ECE 249 Lab 1

Spring 2005

Adapted from

"Short Tutorial on Cadence", Z. Tao and M. Keramat, UCSN-0300-ZBT-03, March 2000
"Symbol Creation of Sub-Circuits in Cadence," Z. Tao and M. Keramat, UCSN-0700-ZBT-10, July 2000

Cadence Software Fundamentals

1 Introduction

This lab aims at introducting you to the Cadence software on the Sun workstation. First, a list of basic UNIX commands is given, and then followed by setting the UNIX environment for the Cadence design tools. Finally, a simple example of simulating a CMOS Inverter is presented.

2 Basic Unix Commands

Some basic UNIX commands that you need to know are listed in Table I.

Table I. Basic Unix commands

List the files in the current directory:	
Ls	Displays the name of the files in the directory.
ls -l	Displays additional information (eg. permissions) about the files in the directory.
ls -a	Displays all files in the directory, including hidden files.
ls -al	Combines the previous two options.
Change (or move between) directories:	
cd <directory>	Moves to the directory specified by *directory*
cd ..	Moves up a directory - two periods represent the parent directory.
cd ~	Moves to your home directory – tilde represents your home directory.
Make a new directory:	
mkdir <directory>	Makes a new directory called *directory*
Display the (path of the) current directory you are in:	
pwd	Displays the name of the current directory you are in.
Copy a file or directory:	

cp <existing file> <newfile>	Creates a new file identical to *existingfile* but with the name *newfile*
cp -R <existing Dir> <newDir>	Creates a new directory identical to *existingDir* but with the name *newDir*
Rename or move a file:	
mv <oldfile> <newfile>	Renames *oldfile* to *newfile* within the same directory.
mv <oldfile> <directory/newfile>	Moves *oldfile* to a different *directory* with the name of *newfile*
Remove (delete) a file:	
rm <file>	Deletes *file*. The file cannot be recovered.
Change password:	
yppasswd	Change current password.

3 Setting up an Account to Run the Cadence Tools

You need to login a UNIX workstation and set up your UNIX environment to run the Cadence software. Depending on when your account was created, you may be using csh as your default login shell. The current recommended shell for ECS users is bash. To check if you are using, type "`echo $shell`" at the shell prompt. If the reply is "/bin/csh", you will need to change to use the bash shell by typing "`exec bash`" at the shell prompt. Once you have established that you are using bash, proceed with the following steps to setup the Cadence tools.

Step 1:> `mkdir cadence`

Step 2:> `mkdir cadence/models`

Step 3:> `mkdir inbox`

Step 4: The following command will overwrite your `.bashrc` file in your home directory. If you have something important in your `.bashrc` file, save it to a temporary copy.

```
> bash /apps/ecs-apps/software/ece/cdssetup/cdsupdate2005
```

Add in your `.bashrc` changes if necessary.

Step 5:> `source .bashrc`

4 Starting Cadence Software

Step 1: > cd ~/cadence (current directory must be cadence)

Step 2: > icfb &

A Command Interface Window (**CIW**) window appears on the screen, shown in Fig. 1.

Fig. 1. Main window of Cadence tools.

5 Creating Inverter Schematic

Step 1: In the ***CIW*** window, go to ***File…→ New…→ Library***

A ***New Library*** window appears. Fill out the pop-up form according to Fig. 2, and click ***OK***.

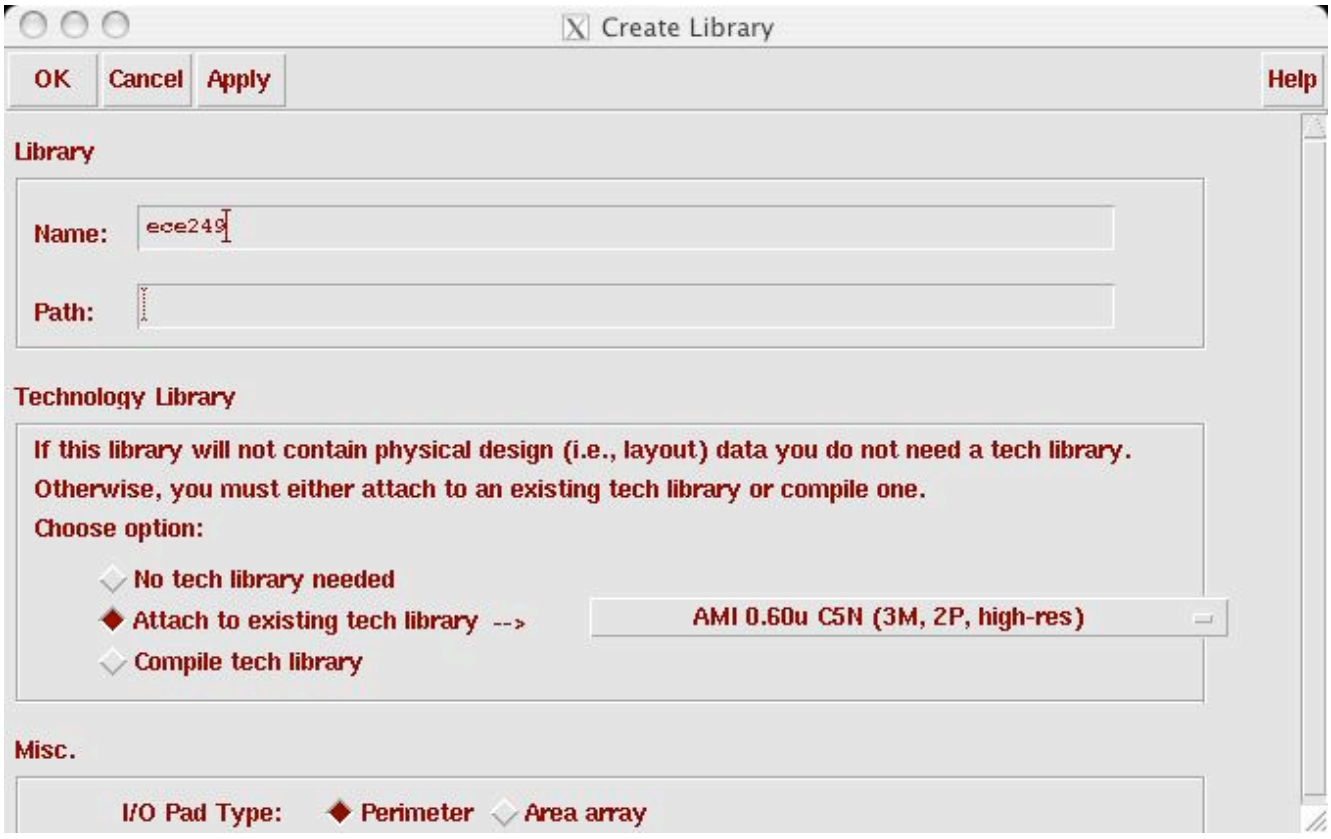

Fig. 2. Creating a new library.

Step 2: In the ***CIW*** window, go to ***File…→ New…→ Cellview***

A ***Create New File*** window appears, in Fig. 3. Fill it out and click ***OK***.

Fig. 3. Creating a new file.

Then a new ***Virtuoso Schematic Editing*** window appears as in Fig. 4.

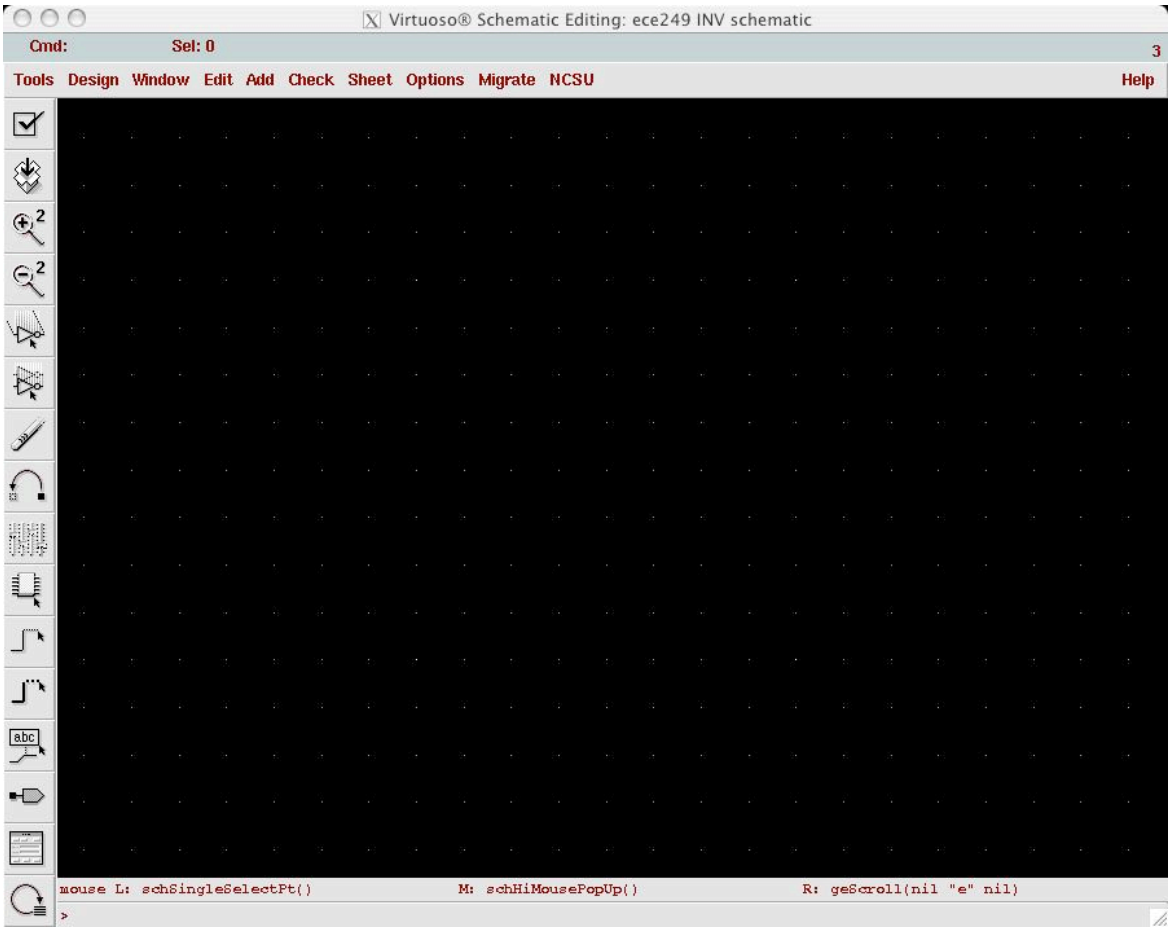

Fig. 4. The Virtuoso Schematic Editing window.

