



HSPICE Manual

ChangNoh Yoon & Prof. Jinsang Kim
CSA & VLSI Lab.



Environment Setup

1. First step is to install Linux-like environment in your PC. Download freeware cygwin tool. If it is already installed, skip this process

www.cygwin.com

Cygwin Home
Cygwin/X Home
Red Hat Cygwin Product
Community
• Reporting Problems
• Mailing Lists
• Newsgroups
• Gold Stars
• Mirror Sites
• Donations
Documentation
• FAQ
• User's Guide
• API Reference
• Acronyms
Contributing
• Snapshots
• Source in CVS
• Cygwin

© GNU + Cygnus + Windows = **cygwin™**

What Is Cygwin?
Cygwin is a Linux-like environment for Windows. It consists of two parts:

- A DLL (cygwin1.dll) which acts as a Linux API emulation layer providing substantial Linux API functionality.
- A collection of tools which provide Linux look and feel.

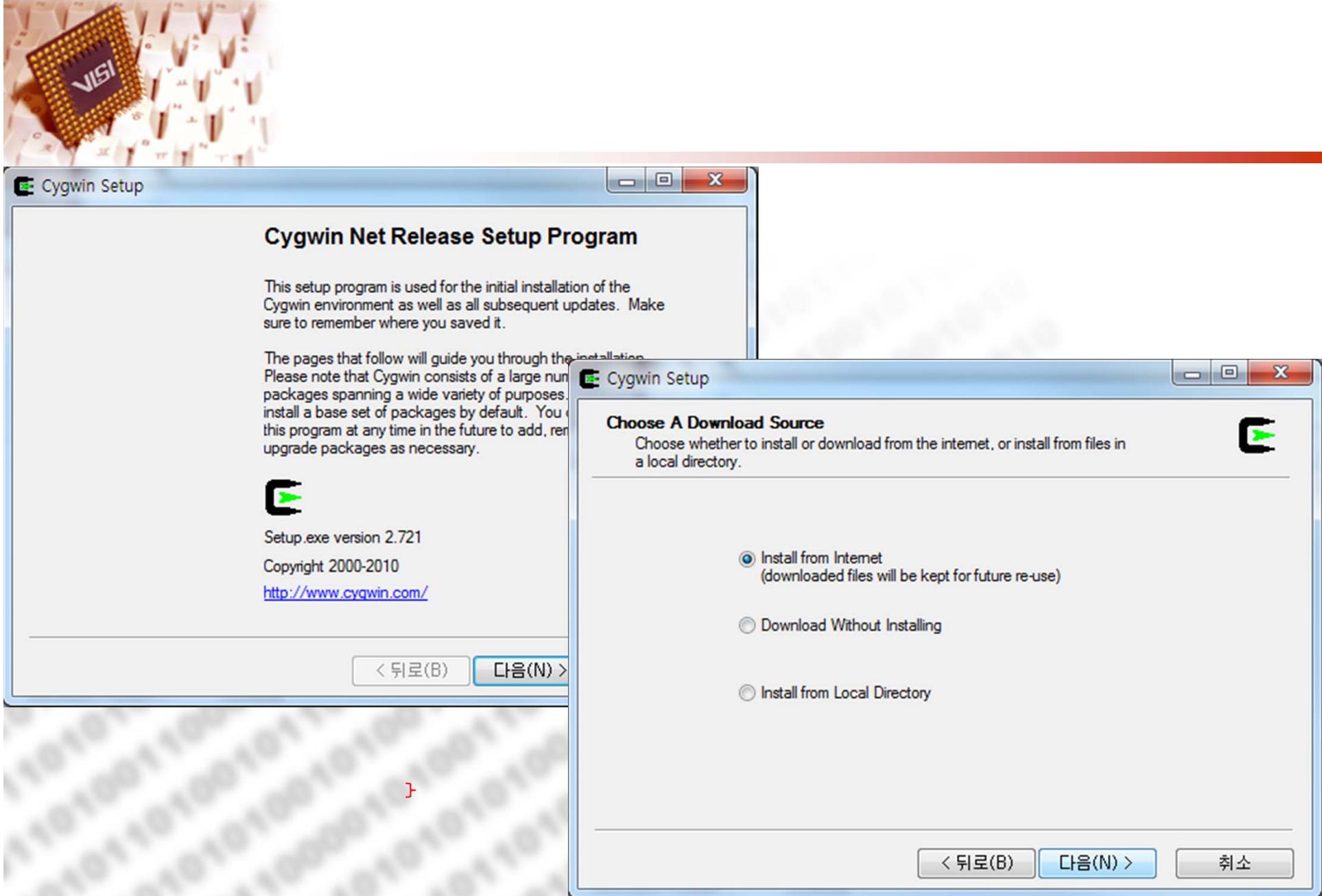
The Cygwin DLL currently works with all recent, commercially released x86 32 bit and 64 bit versions of Windows, with the exception of Windows CE.
Note that the official support for Windows 95, Windows 98, and Windows Me will be discontinued with the next major version (1.7.0) of Cygwin.

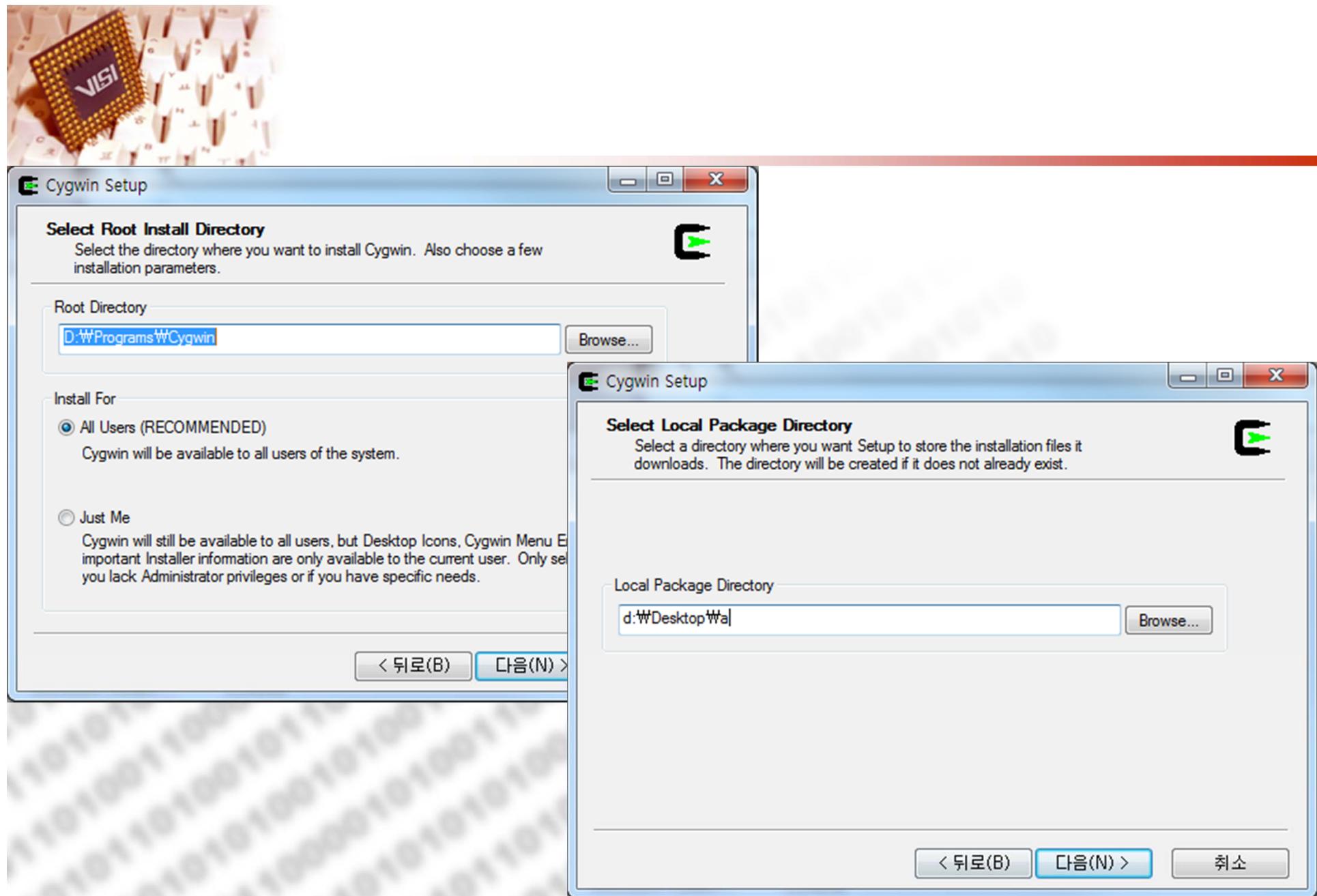
What Isn't Cygwin?

