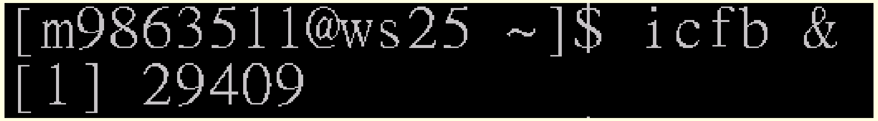
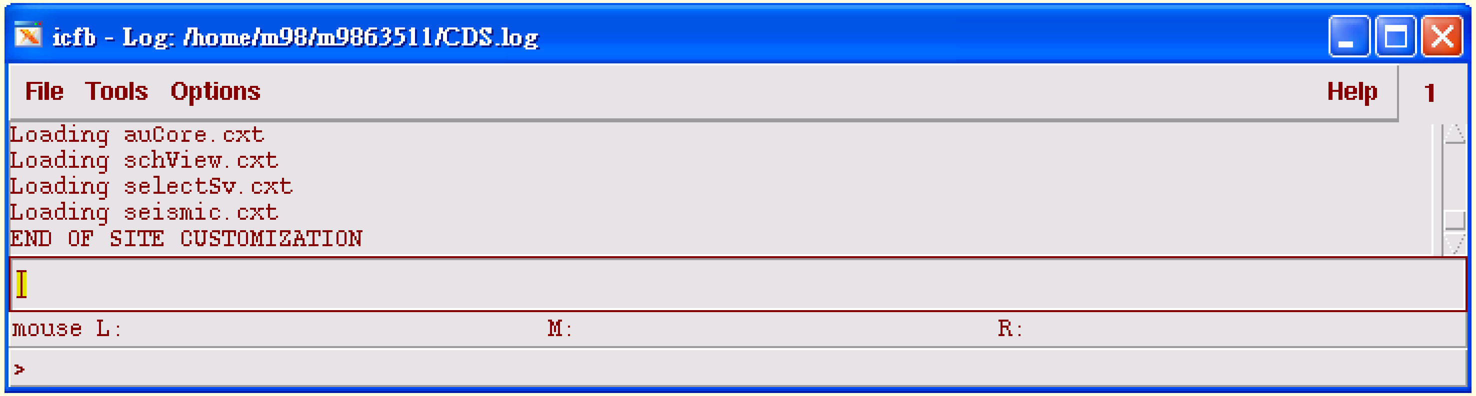
Tutorials of Composer and Spectre

AIC 2010

Get started

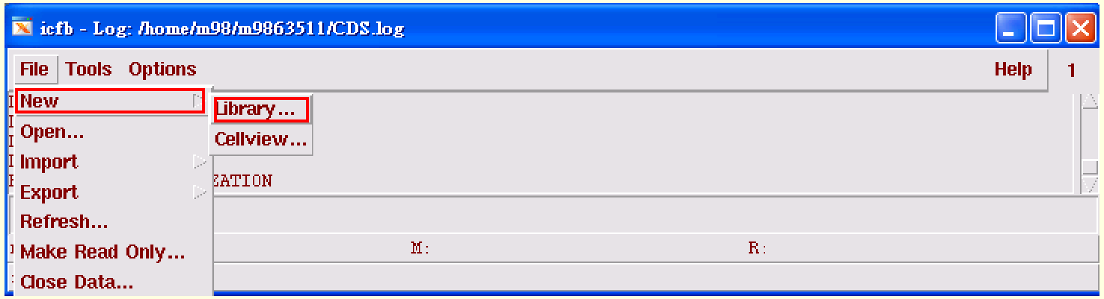
Enter one workstation.Enter icfb & at terminal, then the main window appears

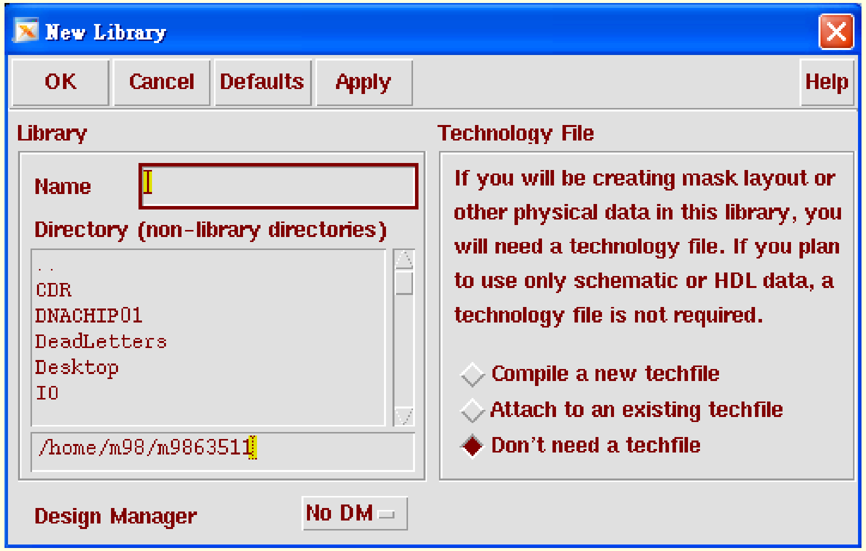




Get started

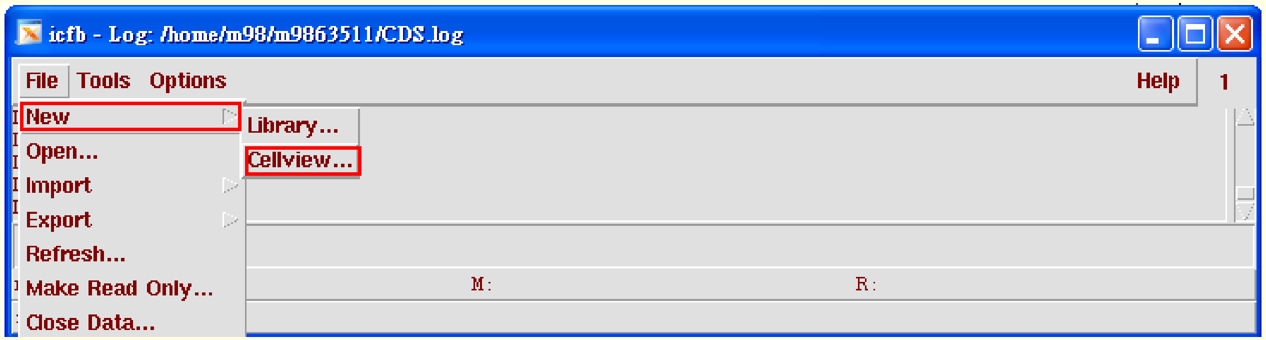
Create a new library just like what you did in Laker.Choose "Don't need a techfile unless you want to draw layout by Virtuoso

