




# Tutorials of Composer and Spectre

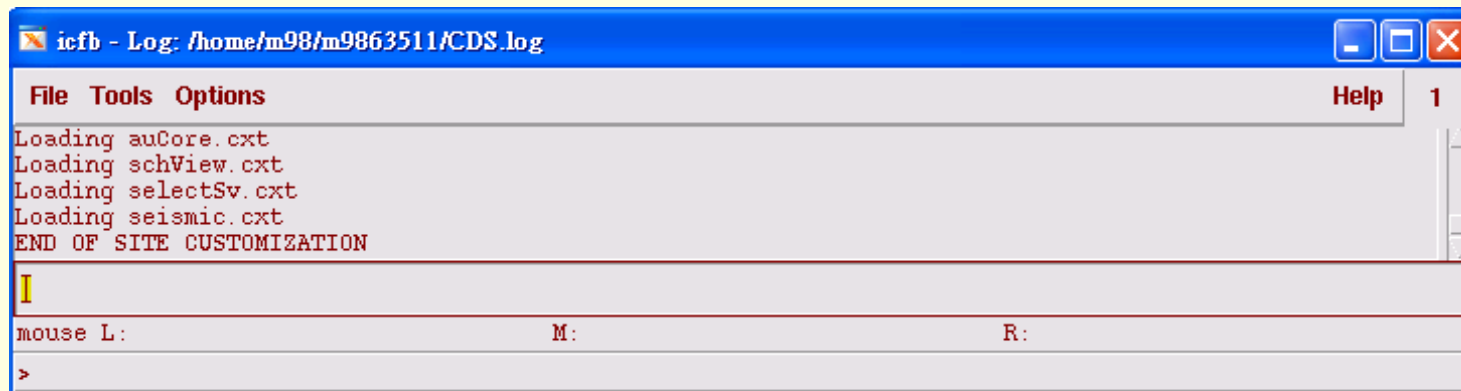


AIC 2010

# Get started

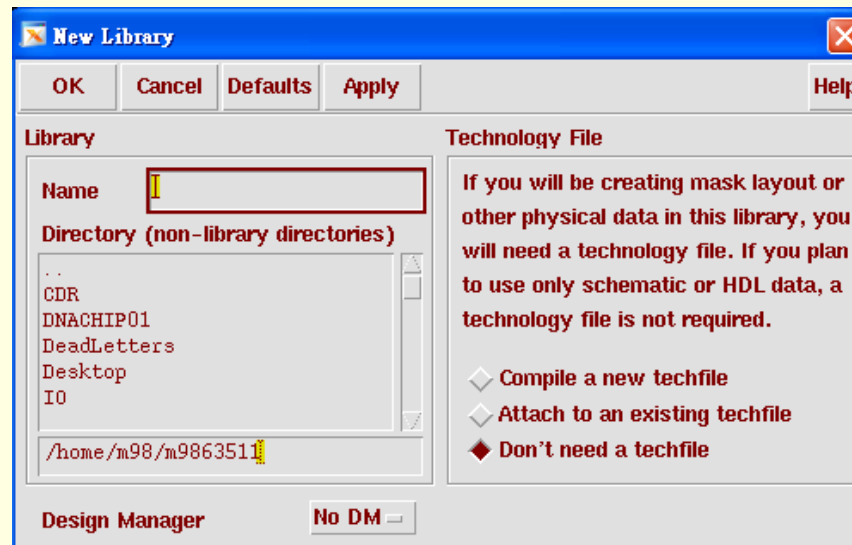
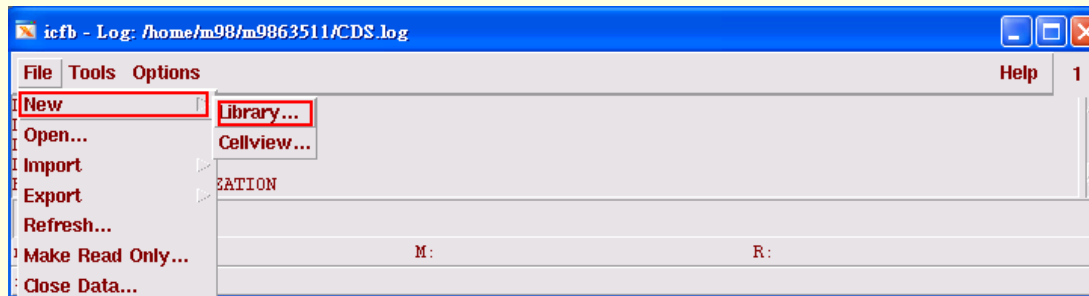
Enter one workstation. Enter icfb & at terminal, then the main window appears

```
[m9863511@ws25 ~]$ icfb &  
[1] 29409
```



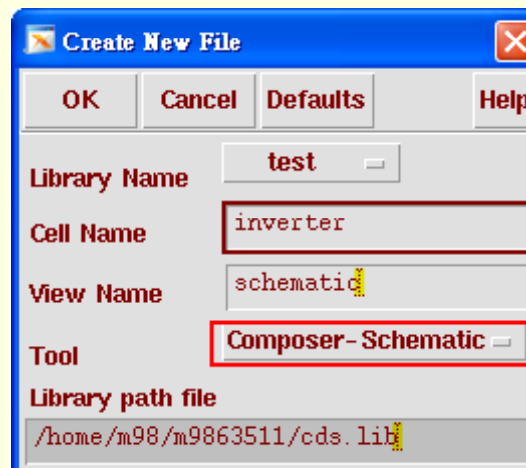
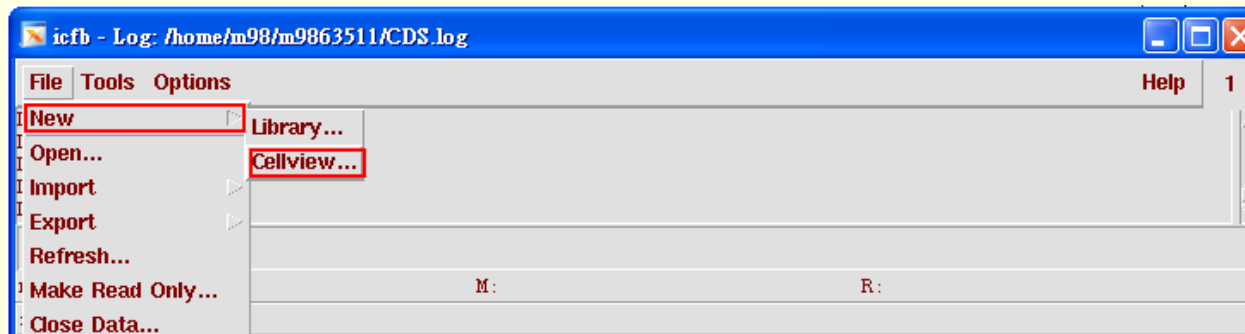
# Get started