Step 3: In the ***Virtuoso Schematic Editing*** window, go to ***Add… → Instance …***

An ***Add Instance*** window will appear. Click ***Browse***, and a ***Library Browser*** window appears as shown in Fig. 5.

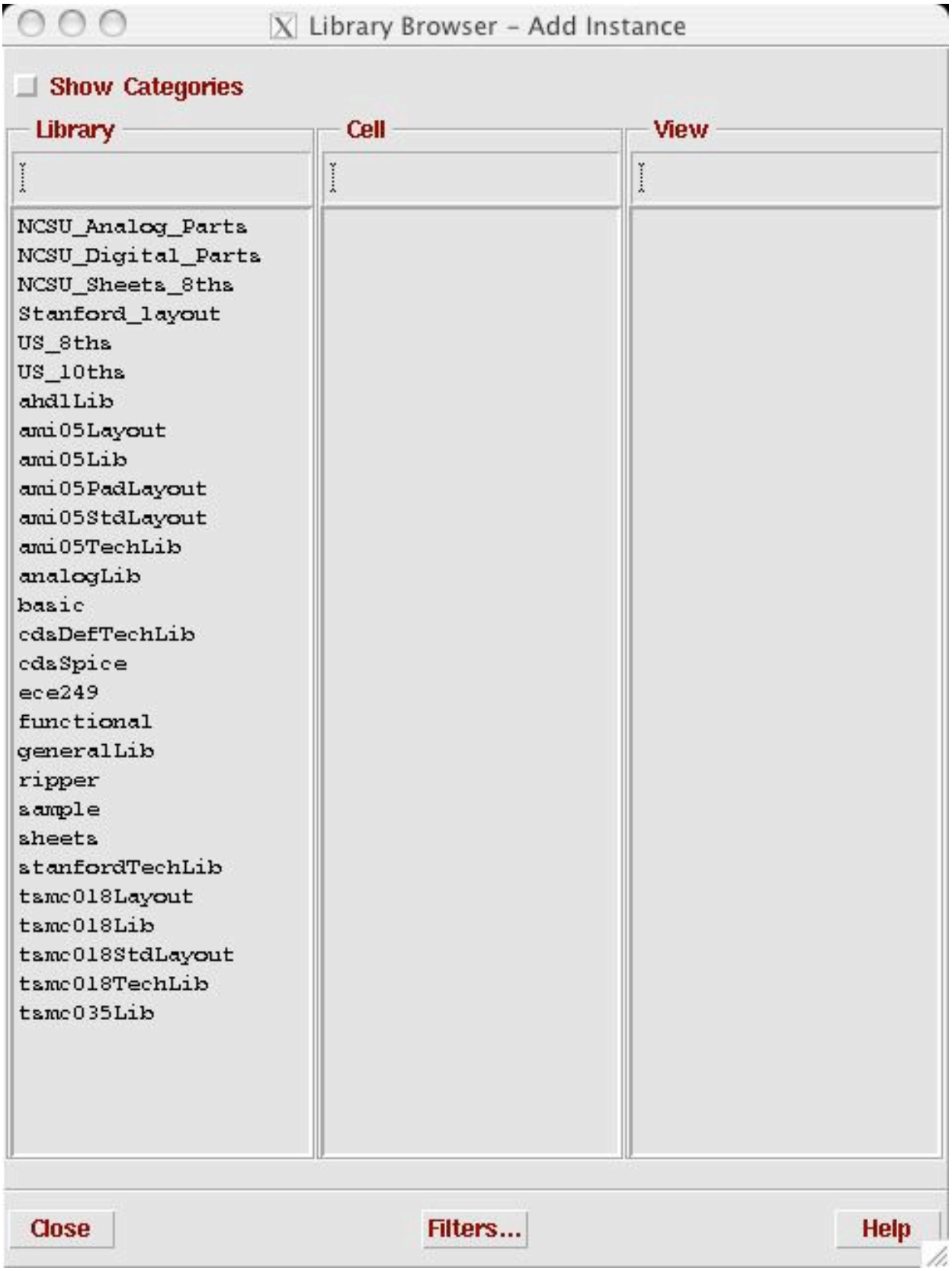

Fig. 5. Instance window.

Step 4: Select ***ami05Lib*** in ***Library***, ***pmos4*** in ***Cell***, and ***symbol*** in ***View***. The Add instance window should now have the pmos information filled in as in Fig. 6. Stamp a ***pmos*** device symbol in the ***Virtuoso Schematic Editing*** window, as shown in Fig. 7.

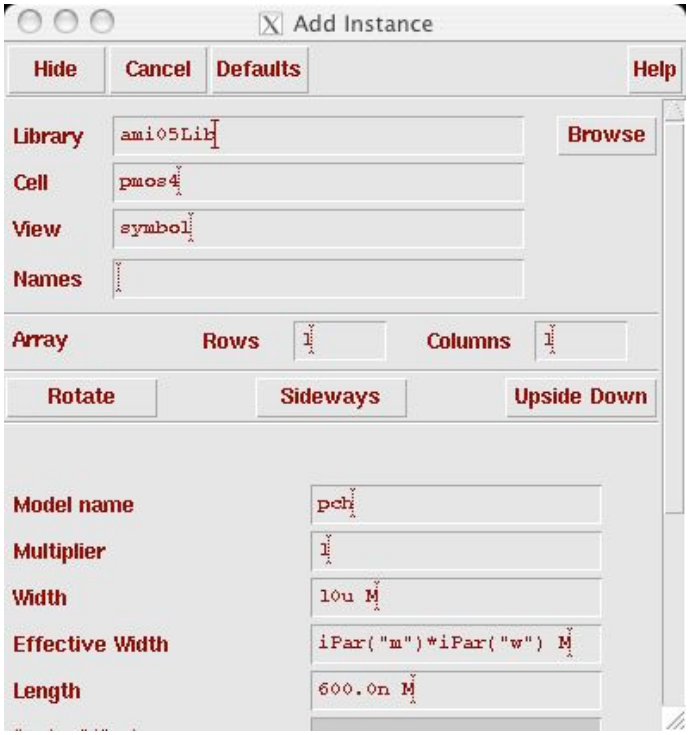

Fig. 6. Add instance window.

