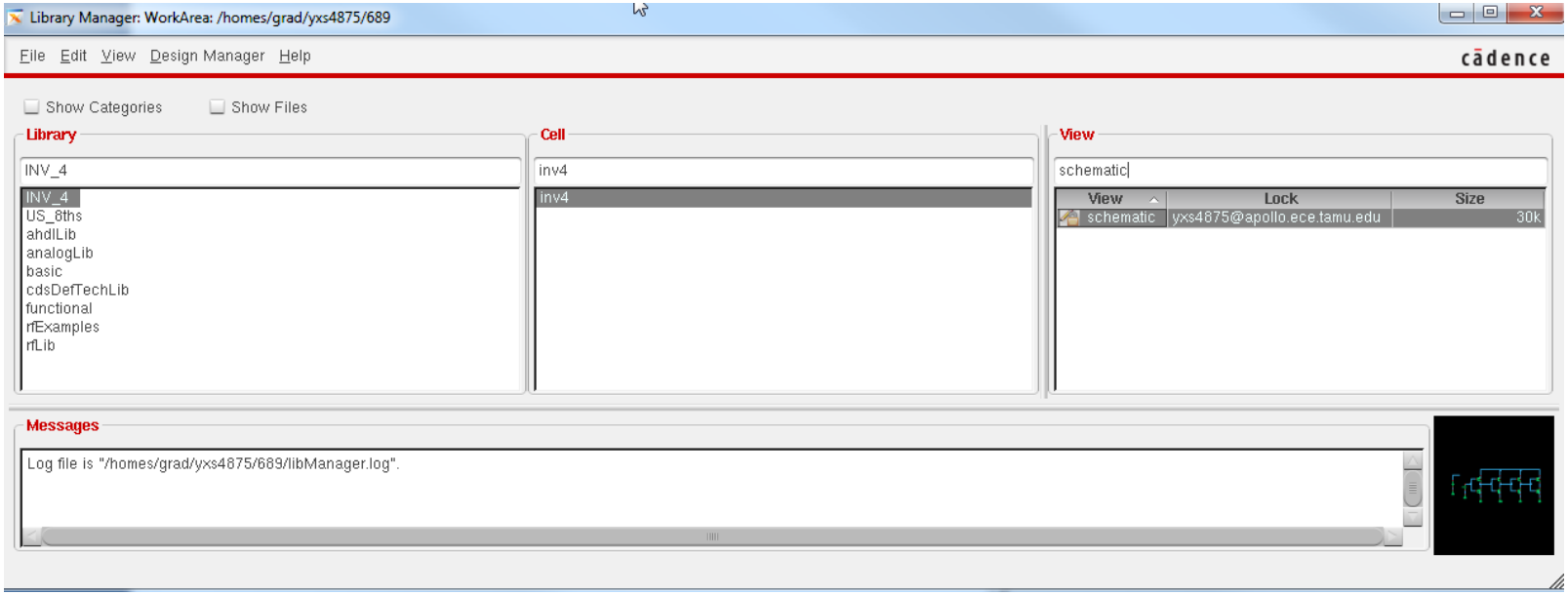
Copy files to your work folder

Check if you have all the files under your work folder

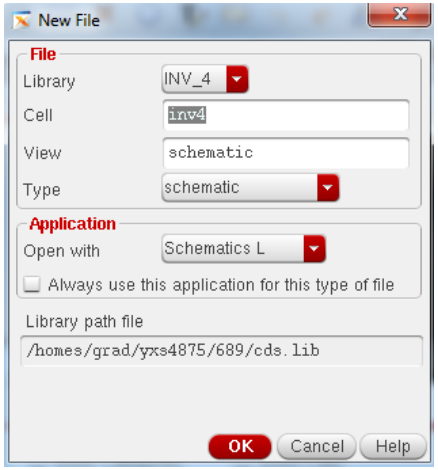
Start Cadence

Creating a Library

1. From the CIW select Tools → Library Manager to load the Library Manager



2. Do not Attach to an existing techfile due to using IBM90nm Model which is not support by cadence.

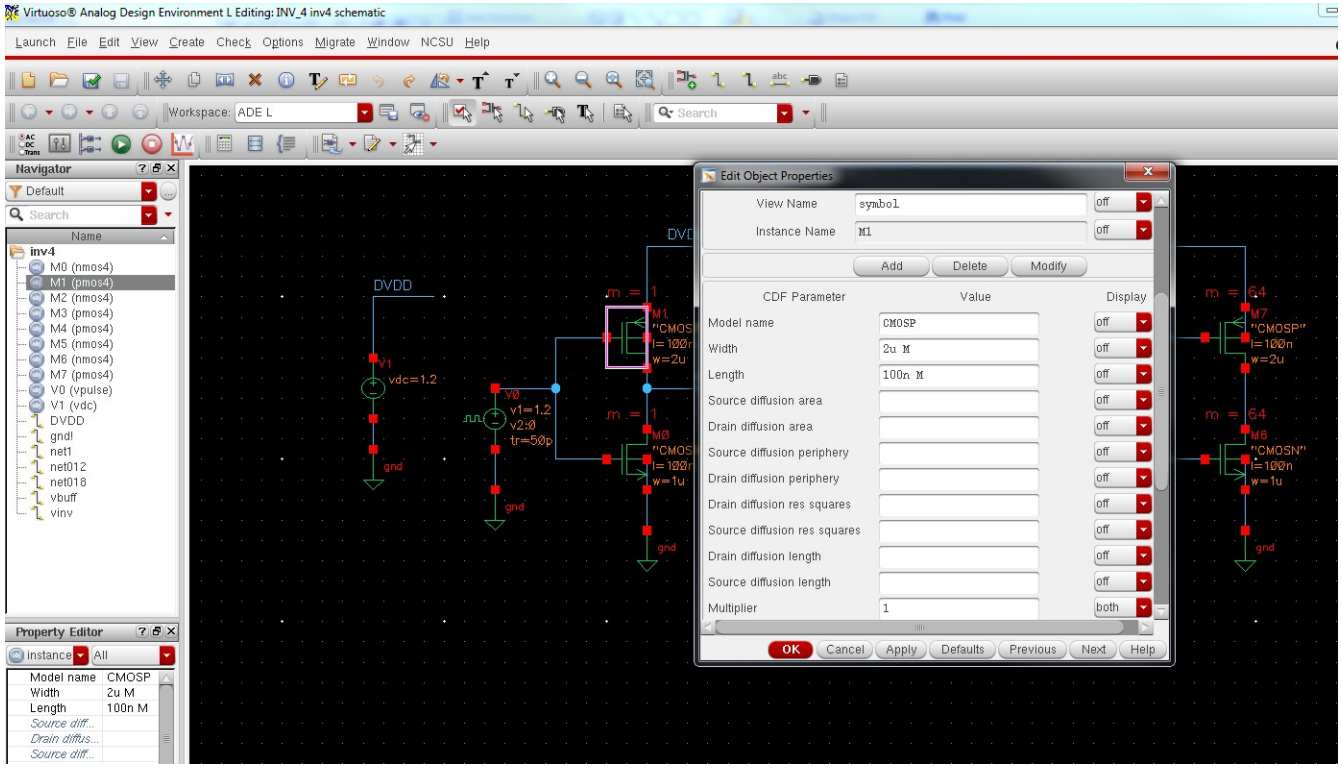


Creating a Schematic

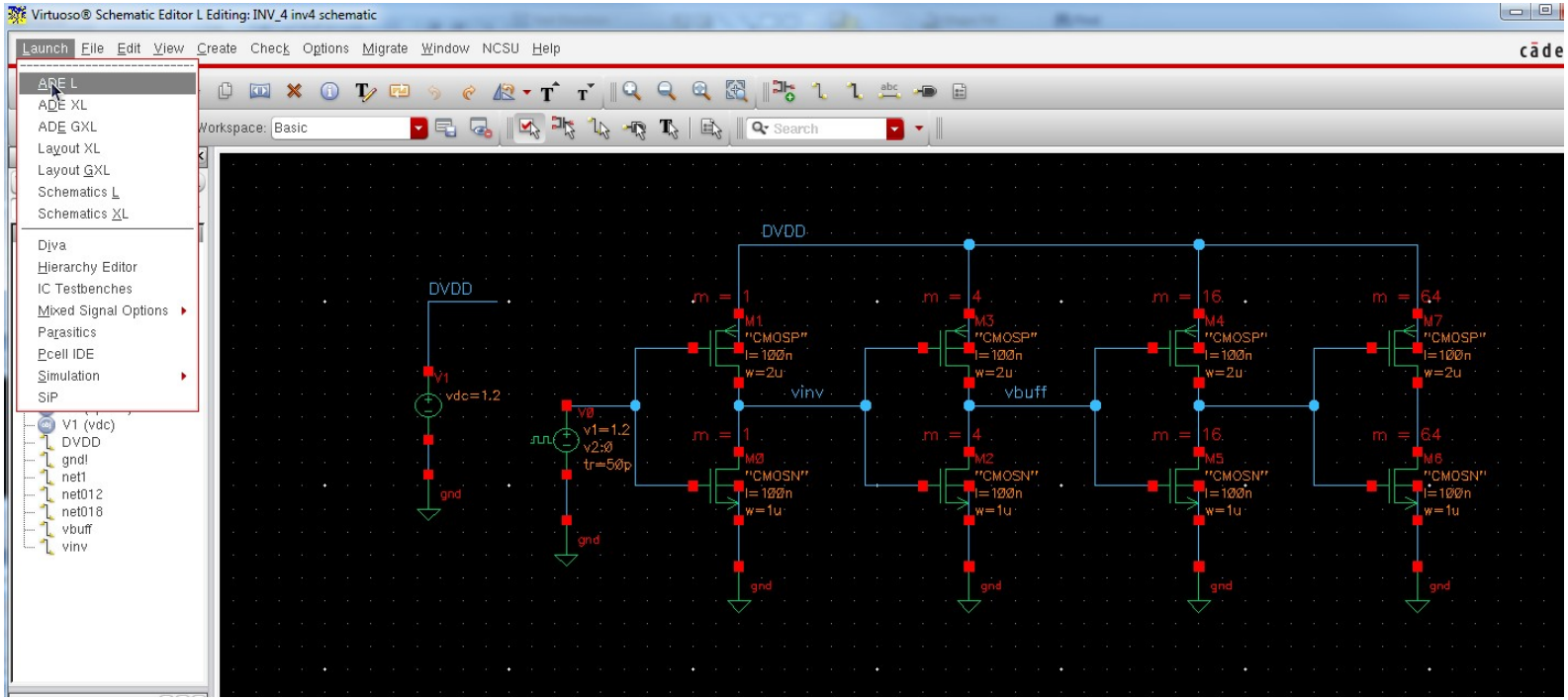
The first circuit we will design is a simple inverter. Select which library you want to put the cell into, in this case “INV4”, and then File → New → Cell. Name your cell inverter. The tool you want to use here is Composer-Schematic

After selecting OK, the schematic window opens. We wish to add two transistors so that we can make an inverter. To do this we need to add an instance. You can do this by either clicking Add → Instance or by pressing “i” on the keyboard. A window titled “Component Browser” should pop up. Make sure that the library analogLib is selected. Select N_Transistors and then nmos4. Go back to the schematic and select where you would like to add the NMOS transistor. Go back to the Component Browser and select P_transistors and then pmos4. Add this transistor to your schematic. Hit ESC to exit the Add Instance mode. Connect components together using wires. You can select Add → Wire or use the “w” hotkey. To change the properties of a device use Edit → Properties → Objects or use the “q” hotkey.

