CMOS Analog Integrated Circuit Design

Lab 1
Introduction to Cadence

Department of Electrical and Computer Engineering

The University of Texas at Austin

Fall 2006

In this lab, the students are introduced to Cadence integrated circuit design environment, the tool set that will be extensively used in the labs of *CMOS Analog Integrated Circuit Design*. Main contents of this lab include,

1. How to log in and log out SUN workstations and some basic UNIX commands.
2. Introduction to Cadence Design System for analog integrated circuit design, including, Cadence environment set up, schematic capture, creating symbols, starting analog environment, and circuit simulation with spectreS.

The TA will verify and evaluate your schematic and simulation results. Evaluation sign-up sheets will be available 3-4 days before the due date.

1. Objectives:

1. To learn how to use Cadence program.
2. To draw a simple inverter schematic with Virtuoso Schematic Composer.
3. To simulate the inverter schematic in Analog Environment.
4. Plot and view simulation waveforms in Waveform window.

2. Using UNIX Sun Workstations:

1. Log in: The Unix terminals in the Learning Resources Center, ENS 507, are used for the labs. You should have a login account to the system if you are a registered student. Check with the proctors in ENS 507 if you have any problems to log in.

   Please choose “Common Desktop Environment (CDE)” when you log in Sun machine. Otherwise, the figures shown in this lab manual may look different. Click “Options” on the login window and go to “Sessions” and choose “Common Desktop Environment (CDE)”.

2. Basic UNIX commands: If you need to learn basic UNIX commands, tutorial in the following URL will be more than enough -- `http://www.unixtools.com/unixprimer.html`. Some basic commands are summarized in the table below.

Basic UNIX Commands

Commands	Explanation
man *command_name*	print **man**ual for *command_name* on screen. Type space to see the following pages if one screen can not hold all the information. Type ‘b’ to go back to the previous screen. Type ‘q’ to quit viewing the manual if there are more than one screens. For example, you can try “man pwd” for information of command “pwd”.
pwd	**p**rint **w**orking **d**irectory, print current working directory that you are in.

cd *directory_name*	**c**hange **d**irectory to *directory_name*. If only "cd" is typed without *directory_name*, change to your home directory.
mkdir *directory_name*	**m**a**k**e a new **dir**ectory named *directory_name*
rmdir *directory_name*	**r**e**m**ove **dir**ectory named *directory_name*
ls	**l**i**s**t files in one directory. "ls -l" list file information in a long format.
mv *file_name new_file_name*	**m**o**v**e *file_name* to *new_file_name*. It can also be used to move a file to a different directory using "mv *file_name directory_name*".
cp *file_name new_file_name*	**c**o**p**y *file_name* to *new_file_name*. It can also be used to copy a file to a different directory using "cp *file_name directory_name*".
rm *file_name*	**r**e**m**ove *file_name*. Unlike windows, you can NOT recover a removed file. Be cautious when you remove a file. You can add one line (between " and "), "`alias rm 'rm -i'`", into the .cshrc file in your home directory.
textedit *file_name*	start text editor program to edit *file_name*.
quota -v	check disk quota.
command &	Run *command* in the background.

3. Log out: If you are using "Common Desktop Environment (CDE)" desktop, click "EXIT" button on the tool bar at the bottom of the desktop.

3. Using the Cadence Tool for IC Design

The Cadence Design System includes several software packages for integrated circuit design, such as, schematic composer, circuit simulators, layout editor, and layout extraction and verification tools. Cadence design framework manages the process for development of analog, digital, and mixed-signal (with both analog and digital) integrated circuits. In this course, we will only use the tools that are involved in analog integrated circuit design. This section has the following contents:

1. Introduction to Cadence
2. Setting up the Environment
3. Starting Cadence
4. Library Manager
5. Schematic Composer
6. Creating symbols

7. Starting the Analog Environment
8. Simulation in the Analog Environment (DC and transient analyses)
9. Plot and view waveforms in waveform window

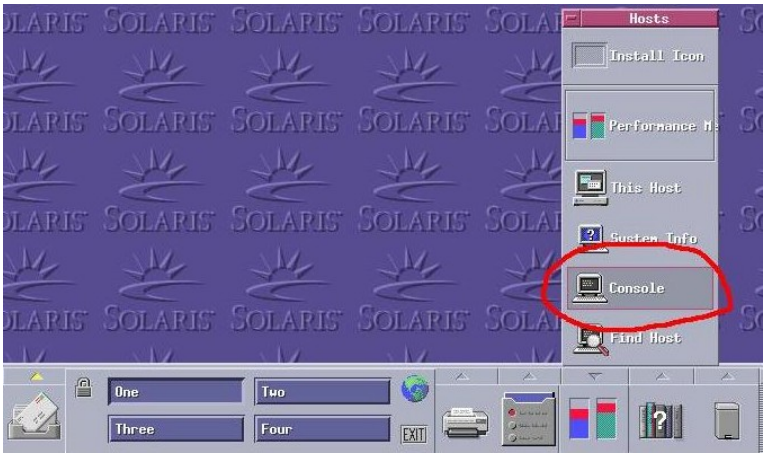
3. 1. Introduction to Cadence

The Cadence program that we are using is IC 5.0.33 with NCSU_CDK 1.2. Menus are generally customized. We can use CMOS processes available through MOSIS (`http://www.mosis.org/`) for schematic and layout design. The tools we are going to use in our labs include: 1. Virtuoso schematic composer, 2. Analog environment with SpectreS circuit simulator, 3. Virtuoso layout editor, 4. DIVA - DRC, Extraction and LVS verification tools.

In this lab, we will learn Virtuoso Schematic Composer and Analog Environment with SpectreS circuit simulator. Virtuoso layout editor, DRC, Extraction, LVS verification, and post layout simulation will be introduced in Lab 2.

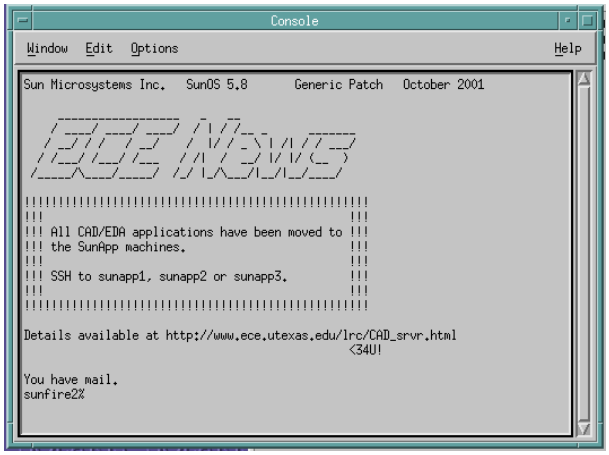
3.2. Setting up the Cadence Environment

First of all, open the console on the `sunfire` machine (i.e. `sunfire1` or `sunfire2`) [†].



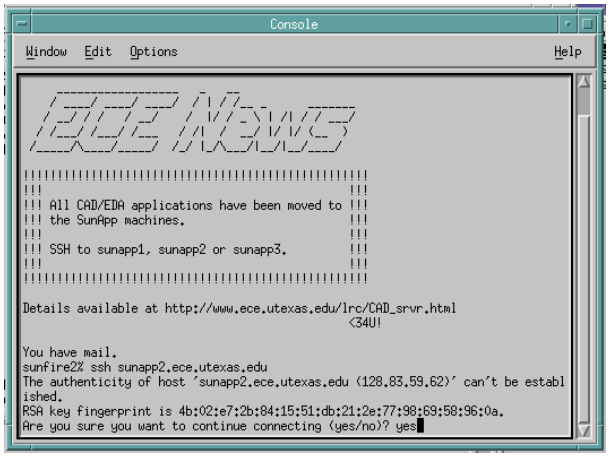
Then a console window pops up as shown below.

[†] `sunfire` machines are `sunfire1` and `sunfire2`. `sunapp` machines are `sunapp1`, `sunappp2`, and `sunapp3`. These are the UNIX application servers at ECE LRC. Different applications, such as web browser `netscape` and PDF viewer `acroread`, are available on `sunfire` servers. You can not start Cadence on `sunfire` machines. Instead, Cadence should run on `sunapp` machines. Web browser `netscape` and PDF viewer `acroread` are not available on `sunapp` machines.



As you see on the screen, “all CAD/EDA applications have been moved to the SunApp machines”, you have to connect to `sunapp1`, `sunapp2`, or `sunapp3` machine to start Cadence. Type “`ssh sunapp2`” in the console command and press <ENTER> to connect to `sunapp2` as shown in the following screen snapshot. Please make sure that there is a space between “`ssh`” and “`sunapp2`”.

If it is the first time you login to `sunapp2`, you will be asked, “`Are you sure you want to continue connecting (yes/no)`?” Type “`yes`” and press <ENTER>. Then, you are prompted for your password to access `sunapp` machine. Type your password and press <ENTER>. (Note: Once you answered “yes” from the machine that you are using now, it won’t ask you the same question again if you connect to the same machine next time. Instead, you will be directly asked to enter your password.)



Now, you see the console command line prompt changed to “`sunapp2%:`” showing that you have connected to `sunapp2`. If you connect to `sunapp1` or `sunapp3`, then the console will display “`sunapp1%` ” or “`sunapp3%` ” instead. Please understand that all of the following commands related to Cadence have to run on `sunapp` machines. Cadence is not accessible on `sunfire` machines. Other applications may or may not run on `sunapp`, but are available on `sunfire`.

You should create a directory under your home directory called “`Analog`”. Type “`mkdir Analog`” (“`mkdir`” is the UNIX command to make a directory) and press <ENTER>. Don’t forget the space between “`mkdir`” and “`Analog`”. Note that an <ENTER> key stroke following the command text is

always needed to execute a command. For simplicity, we will not repeat saying the <ENTER> key stroke after the command line in the following sections. Please add an <ENTER> key stroke at the end of any command line by yourself to execute that command.

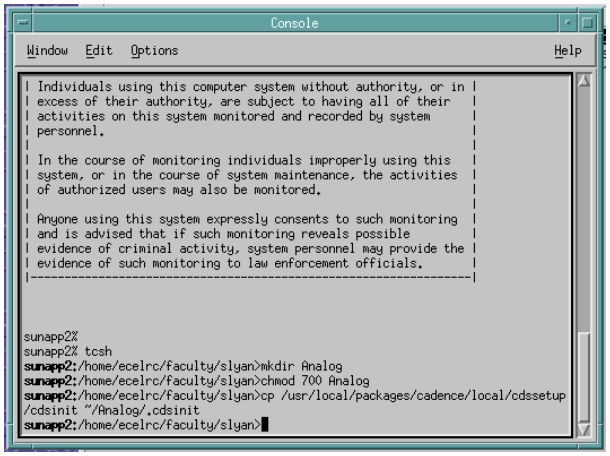
Type "`chmod 700 Analog`" to change the permission of "`Analog`" directory, thus other students can not see your design files. If you leave your directory open and some one else copied your work, you will have equal responsibility for copying of lab work.