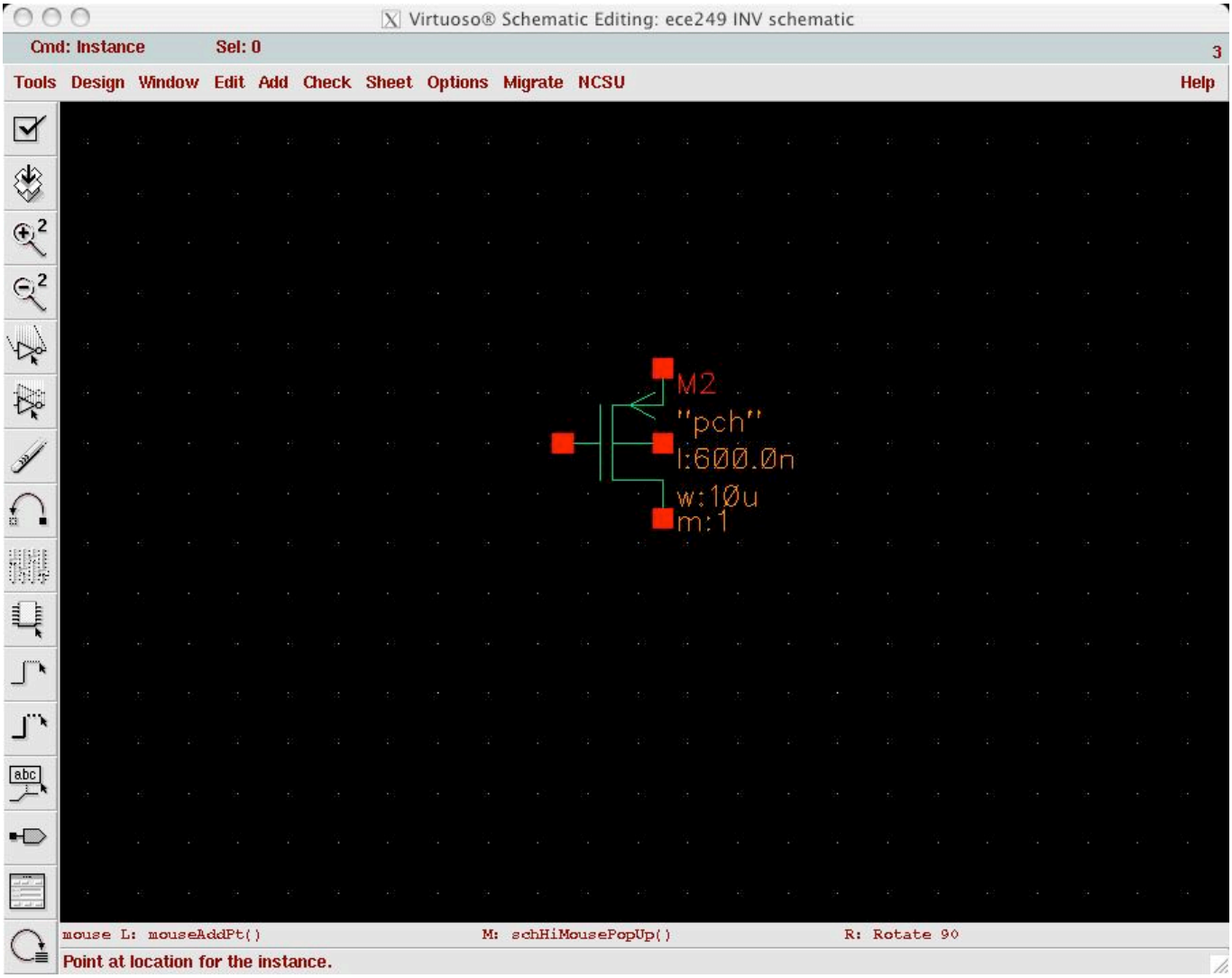

Fig. 7. Stamping ***pmos*** device.

Step 5: Repeat step 3–4, but adding an ***nmos4*** transistor.

Step 6: In the ***Virtuoso Schematic Editing*** window, go to ***Add… → Pin***. Name the pin 'A'.

Step 7: Repeat step 11 for a pin named 'B'.

Step 8: Repeat step 11 for a pin named 'vdd'.

Step 9: Repeat step 11 for a pin named 'gnd'.

Step 10: In the ***Virtuoso Schematic Editing*** window, go to ***Add… → Wire*** and connect the instances as shown in Fig. 8.

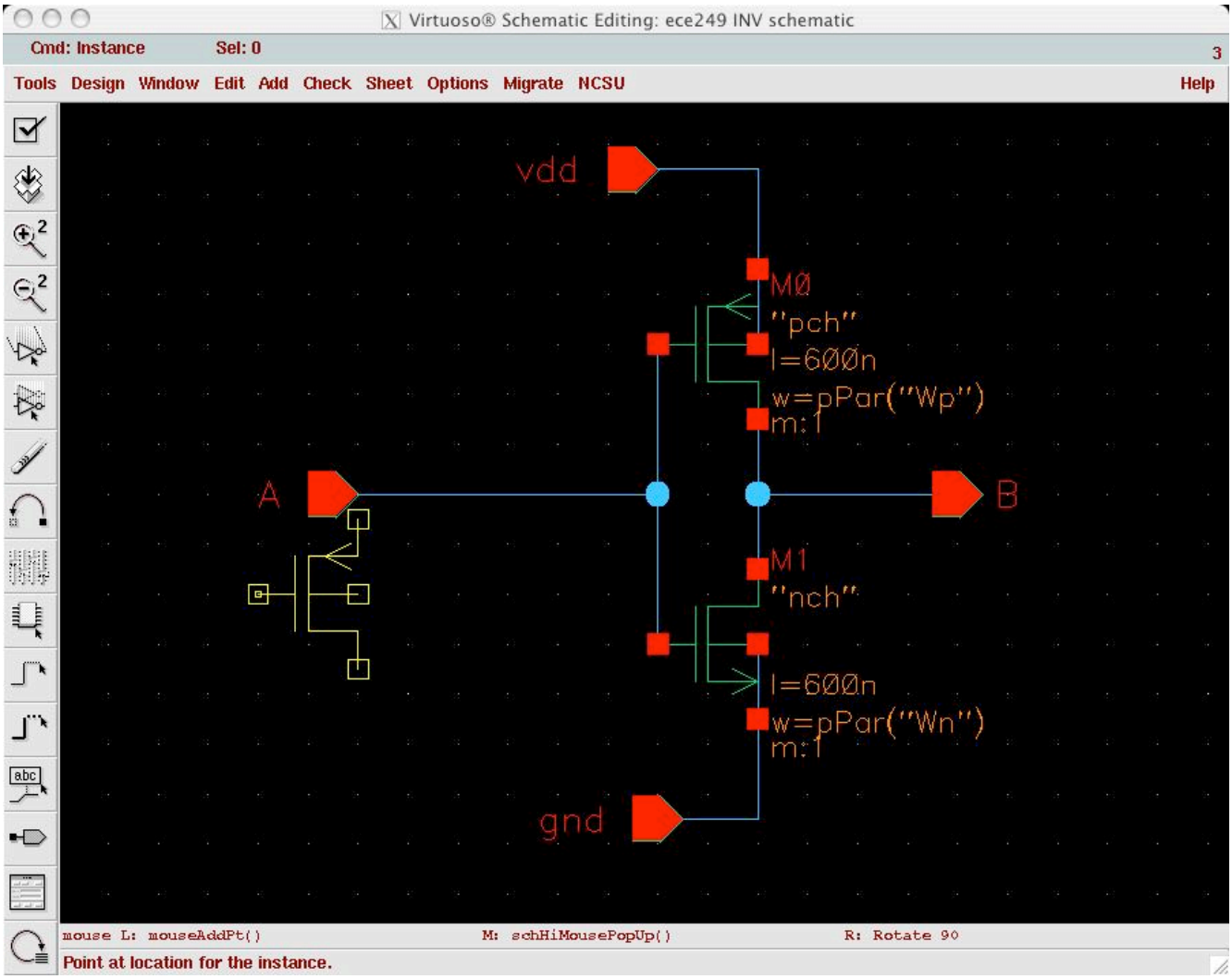

Fig. 8. Inverter schematic

Step 11: In the ***Virtuoso Schematic Editing*** window, go to ***Design… → Check and Save***. You should have no errors or warnings.

6 Symbol Creation of Sub-Circuits in Cadence

If a design is complicated, generally we use the top-down design method, or hierarchy structure. In this case, it is very beneficial to assign each sub-circuit a corresponding symbol (or icon) to represent that module. This step largely simplifies the schematic representation of the overall design. Thus, the "symbol" view of a circuit module is an icon that stands for the collection of all components within the module.

Step 1: In the ***Virtuoso Schematic Editing*** window, go to ***Design… → Create Cellview → From Cellview***, the following window will pop up.

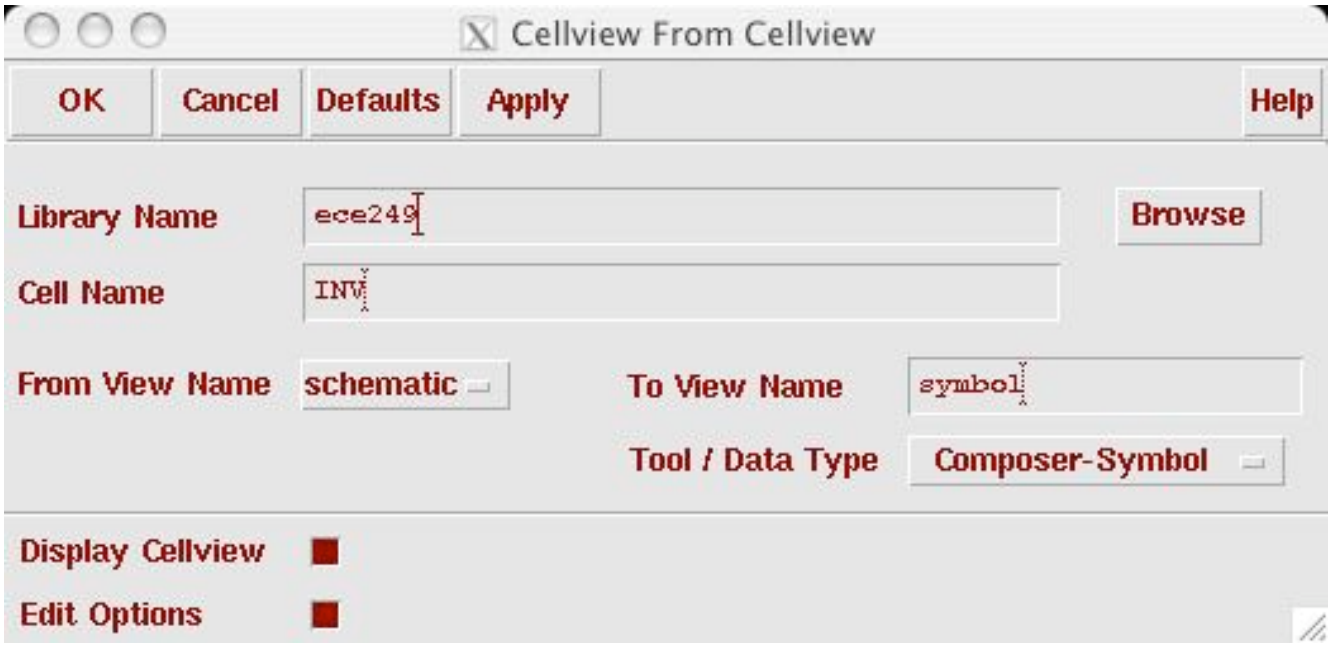

Fig. 9. "Cellview From Cellview" form.