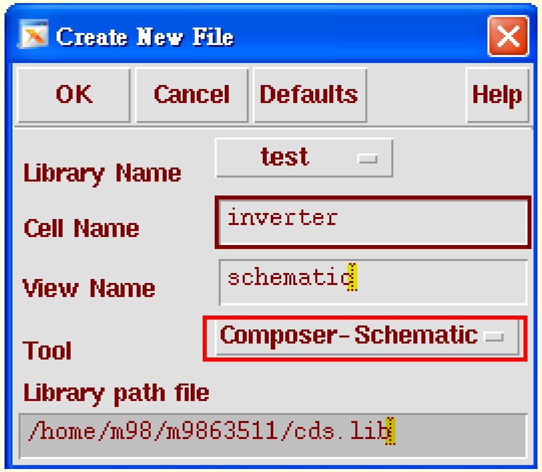




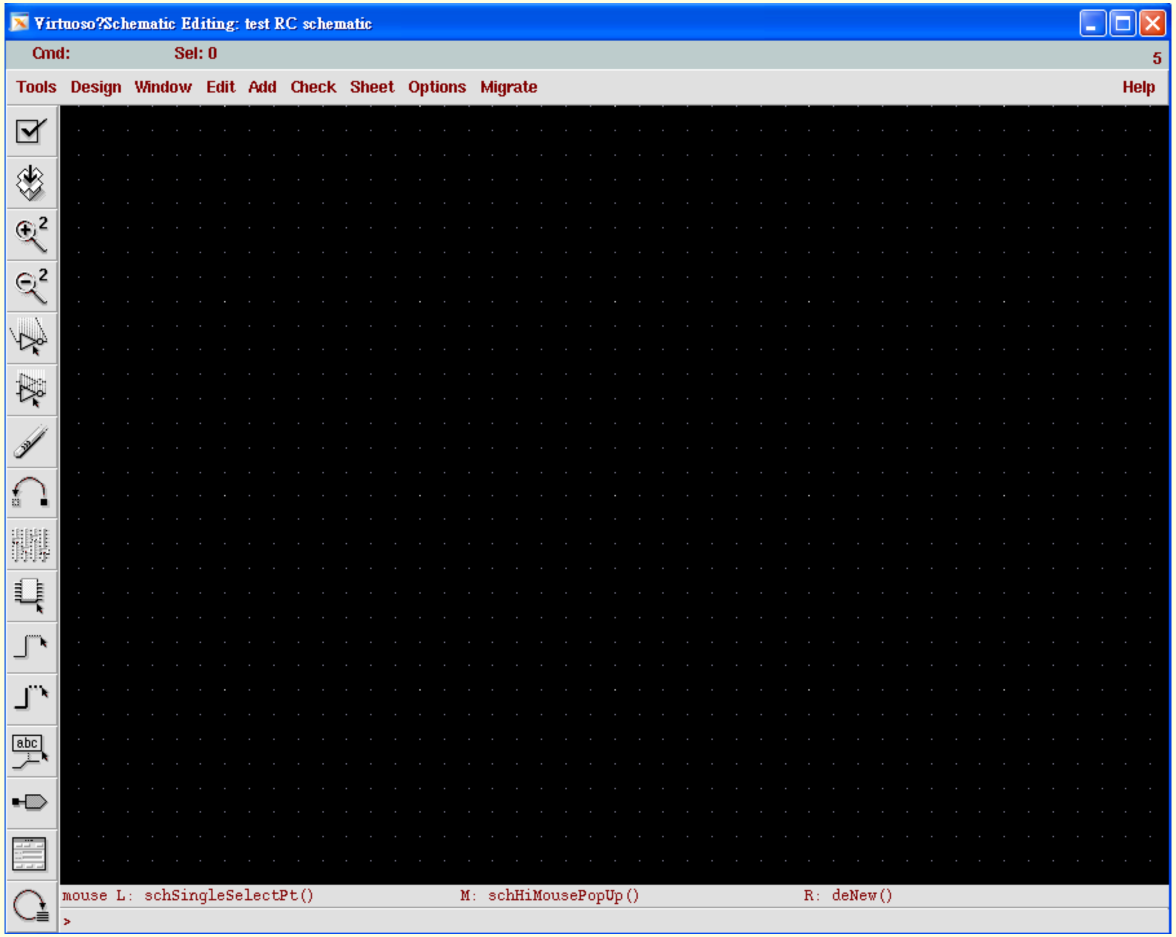
Get started

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



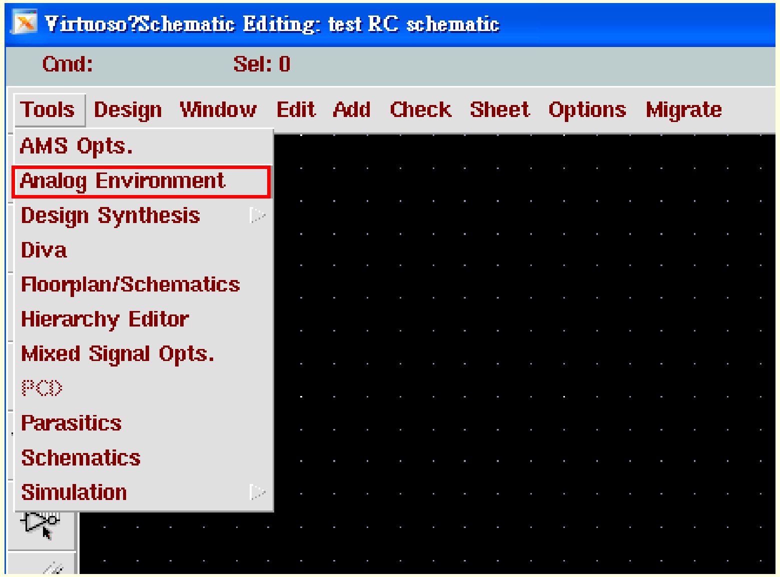