Create a new library just like what you did in Laker. Choose “Don’t need a techfile unless you want to draw layout by Virtuoso

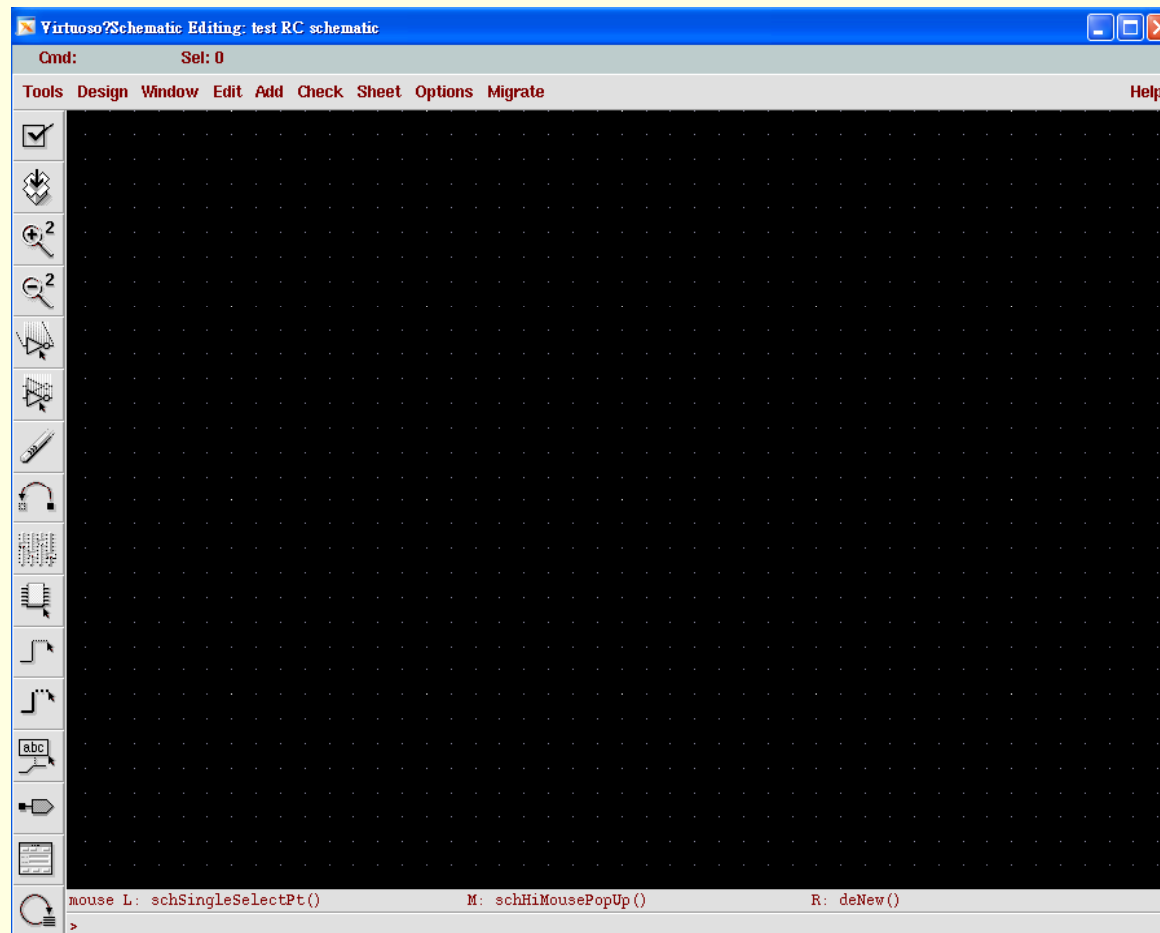


# Get started

Back to the main window, create a new cellview in the library you made. Choose Composer-Schematic.

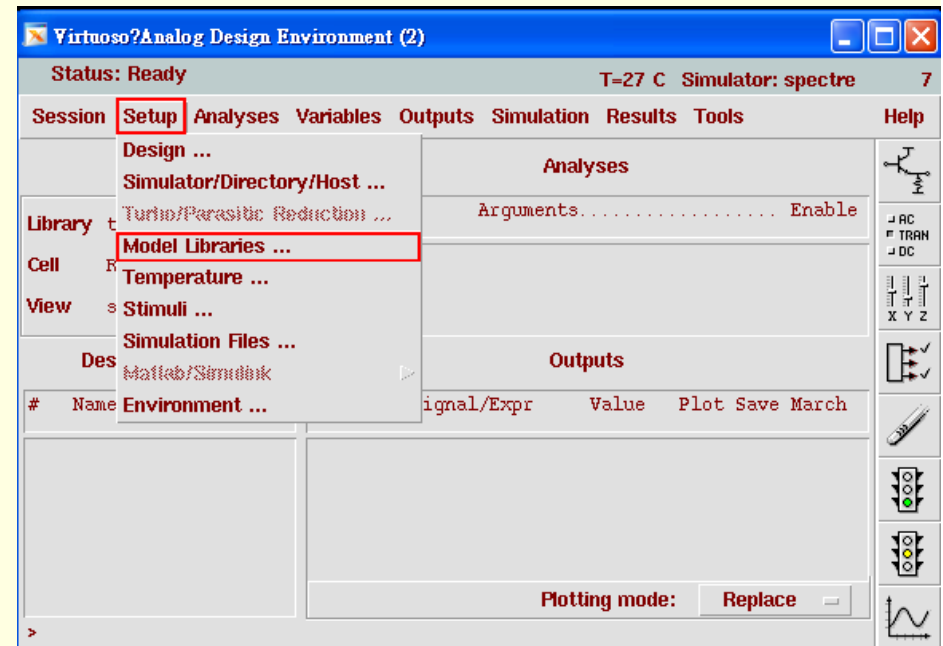
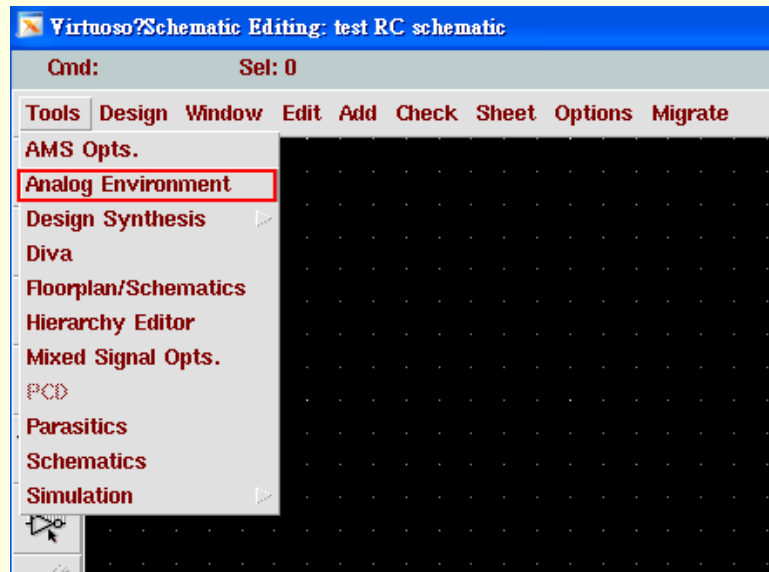