Step 2: Check the view names and click ***OK***. Make sure that the target view name is symbol, which is indicated in "*To View Name*".

Step 3: In a new window, a symbol is automatically generated. The default shape of the symbol icon is a rectangle as shown in Figure 10.

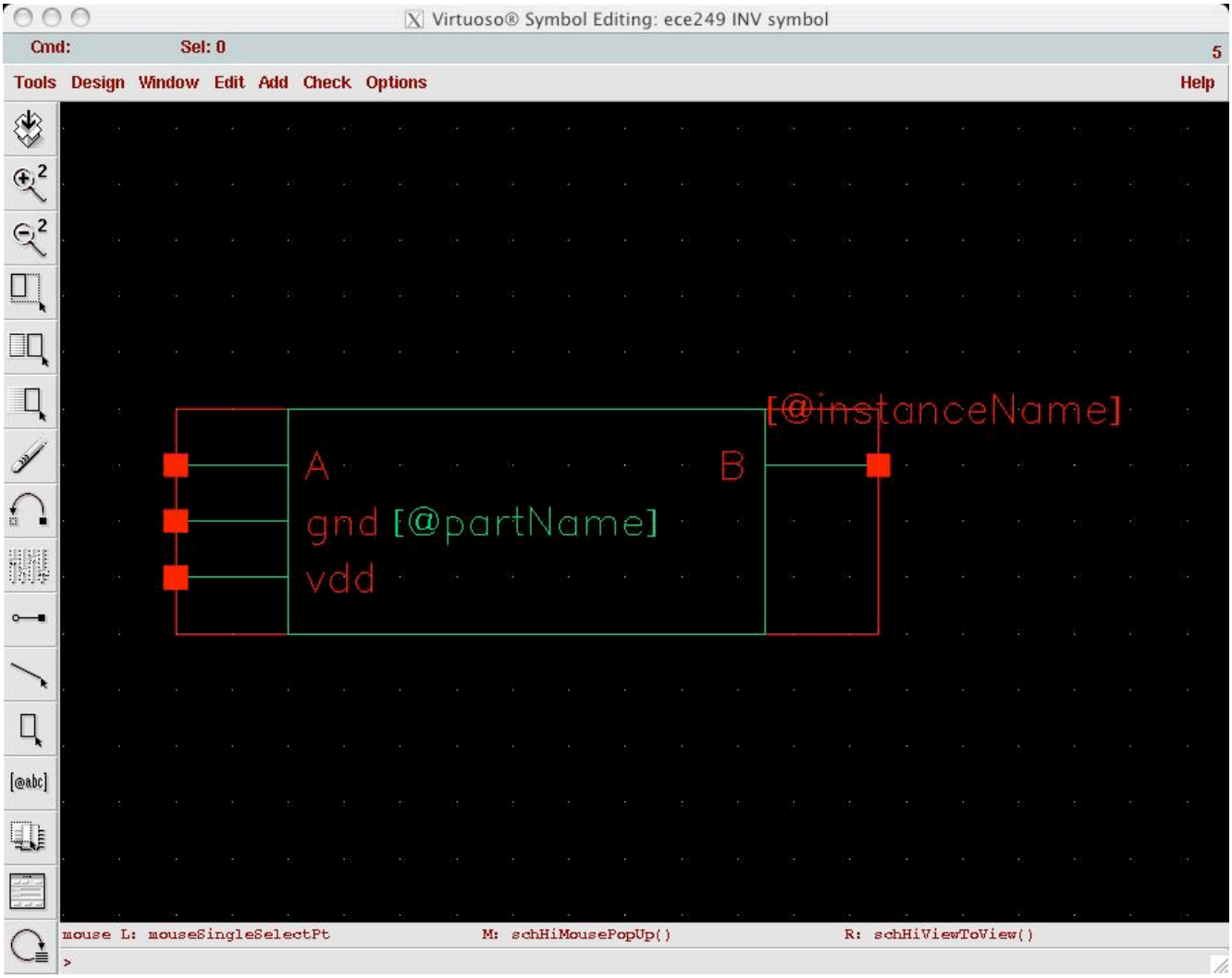

Fig. 10. Automatically generated inverter symbol.

Step 4: Editing the shape of the symbol icon. You can do the following operations on your symbol:
(1) Deleting/replacing some existing parts
(2) Adding new geometric shapes
(3) Changing the locations for pins and instance name
(4) Adding new labels

Figure 11 is an example for a manually created inverter symbol, which was obtained by editing the symbol shown in Fig. 10.

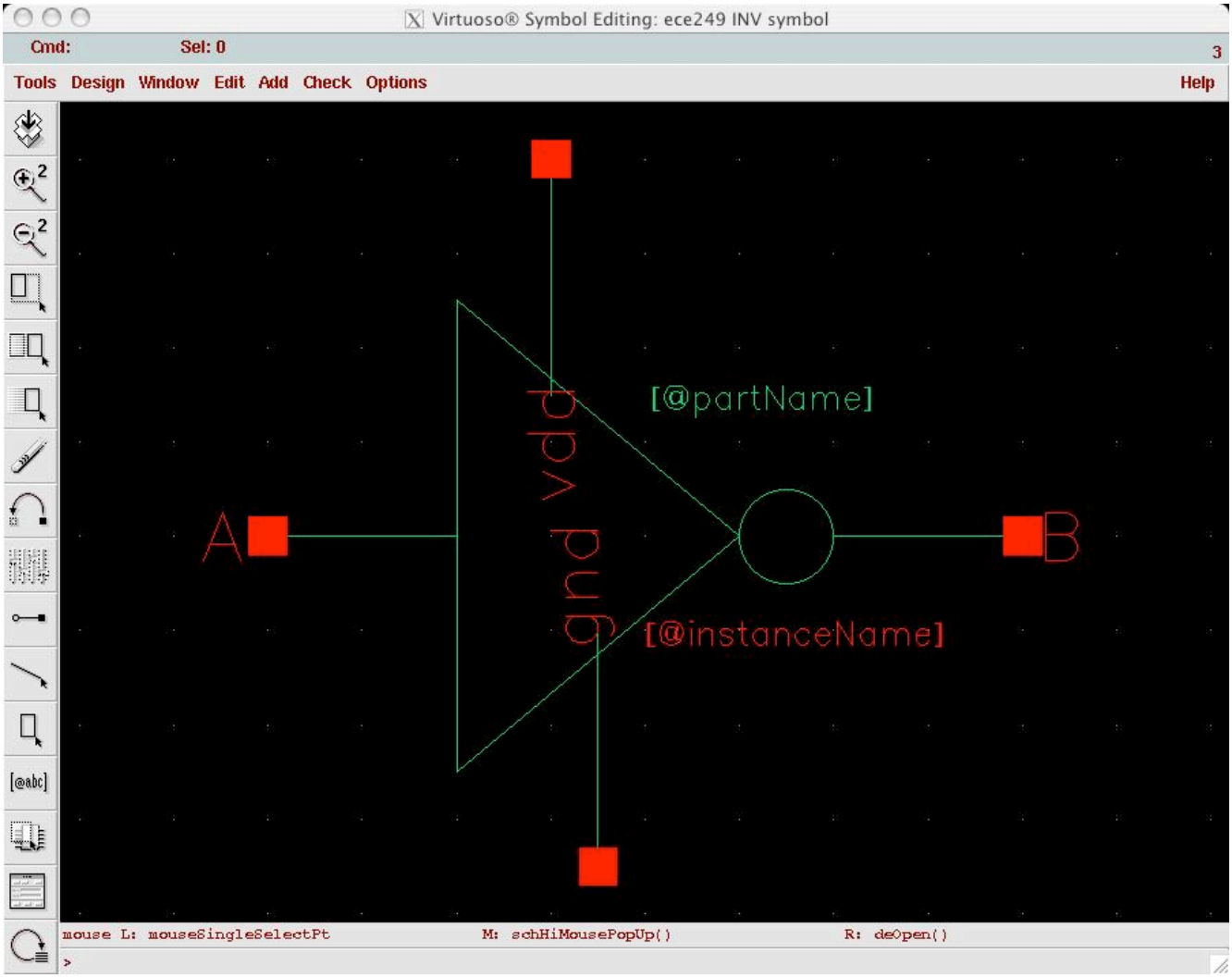

Fig. 11. The inverter symbol after modification.

Step 5: In "***Virtuoso Symbol Editing***" window, go to ***Design… → Check and Save.*** Thus, you have created your own symbol for future use.

7 Simulation

Now we are going to simulate the inverter we just created to confirm that it operates as we expect.

Step 1: In the ***CIW*** window, go to ***File…→ New…→ Cellview***

A ***Create New File*** window appears, in Fig. 12. Fill it out and click ***OK***.