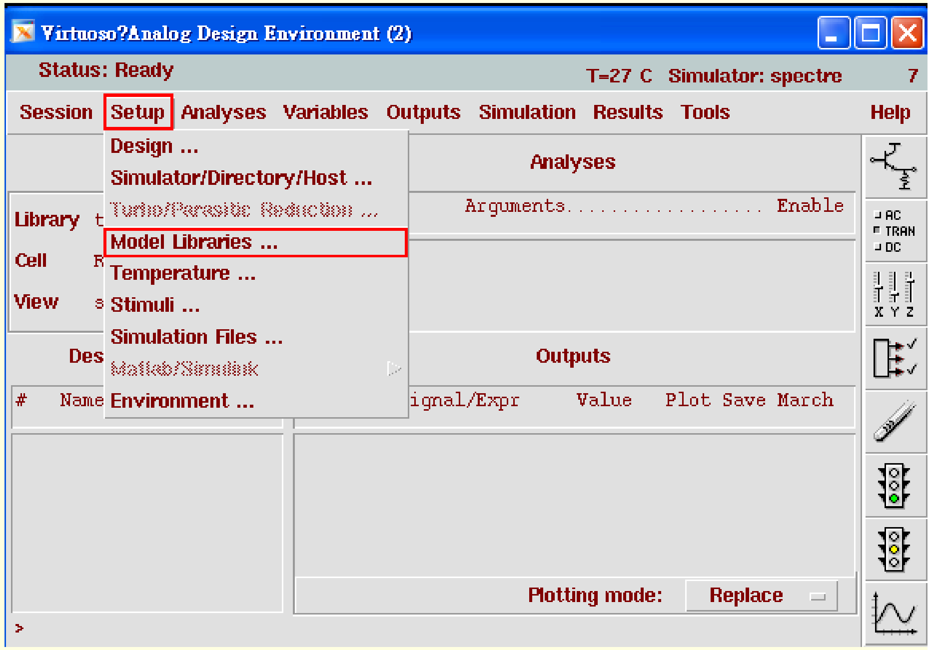
Composer schematic window



Library File

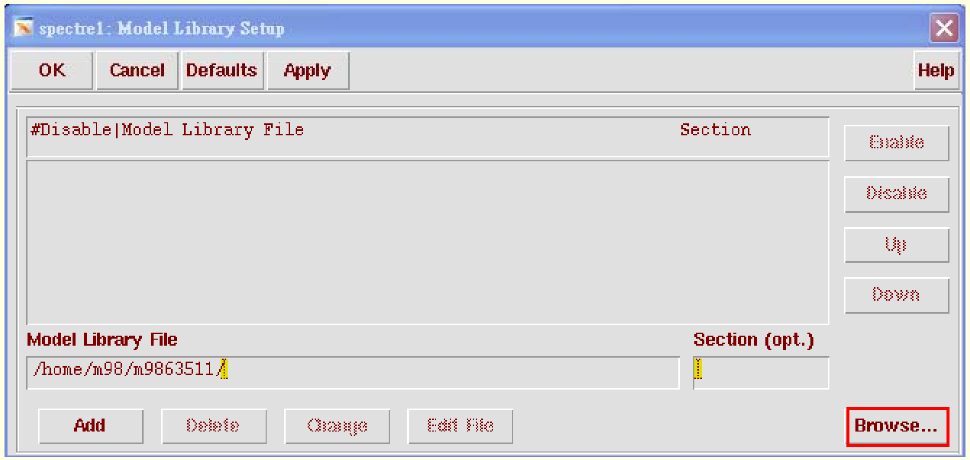
Coose tools →Analog Enviroment →setup → Model Libraries