The user settings should be copied from `/usr/local/packages/cadence/local/cdssetup` directory. There are three files you should copy from that directory, `cdsinit`, `cdsenv`, and `simrc`.

1. The first file is `.cdsinit` . Type the following command.

```
cp /usr/local/packages/cadence/local/cdssetup/cdsinit ~/Analog/.cdsinit
```

Note: The source file is `cdsinit` while the destination is `.cdsinit` (with a leading dot). You can also copy-and-paste the above command line into the console.



2. The second file is `.cdsenv` . Type the following command.

```
cp /usr/local/packages/cadence/local/cdssetup/cdsenv ~/Analog/.cdsenv
```

3. The last file is .simrc . Copy the file by typing

```
cp /usr/local/packages/cadence/local/cdssetup/simrc ~/Analog/.simrc
```

4. Then, revise `.cshrc` file in your home directory (not "`Analog`" directory). Add the lines inside the frame on next page into the `.cshrc` file at your home directory, `~/.cshrc` ("`~/`" is your home directory) .

To avoid confusion, you are required to use `tcsh` shell which probably is not your login shell. To invoke the `tcsh` shell, type the following command.

```
tcsh
```

A sample `.cshrc` file can be found at (don't forget the leading dit in the file name)

http://www.ece.utexas.edu/~slyan/labs/setup/.cshrc

which is also shown below.

```
# Sample .cshrc file for students in Analog Integrated Circuit Design

alias setprompt 'set prompt = "%B%m%b:`pwd`>"'
if ($?prompt) then
       setprompt
       set history = 40
       alias h history
endif
set path = (/bin /usr/local/bin /usr/local/gnu/bin /usr/bin /usr/sbin /usr/bin/X
11 /usr/X/bin .)

##################################################################
# Cadence Tools
##################################################################

# settings for cadence

setenv CDS_LIC_FILE /usr/local/packages/cadence/license.dat
setenv INSTALL_DIR /usr/local/packages/cadence
setenv LM_LICENSE_FILE /usr/local/packages/cadence/license.dat
setenv   PATH   ${PATH}:/usr/local/packages/cadence/ic/tools/bin:/usr/local/packages/
cadence/ic/tools/dfII/bin:/usr/local/packages/cadence/ldv/tools/simvisdai/bin:/usr/
local/packages/cadence/ldv/tools/verilog/bin:/usr/local/packages/hspice/current/bin
setenv CLS_CDSD_COMPATIBILITY_LOCKING NO

# settings for NCSU CDK

setenv IC $INSTALL_DIR/ic
setenv CDS_SITE /usr/local/packages/cadence/local
setenv USE_NCSU_CDK 1
setenv CDS_Netlisting_Mode Analog
setenv SIMRC /usr/local/packages/cadence/local/skill

set prompt = "%B%m%b:`pwd`>"
```

```
alias ls 'ls -F'
alias ll 'ls -l'
alias rm 'rm -i'
alias cp 'cp -i'
alias cd 'cd \!* ; set prompt = "%B%m%b:`pwd`>"'
```

Your `.cshrc` file may look very similar to the above example `.cshrc` file. Just add the framed lines to your original .cshrc file by typing or copy-and-paste. Please be careful that copy-and-paste may cause line misalignment problem. For example, the command started with "`setenv PATH ${PATH} ...`" appears to have several lines, but they all are in the same long physical line in the file, and are not separated with "RETURN" characters. When you copy-and-paste that command, several "RETURN" characters might be inserted. Correct that problem if it happens.

The following lines are recommended for your `.cshrc` file.

```
alias rm 'rm -i'
alias cp 'cp -i'
```

If you have the following command aliases, when you delete a file or multiple files using 'rm', you will be prompted "rm: remove file_name (yes/no)?" If you type 'y' or 'yes', the file will be deleted. Otherwise, it will not. Thus you are given the last chance to make a decision whether the file should be kept or deleted . 'cp -i' has the same effect that, you are prompted to decide whether to overwrite a file if you copy a file to an existing file.

Or, if your original `.cshrc` file is the same as the one given above except the lines for the cadence setup, directly saving the `.cshrc` file to overwrite the original one could be a better option to avoid copy-and-paste problems. To do this, follow the steps below.

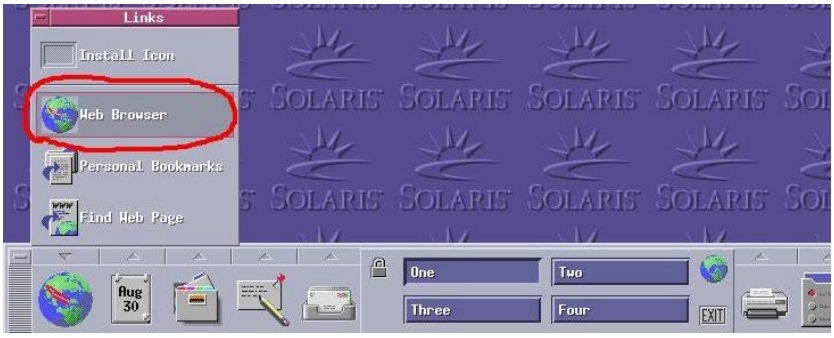
Rename the original `.cshrc` file to a different file name, such as `.cshrc_old`, by typing

```
mv .cshrc .cshrc_old
```

In case that you need the original `.cshrc` file later on, it is still available as `.cshrc_old`.

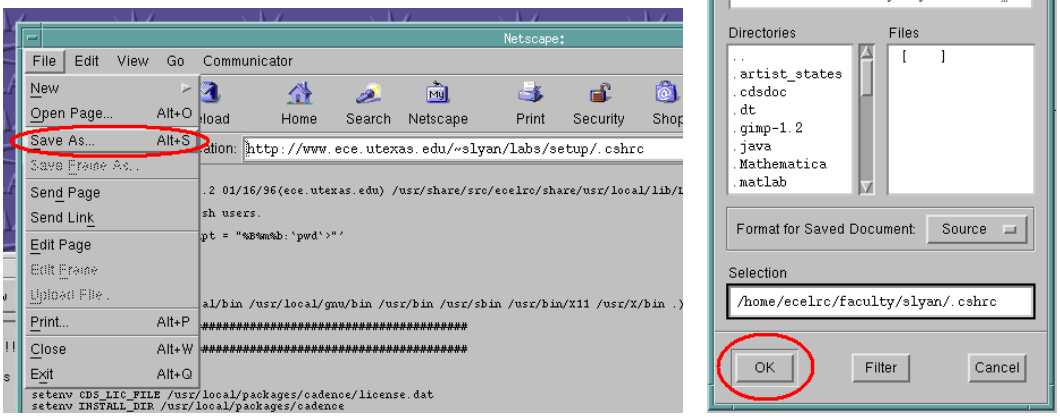
You can download `.cshrc` file from the above link. Before download the new `.cshrc` that may overwrite the original `.cshrc` file, please rename your original `.cshrc` to `.cshrc_old`.

Start the web browser as shown below from CDE.



Type or copy the link `http://www.ece.utexas.edu/~slyan/labs/setup/.cshrc` to web browser "Location" field, and press <ENTER>.

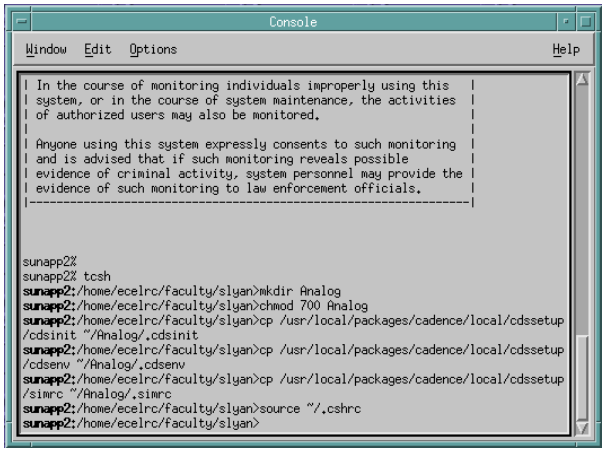
Save the file by choosing "File → Save As", as shown below. A new "Save As…" window appears. You may need to change the location. Confirm the file location and click OK.



Now, you have a new `.cshrc` file in you home directory. If necessary, you can copy some of the command lines from you old .cshrc file `.cshrc_old` to your new `.cshrc` file.

Type the following command to load the new `.cshrc` file to the command shell.

```
source .cshrc
```

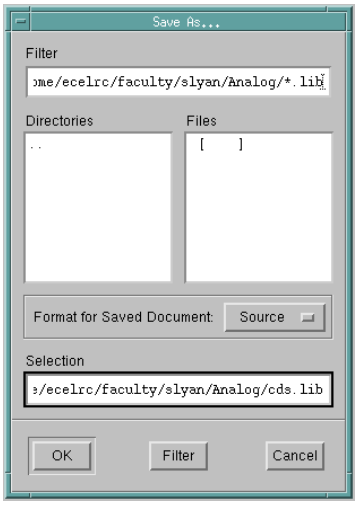


5. Use any text editor to create `~/Analog/cds.lib`. Include the following lines in the `cds.lib` file.

```
-- cds.lib example file for Analog Integrated Circuit Design
INCLUDE /usr/local/packages/cadence/local/cdssetup/ncsu.lib
```

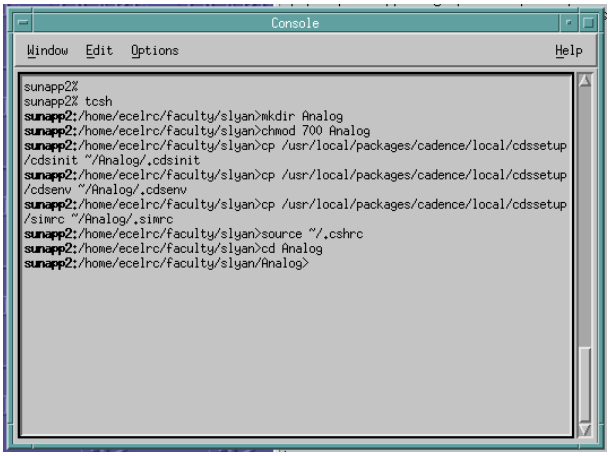
Or you can download `cds.lib` file from `http://www.ece.utexas.edu/~slyan/labs/setup/cds.lib`,

following the similar steps as above. Save the `cds.lib` file in your "`Analog`" directory instead of your HOME directory! Make sure that the path is similar to the one in the screen snapshot below.