- Cygwin is not a way to run native linux apps on Windows. You have to rebuild your application *from source* if you want it to run on Windows.
- Cygwin is not a way to magically make native Windows apps aware of UNIX® functionality, like signals, ptys, etc.

Install Cygwin Now!

press this icon and press run
Press "next" until







Cygwin Setup

Select Your Internet Connection

Setup needs to know how you want it to connect to the internet. Choose the appropriate settings below.

Direct Connection

Use IE5 Settings

Use HTTP/FTP Proxy:

Proxy Host

Port

< 뒤로(B) 다음(N) >

Cygwin Setup

Choose A Download Site

Choose a site from this list, or add your own sites to the list

Available Download Sites:

- http://ftp.itm.ac.in
- ftp://ftp.jaist.ac.jp
- http://ftp.jaist.ac.jp
- ftp://ftp.ijj.ad.jp
- http://ftp.ijj.ad.jp
- ftp://ring.aist.go.jp
- ftp://ftp.kaist.ac.kr**
- http://cygwin.xd-mirror.nl
- ftp://ftp.eq.uc.pt
- http://ftp.eq.uc.pt
- ftp://mirrors.fe.up.pt
- http://mirrors.fe.up.pt
- ftp://ftp.sunet.se

User URL: Add

< 뒤로(B) 다음(N) > 취소

ftp://ftp.kaist.ac.kr



Select Packages
Select packages to install

Search **ssh** Clear

Keep Prev Curr Exp View Category

Category	New	B..	S..	Size	Package
<input type="checkbox"/> All	<input checked="" type="checkbox"/> Default				
Net <input checked="" type="checkbox"/> Default					
<input checked="" type="checkbox"/> 1.4b-1	<input type="checkbox"/>	<input type="checkbox"/>	26k	au	
<input checked="" type="checkbox"/> 1.2,2-1	<input type="checkbox"/>	<input type="checkbox"/>	?	lib	
<input checked="" type="checkbox"/> Skip	<input type="checkbox"/>	<input type="checkbox"/>	155k	lib	
<input checked="" type="checkbox"/> Skip	<input type="checkbox"/>	<input type="checkbox"/>	67k	lib	
<input checked="" type="checkbox"/> 5,6p1-2	<input type="checkbox"/>	<input type="checkbox"/>	748k	op	

< 뒤로(B)

Hide obsolete packages

Select Packages
Select packages to install

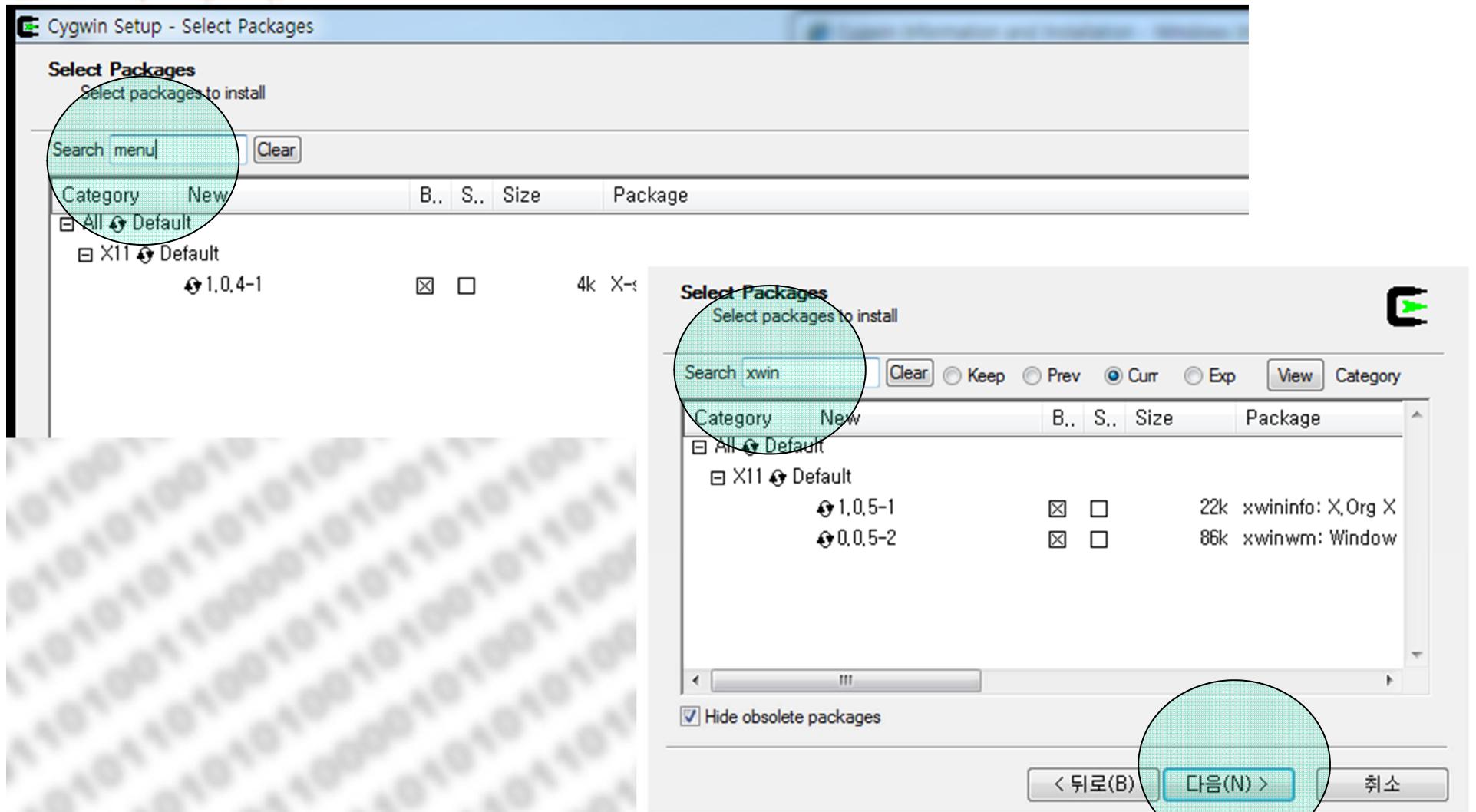
Search **window** Clear

Keep Prev Curr Exp View Category

Category	New	B..	S..	Size	Package
<input type="checkbox"/> All	<input checked="" type="checkbox"/> Default				
X11 <input checked="" type="checkbox"/> Default					
<input checked="" type="checkbox"/> 0,92,1pre2-1	<input type="checkbox"/>	<input type="checkbox"/>	1,704k	WindowMaker: Wiri	
<input checked="" type="checkbox"/> Skip	<input type="checkbox"/>	<input type="checkbox"/>	?	libWindowsWM: X,	
<input checked="" type="checkbox"/> Skip	<input type="checkbox"/>	<input type="checkbox"/>	9k	libWindowsWM-de	
<input checked="" type="checkbox"/> Skip	<input type="checkbox"/>	<input type="checkbox"/>	8k	libWindowsWM7: >	
<input checked="" type="checkbox"/> Skip	<input type="checkbox"/>	<input type="checkbox"/>	6k	windowswmproto:	