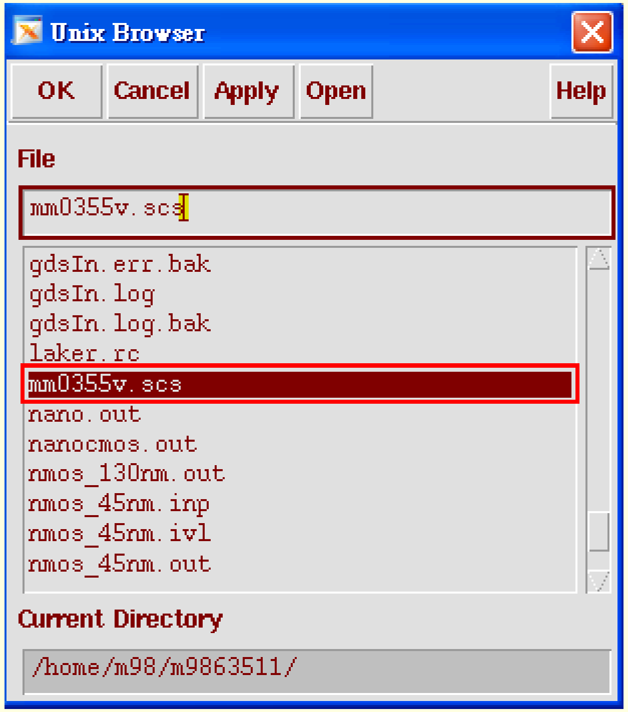




Library File

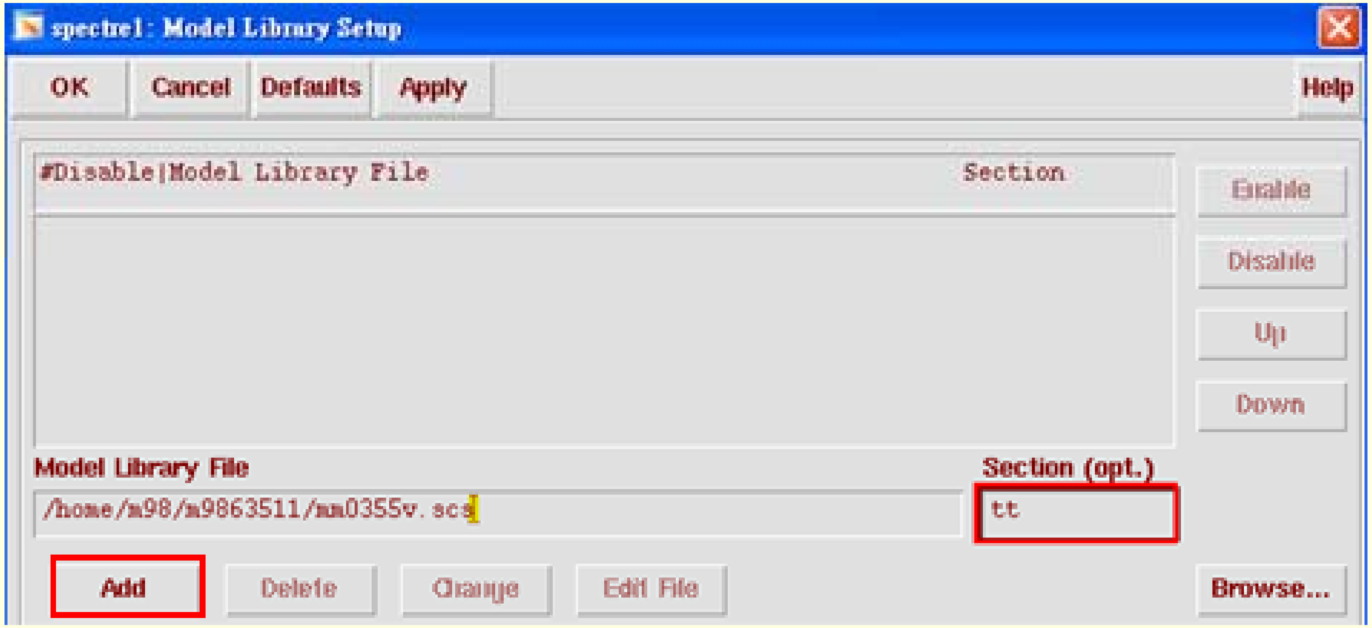
Choose Browse → mm0355v.scs





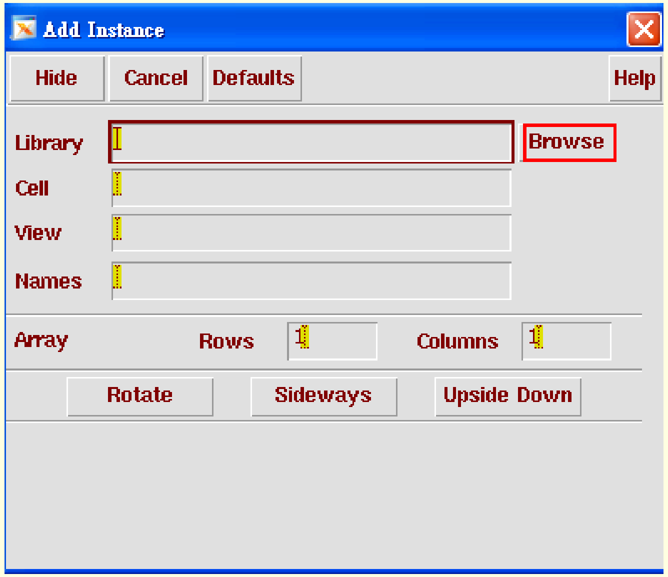
Library File

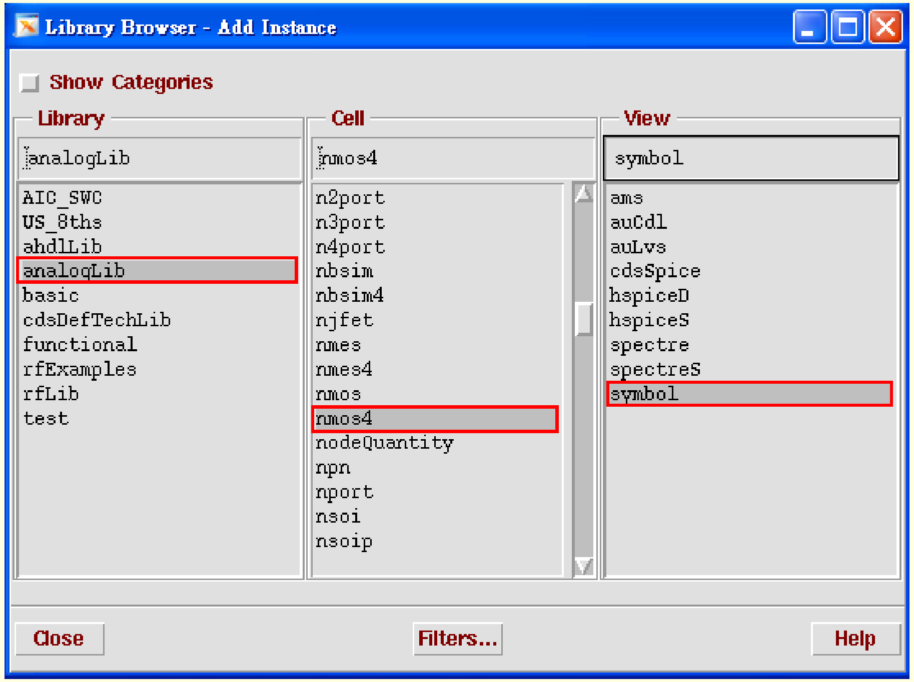
Key “tt” in section(change to ff fs sf ss for five corner simulation).Cick Add.