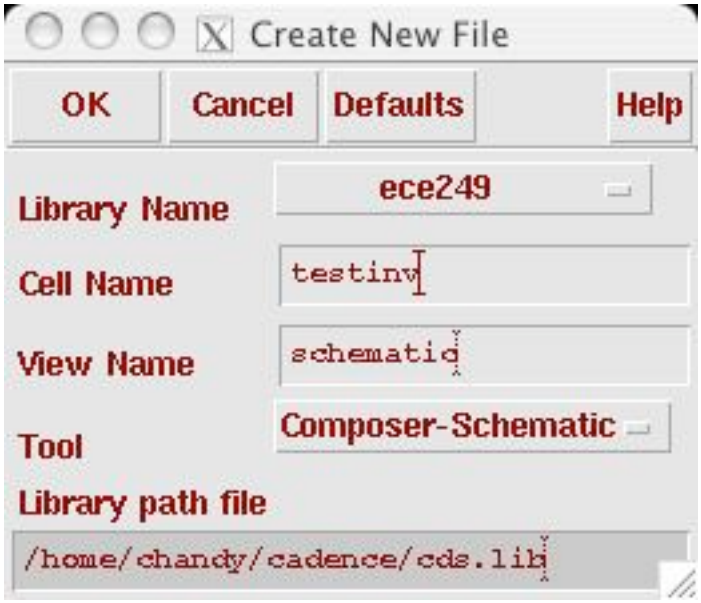

Fig. 12. Creating a new file.

Then a new ***Virtuoso Schematic Editing*** window appears as in Fig. 4.

Step 2: Add an instance of the new ***INV*** symbol that we just created as shown in Fig. 15.

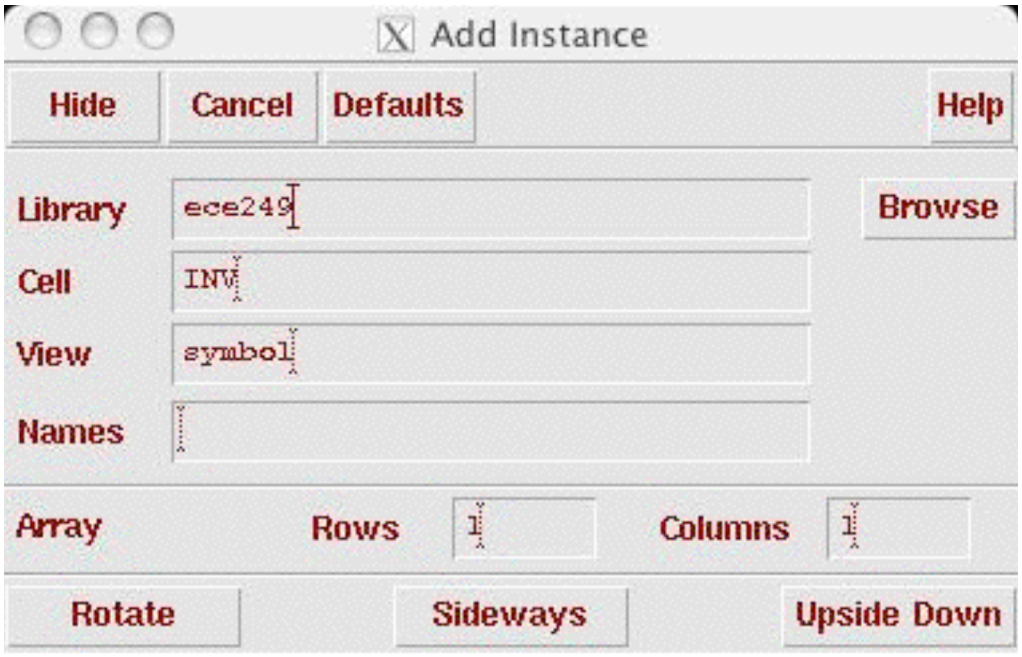

Fig. 13. Adding inverter instance

Step 3: Add a ***vpulse*** instance from the ***analogLib*** library with properties as shown in Fig. 14.

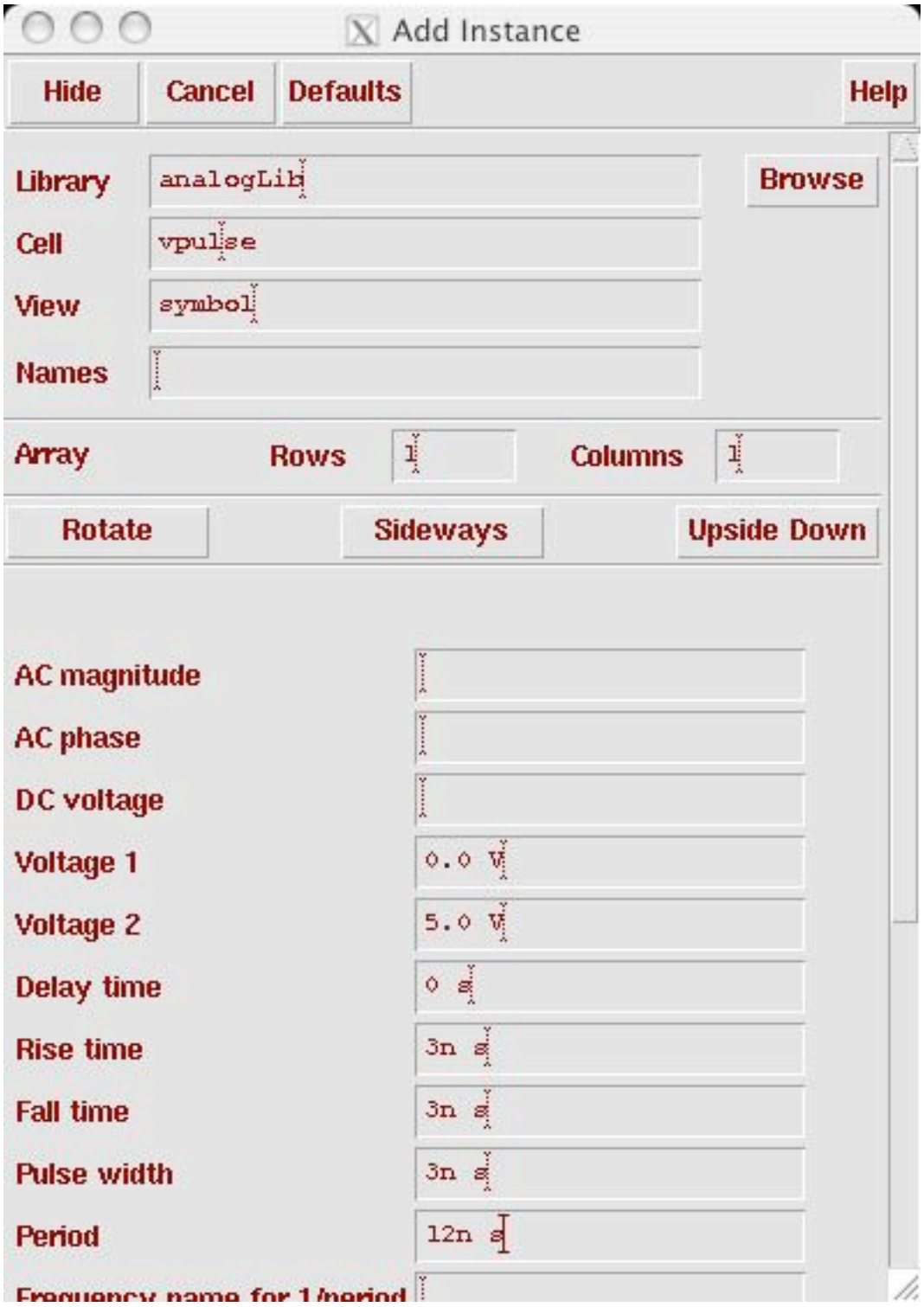

Fig. 14. Properties of *vpulse*.

Step 4: Add a ***vdc*** instance from the ***analogLib*** library with properties as shown in Fig. 15.