6. Change current directory to "`Analog`" directory by typing the following command,

```
cd Analog
```



Type "`ls -la`" to check if you have these 4 files: `.cdsinit , .cdsenv , .simrc , cds.lib .`

```
sunfire2:/home/ecelrc/faculty/slyan/Analog>ls -al
total 152
drwx--l---   2 slyan    students    4096 Sep 10 17:23 ./
drwx--s--x  23 slyan    students   28672 Sep 10 17:17 ../
-rw-------   1 slyan    students    6848 Sep 10 17:23 .cdsenv
-rw-------   1 slyan    students   29418 Sep 10 17:20 .cdsinit
-rw-------   1 slyan    students    2475 Sep 10 17:23 .simrc
```

Now we are ready to use Cadence tools. Type "`icfb &`" command. The '`&`' at the end means we run icfb

at the background.

```
icfb &
```

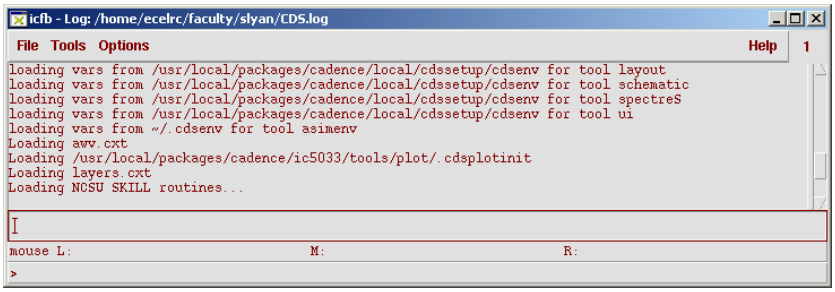
Next time when you run Cadence, type the following commands.

```
tcsh
cd Analog
icfb &
```

3.3. Starting Cadence

You should always start Cadence from the directory that contains the Cadence user setup information. In our case it is your "`~/Analog`" directory. You can start Cadence by typing the following command (assuming you start from your HOME directory).

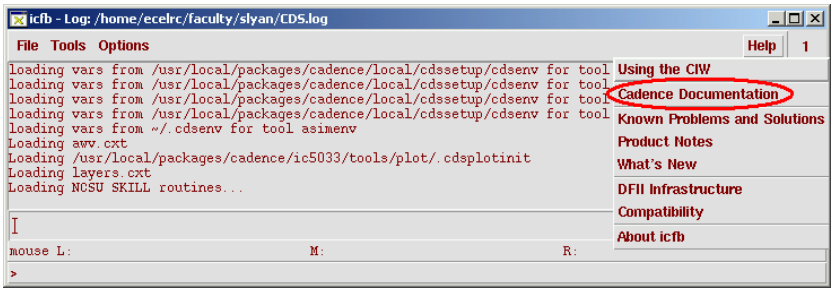
```
tcsh
cd Analog
icfb&
```

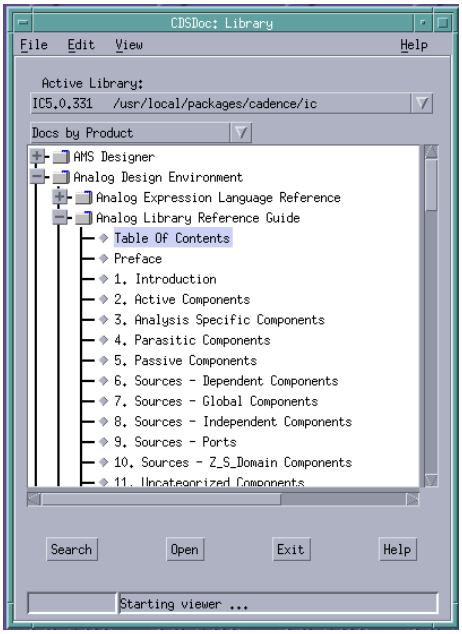


After a short while, a window called "CIW" (Command Interpreter Window) appears as shown above. CIW displays Cadence log file "`CDS.log`" at your home directory. When you use a specific Cadence tool (e.g. Virtuoso) and run a task (e.g. DRC), most of the times the results will be displayed in CIW. Thus you should check back on this CIW window very often to know what is happening for the tasks that you have initiated.

Your Cadence designs (schematic, layout, ...) are organized in libraries. You can go to menu "Tools → library manager" to start the library manager, which usually automatically starts after CIW appears, thus you may not always need to click menu "Tools → library manager". How to use library manager will be explained in Section 3.4.

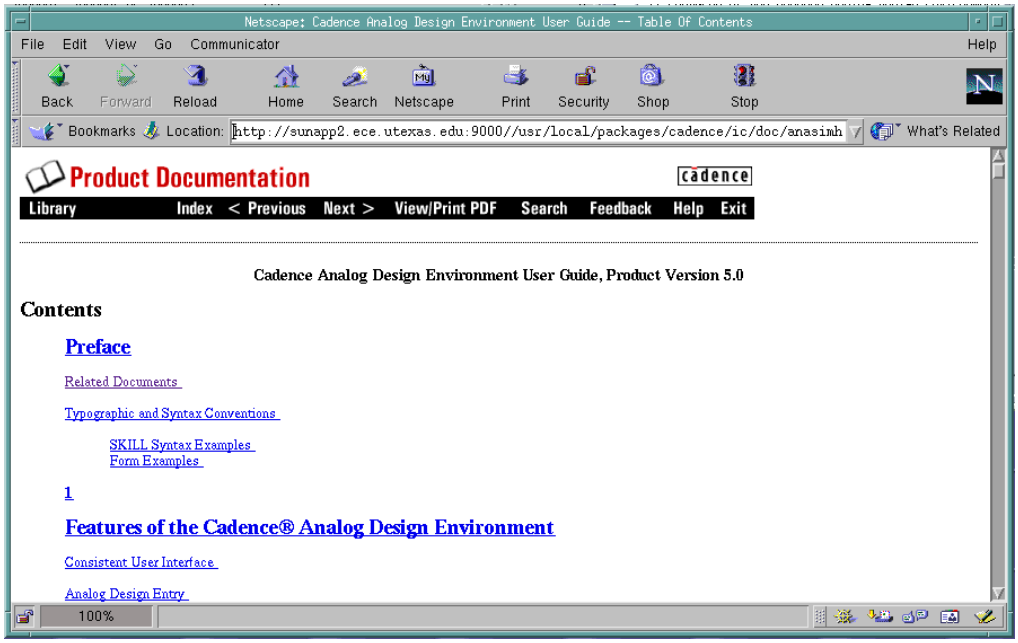
You can read Cadence documents through the Help menu of CIW as shown below. Or, your can type "`cdsdoc`" (for IC 5.0.33 package) on the command Console of UNIX (not in CIW) to read the documents for all Cadence tools.





The above cdsdoc window will appear. Double click the item that you would like to open, or highlight the item, and click “Open” button, you may expand or collapse the document tree, or open the document html webpage in the web browser, such as netscape. You may need to start web browser on sunfire machine before starting ‘cdsdoc’ command on sunapp machine.

You can open or save PDF version of the documentation by click “View/Print PDF” button in the web browser. You may also find the PDF files in the sub-directories under /usr/local/packages/cadence/ic/doc on `sunapp` machines. Note that /usr/local/packages/cadence directory is not accessible on `sunfire` machines. Netscape and acroread (PDF file viewer) are only available on `sunfire` machines. Thus if you want to read the PDF documentation directly in acroread or other PDF viewer/editor applications, you may have to copy the PDF files from `sunapp` to `sunfire` or other computers, and view the files on `sunfire` or other computers.



3.4. The Library Manager

CIW and Library Manager are probably the two most important and most frequently used windows in Cadence tools. CIW shows the log information; Library Manager helps you to manage your designs which are generally organized in a hierarchy of libraries/cells/cell_views. When you start a design in Cadence for the first time, you have to create a library where you can store your designs (cells/cell_views). Every library is associated with a technology file that supplies all the color maps, layer maps, design rules, extraction parameters required to view, design, simulate and fabricate your design.

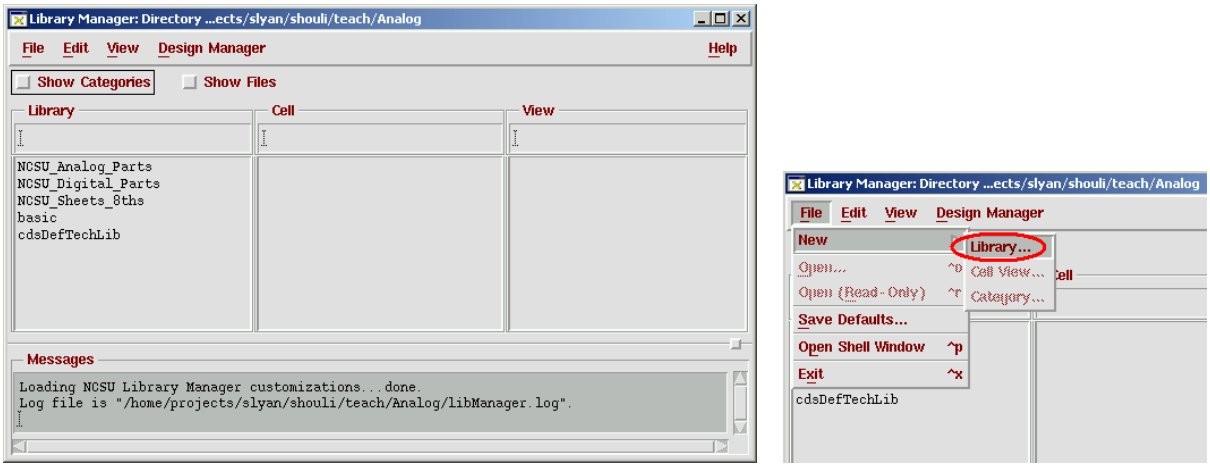
1. Starting Library Manager

The Library Manager usually automatically starts after CIW window appears. In case that the library manager has not started, or accidentally was closed, go to CIW window, click on "Tools→Library Manager". Library Manager looks like the window at the left (see next page).