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)

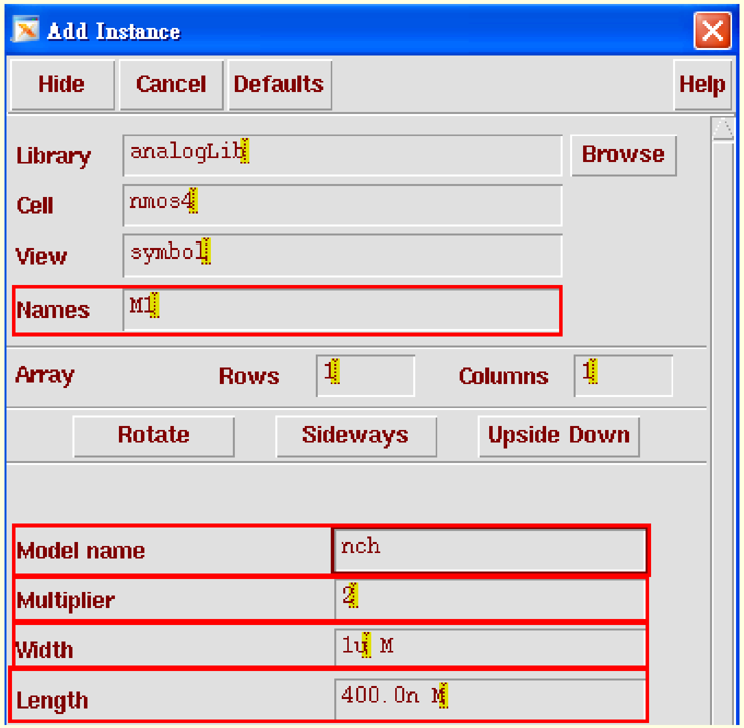


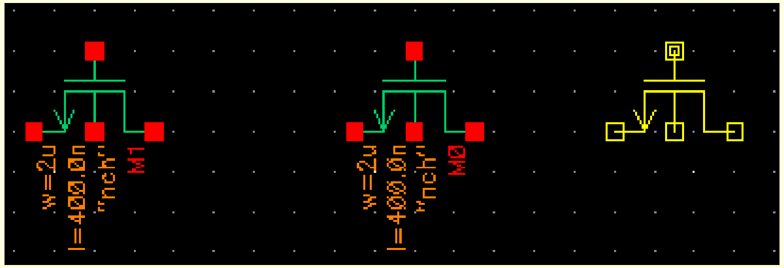


An example-SWC RC filter

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

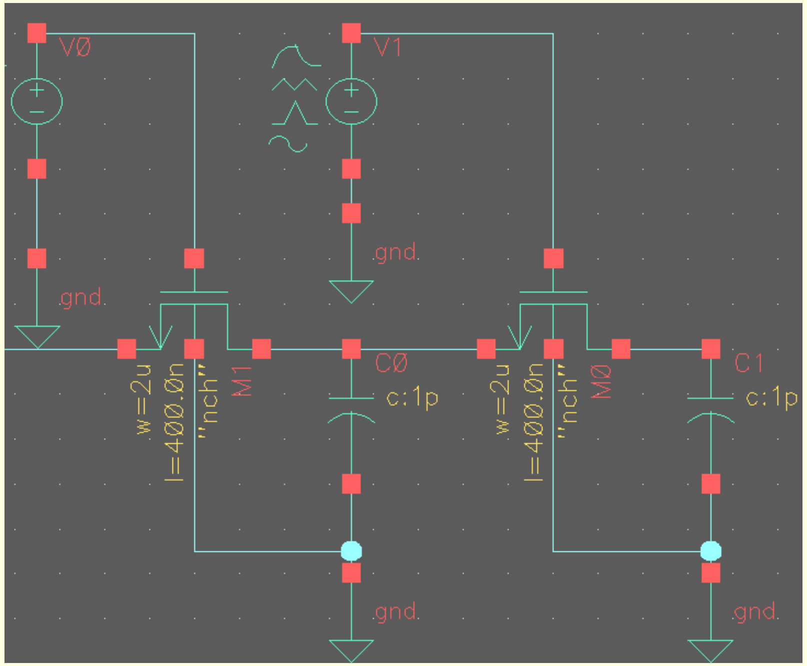
Click “Hide”, add your device in main window





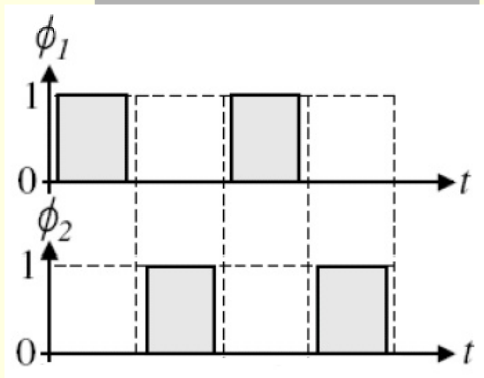
An example-SWC RC filter

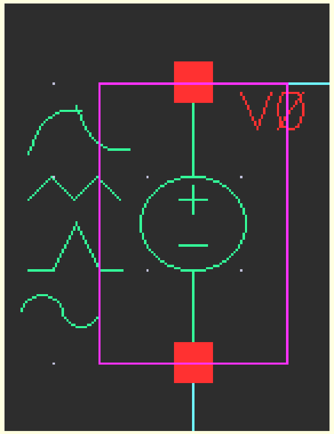
Add analogLib → cap, gnd ,Vsource.Press “W” to make wire connection. (Vsource is used as signal source for transient and pss pac analysis. Use Vdc instead when doing ac analysis.)