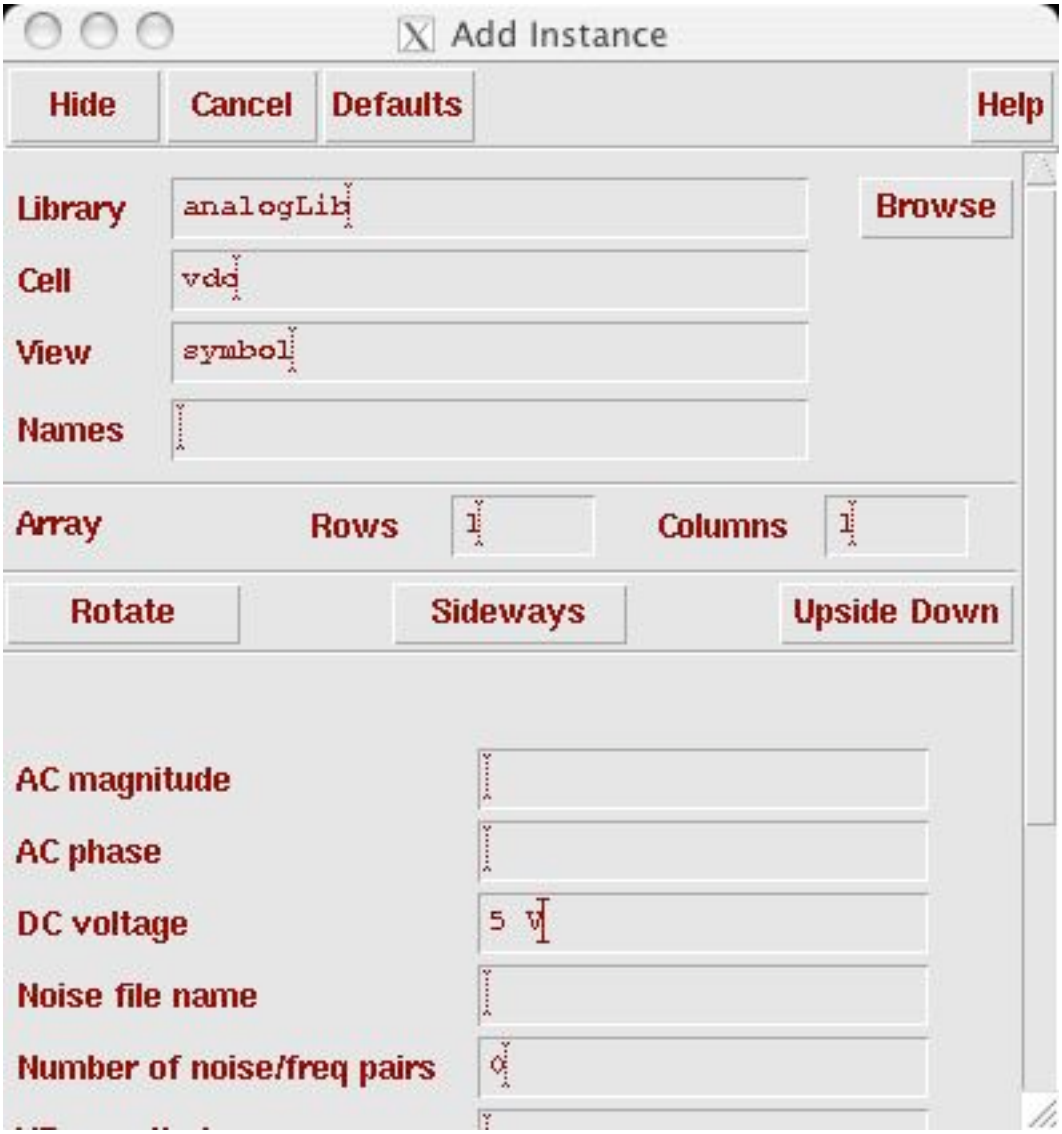

Fig. 15. Properties of *vdc*.

Step 5: Add an output pin named **Vout.**

Step 6: In ***Virtuoso Schematic Editing*** window, go to ***Add… → Wire***, connecting all the components as shown in Fig. 16. The vdd and gnd symbols are in the analogLib library.

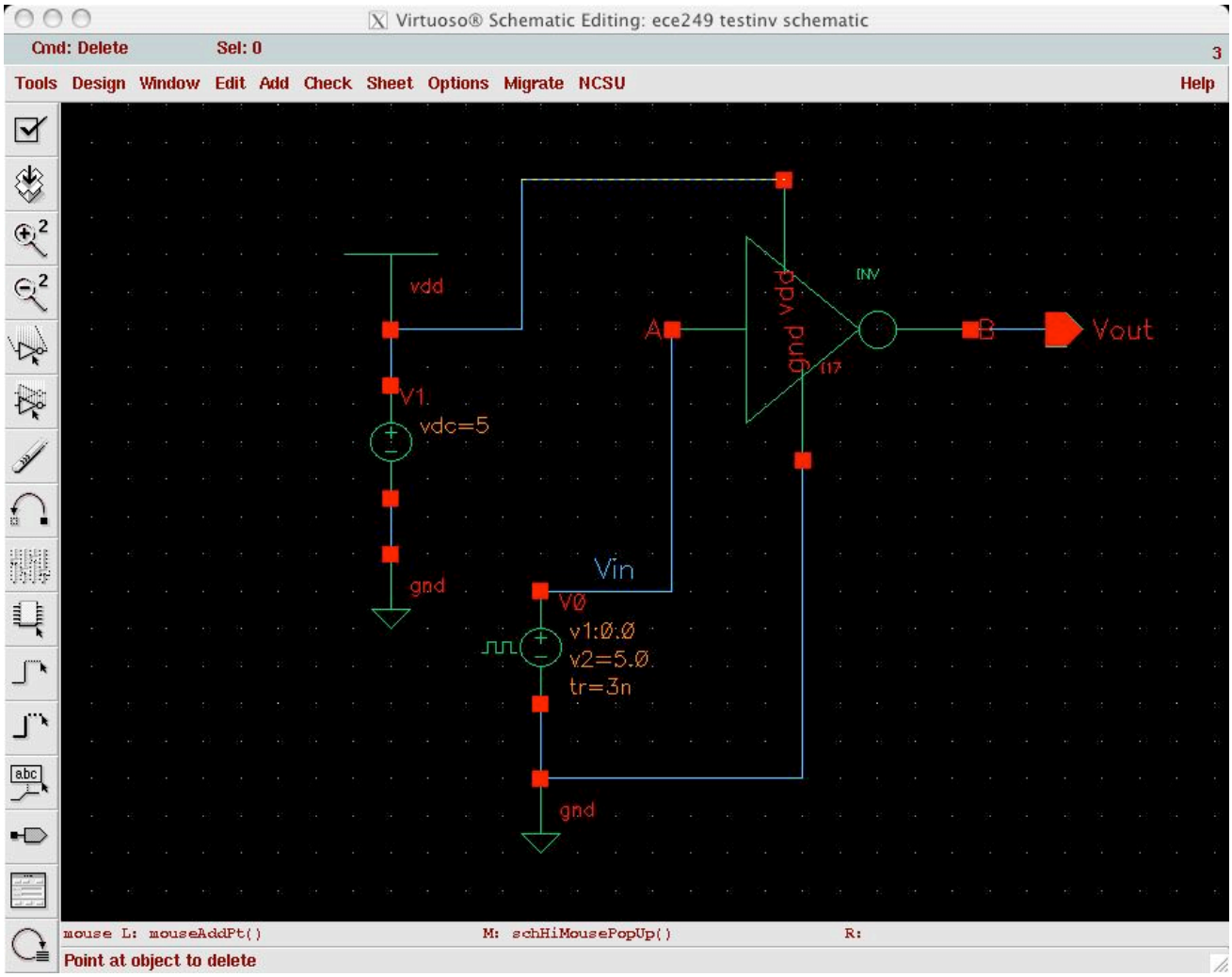

Fig. 16. The completed testinv schematic.

Step 7: In the ***Virtuoso Schematic Editing*** window, go to ***Add… → Wire Name***, and name the input wire to the inverter **Vin**.

Step 8: In the ***Virtuoso Schematic Editing*** window, go to ***Design… → Check and Save***. Correct any errors in your schematic figure.

Step 9: In the ***Virtuoso Schematic Editing*** window, go to ***Tools… → Analog Environment***, an ***Analog Circuit Design Environment*** window appears as shown in Fig. 17.