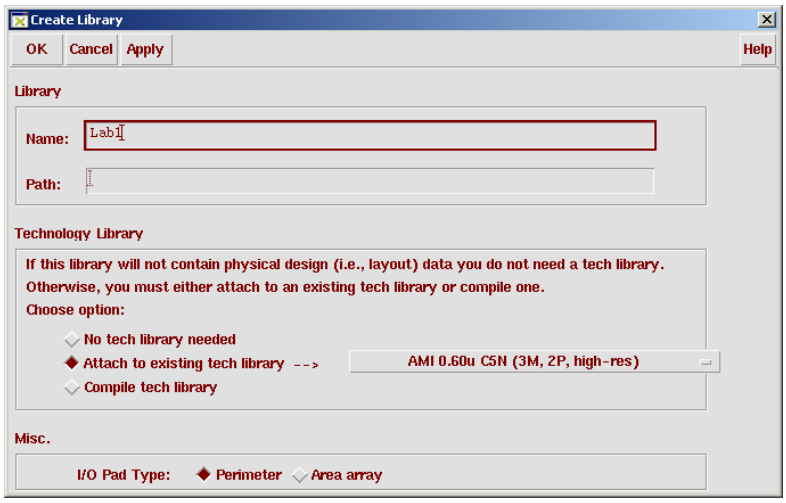
2. Create a new library

In Library Manager window, click on "File→New→Library..."

A dialog window pop up, as shown in the right. At the "`Name`" field, enter the name of the library that you want to create, in our case "`Lab1`", to store your design cells.

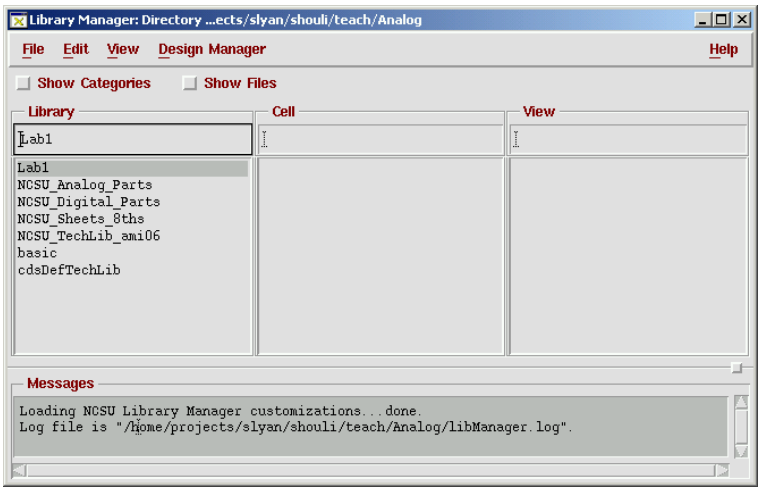


Attach this library to an existing technology (example shown: AMI 0.6 µm technology) by clicking the radio button "`Attach to existing tech library -->`" and choosing "`AMI 0.60u C5N (3M, 2P, high-res)`", thus the Cadence tools would know the technology information of your design (like SPICE models, DRC rules, ERC rules, etc.). Please do not enter or change any other fields or options.



*<***For information only***>* You can also use "`compile tech library`". This creates a local copy of the technology library in your run directory, thus it is not very efficient in the sense that, every user has a copy of an identical tech library, and if there is any change made to the main technology library, each user has to update his copy of the compiled tech library. Because of these two reasons, we prefer "`attach to existing tech library`" to "`compile tech library`".

Now your library manager window should look like,



You have not only created a library "`Lab1`" of your own, but also included its associated technology library "`NCSU_TechLib_ami06`" in your library path.

3.5. Editing Schematics Using Virtuoso Schematic Composer

At this point, you have created a library (in the example "`Lab1`") to store your design and can start the design process. For a full custom design, we start from creating a schematic. Then we simulate this design to verify its functionality and analyze its behavior to optimize the performance. After that we will do layout of the design. Layout will be introduced in Lab 2.

You can usually initiate a command in Cadence Schematic Composer and Layout Editor in three ways, (1) choosing a menu from the top menu bar; (2) clicking a button on the left toolbar of the window, or (3) pressing a *hot key*. All of three will give the same results. You can choose what you feel most convenient. For example, to add an instance, you can choose "Add → Instance …" from the menu bar, or you can click the button  from tool bar at the left side of the window, or you can simply press the hot key "i" from the keyboard.

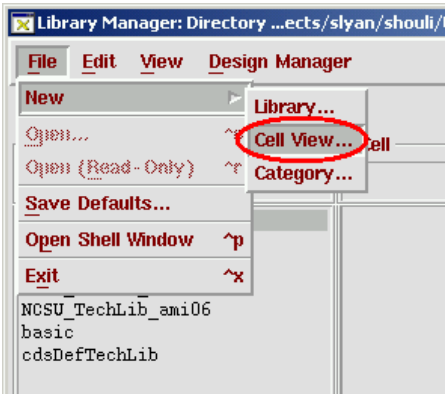
1 .Hot keys, menus, and tool buttons

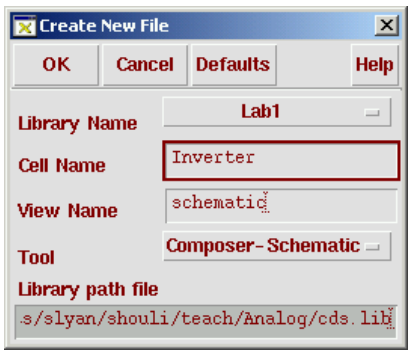
Function	Hot key	Menu	Tool button
Check and save	F8	Design → Check and Save	
Save	F2	Design → Save	
Add instances	i	Add → Instance…	
Edit properties	q	Edit → Properties	

Add wires	w	Add → wire (narrow)	
Label a wire	l	Add → Wire Name…	
Delete (a pin, label, wire, instance…)	DEL	Edit → Delete	
Add a pin	p	Add → Pin…	
Undo	u	Edit → Undo	
Zoom in	z	Window → Zoom → Zoom In	
Zoom in by 2X	^z	Window → Zoom → Zoom In By 2	
Zoom out by 2X	Z	Window → Zoom → Zoom Out By 2	
Fit the schematic	f	Window → Fit	

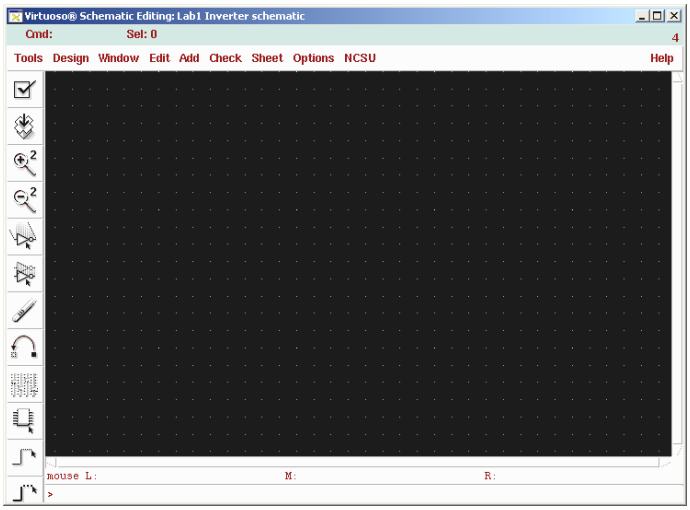
2. Creating a new schematic

In your Library Manager window, click on the "File→New→Cellview". A pop-up dialogue window appears. Click on the library name button and select "`Lab1`". Click on the "Tool" button to select "Composer-Schematic". The "View Name" will automatically change to "schematic". Enter the name of the cell you wish to design in "Cell Name". For this lab, we will design a CMOS inverter, with the cell name of "`Inverter`". Click "OK" after you type in the cell name.





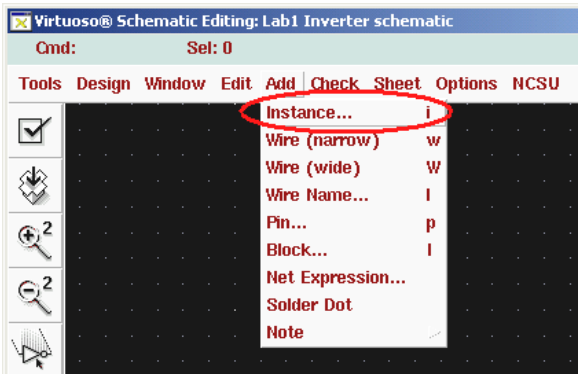
An empty schematic edit window will appear as shown below.

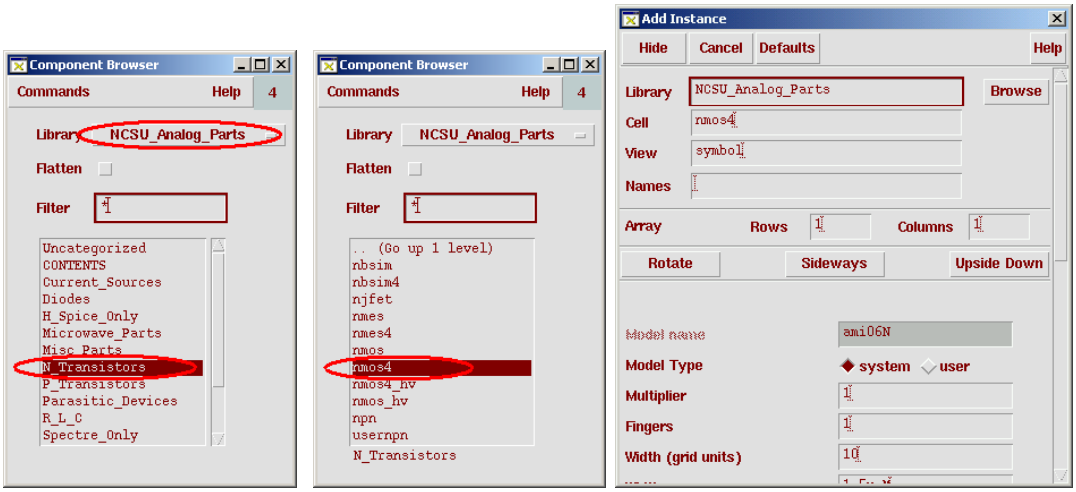


In the following steps, we will learn how to create an inverter schematic.

3. Adding an instance

To place an instance, e.g. an NMOS device, in your schematic, activate the schematic window, then click on menu “Add→Instance” (or type “i” hot key, or click  on the left side tool bar). The “Add Instance” dialogue box appears together with the “Component Browser” dialogue window. In case the “Component Browser” window does not appear, click on the “Browse” button in the “Add Instance” window to start it.