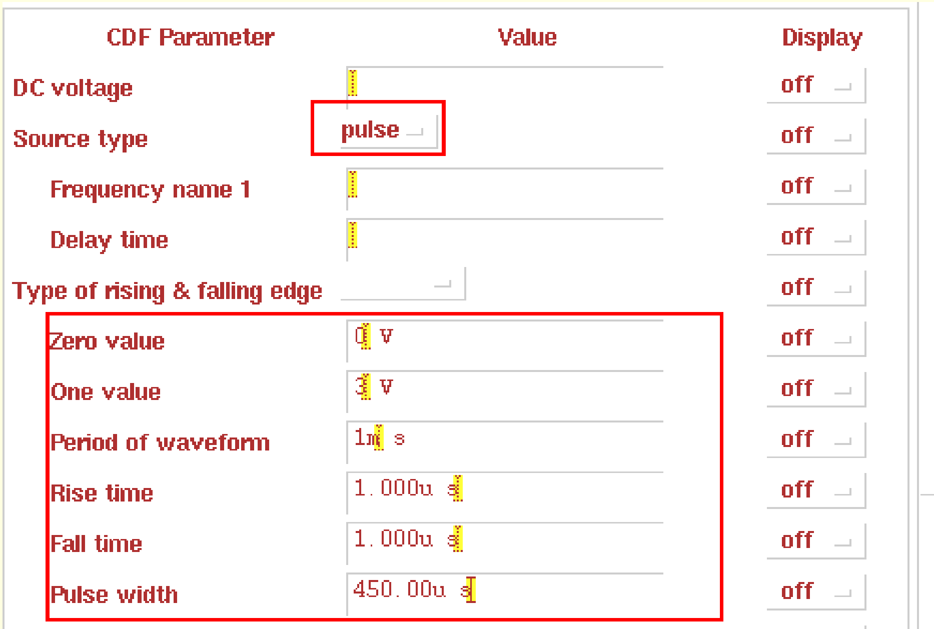


An example-SWC RC filter

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.

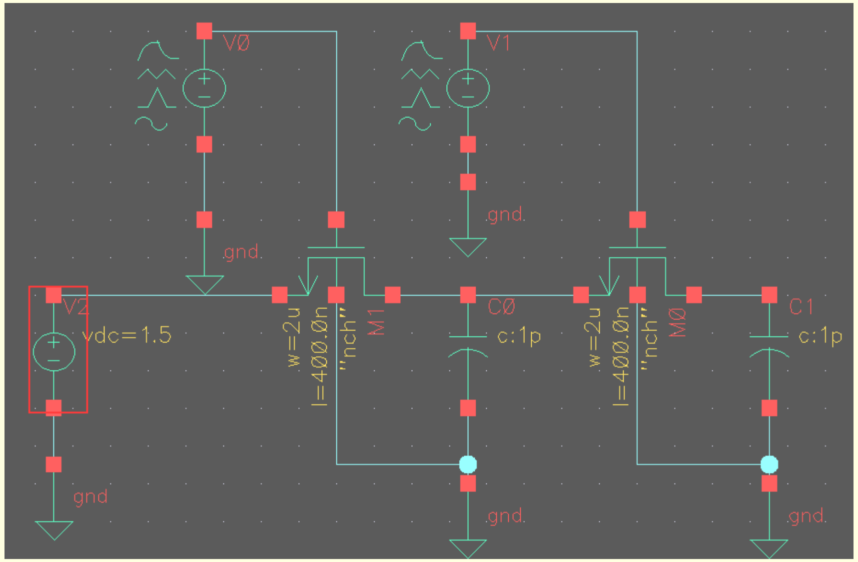


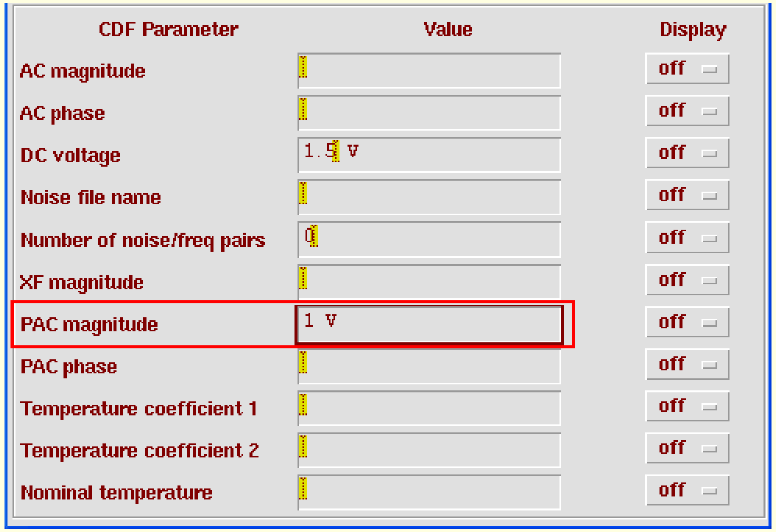




An example-SWC RC filter

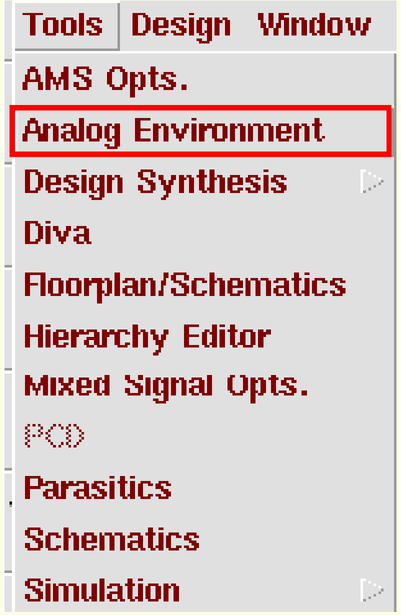
To simulate frequency response of the switched capacitor filter,add analogLib → Vdc as input. Choose PAC magnititude =1.