MINIMUM W/L - 0.12um/0.10um



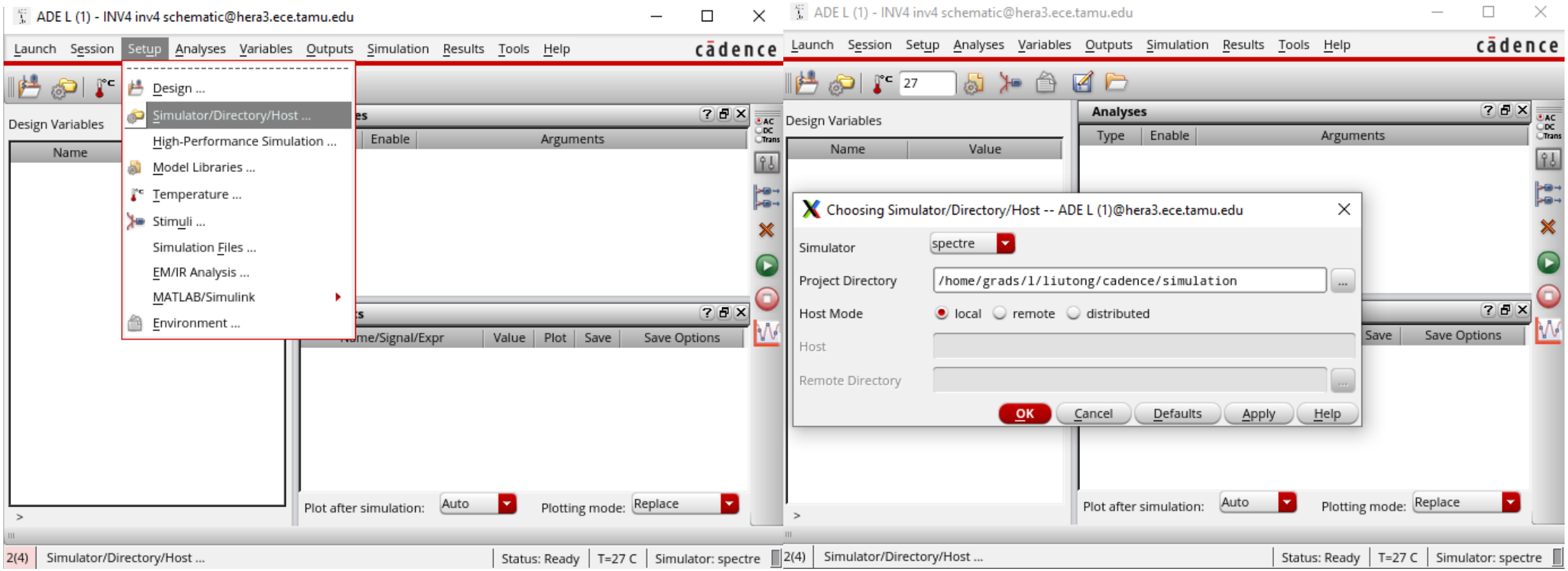
When finished, your schematic should resemble shown Figure to measure FO4 Delay. Select Design → Check and Save to save your schematic and make sure that there are no errors or warnings.



Simulating the Schematic

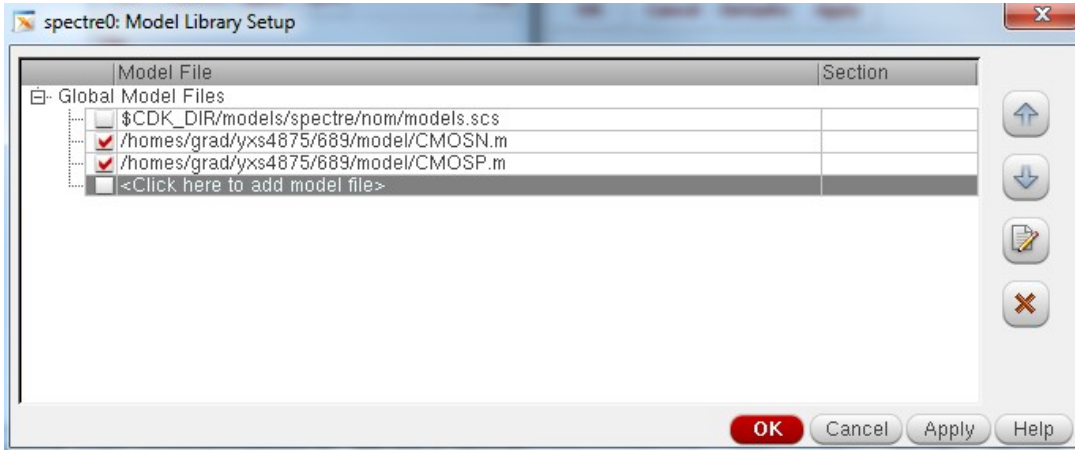
Start the simulator environment by selecting Launch → ADE L