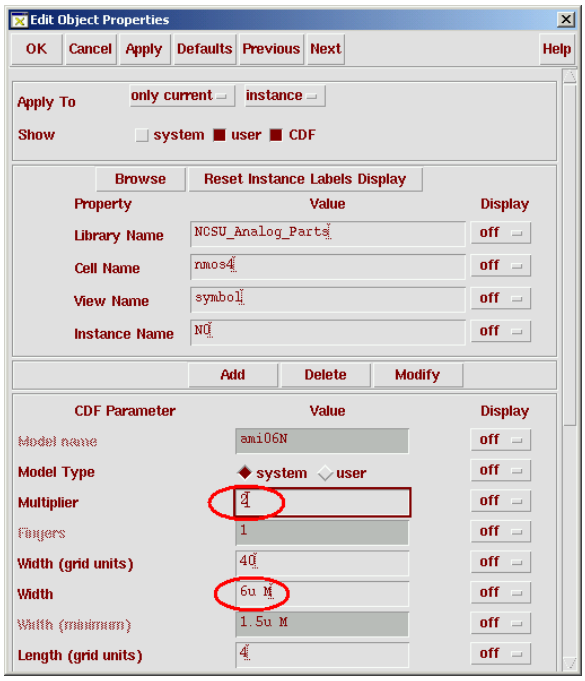




In “Component Browser” window, click on “Library” multi-selection button and choose “`NCSU_Analog_Parts`”. Click on “`N_Transistors`” and choose “`nmos4`” as your NMOS device (Note

that “nmos4” has 4 terminals. While “nmos” only shows 3 terminals, drain, gate, and source. The bulk terminal is hidden, and is specified in the properties window to a certain wire name through the wire label /name. Use of “nmos4” is encouraged to avoid some hidden mistakes. To place the instance, activate the schematic window and click the left mouse button to put the instance at the place desired.

Note in Cadence schematic composers and layout editors, a command will not terminate unless the user cancels or starts a new command. In this case, you can see another instance is ready to be placed right after you placed the first instance. To terminate the current operation (which is “Add Instance” in this case), press <ESC> key on the keyboard. In fact, you can always cancel the current operation in schematic or layout editors by pressing <ESC> key.



4. Setting the properties

To set the properties of the instance that you just placed, or any object in your schematic or layout design, select the object by clicking on it and then go to “Edit→properties→Object...” (or by typing “q” hot key). An object properties editing form will pop up. The following is the properties window of an NMOS Transistor. Change the “Width”, “Length”, and “Multiplier” properties to “Width”=6u, “Length”=0.6u, and “Multiplier”=2, as shown in the right side window snapshot. Keep other parameters unchanged.

In a similar manner, we can add an instance of PMOS transistor. Set the properties of the PMOS transistor as, “Width”=6u, “Length”=0.6u, and “Multiplier”=4.

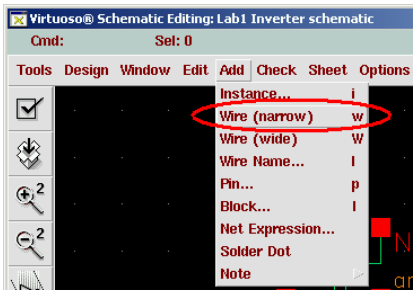
Note that in the “Edit Object Properties” window, “Apply to” multi-selection button shows “only current”. If you want to change the properties of multiple instances at the same time, you may highlight multiple instances first, and change “Apply to” multi-selection button to “all selected”, change the properties in the properties list, and confirm the change by click “Apply” or “OK” button at the top of the “Edit Object Properties” window. The properties of all highlighted instances are changed to the same set of values. The difference between “Apply” or “OK” is that, if you click “Apply” button, the properties in the list will be applied, but the “Edit Object Properties” window remains open for further action; if you click “OK” button, the properties will be applied, and the “Edit Object Properties” window closes. You can always cancel current change and close the window by clicking “Cancel” button. You can clear the entire properties and set to default values (usually bank or empty fields) by clicking “Default” button.

<***For information only***> You may find “Apply”, “Previous” and “Next” function buttons useful when you highlight multiple objects and change the properties of the instances one after another.

Note that you can also change of the properties of the device in the “Add Instance” window as show previously while you place the instance.

5. Wiring the instances

To connect the PMOS and NMOS devices or any other devices, click on “Add→Wire (narrow)” in the schematic window (or type “w” hot key). Click on the terminal where the wire starts, and then click at the terminal where the wire ends, a wire will be automatically added. If you are not satisfied with the automatic wiring, you can remove the wire by selecting the wire and press <DEL> key. The reroute it manually. This time, instead of directly clicking on the terminals where the wire starts/ends, you can click the left mouse button wherever you want to confirm the suggested wiring shown in yellow color.

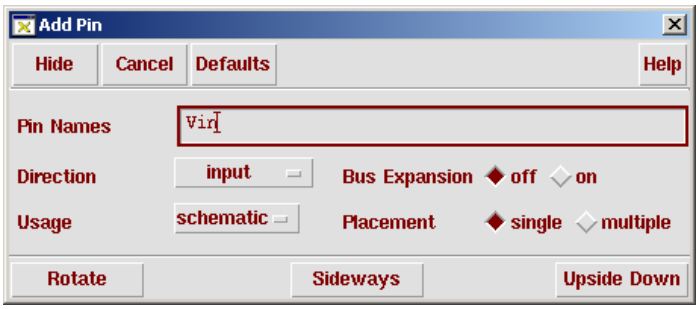


If you want to stop the wire somewhere empty instead of connecting it to a terminal, double click your left mouse button and a dangling wire is created. In general dangling wire should be avoided. However, in some cases, for example, if you want to label this wire or add a pin to this wire, a dangling wire makes sense.

<***For information only***> Click on “Add→Wire Name...” (or type hot key “l” – the lower case of “L” not a number “1”) in the schematic window, the following dialogue box would appear. You can specify a name for each wire, but we generally only add labels for those wires that you may have interest to see signal voltages in simulation, or the wires that you may want to connect with the same wire label, or the global wires like VDD or VSS. An “!” denotes a global wire name, such as “VDD!” and “gnd!”. Global wires are not encouraged in most cases, like global variables are not often used in software programming practice. If you specify the positive and negative power supplies of a cell (such as an inverter or an amplifier) to “VDD!” and “VSS!”, it is not flexible if you want to connect the cell to different power supplies other than “VDD!” and “VSS!”.

6. Creating pins

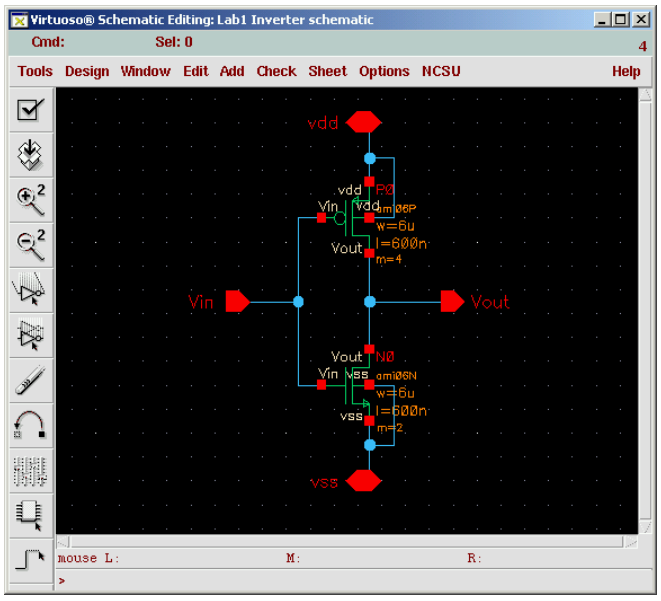
Click on “Add→Pin...” (or type hot key “p”) in the schematic window, “Add Pin” window would appear. We have to specify its input/output behavior for each pin.



For an input pin, choose the “Direction” to be “`input`”. Specify the “Pin Names” (“`Vin`” in the example). Activate the schematic window and click the left mouse button to place the pin in the schematic.

Similarly we can add an output pin (“Vout” in the example) to the output of the cell. The “Direction” of “vdd” and “vss” pins are specified as “InputOutput”.

Please note that Cadence automatically labels internal nodes, usually you don’t have to explicitly label them.



7. Check and save the design

After the design has been completed, click menu “Design→Check and Save” to check and save your design. Go to the CIW window to see if there is any error in your design. The following schematic shows the completed design of the inverter. Use the widths, lengths, and multipliers of the transistors given in the following screen shot.

You can save the design (without checking) by clicking “Design→Save”, or clicking  button, or pressing hot key <F2>. Saving the design frequently is encouraged to avoid loss of work due to unexpected reasons, such as loss of power, network failure, and etc.

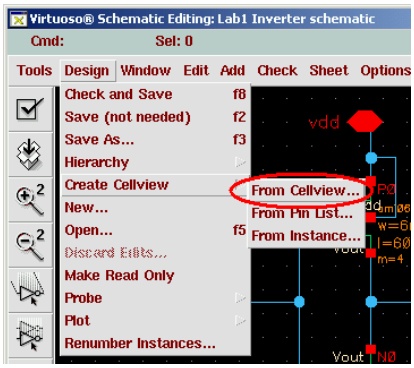
3.6. Creating Inverter Symbol

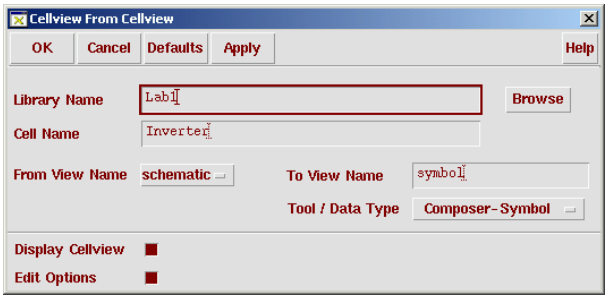
Symbols are useful when the schematic design is done hierarchically. At a higher level of abstraction, we would like to use a symbol to replace the schematic details of a cell. The symbol of a schematic cell should define all the inputs and outputs of that cell.

There are two ways to create a symbol. If you already have a schematic design, you can create its symbol right from the cell. If you don’t have a schematic design yet or you want to manually create a symbol for the cell, you can start from scratch manually.

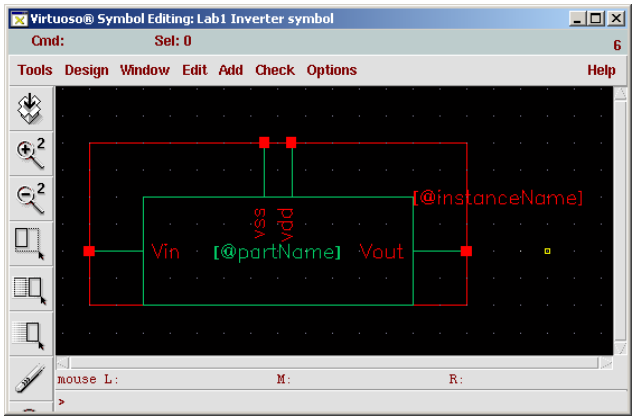
In this section, we introduce how to create a symbol from schematic cell view.

Click on “Design→Create Cellview→From Cellview” menu in the schematic edit window, a pop up dialogue box will appear.