# Composer schematic window



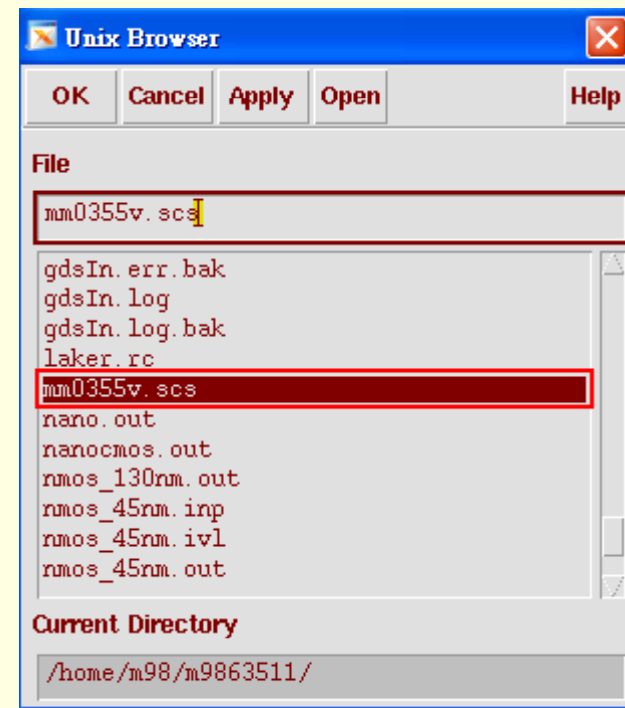
# Library File

Coose tools → Analog Enviroment → setup → Model Libraries



# Library File

Choose Browse → mm0355v.scs



# Library File

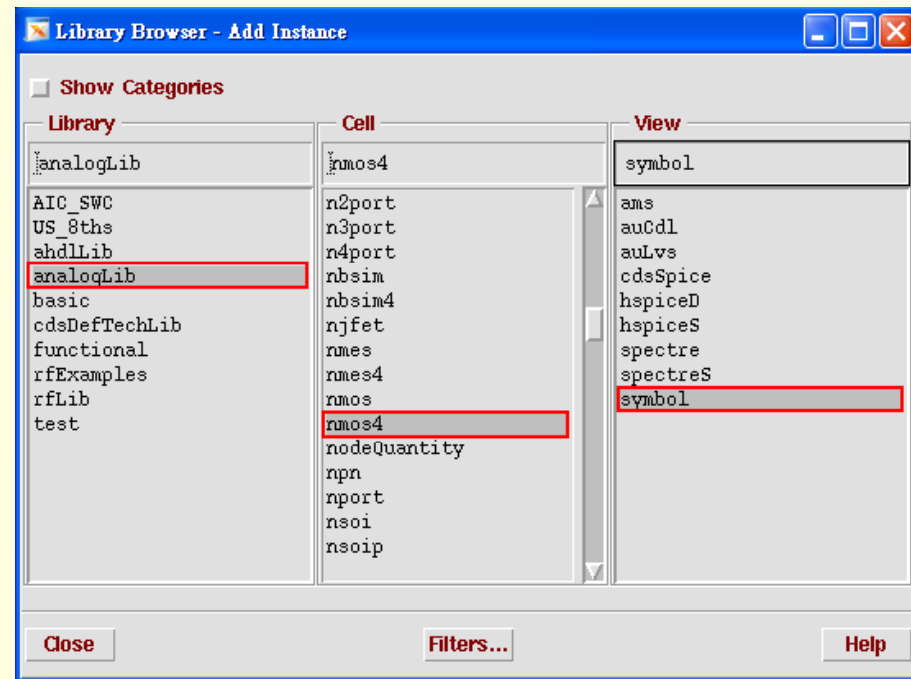
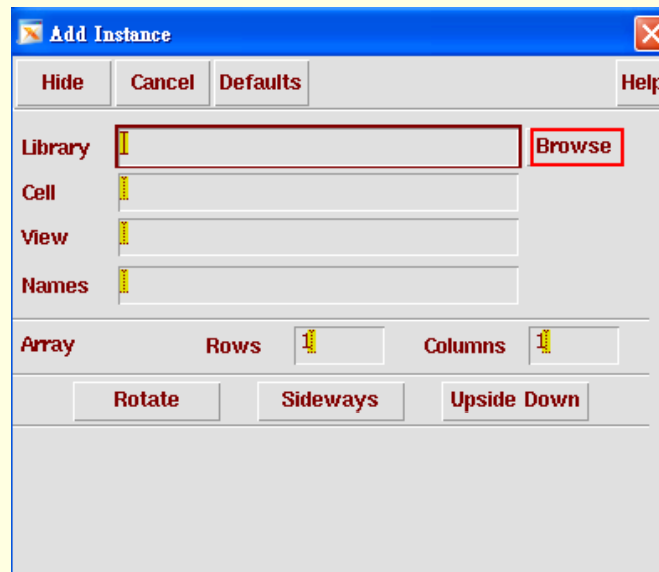
Key "tt" in section(change to ff fs sf ss for five corner simulation).Click Add.





# An example-SWC RC filter

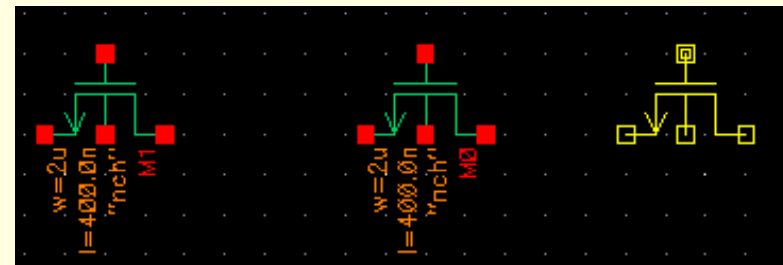
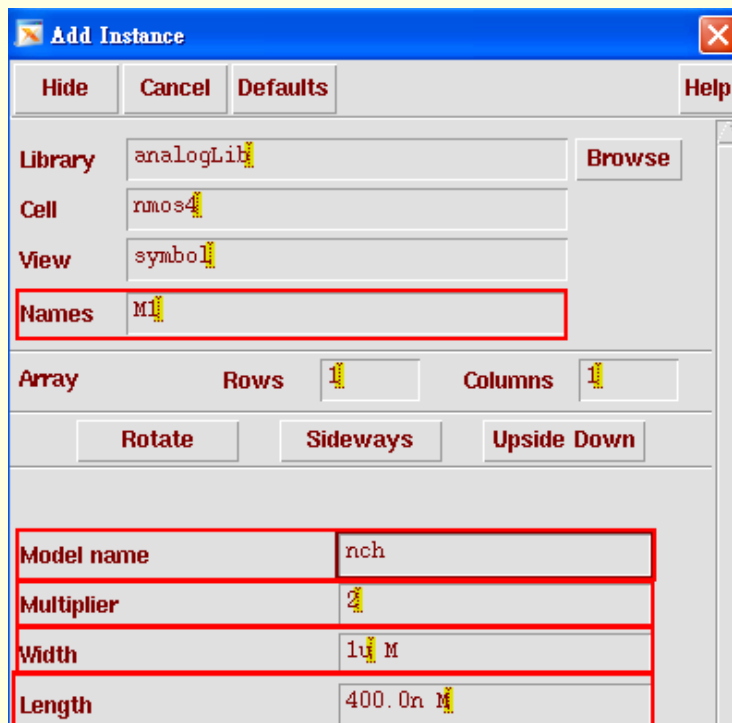
Press “i” in composer main window. Click Browse. Choose analogLib  
→nmos4 →symbol (You can find most components you need including resistors, capacitors, inductors.....in analogLib)