< 뒤로(B) 다음(N) > 취소

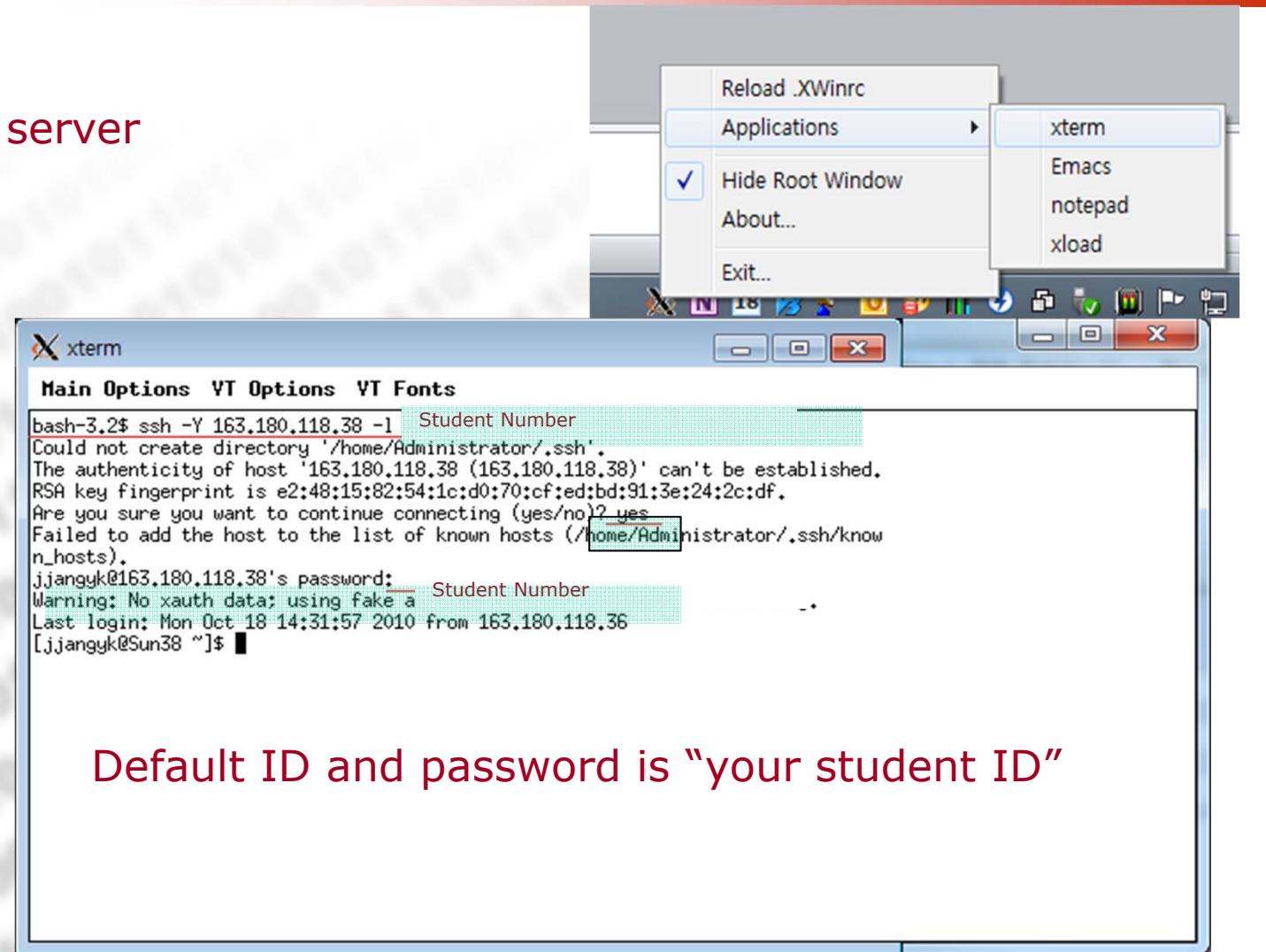
Hide obsolete packages





2. Connect HSPICE server

Cygwin-x->X win server

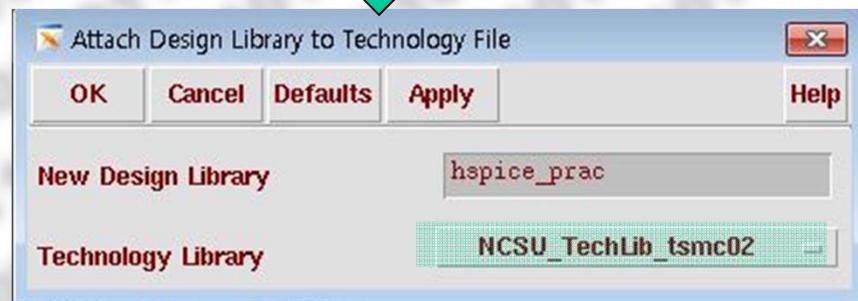
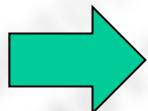


Default ID and password is “your student ID”



3. Attach technology library for HSPICE

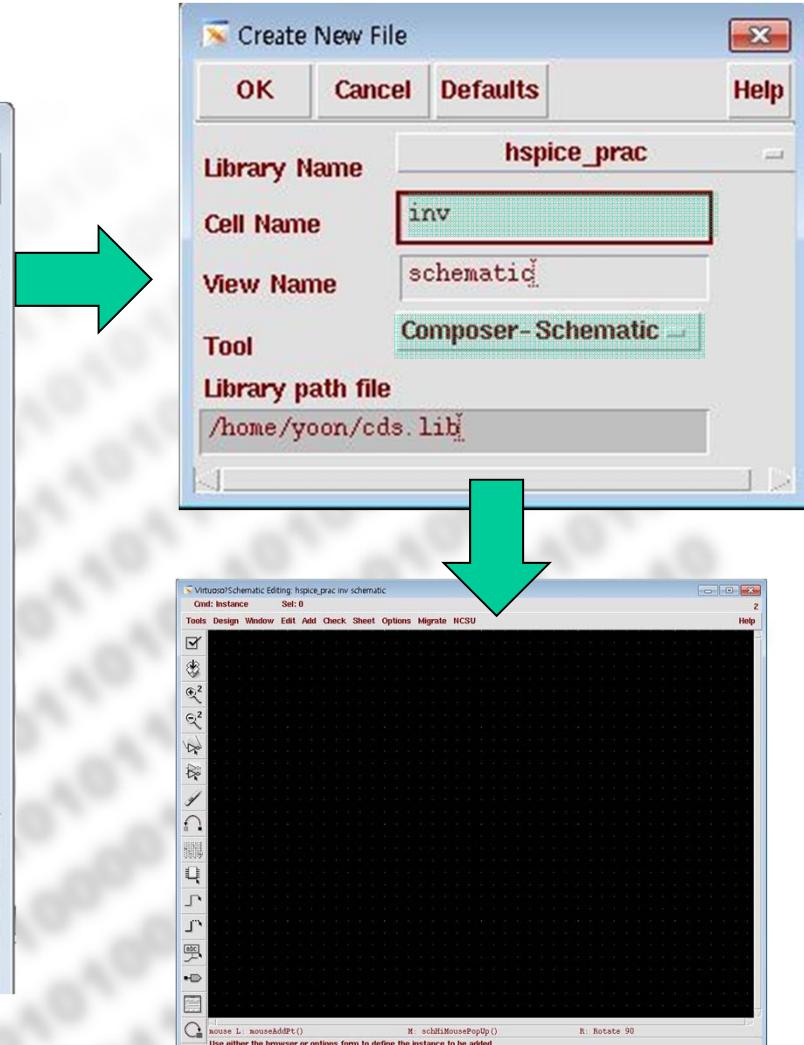
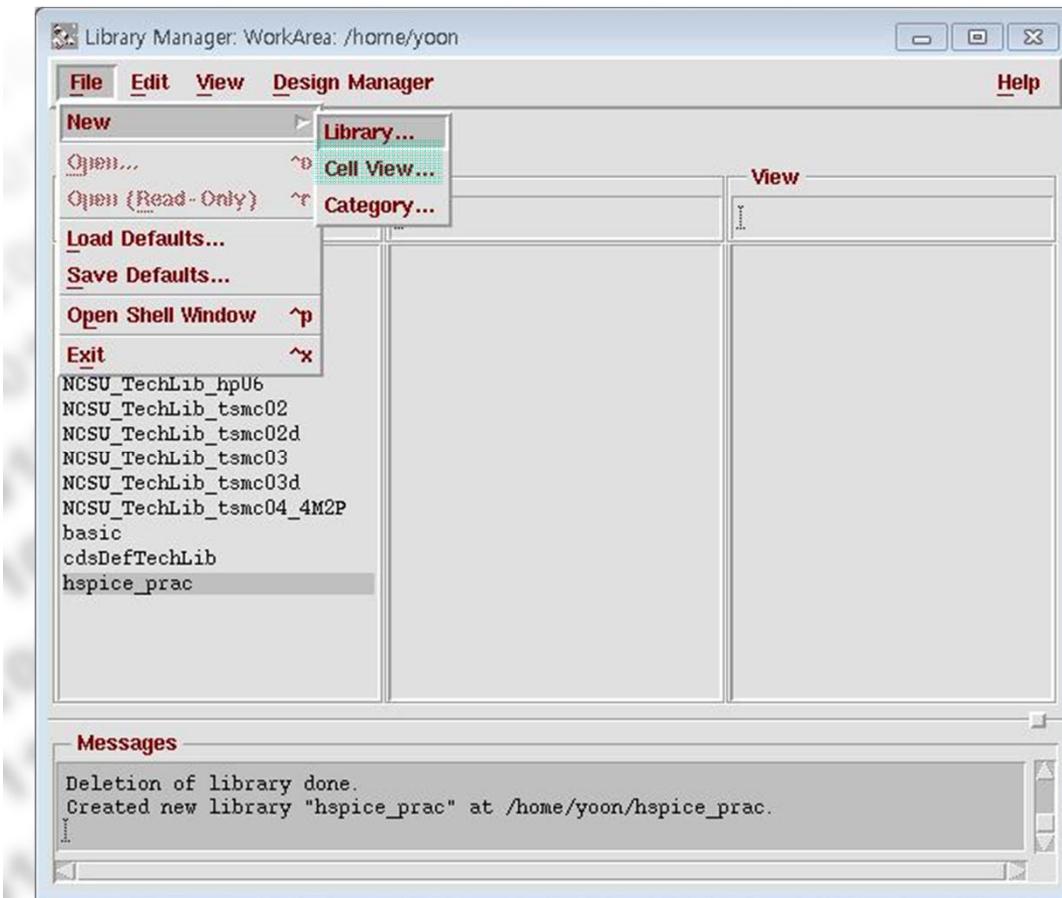
- File->New->Library