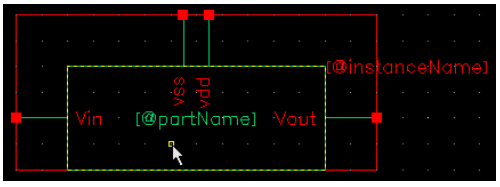


We can use the default setup in this window, thus click “OK”. The symbol edit window will appear that contains a default symbol created automatically by Cadence. It has a red box that encloses the green colored inverter symbol. This red box defines the actual size that a symbol will occupy in a parent schematic. Even though you can change the size of this red box to an arbitrary size, it is preferred to exactly fit the symbol within the red box. The small red squares indicate the pins. [@InstanceName] and [@PartName] are display variable holders for instance name (such as “I0”) and part name (such as “Inverter”) when the inverter symbol is placed in a parent schematic. The following figure shows the symbol.

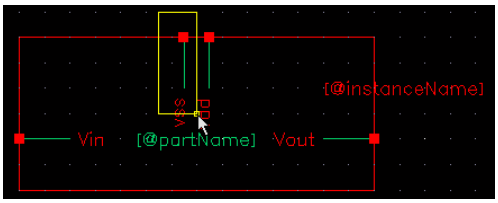


If you don’t like this rectangular symbol that is automatically created by the tool, you can edit it or create a symbol of your own. In the following paragraph, we will explain how to edit a symbol.

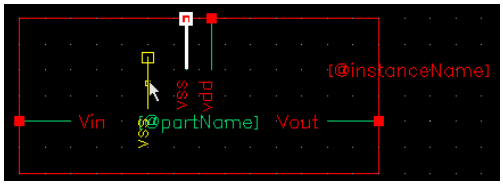
When the pointer is moved to above the rectangle, a yellow dashed rectangle appears along with the green rectangle as shown in the following figure. The yellow dashed rectangle indicates the object under the focus of the pointer. If you click the left mouse button, the object under focus will be selected and highlighted. Highlight and select the green rectangle by clicking the left button. Press <DEL> key to delete it.



If you press and hold the left mouse button and move the pointer, a yellow rectangle will appear. Any objects inside of the yellow rectangle will be selected (highlighted) when you release the left mouse button. Select (and highlight) the Vss pin.

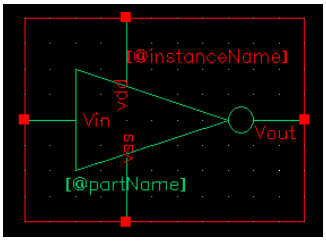


Select “Edit→Move” or press the hot key “m”, the following window will appear. Click “Rotate” button twice. Note that you may not see the button is pressed if you click too fast. Move the pointer the symbol edit window and click the “vss” pin, you may see as the following figure.



Move the vss pin to wherever you like, and click the left mouse button, the vss pin is dropped to the new location.

Choose menu “Add→Shape→Line” or click the tool bar icon  , and draw a triangle. Choose menu “Add→Shape→Circle” to add the small circle of the inverter. Move the pins, triangle, and the small circle, if necessary, by selecting and moving them one by one. Resize the red rectangle if necessary by choosing menu “Edit→Stretch” or by clicking the tool bar icon  , and stretch one side of the red rectangle. You finally have a symbol as shown below.

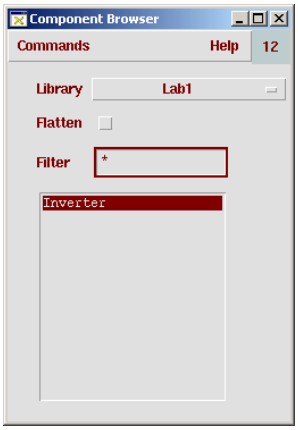


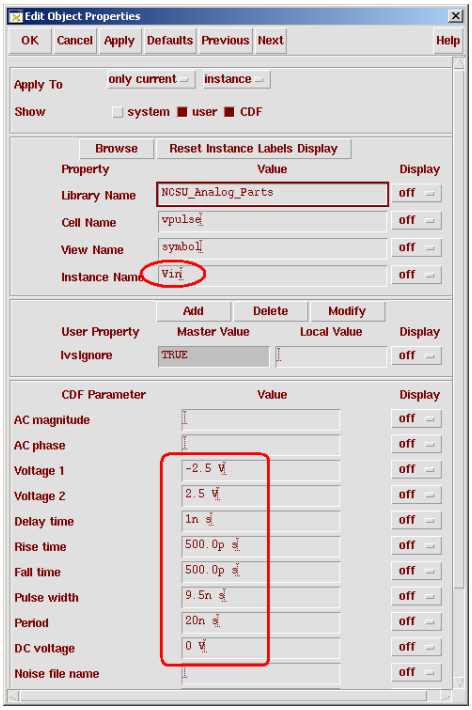
3.7. Create Inveter_tst Schematic Cell

Create another schematic cell, “`Inveter_tst`”, to test the inverter that we just created.

Following the procedures to create “`Inverter`” schematic, create a new schematic “`Inveter_tst`”. In the new “`Inveter_tst`” schematic editing window, create an instance of the inverter by choosing menu

"Add→Instance". Component Browser window appears, and choose "Lab1" library and "Inverter" cell, as shown in the right figure. Place the inverter symbol in "`Inveter_tst`" schematic.





Similarly, place positive power supply source Vdd, negative power supply source Vss, and input voltage source of the inverter, Vin. Vdd and Vss are "vdc" cell from NCSU_Analog_Parts library. Both Vdd and Vss have a DC voltage of 2.5V. Please pay attention to the polarities of the supply voltages. Vin is "vpulse" cell from NCSU_Analog_Parts library. Specify parameters of Vin as shown in the properties window in the right.

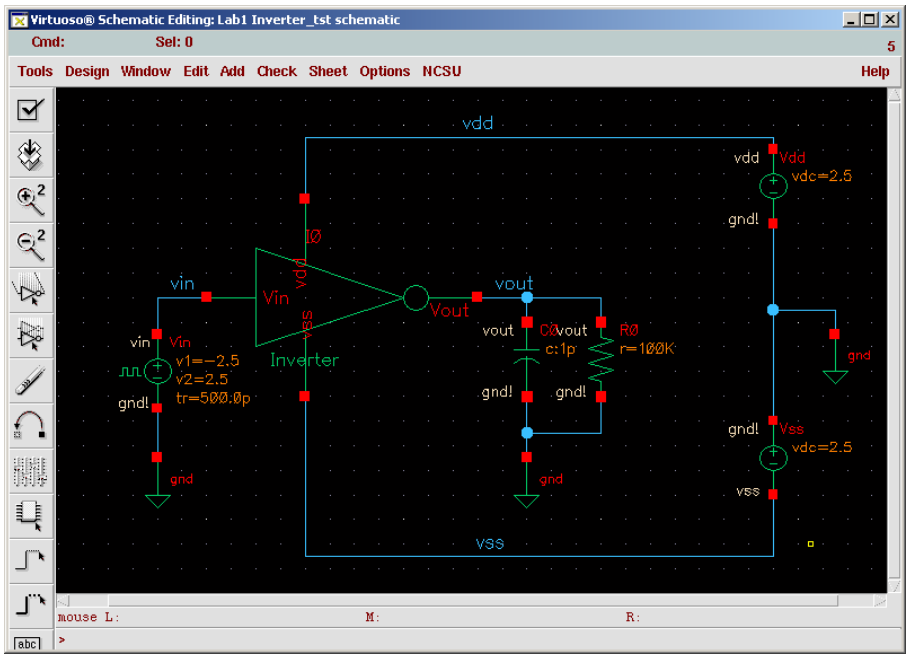
Add R0 and C0 as the load resistor and capacitor of the inverter.
We choose R0 = 100 kΩ, and C0 = 1 pF. Both of them are from NCSU_Analog_Parts library.

All components are summarized in the table below.

Instance Name	Library	Category	Cell	Comments
I0	Lab1		Inverter	
Vin	NCSU_Analog_Parts	Voltage_Sources	Vpulse	Properties in the above figure.
Vdd	NCSU_Analog_Parts	Voltage_Sources	vdc	DC voltage = 2.5V
Vss	NCSU_Analog_Parts	Voltage_Sources	vdc	DC voltage = 2.5V
R0	NCSU_Analog_Parts	R_L_C	res	Resistance = 100 kΩ
C0	NCSU_Analog_Parts	R_L_C	cap	Capacitance = 1 pF

Wire the components together. The `Inveter_tst` final schematic looks like below.

*<***For information only***>* You may find the Composer Tutorial from Cadence useful at /usr/local/packages/cadence/ic/doc/comptut/comptut.pdf on sunapp machines.



3.8. The Analog Environment

1. Preparing the model files for simulation

The simulator we will use will be specterS. Before proceeding further, we need to copy the model files for the NMOS and the PMOS transistors to `~/cadence/models/spectre` directory. By default the spectreS simulator searches device models in that directory.

Create `~/cadence/models/spectre` if the directory does not exit. Note that "~" is your HOME directory. If you are not sure how to create `~/cadence/models/spectre` directory, follow the following procedures. (1). Type "cd" in your UNIX shell command line to change current directory to your home directory. (2). Create `~/cadence` directory by typing "mkdir cadence" if you do not have `~/cadence`. (3). Type "mkdir cadence/models" to create `~/cadence/models` directory. (4). Type "mkdir cadence/models/spectre" to create `~/cadence/models/spectre` directory.

The CMOS technology that we use is AMI 0.6u. The NMOS and the PMOS model files are named as ami06N.m and ami06P.m respectively. You can view the properties of NMOS and PMOS transistors in schematic Inverter (cell Lab1/Inverter). Note that the "model" properties of NMOS and PMOS are ami06N and ami06P respectively.

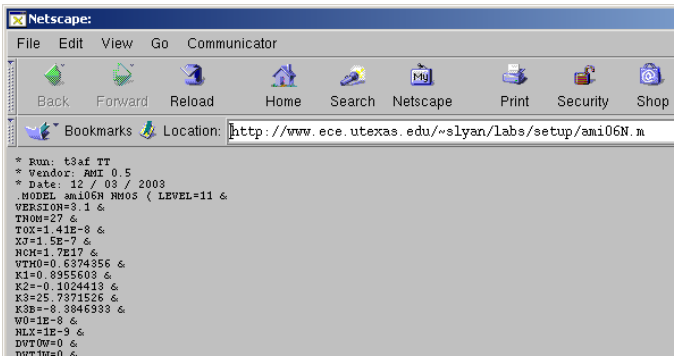
Download the ami06N.m and ami06P.m files from the following http links.