# An example-SWC RC filter

Specify device parameters similar to HSPICE. Only need to specify device name, Model name, Multiplier, Width, Length.

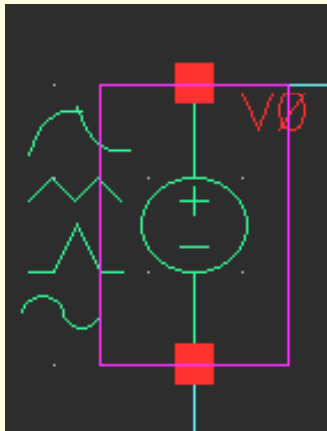
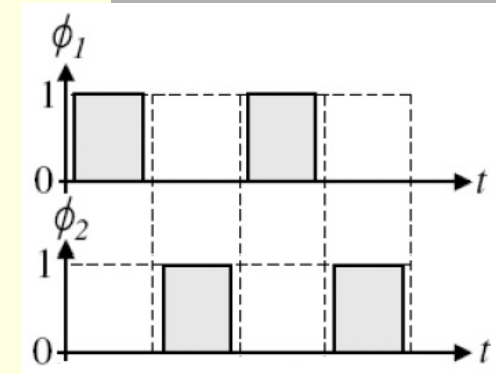
Click “Hide”, add your device in main window





# An example-SWC RC filter

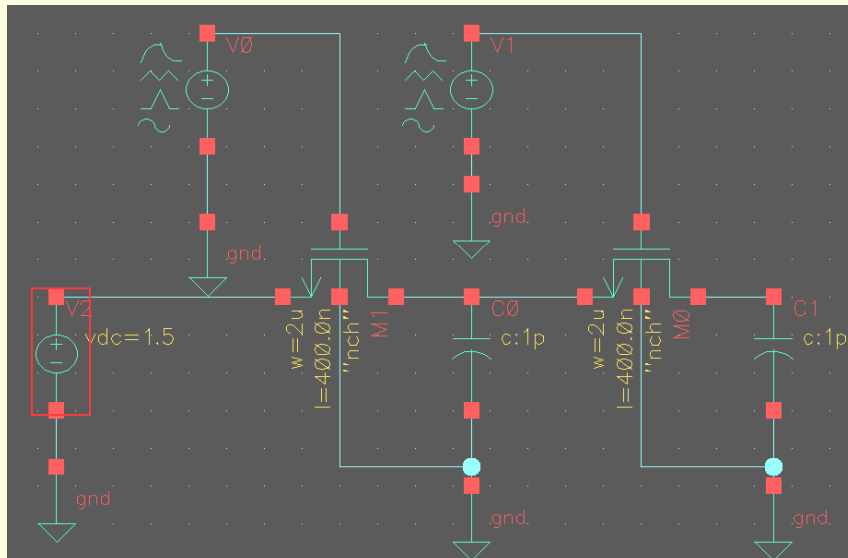
Click on Vsource. Press “Q” to edit its properties. Create two non-overlapping clock signals as in the right figure. Press “X” to check and save.



CDF Parameter	Value	Display
DC voltage		off <input type="checkbox"/>
Source type	pulse <input type="text"/>	off <input type="checkbox"/>
Frequency name 1		off <input type="checkbox"/>
Delay time		off <input type="checkbox"/>
Type of rising & falling edge		off <input type="checkbox"/>
Zero value	0 V	off <input type="checkbox"/>
One value	3 V	off <input type="checkbox"/>
Period of waveform	1n s	off <input type="checkbox"/>
Rise time	1.000u s	off <input type="checkbox"/>
Fall time	1.000u s	off <input type="checkbox"/>
Pulse width	450.00u s	off <input type="checkbox"/>

# An example-SWC RC filter

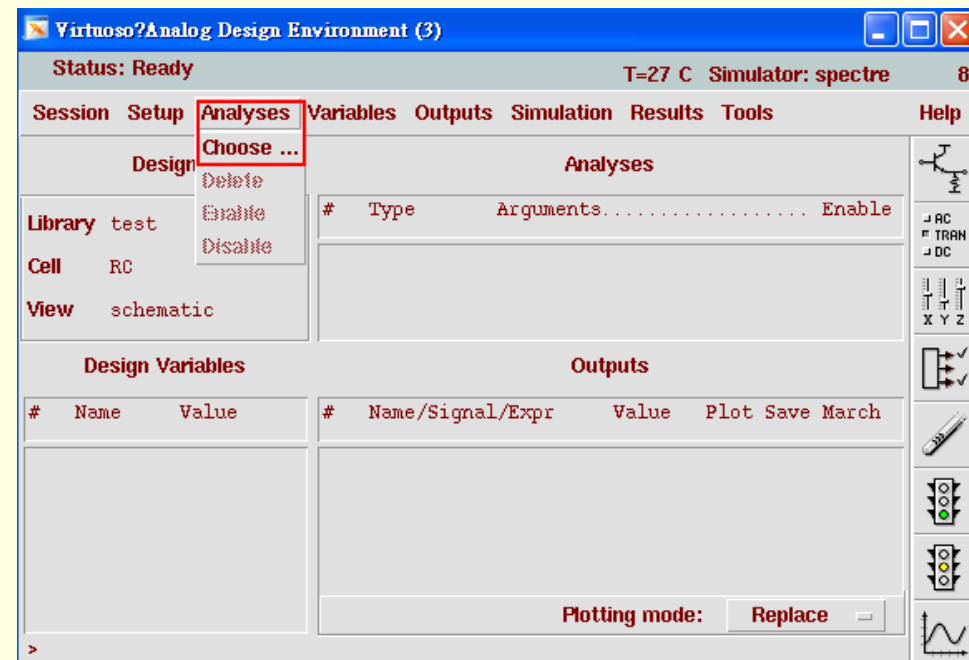
To simulate frequency response of the switched capacitor filter, add analogLib → Vdc as input. Choose PAC magnitude = 1.