4. Open a Window for Schematic Capture

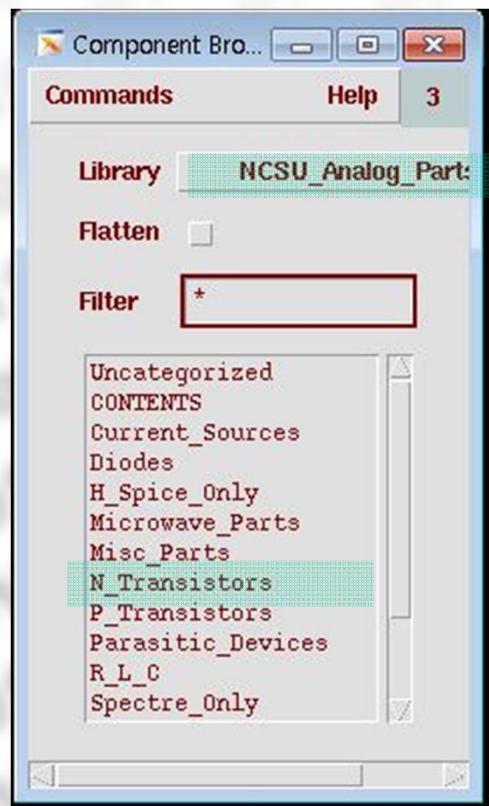
- File -> New -> Cell view



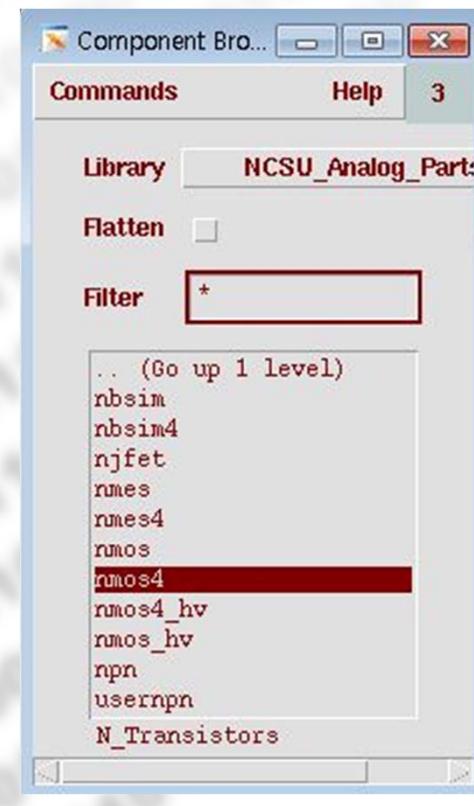
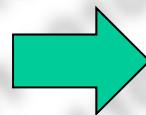