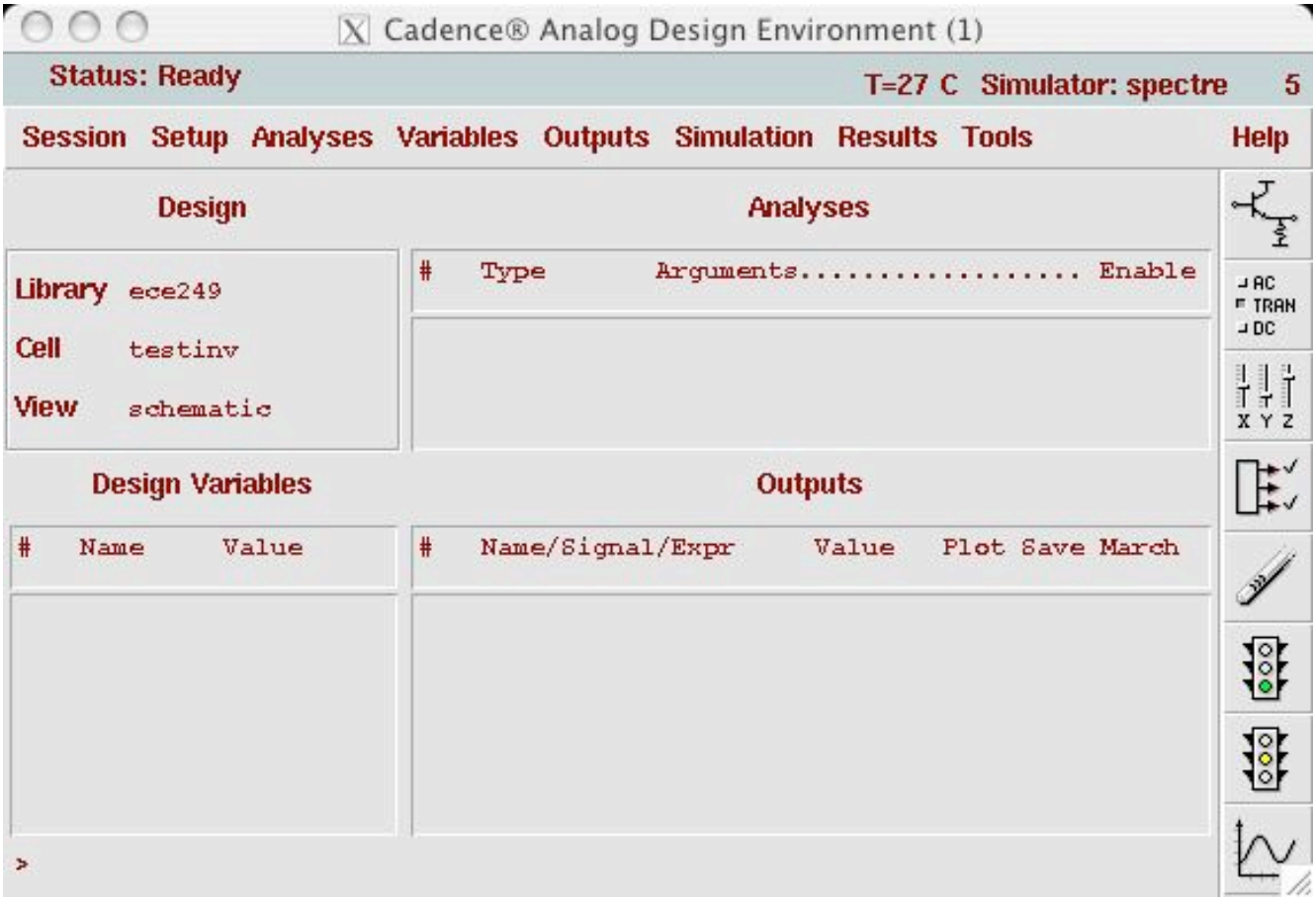

Fig. 17. Analog Circuit Design Environment window.

Step 10: In the ***Analog Circuit Design Environment*** window, go to ***Setup… → Model Libraries…***, a ***Model Library Setup*** window appears. Click the ***Browse…*** button and then choose the models directory and then choose the ami05.scs model. Click ***Add***, as in Fig. 18 and then click ***OK***.

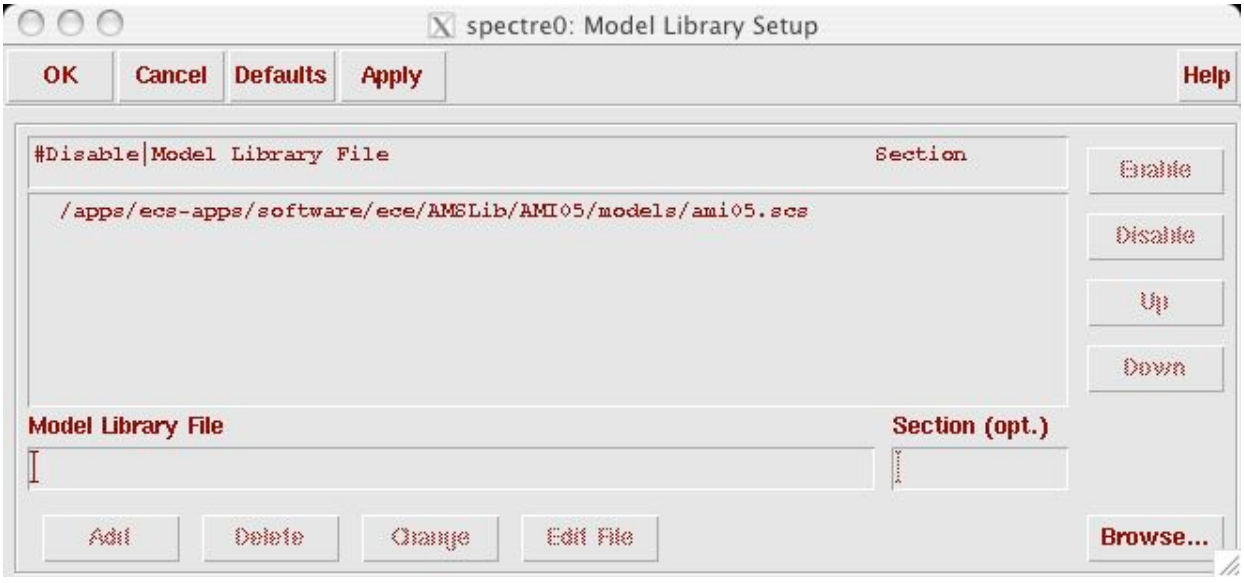

Fig. 18. Adding model path.

Step 11: In the ***Analog Circuit Design Environment*** window, go to ***Analysis…→ Choose…***, and fill out the form according to Fig. 19.

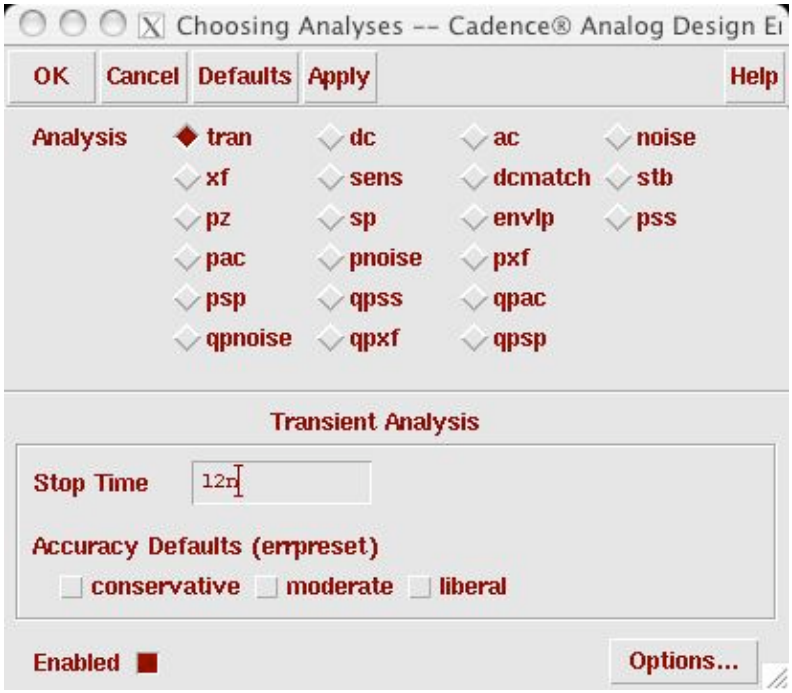

Fig. 19. Setting up transient analysis.

Step 12: In the ***Analog Circuit Design Environment*** window, go to ***Outputs…→ To be Plotted… → Select on Schematic.*** Then go back to the ***testinv*** schematic window and select the **Vin** and **Vout** wires. Note, the wires should change color, and the signals should be added in the outputs window, as in Fig. 20.

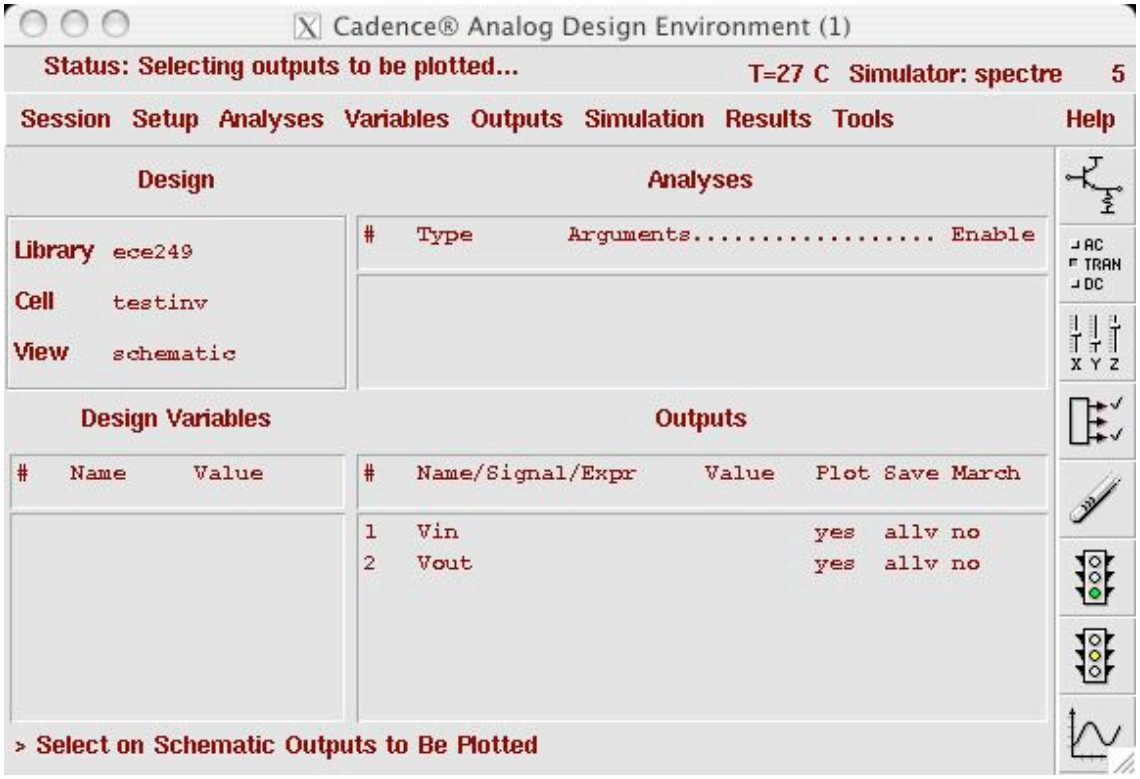

Fig. 20. Analog Circuit Design Environment ready for a simulation.

Step 13: In the ***Analog Circuit Design Environment*** window, go to ***Simulation…→ Netlist and Run.*** The plot of the simulation should appear in a ***Waveform Window***, like Fig. 21.