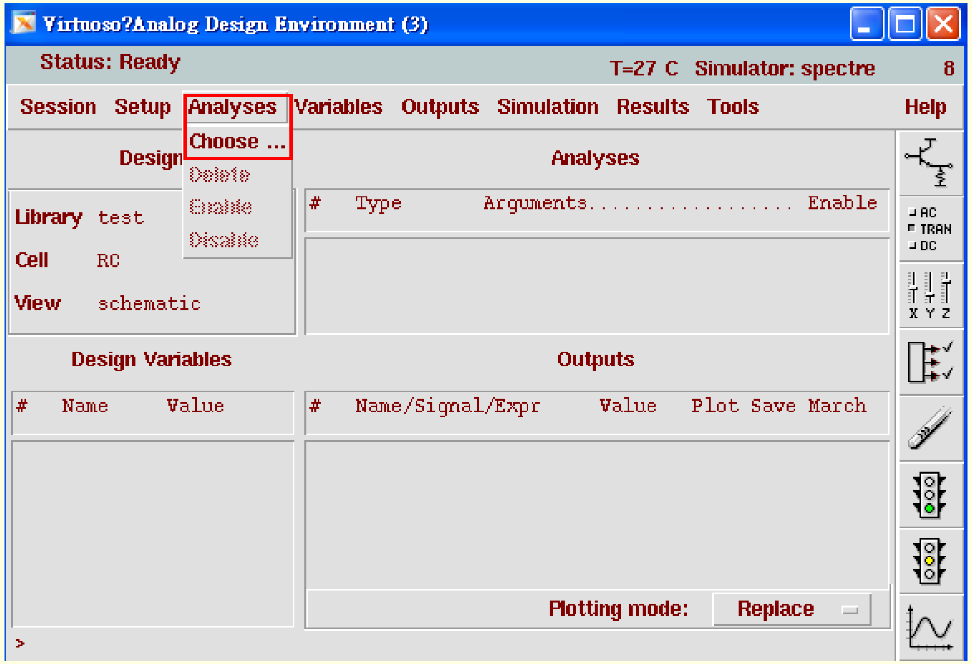




An example-SWC RC filter

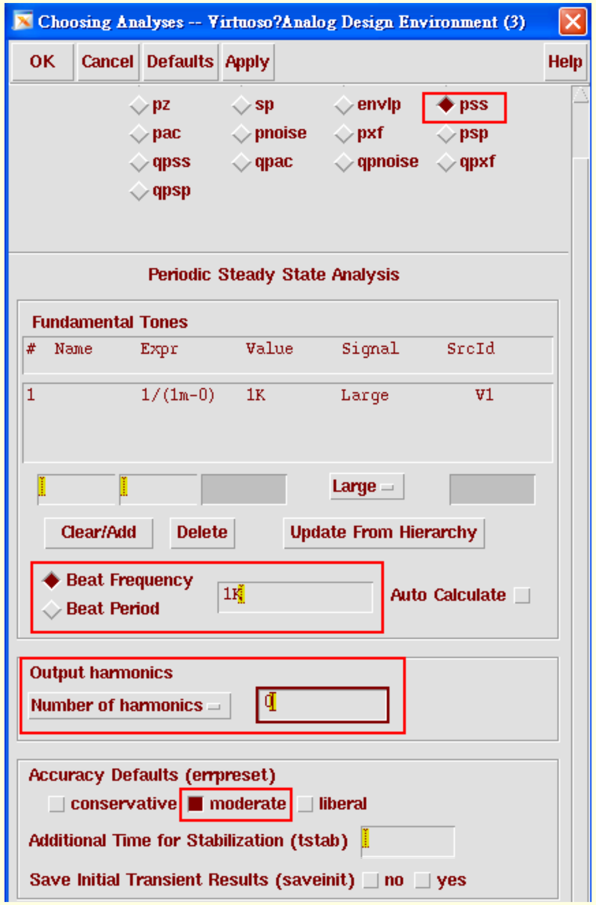
Click Tools → Analog Environment.Click Analysis → Choose.

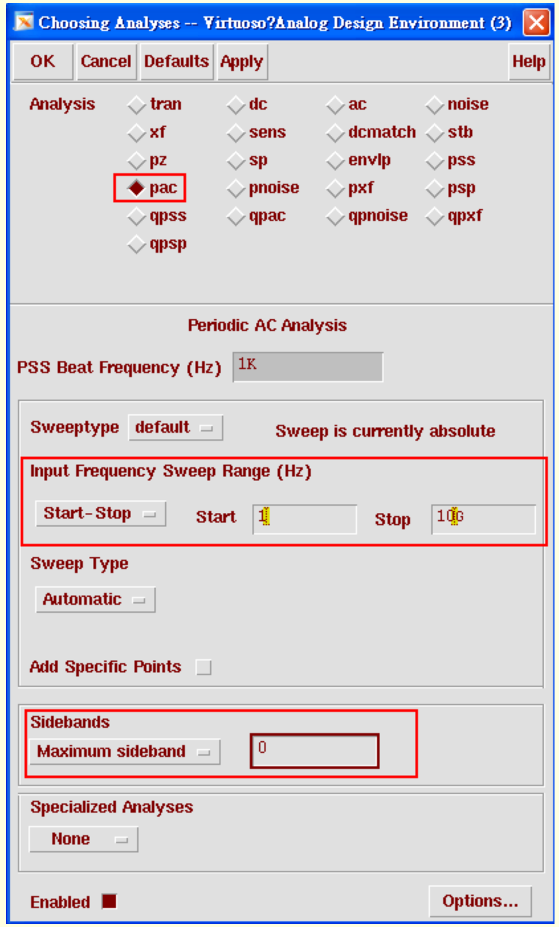




An example-SWC RC filter

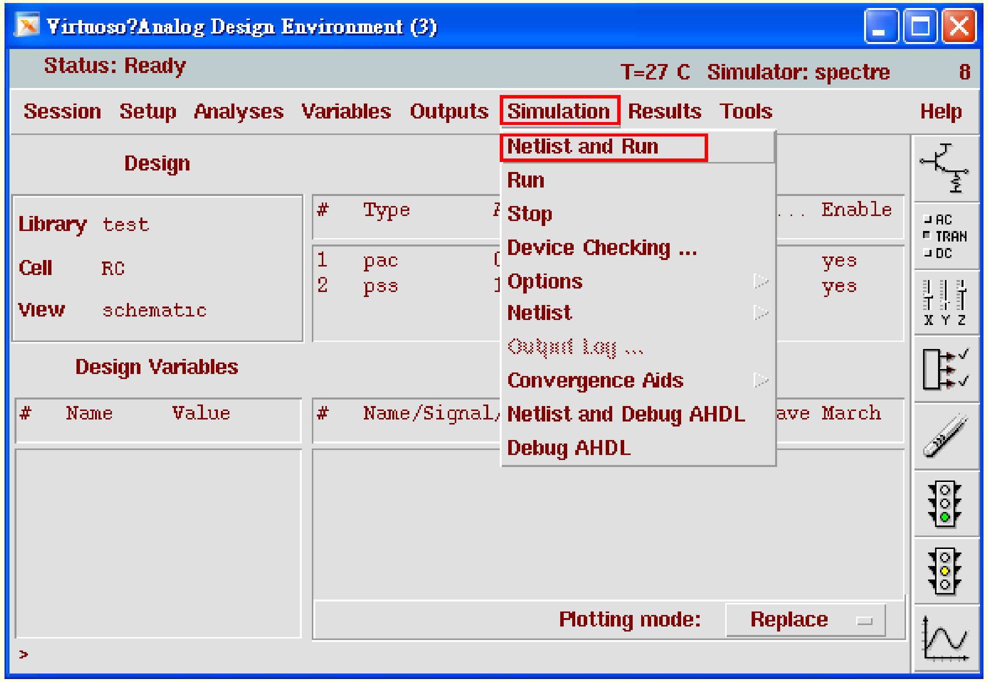
Use pss and pac analysis. Set Beat Frequency = your clock frequency. Output harmonics = 0.Accuracy = moderate.Set your interested frequency range in pac analysis and maximum sidebands=0.Click “OK” when all settings are done.