5. Load the components in your schematic

- Press 'I' (bind key) to load the Component Browser



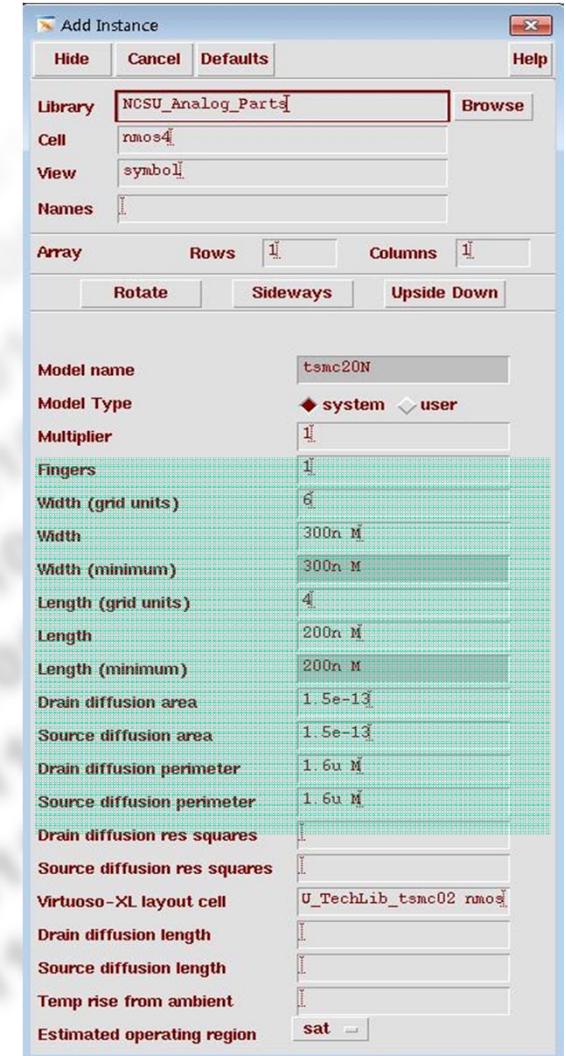
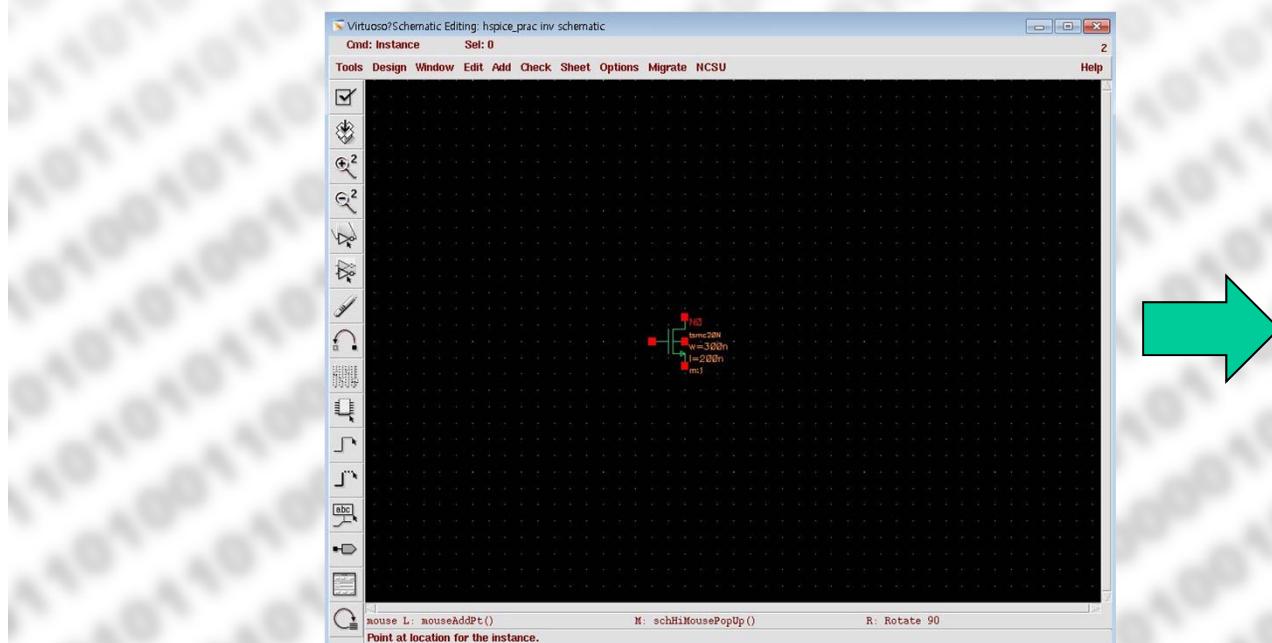
NCSU_Analog_Parts





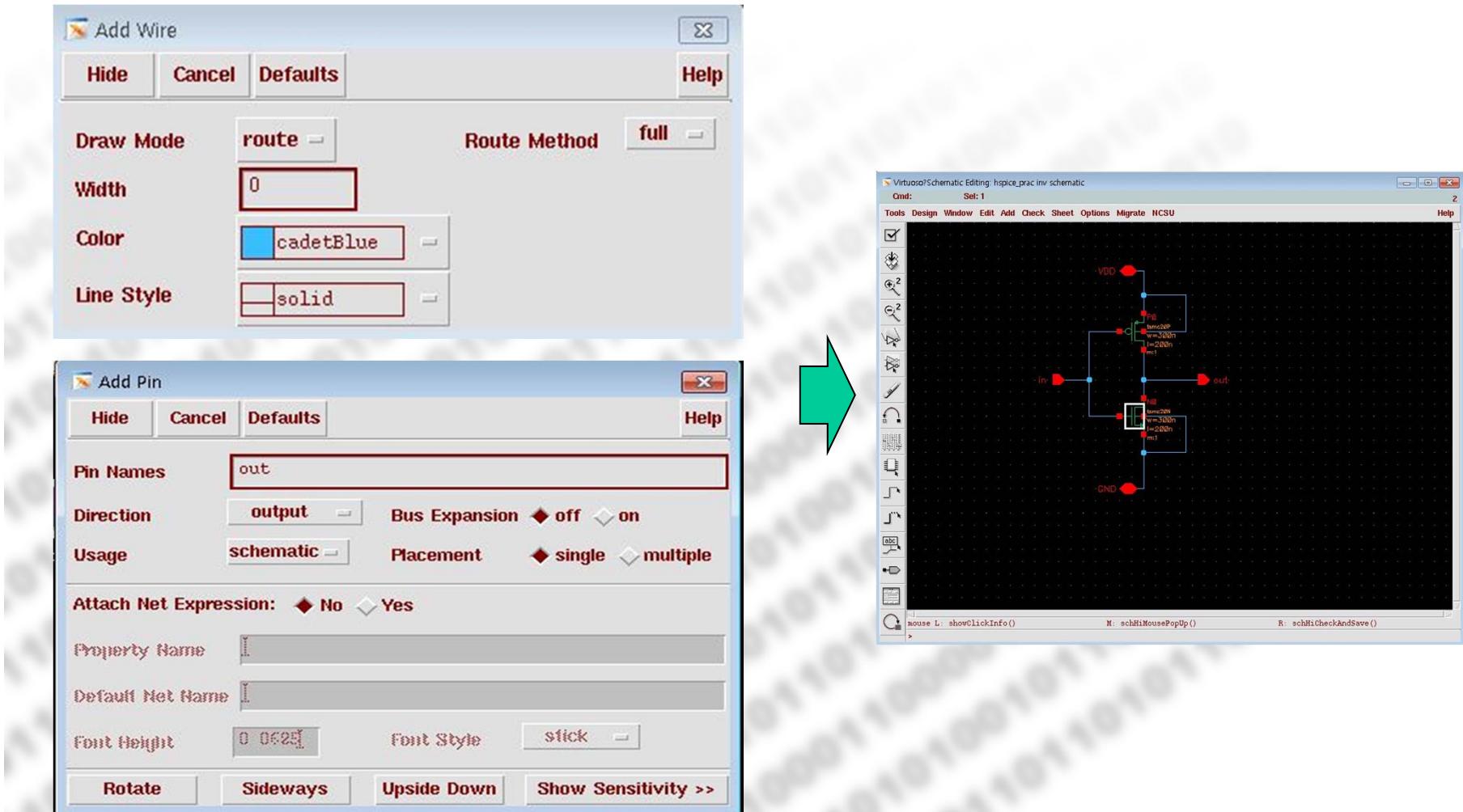
6. Adjust parameters if necessary

- Press 'Q' to change the parameter of TR
- "tsmc20N" provides 200nm feature size.





7. Draw the schematic





Helpful hot keys for drawing schematic

- These are the most frequently used bindkeys...

c: copy

q: property.

m: move

i: instance

p: pin

l (small L) : label (you can define a net name and also connect two separate wires with the same name)