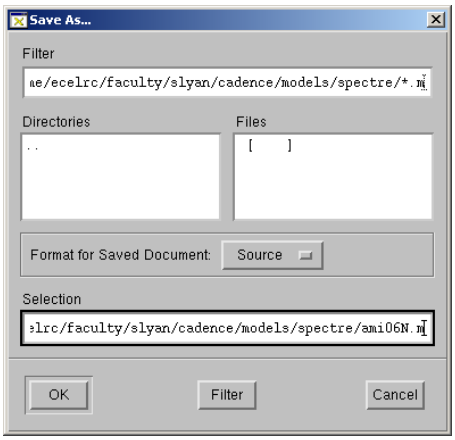
Ami06N.m at `http://www.ece.utexas.edu/~slyan/labs/setup/ami06N.m`

ami06P.m at `http://www.ece.utexas.edu/~slyan/labs/setup/ami06P.m`

You can download the model files using *netscape* web browser on `sunfire` machines, as shown below.

Save the ami06N.m file as ~/cadence/models/spectre/ami06N.m by choose “File → Save As …”, choose the ~/cadence/models/spectre directory in the “Save As …” window as shown below.



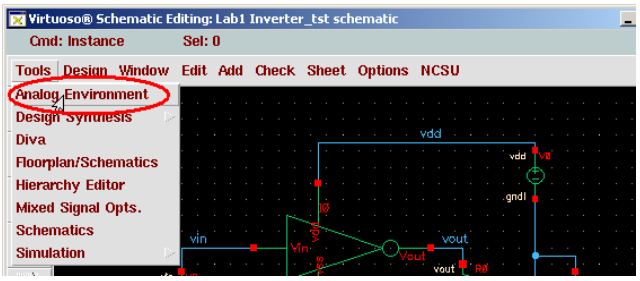


Repeat the above procedures to save the ami06P.m file as ~/cadence/models/spectre/ami06P.m .

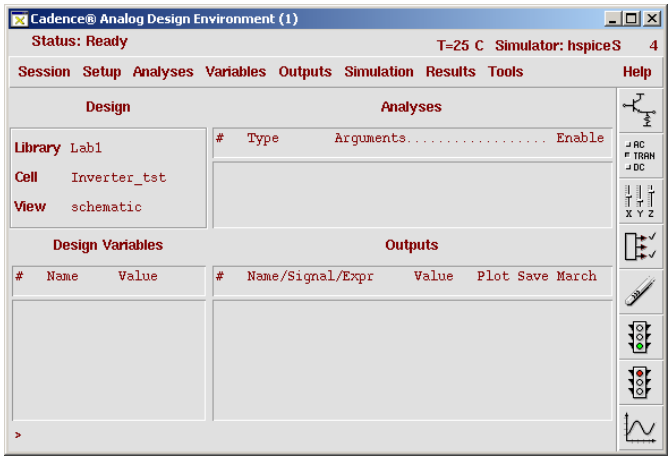
We will open Analog Environment to simulate `Inveter_tst` schematic. We will specify the outputs to be saved, and you will also specify analysis types, do simulations, and plot the simulation results in the waveform window.

2. Starting the Analog Environment from schematic window

In the `Inveter_tst` schematic editing window, choose menu “Tools → Analog Environment”.

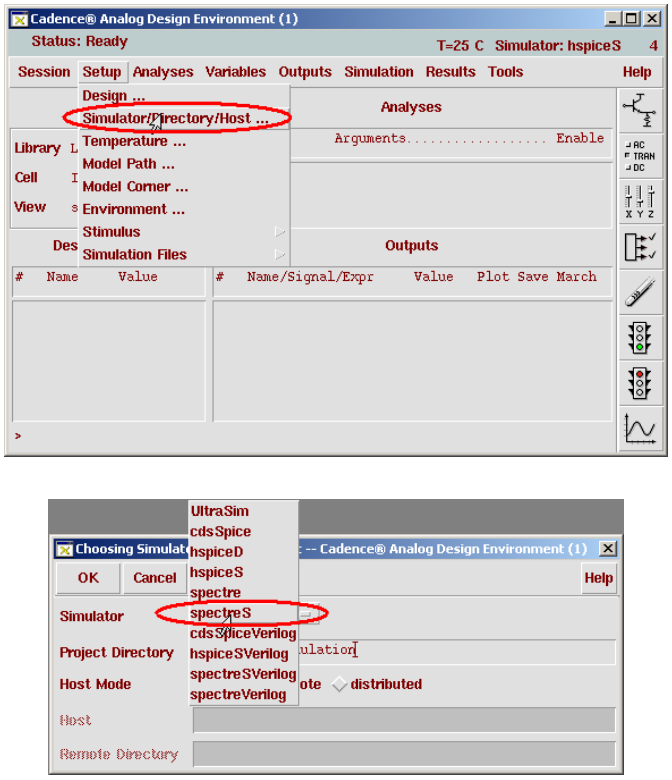


Analog Design Environment window will appear.



3. Selecting the simulator

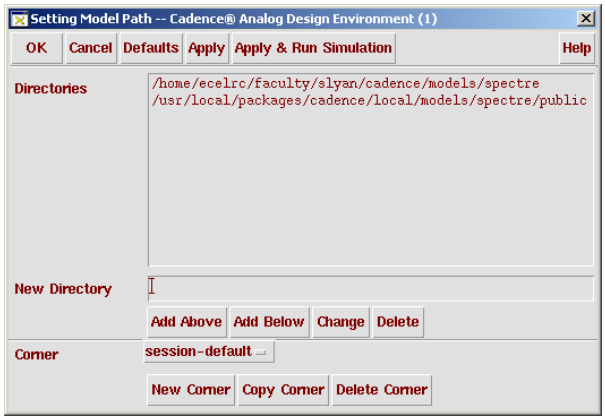
In the Analog Design Environment window, choose menu "Setup → Simulator/Directory/Host …". Simulator/Directory/Host setup window appears and set "Simulator" as "specterS" as shown in the following window screen snapshot.



4. Setting the model search paths

Now we will set the model directory, thus the Analog Design Environment can find the model files for simulation. In the Analog Design Environment window, choose menu "Setup → Model Path". The following window appears. Note that `~/cadence/models/spectre`, the directory we used to store the model files, is the first item in the "Directories". Note that in the Inverter schematic, the "Model name"

in the Properties Window of the NMOS and PMOS transistors is “ami06N” and “ami06P”. When simulating the schematic, the Analog Design Environment will search “ami06N.m” and “ami06P.m” starting from the first directory listed in the “Directories” in the “Setting Model Path” window, until the right model files are found or the last directory in the “Directories” list has been searched.



5. Choose analysis (simulation) types

DC Analysis

In the Analog Design Environment window, click menu “Analyses→Choose…”, the Choosing Analysis window will appear. Following the following steps:

(1) Click “dc” radio button in the “Analysis” section.

(2) In the “DC Analysis” section, select “Save DC Operating Point” button.

(3) Click “Component Parameter” button in the “Sweep Variable” section.

(4) Click “Select Component” button.

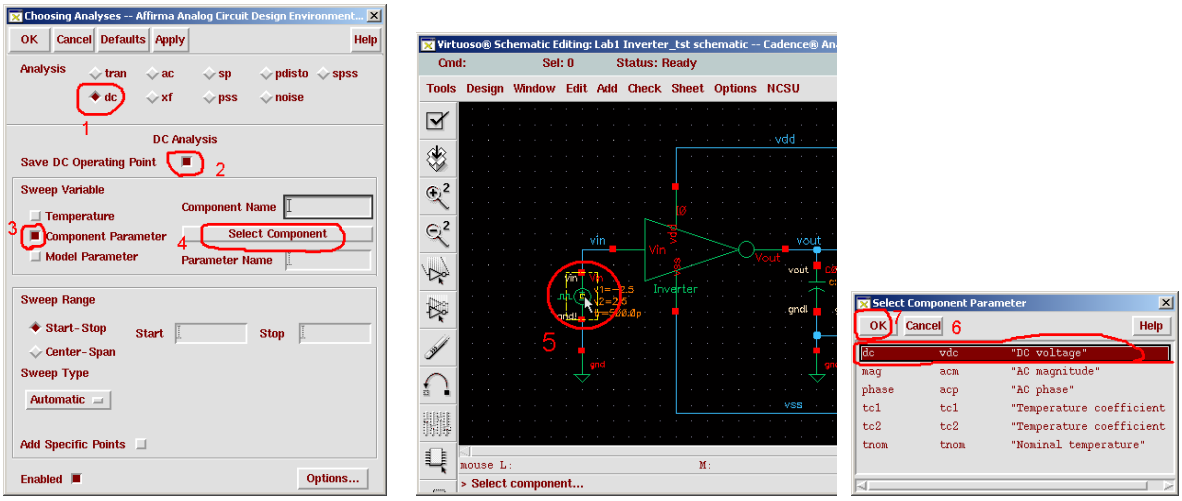
(5) The schematic window rise up, click the input voltage source Vin.

(6) A new window, Select Component Parameter window, appears as shown below in the right. Click “dc – vdc – DC voltage” line.

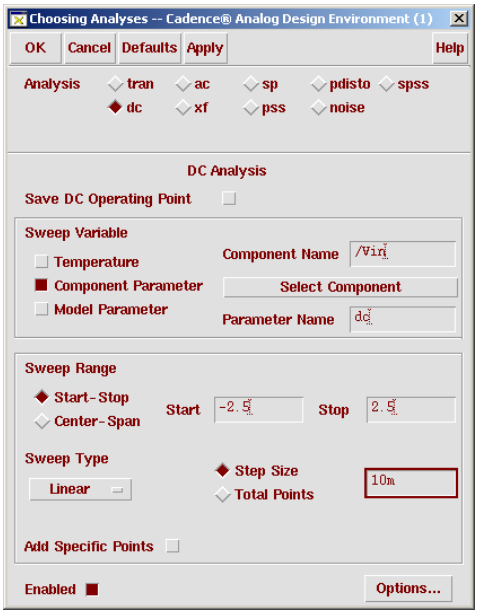
(7) Click OK of the Select Component Parameter window. In the Choosing Analysis window, “Component Name” text field will be filled as “/Vin” and “Parameter Name” text field shows as “dc”. You can also save steps (4) to (6) and type directly “/Vin” in the “Component Name” text, and “dc” in the “Parameter Name” text field.

(8) In the “Sweep Range” section, click “Start-Stop” button. Specify –2.5 as “Start” and 2.5 as “Stop”.

(9) In the “Sweep Type” section, “Automatic” is selected by default. You can change to “Linear”, and specify the “Step Size” as 10m (V).



After the above steps, the Choosing Analysis window will look like,



Click “Apply” button to apply (save) the change.