CDF Parameter	Value	Display
AC magnitude		off
AC phase		off
DC voltage	1.5 v	off
Noise file name		off
Number of noise/freq pairs	0	off
XF magnitude		off
PAC magnitude	1 v	off
PAC phase		off
Temperature coefficient 1		off
Temperature coefficient 2		off
Nominal temperature		off

# An example-SWC RC filter

Click Tools → Analog Environment. Click Analysis → Choose.



# An example-SWC RC filter

Use pss and pac analysis.  
Set Beat Frequency =  
your clock frequency.  
Output harmonics =  
0. Accuracy =  
moderate. Set your  
interested frequency  
range in pac analysis and  
maximum  
sidebands=0. Click “OK”  
when all settings are done.

Choosing Analyses -- Virtuoso?Analog Design Environment (3)

OK Cancel Defaults Apply Help

pz  sp  envlp  pss  
 pac  pnoise  pxf  psp  
 qpss  qpac  qpnoise  qpxf  
 qpasp

Periodic Steady State Analysis

Fundamental Tones

#	Name	Expr	Value	Signal	SrcId
1		1/(1m-0)	1K	Large	V1

Clear/Add Delete Update From Hierarchy

Beat Frequency 1K Auto Calculate   
Beat Period

Output harmonics  
Number of harmonics 0

Accuracy Defaults (errpreset)  
 conservative  moderate  liberal

Additional Time for Stabilization (tstab)

Save Initial Transient Results (saveinit)  no  yes

Choosing Analyses -- Virtuoso?Analog Design Environment (3)

OK Cancel Defaults Apply Help

Analysis  tran  dc  ac  noise  
 xf  sens  dcmatch  stb  
 pz  sp  envlp  pss  
 pac  pnoise  pxf  psp  
 qpss  qpac  qpnoise  qpxf  
 qpasp

Periodic AC Analysis

PSS Beat Frequency (Hz) 1K

Sweeptype default Sweep is currently absolute

Input Frequency Sweep Range (Hz)  
Start-Stop Start 1K Stop 10G

Sweep Type  
Automatic

Add Specific Points

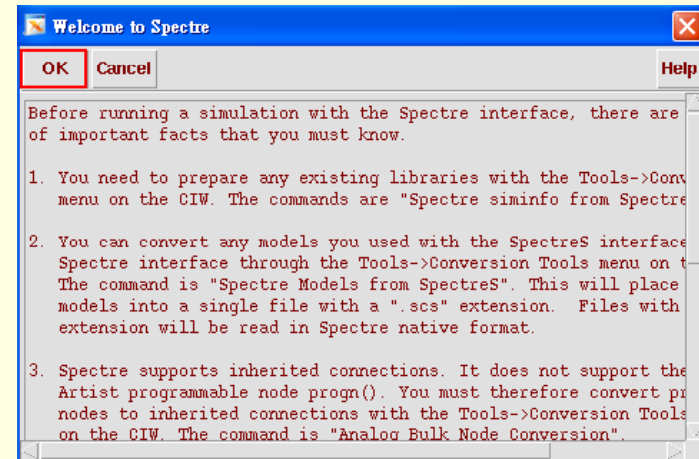
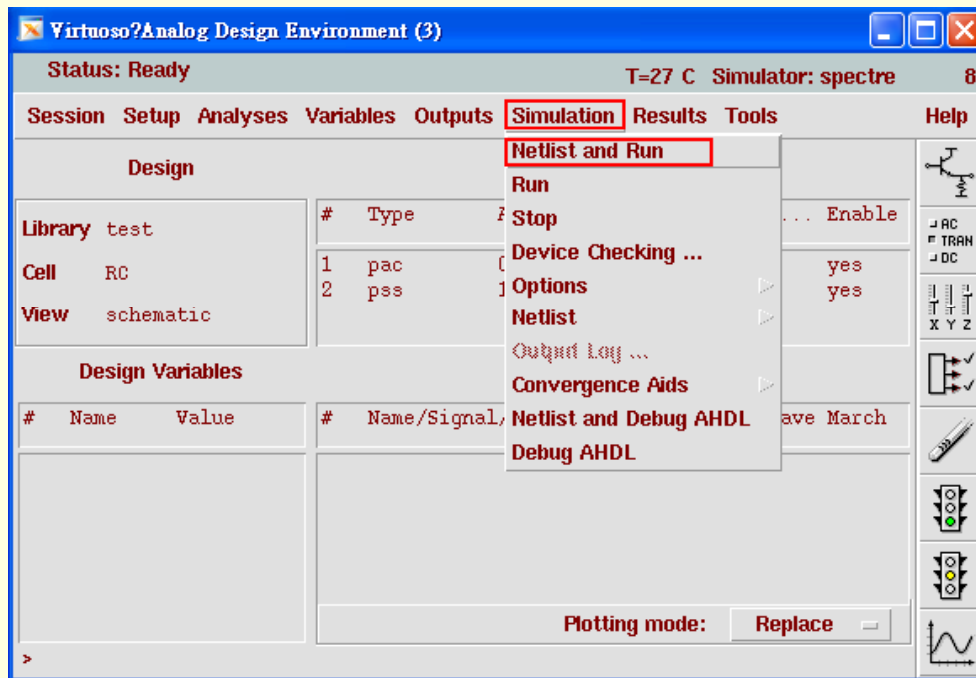
Sidebands  
Maximum sideband 0

Specialized Analyses  
None

Enabled  Options...