9: wire highlight

ctrl-9: erase all highlight

shift-9: only erases nets that are clicked on

f: zoom to fit

shift-z : zoom out

ctrl-z : zoom in

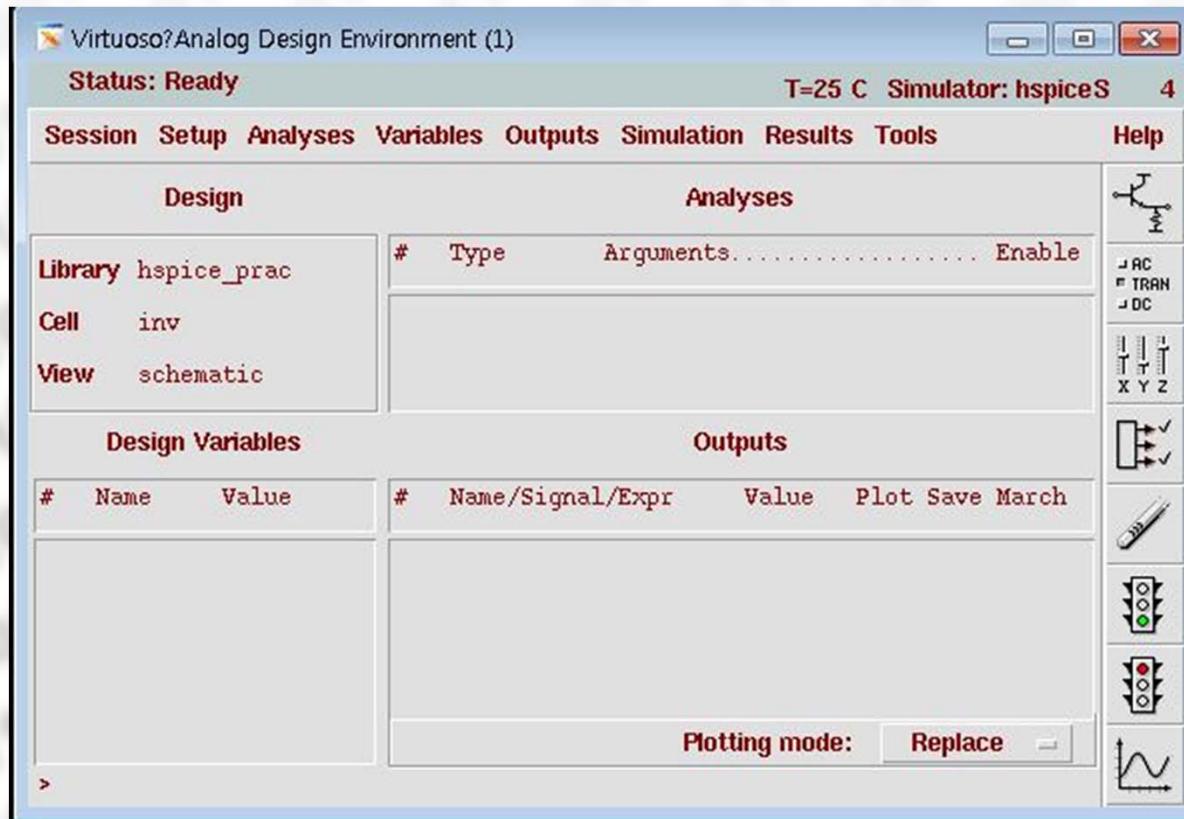
arrows: panning



8. HSPICE Environment Setup

- Now, you are ready to simulate your schematic. Before doing that, you need to preset environments for HSPICE. HSPICE simulation uses analog environment even if your design is digital.

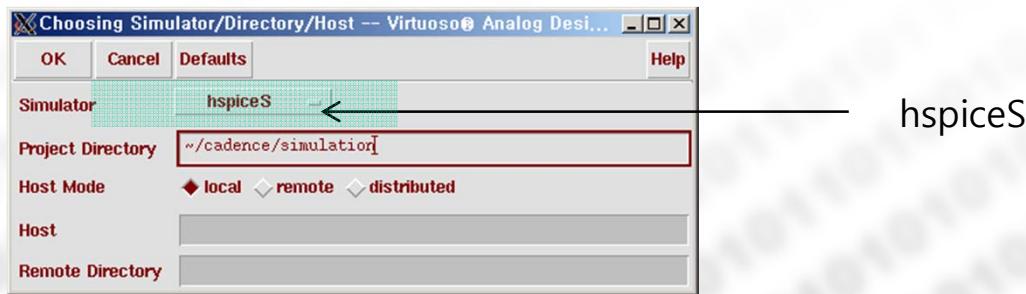
Tools -> Analog Environment





8. HSPICE Environment Setup

- Analog Design Environment
- 1. Setup->Simulator/Directory/Host...

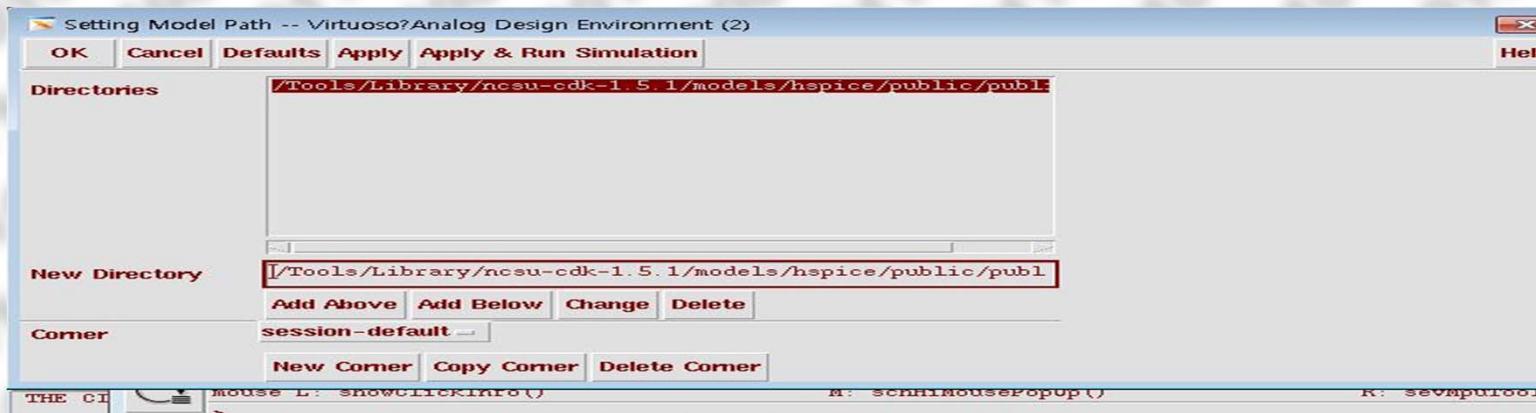


hspiceS

- 2. Setup->Model Path

make sure that the path is set by as follow.

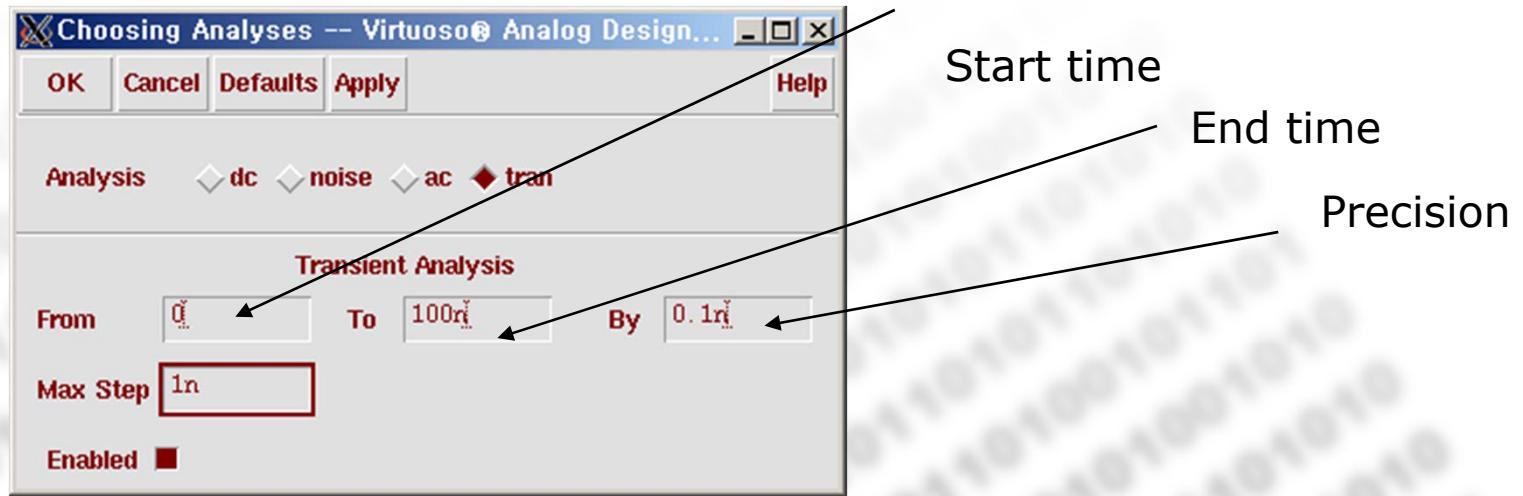
/Tools/Library/ncsu-cdk-1.5.1/models/hspice/public/publicModel