Simulator



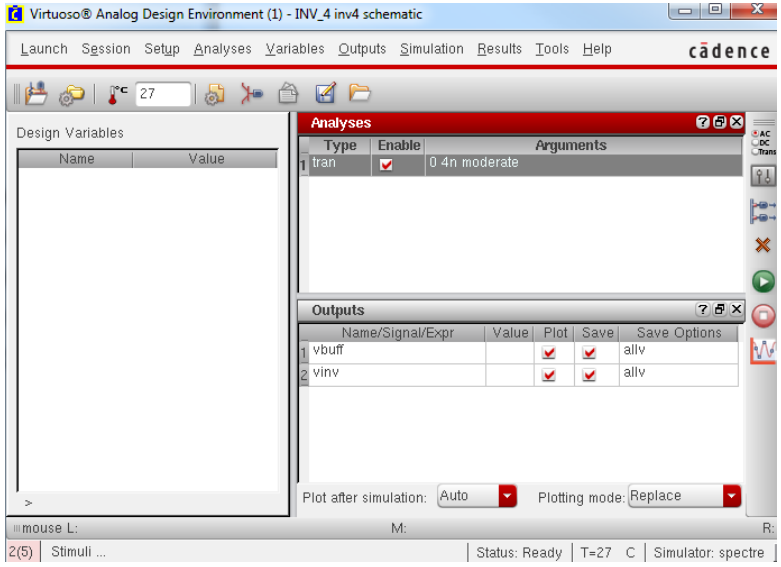
- Change simulator to “spectre”

Select Setup → Model path and add CMOSN.m and CMOSP.m



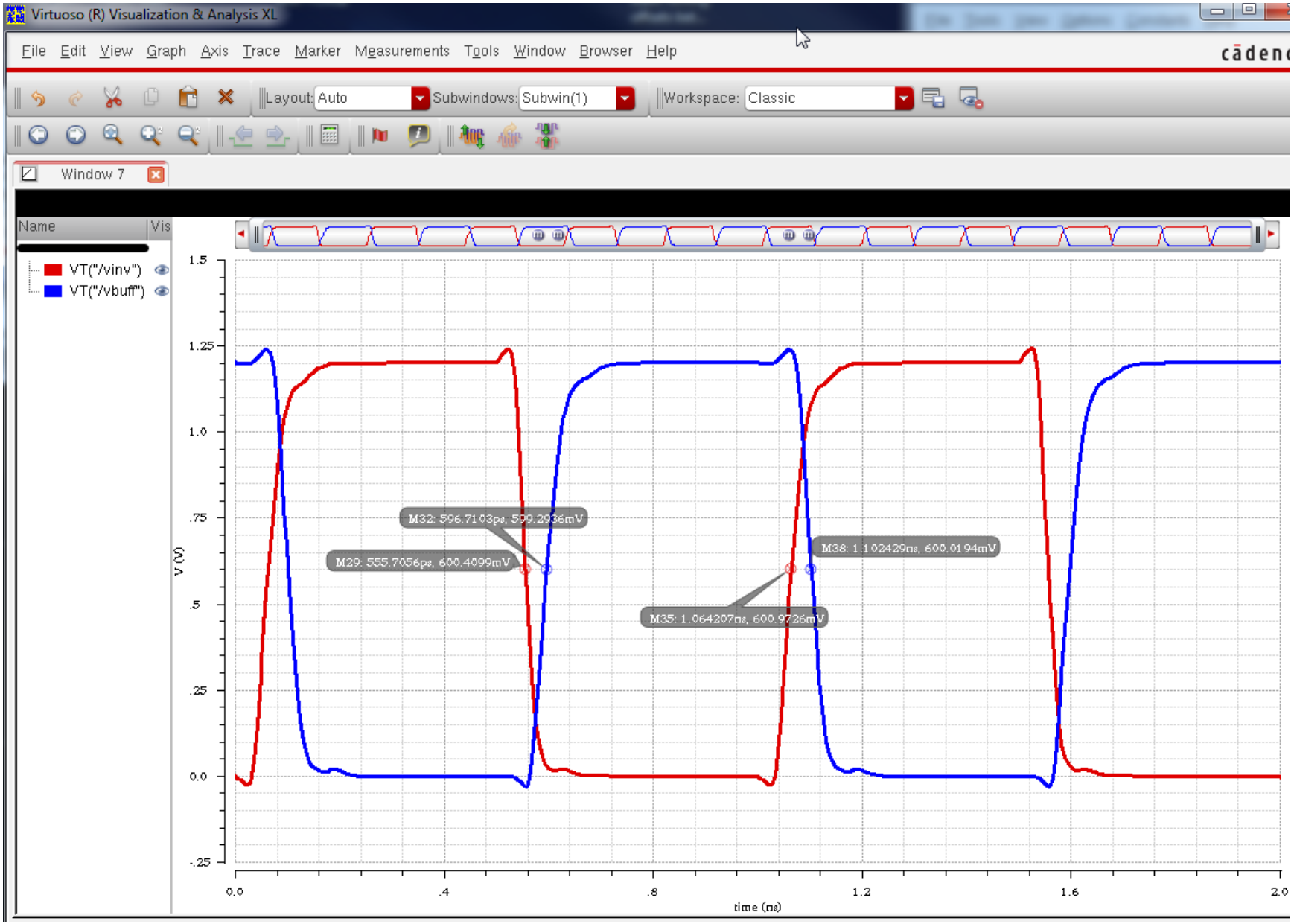
Next we need to configure the environment to run our first simulation. In the Analog Environment window select Analyses → Choose. Select “tran”