# An example-SWC RC filter

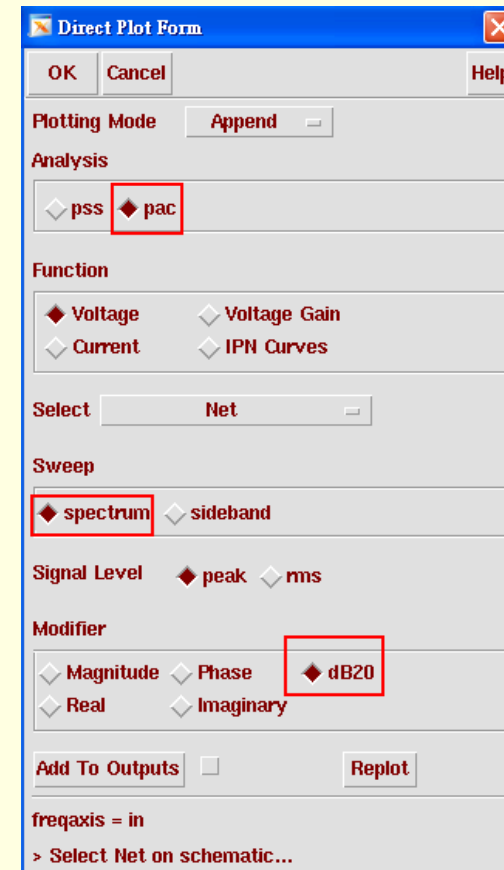
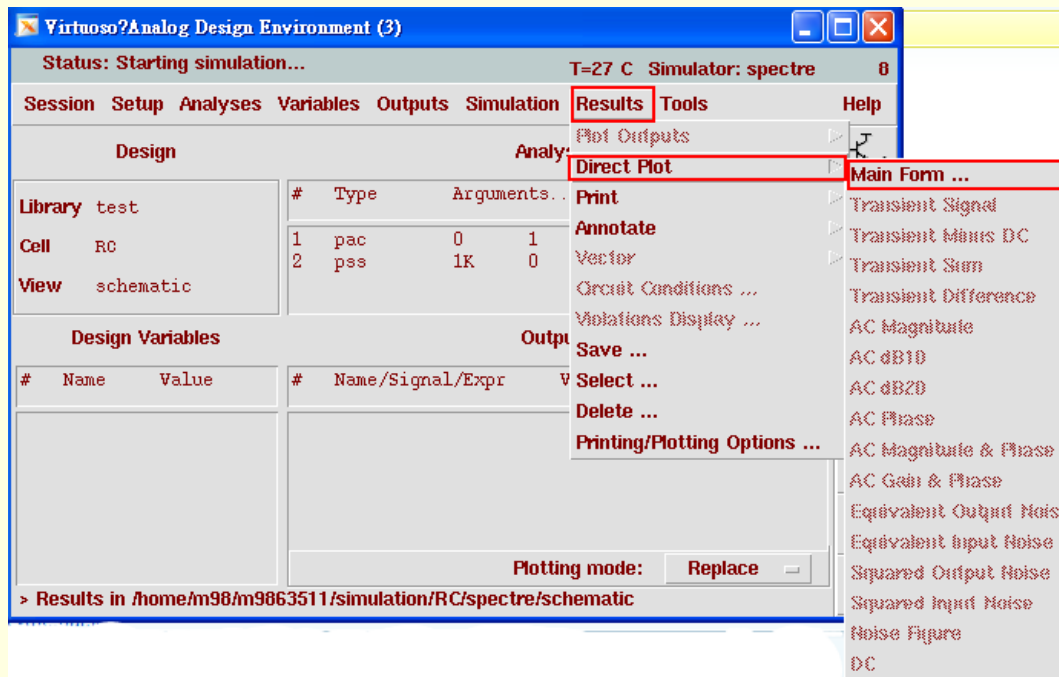
Back to ADE window. Click Simulation → Netlist and Run. Click OK.





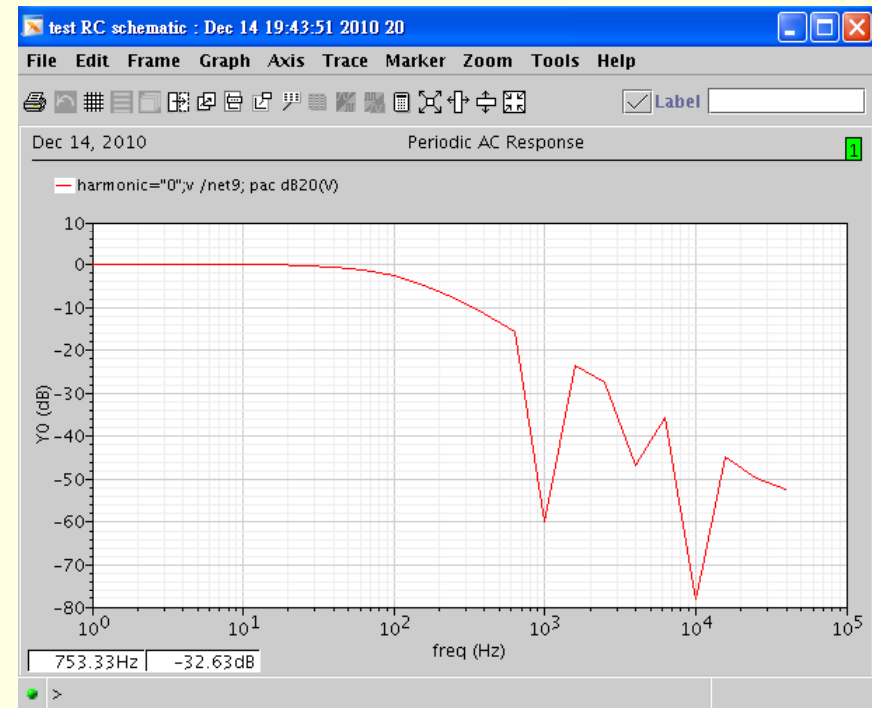
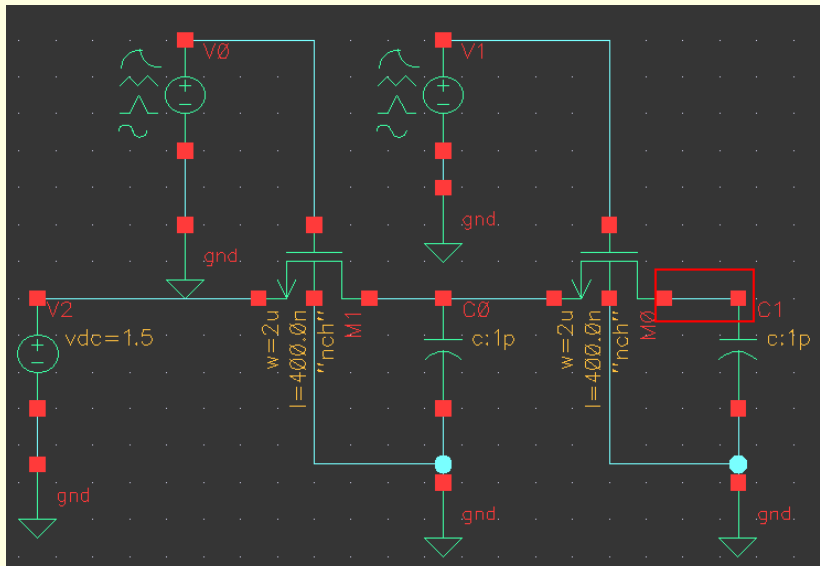
# An example-SWC RC filter

Click Results → Direct Plot → Main Form. Choose pac → spectrum → dB20.