8. HSPICE Environment Setup

- 3. Analysis -> Choose



- 4. Outputs -> Save All... -> ok





9. Create netlist of the schematic

- 5. Simulation -> Netlist -> Create Final
 - ✓ Create Hspice netlist.

```
* # FILE NAME: /HOME/YOON/CADENCE/SIMULATION/INV/HSPICES/SCHEMATIC/NETLIST/
* INV.C.RAW
* NETLIST OUTPUT FOR HSPICES.
* GENERATED ON OCT 20 00:35:05 2010

* FILE NAME: HSPICE_PRAC_INV_SCHEMATIC.S.
* SUBCIRCUIT FOR CELL: INV.
* GENERATED FOR: HSPICES.I
* GENERATED ON OCT 20 00:35:05 2010.

M0 OUT IN VDD VDD TSMC20P L=200E-9 W=300E-9 AD=150E-15 AS=150E-15 PD=1.6E
+PS=1.6E-6 M=1
M1 OUT IN GND GND TSMC20N L=200E-9 W=300E-9 AD=150E-15 AS=150E-15 PD=1.6E
+PS=1.6E-6 M=1

* INCLUDE FILES

* END OF NETLIST
.TRAN 1.00000E-10 1.00000E-07 START= 0.0000
.TEMP 25.0000
.OP
.save
.OPTION INGOLD=2 ARTIST=2 PSF=2
+
+ PROBE=0
+ DELMAX = 1.00000E-09
.END
```



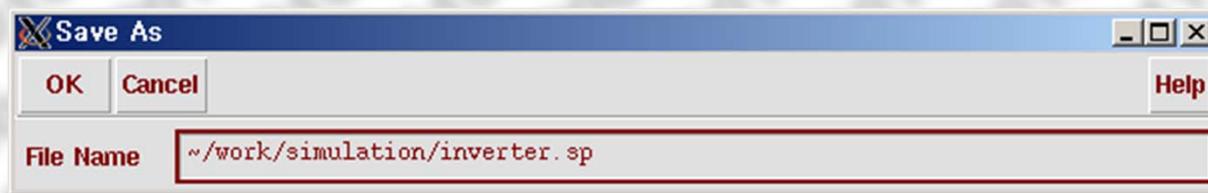
10. Save the HSPICE program

- 6. we need to modify this netlist to be simulated.

- ✓ Make the folder to save the netlist file
 - » > mkdir work
 - » > cd work
 - » > mkdir simulation

```
[choym016@sun30 ~]$ mkdir work
[choym016@sun30 ~]$ cd work
[choym016@sun30 ~/work]$ mkdir simulation
[choym016@sun30 ~/work]$
```

- ✓ File -> Save as
 - ✓ Save the netlist as : ~/work/simulation/inverter.sp



11. Edit the HSPICE program

- 7. in Xterm

```
> cd ~/work/simulation  
> ls  
> vi inv.sp
```

```
* FILE NAME: /HOME/YOUR/CALENDAR/2010FALL/CSA/PROJECTS/SCHEMATIC/INV1.LSTV  
* INV1.C.RAN  
* INVERTER OUTPUT FOR INVICKS.  
* GENERATED ON OCT 20 00:35:08 2010  
*,GLOBAL VDD  
* FILE NAME: HSPICE_PRAC_INV_SCHEMATIC.S.  
* STRUCTCIRCUIT FOR CELL: INV.  
* GENERATED FOR: HSPIKE.  
* GENERATED ON OCT 20 00:35:08 2010.  
  
MPO OUT IN VDD WDD TSMC20P L=200E-9 W=300E-9 AD=1.50E-15 AS=1.50E-15 PD=1.6E-6  
+PS=1.6E-6 M=1  
MNO OUT IN GND GND TSMC20N L=200E-9 W=300E-9 AD=1.50E-15 AS=1.50E-15 PD=1.6E-6  
+PS=1.6E-6 M=1  
  
.lib "/Tools/Library/ncsu-cdk-1.5.1/models/hspice/public/publicModel/tsmc20P" PMOS  
.lib "/Tools/Library/ncsu-cdk-1.5.1/models/hspice/public/publicModel/tsmc20N" NMOS  
  
* INPUT VECTORS  
  
vdd vdd gnd 1.8  
vin in gnd pulse 0 1.8 10n 0.1n 0.1n 20m 40n  
  
* END OF NETLIST  
.TRAN 1.0000E-10 1.0000E-07 START= 0.0000  
.TEMP 25.0000  
.OP  
.SAVE  
.OPTION INGOLD=2 ARTIST=2 PSF=2  
+ PROBE=0  
+ DELMAX = 1.00000E-09  
.END
```

Input Vector
Defined

Deactivate
.save

Check the Library path!

```
.lib "/Tools/Library/ncsu-cdk-1.5.1/models/hspice/public/publicModel/tsmc20P" PMOS  
.lib "/Tools/Library/ncsu-cdk-1.5.1/models/hspice/public/publicModel/tsmc20N" NMOS
```

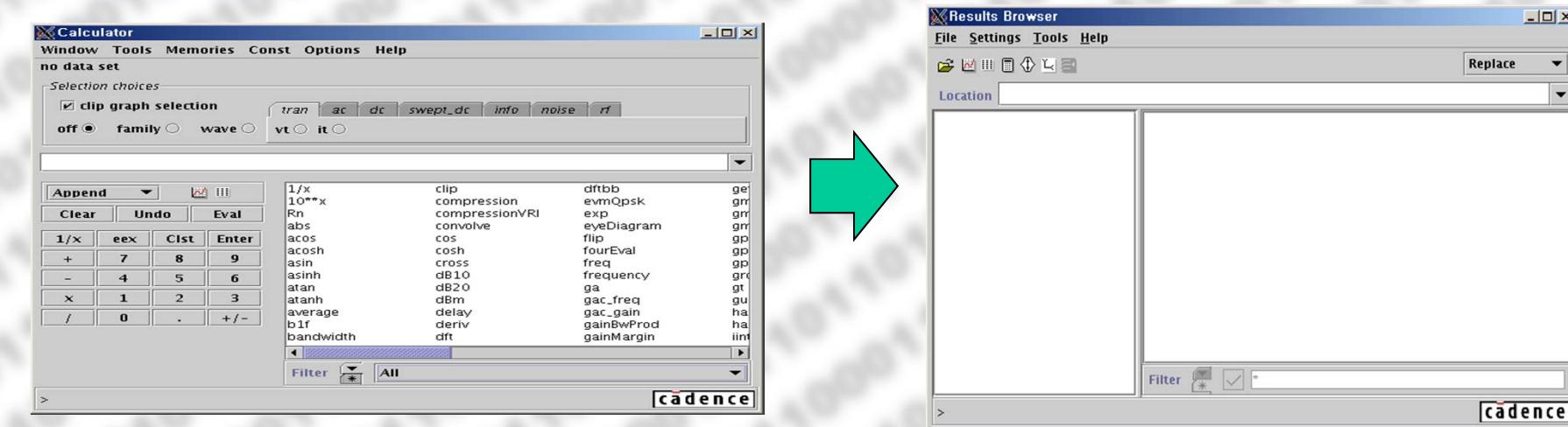


12. Run the HSPICE program

- 8. in Xtern > hspice inverter.sp -o (run hspice simulation)

```
[choym016@sun30 simulation]$ hspice inverter.sp -o
Using: /usr/bin/time -p /tools/synopsis/hspice/hspice/linux/hspice inverter.sp -o
>info: ***** hspice job concluded ← 이 메시지 확인!
real 0.20
user 0.03
sys 0.01
[choym016@sun30 simulation]$ █
```