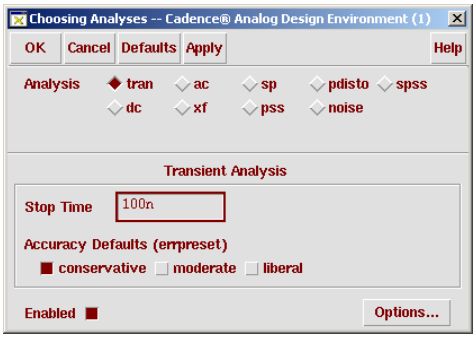
Transient Analysis

Following the following steps to specify transient analysis options in Choosing Analysis window.

(1) Click “tran” radio button in the “Analysis” section.

(2) In the “Transient Analysis” section, type “100n” as the “Stop Time”.

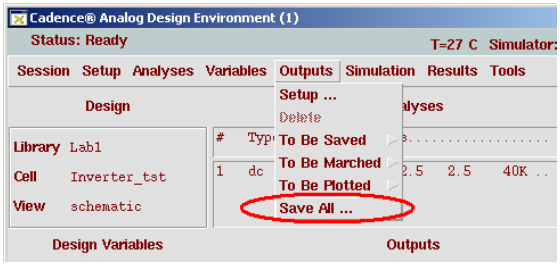
(3) Click “conservative” button as the “Accuracy Defaults (errpreset)”.

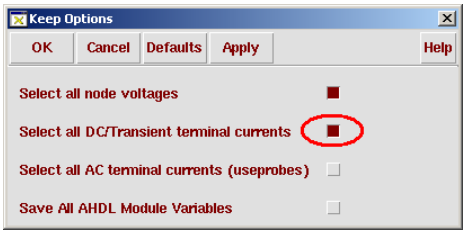


Click “OK” button to apply (save) the change and close the Choosing Analysis window.

6. Specify the outputs to be saved

In the Analog Design Environment window, choose menu “Outputs → Save All …”. The Save All window pops up as shown below. The option button of “Select all node voltages” is on by default. Click “Select all DC/Transient terminal currents” option button to select it. Click OK button to save the change and close the window.

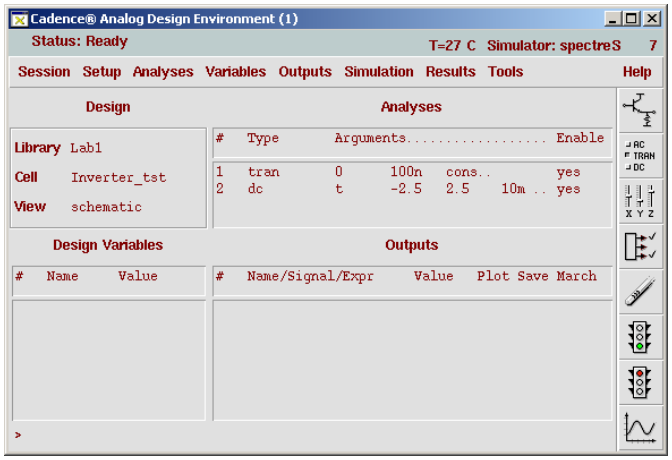




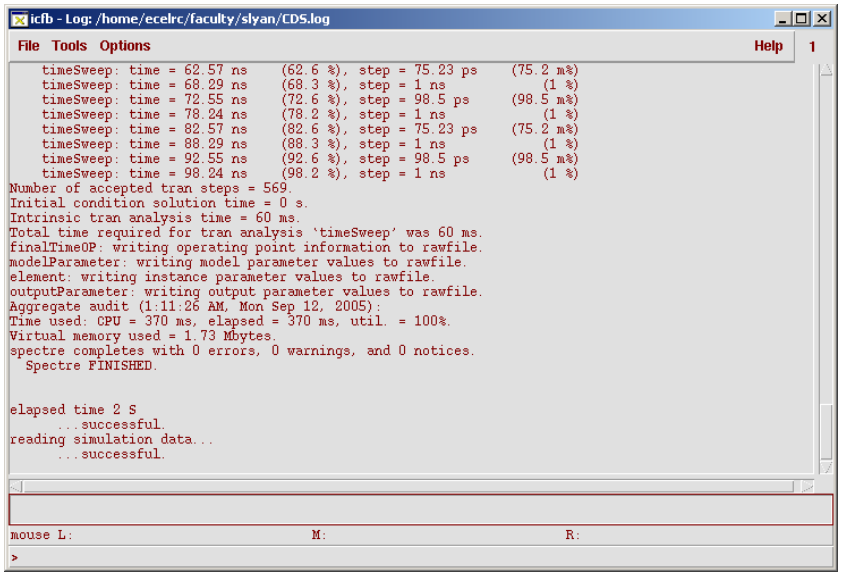
<***For information only***> Or you can choose “Outputs → To Be Plotted → Select from Schematic” and then the schematic window will rise up and becomes active. Click a wire to select a voltage ouput and a device pin to select a current output, when finished hit <ESC>.

7. Run simulations

The Analog Design Environment window should look like the following snapshot. You can start simulation by selecting menu “Simulation → Run” or simply clicking the button  at the right side of the Analog Design Environment window.



Check the CIW window for the simulation log information.



8. Waveform window

View/plot simulation results

You can plot the simulation results in Waveform window through the following steps.

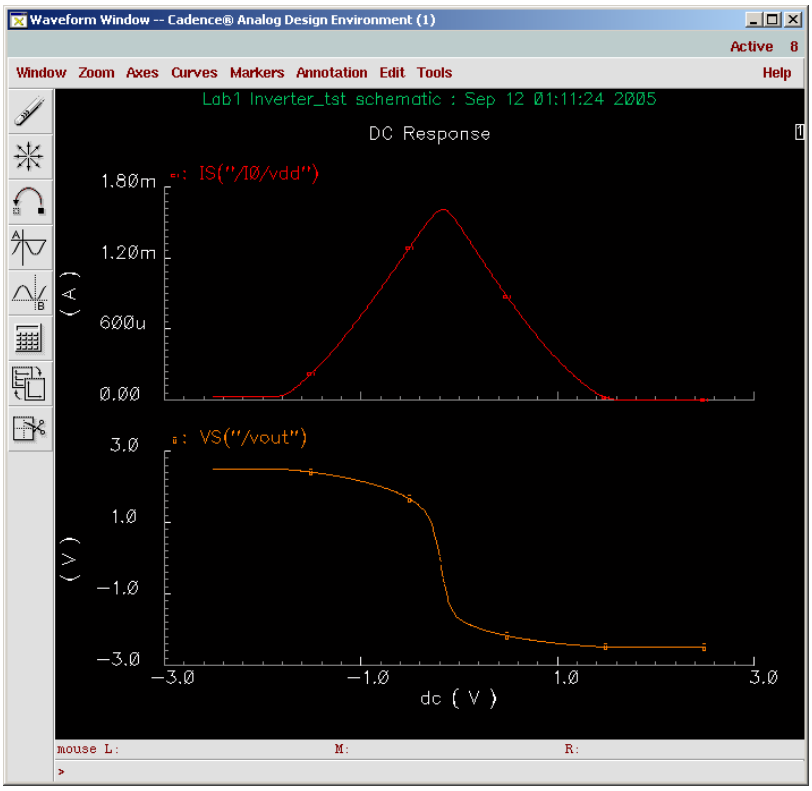
(1) Choose "Results → Direct Plot → DC" or "Results → Direct Plot → Transient Signal".

(2) The schematic window rise up. You can plot a voltage waveform by clicking a net and/or a current waveform by clicking the pin of a component. After you have selected all the voltages/currents, press <ESC> key.

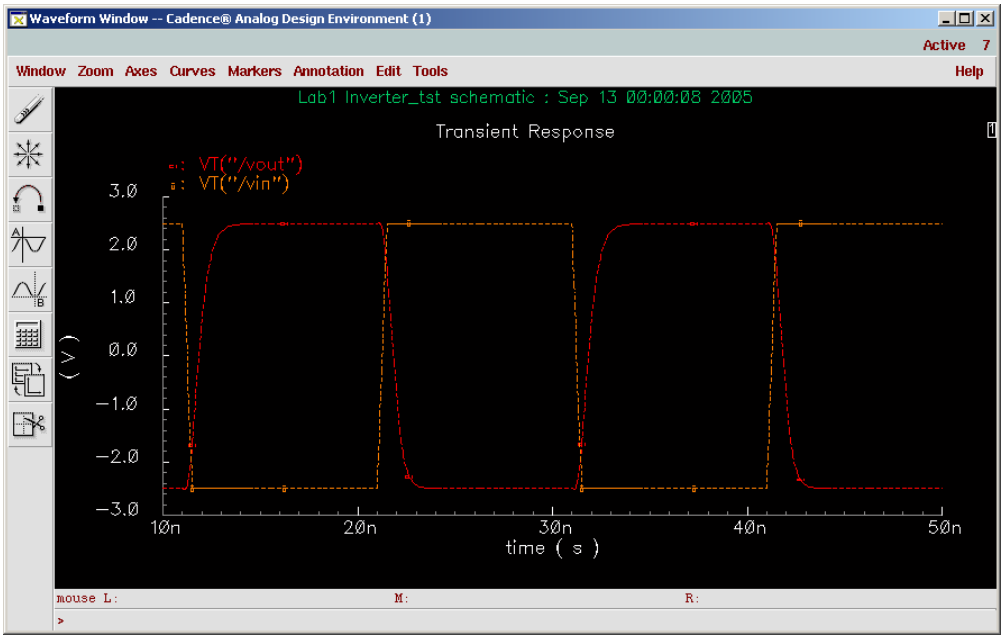
If you use "Outputs → To Be Plotted → Select from Schematic", after successful simulation, the required signals are automatically plotted.

The following figure show inverter supply current and output voltage, with input voltage swept from –

2.5V to 2.5V. The waveform can be obtained by choosing “Results → Direct Plot → DC” and selecting the output voltage of the inverter and the current through “vdd” pin (the small red square) of the inverter.



The input/output voltages of the inverter in the transient simulation are shown below. The waveform can be obtained by choosing “Results → Direct Plot → Transient Signal” and …



Some useful commands of the waveform window

Function	Hot key	Menu	Tool button
Fit all	f	Zoom → Fit	
Zoom in	z	Zoom → Zoom In	
Zoom out	Z	Zoom → Zoom Out	
Marker A	a	Markers → Crosshair Marker A	
Marker B	b	Markers → Crosshair Marker A	
Switch axis mode			
Open calculator window		Tools → Calculator…	
Print hard copy		Window → Hard Copy	
Update/redraw the waveform(s) from the simulation output data *	Window → Update Result		

* Waveform window will not automatically update the waveform if you have re-run the simulation (with possibly changed schematic parameters).

Last updated on Feb. 11, 2006

References:

[1] ELEN474 Analog Integrated Circuit Design Lab Manual, Texas A&M University, 1998

[2] Cadence on-line documentation, command “cdsdoc” in UNIX shell.