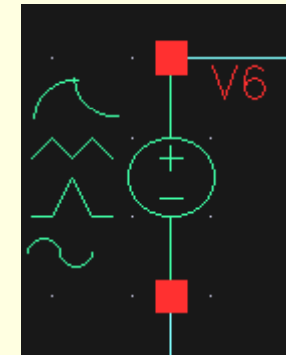
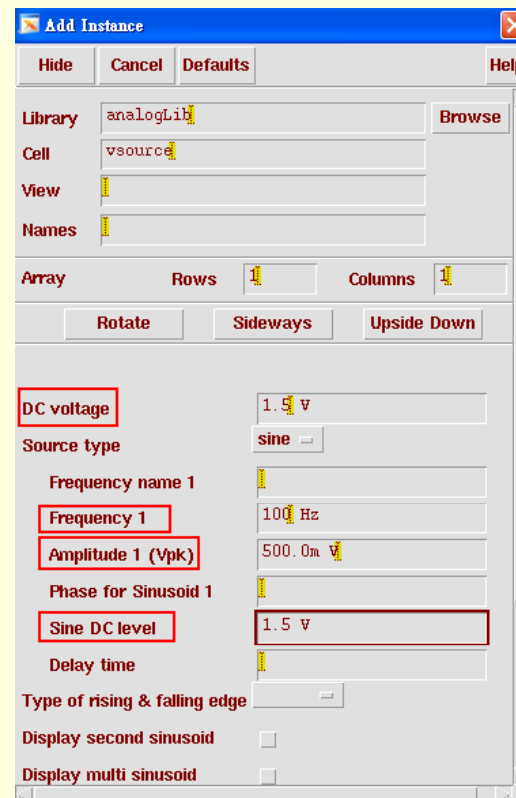
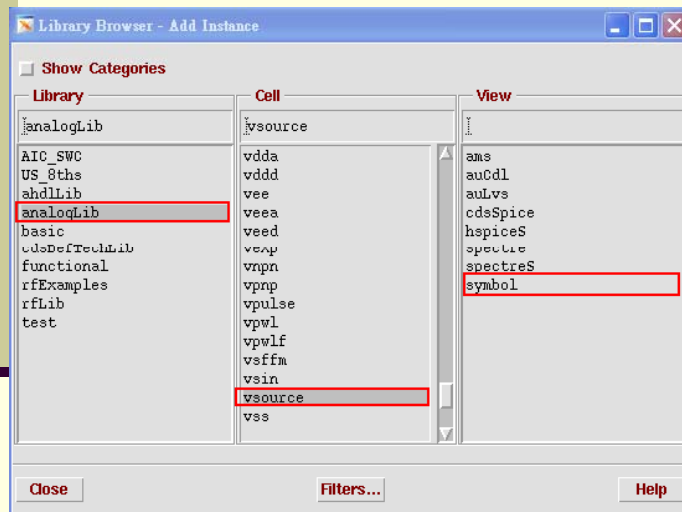
# An example-SWC RC filter

Click on the output node, and you'll see the frequency response.



# An example-SWC RC filter

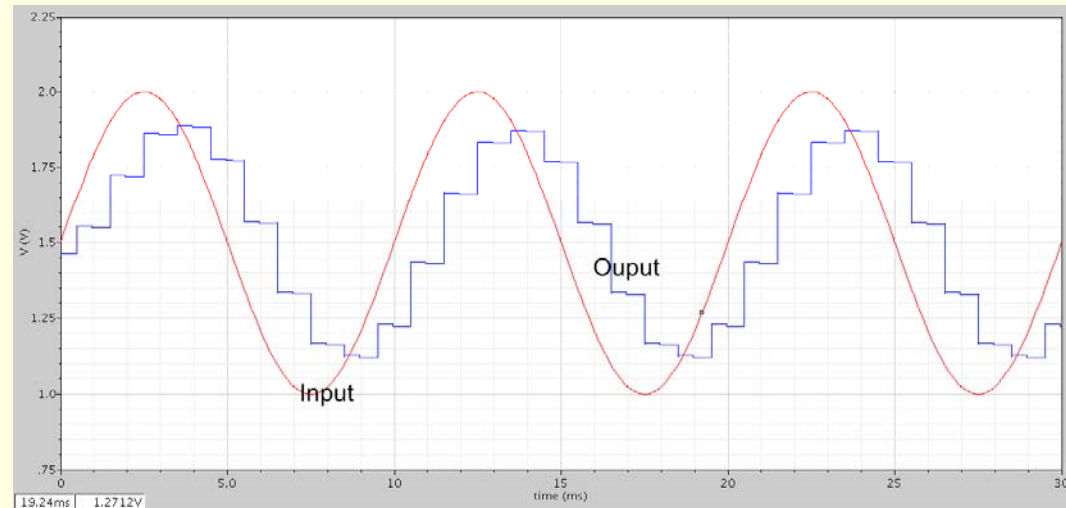
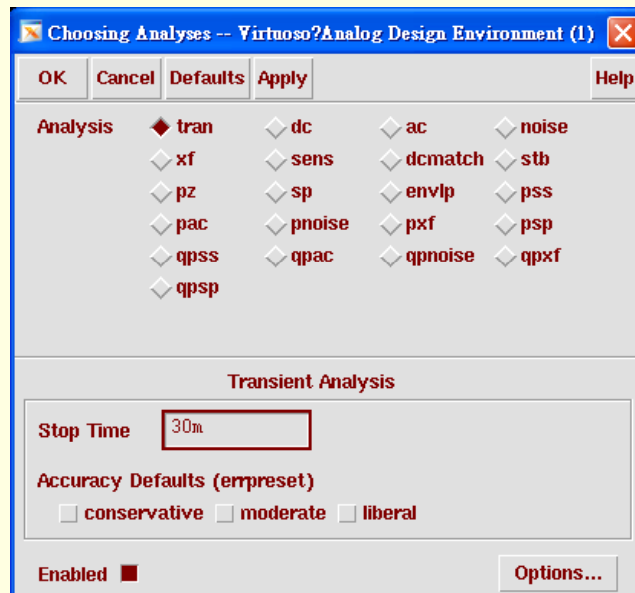
To do transient simulation, replace Vdc by Vsource as input signal