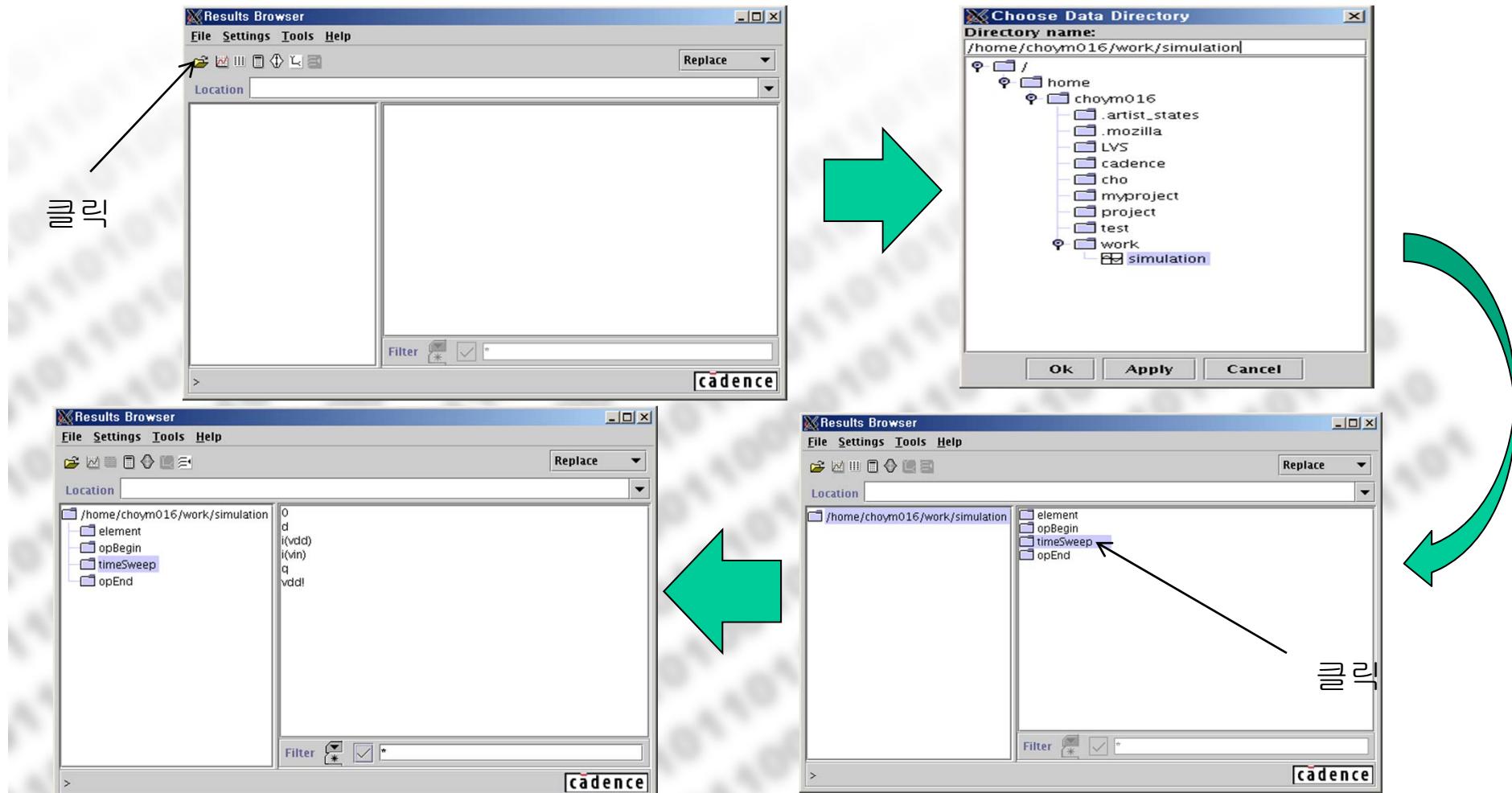
- ✓ If job aborted, see the inv.lis file to analyze the reason and the modify the netlist(.sp)
- ✓ And simulate it again.
- 9. in CIW window
- Tools -> Analog Environment -> Calculator..
 - ✓ Tools -> Browser.. In Calculator window





13. See the result waveforms

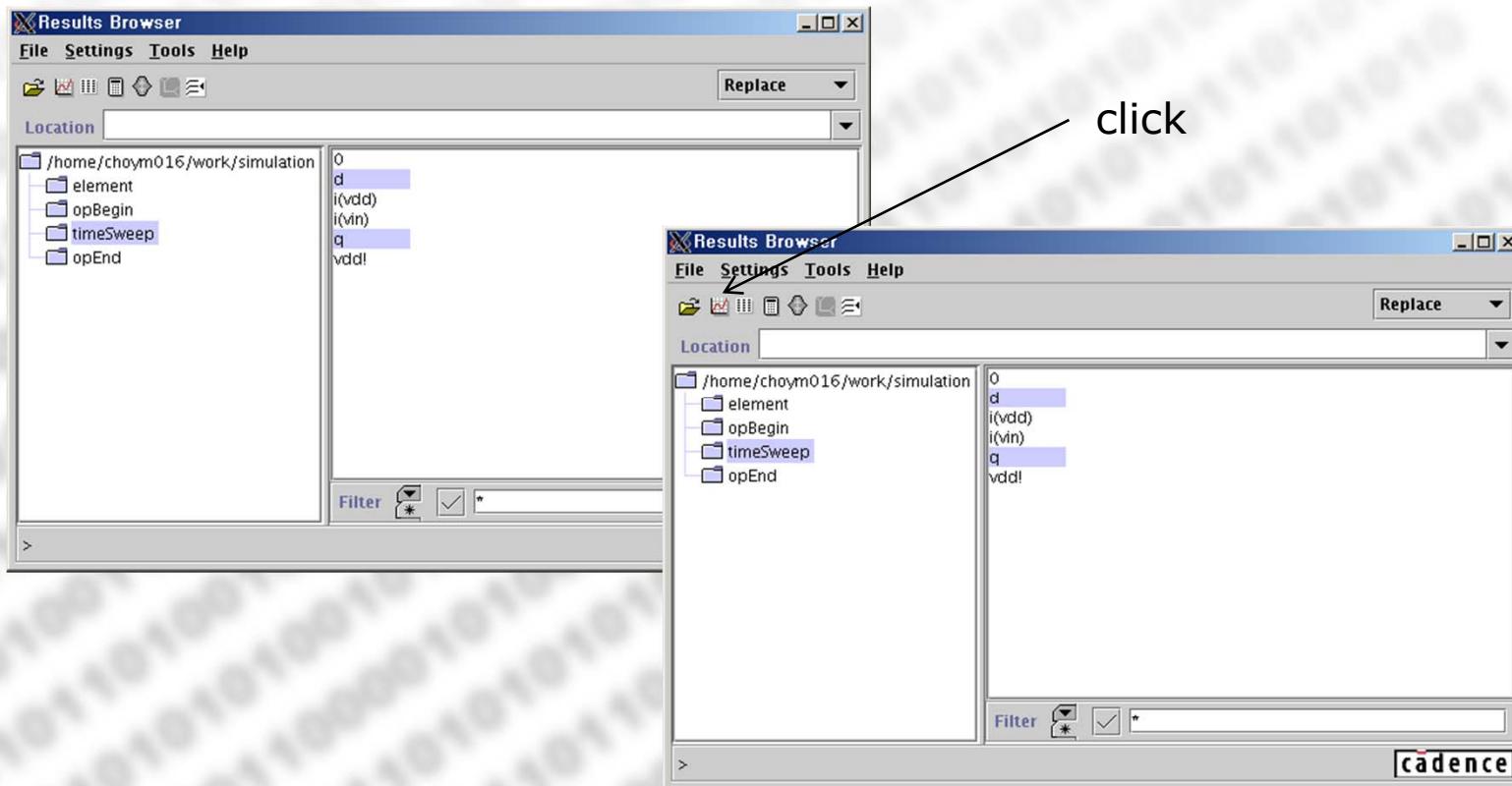
- 10. go to the folder that the simulation results exist.





13. See the result waveforms

- 11. select Input & output node
 - ✓ Plot signal click





13. See the result waveforms

- Activate Graph Window
 - Strip Chart Mode click to distinguish the input and output signal