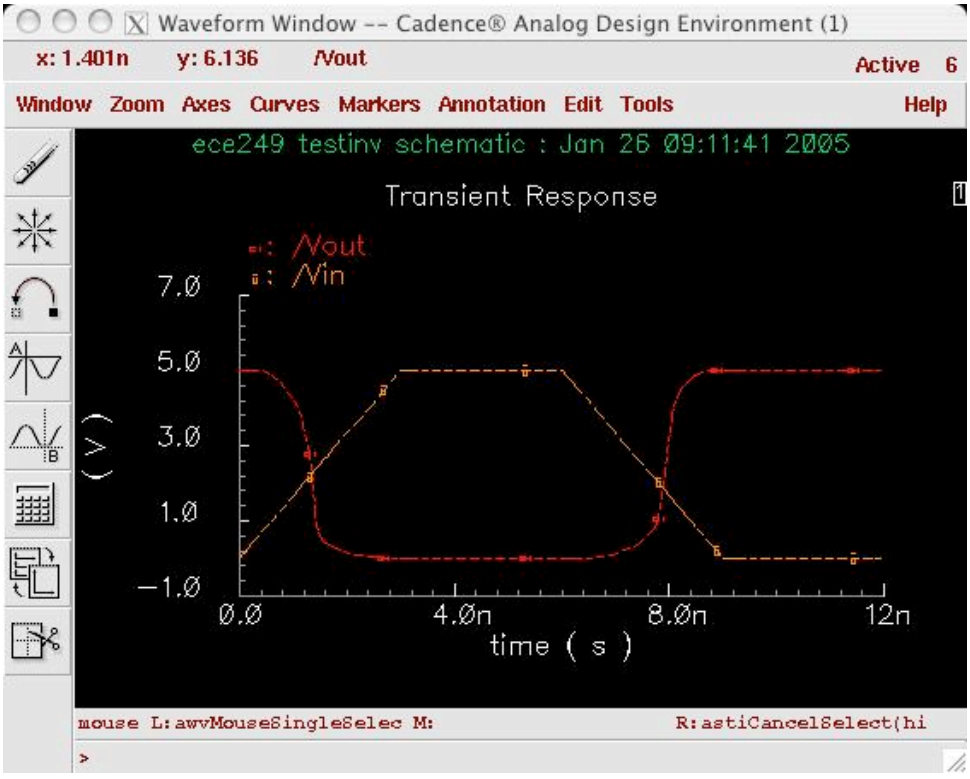

Fig. 21. testinv simulation result.

Step 14: In the ***Waveform Window***, go to ***Axes…→ Xaxis*** and change ***Plot vs independent variable*** to ***Plot vs 1/Vin***, like Fig. 22, and click ***OK.*** The final plot should look like Fig. 23.

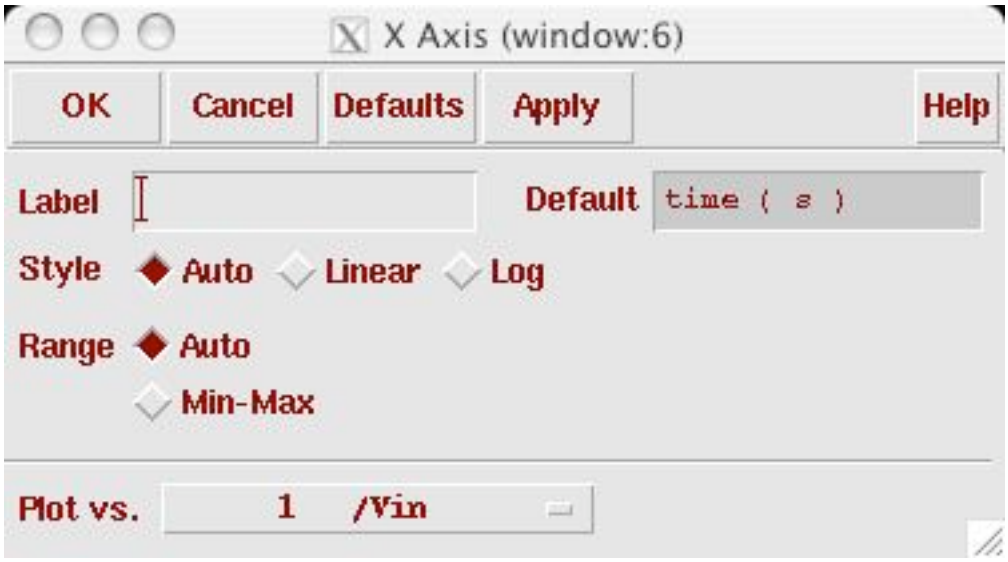

Fig. 22. Axis form.

Step 15: Exit the ***Analog Circuit Design Environment Simulation*** window, and exit the ***Virtuoso Schematic Editing*** window.

Step 16: In the ***CIW*** window, go to ***File…→ Exit***.

8 Homework

1. What is the switching threshold of the inverter that you just created? Use the point at which Vin=Vout on the lower hysteresis curve to determine the switching threshold. Hand in Cadence Vout vs. Vin plots for this configuration and mark the switching threshold point.

2. What values of W or L for the pMOS transistor would change the switching threshold to exactly 2.5V? What does this tell you about ρ the ratio between the k's for the two transistors? Hand in Cadence Vout vs. Vin plots for this configuration.

3. What values of W or L for the nMOS transistor would give a switching threshold of 1.5V? Hand in Cadence Vout vs. Vin plots for this configuration.

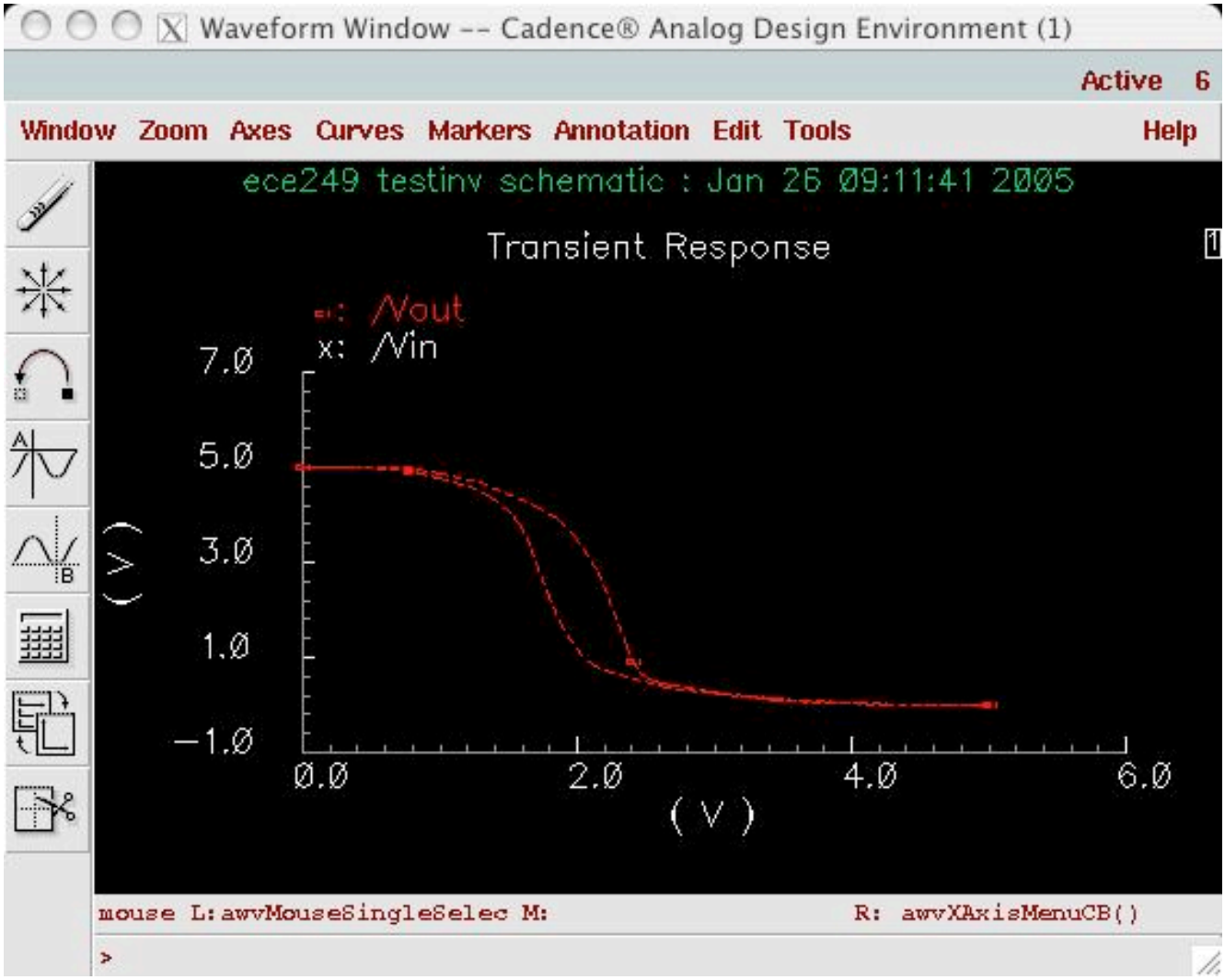

Fig. 23. Output voltage vs input voltage for a CMOS inverter.