# An example-SWC RC filter

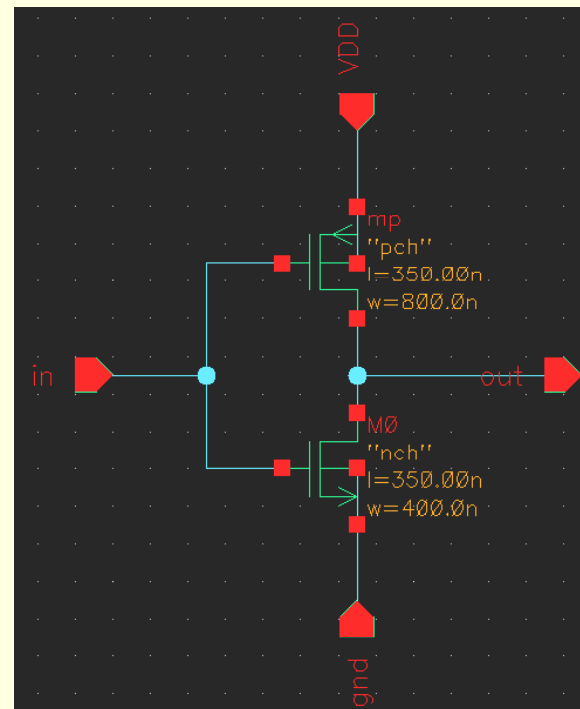
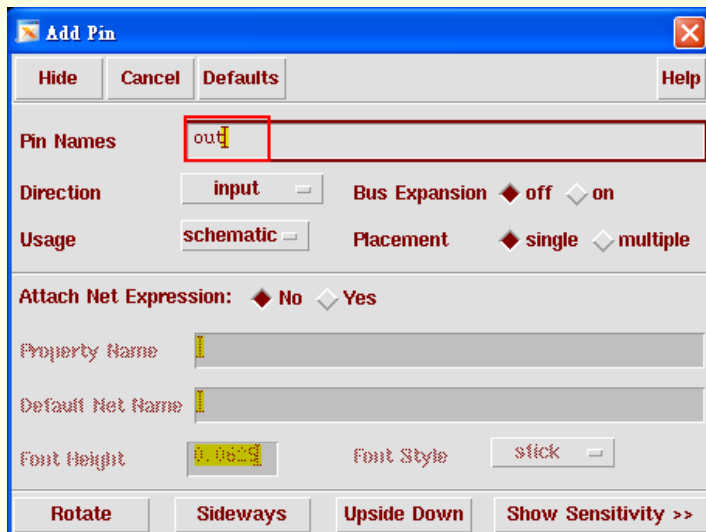
Simulation results when clock frequency=1KHZ and input is a 100HZ Sinusoidal wave.

Transient analysis and AC analysis are the same as HSPICE. Just keep in mind that use Vdc for AC and Vsource for transient.



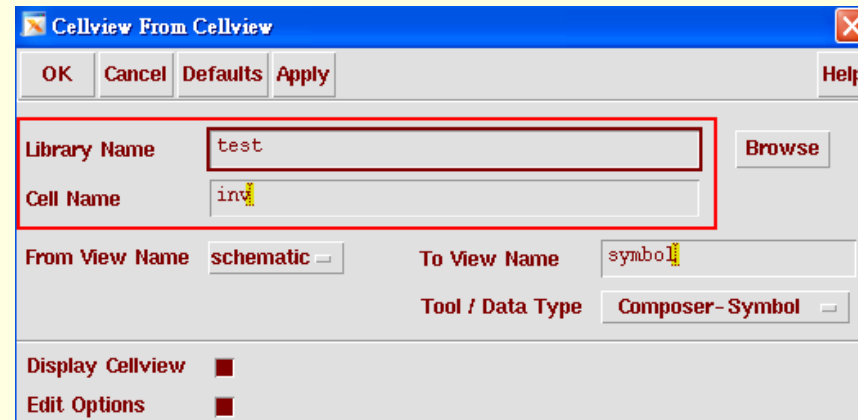
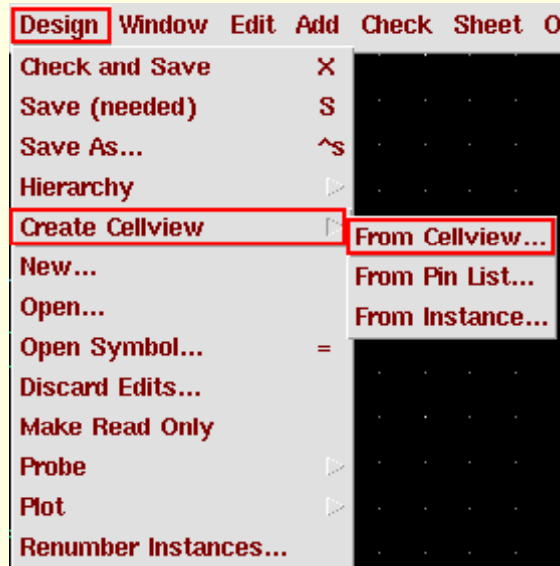
# Create symbol

Use an inverter as example. Press “p” to create pin. Define your pin names. Connect them to the corresponding nodes



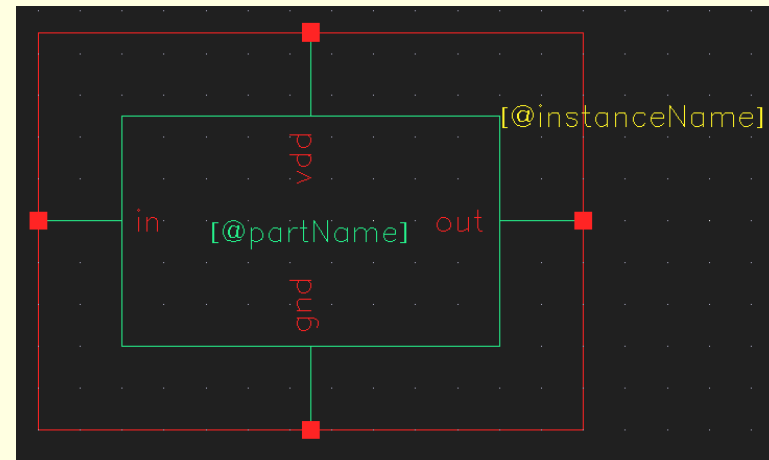
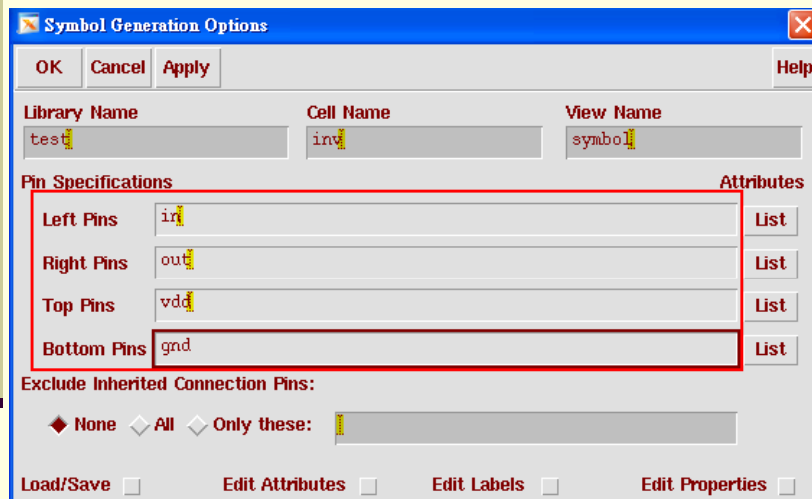
# Create symbol

Click Design → Create Cellview → From Cellview → key in your library and cell name



# Create symbol

Decide your positions of pins in the symbol(left, right, top, bottom).Check the resulted graph



# Create symbol

You can use this cell repeatedly in other cells just like subcircuits in HSPICE