Select Simulation → Run or click on the green light in the bottom right corner. Once the simulation has completed, we can plot any outputs that we wish. To do this we use the calculator. To access the calculator, select Tools → Calculator in the Analog Environment.



Simulation Result

FO4 delay for IBM90nm => 40ps



How to use PRBS generator in Cadence

PRBS generator can be found in ahdlLib. It is called rand_bit_stream. Please specify a PRBS generator as shown in Figure 1. Please set seed to 128 for 7 bit PRBS.

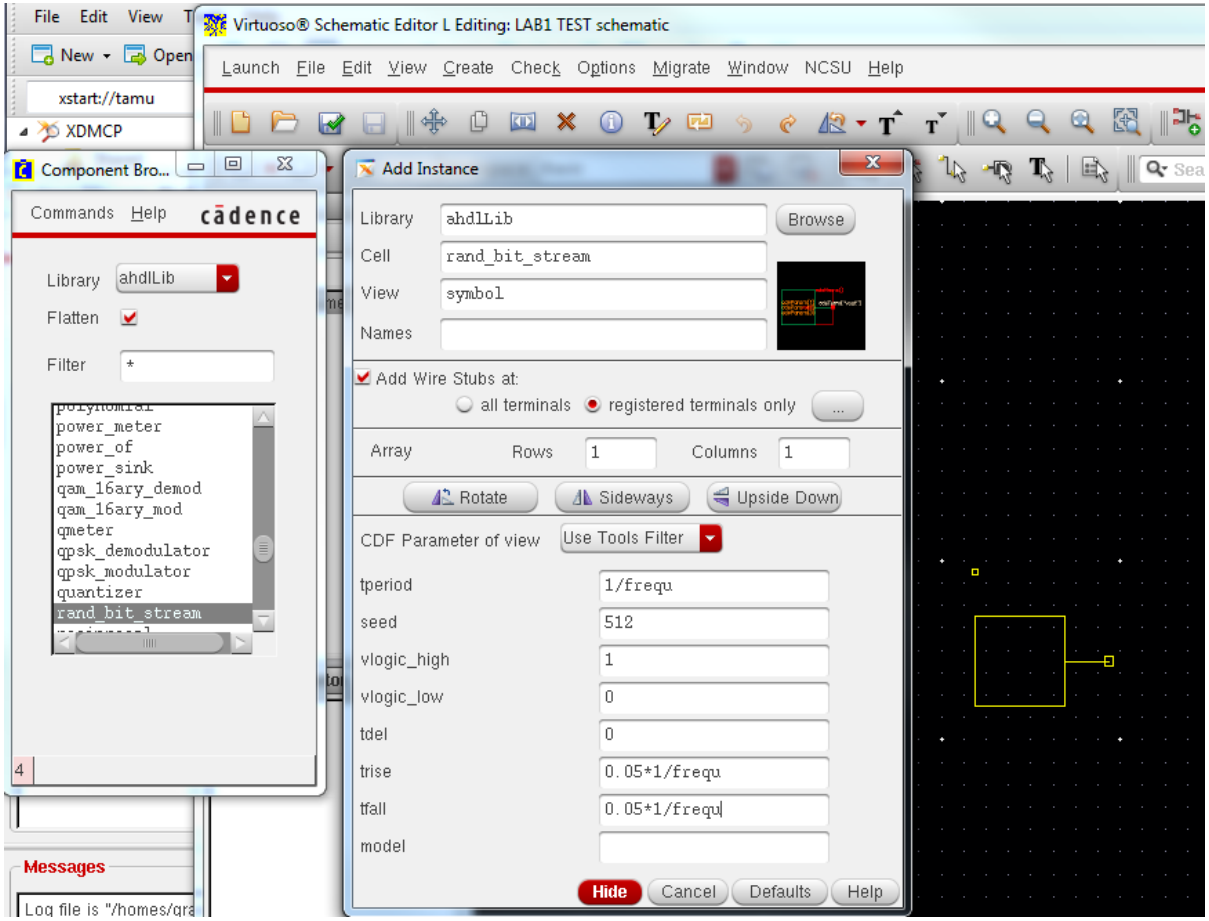
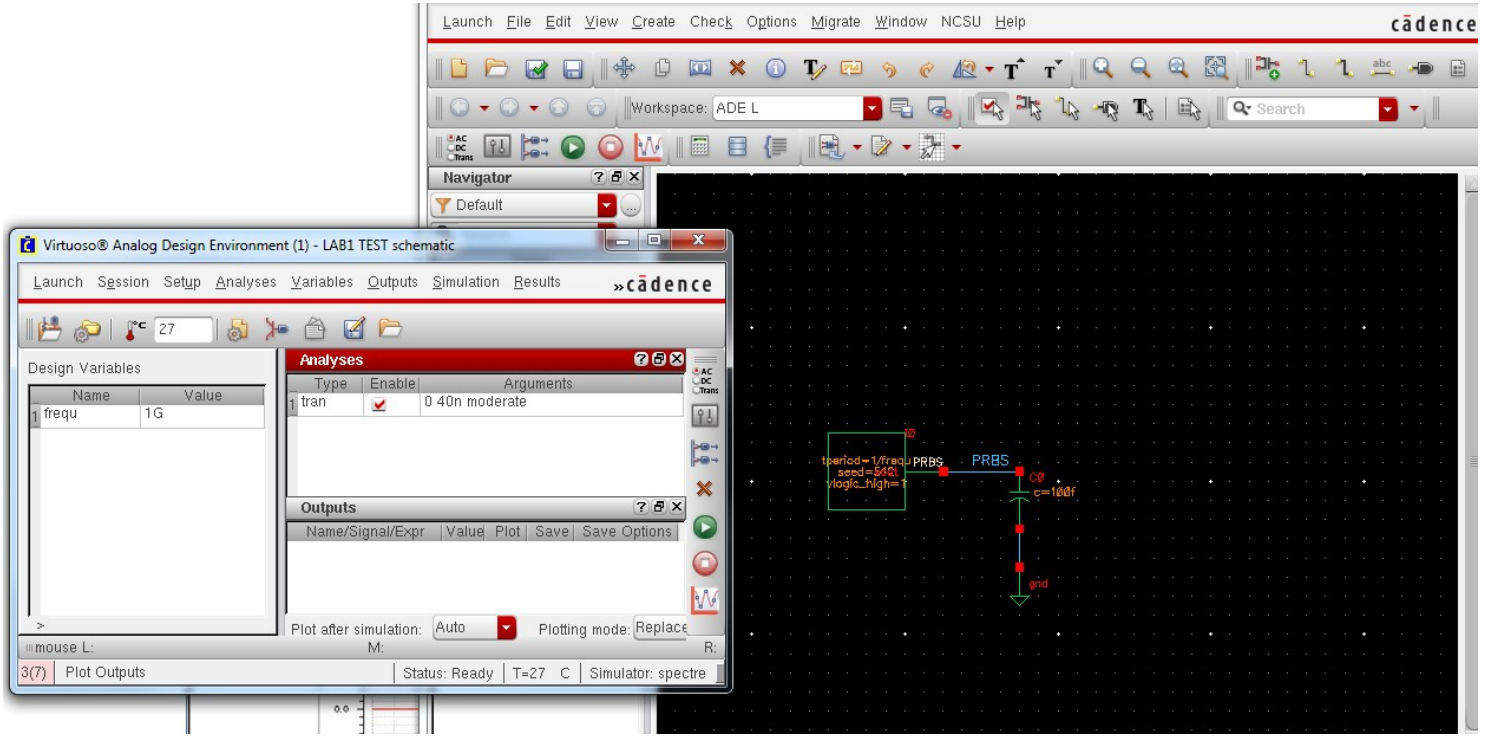


Figure 1 PRBS Generator Property

Cadence Setup



Simulation Result