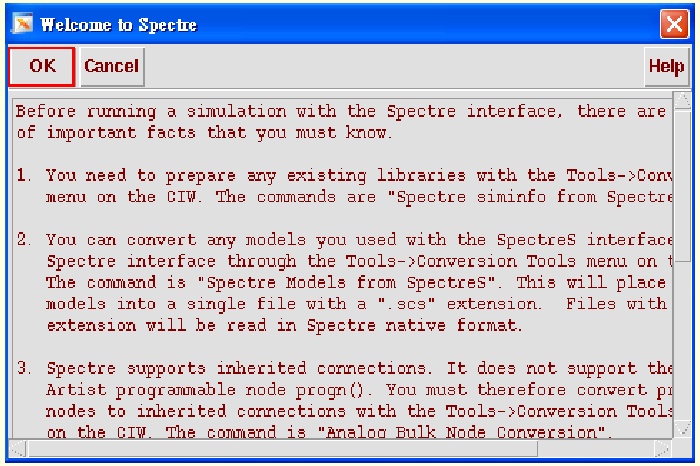




An example-SWC RC filter

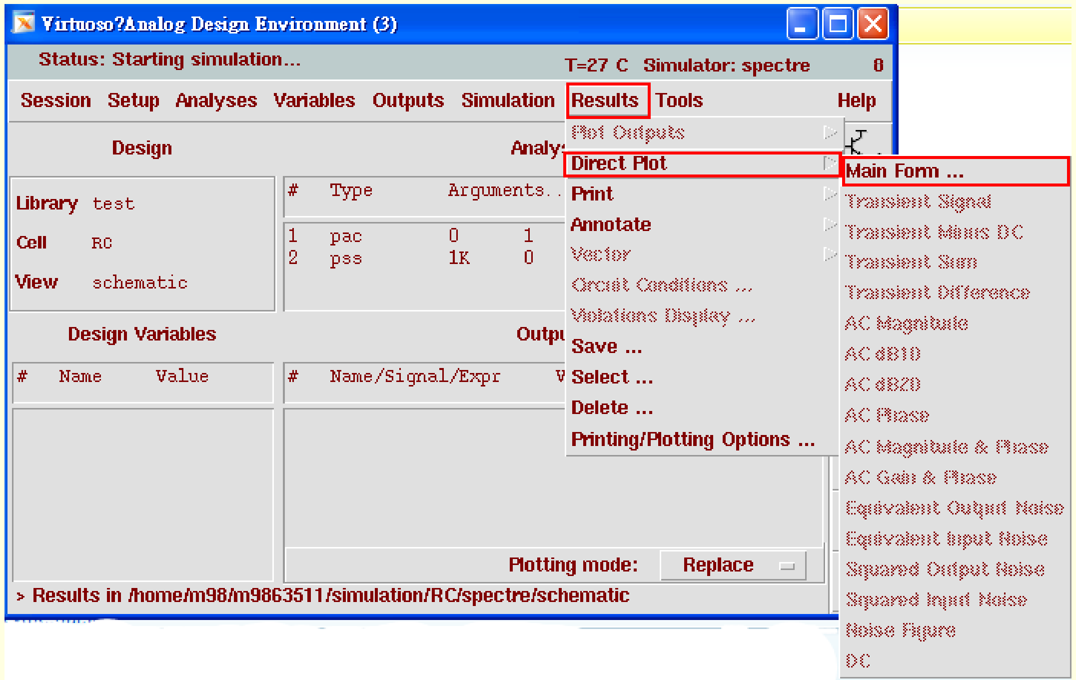
Back to ADE window.Click Simulation → Netlist and Run.Click OK.

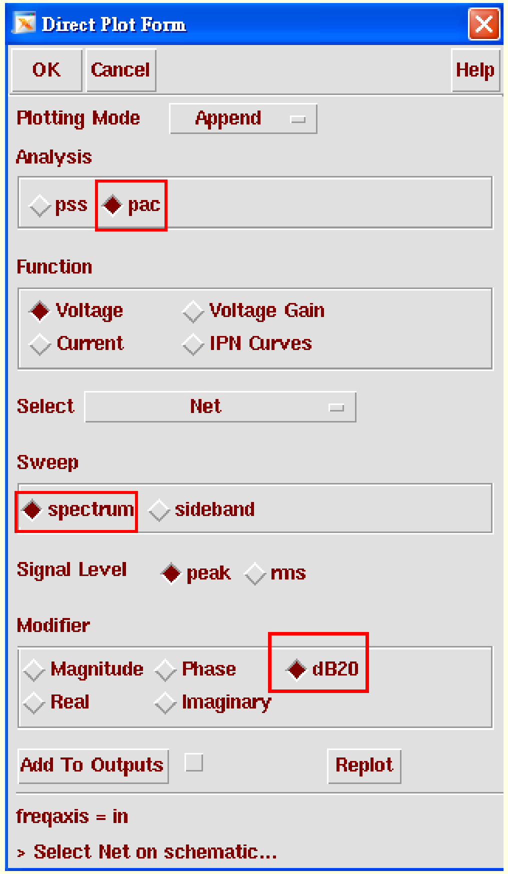




An example-SWC RC filter

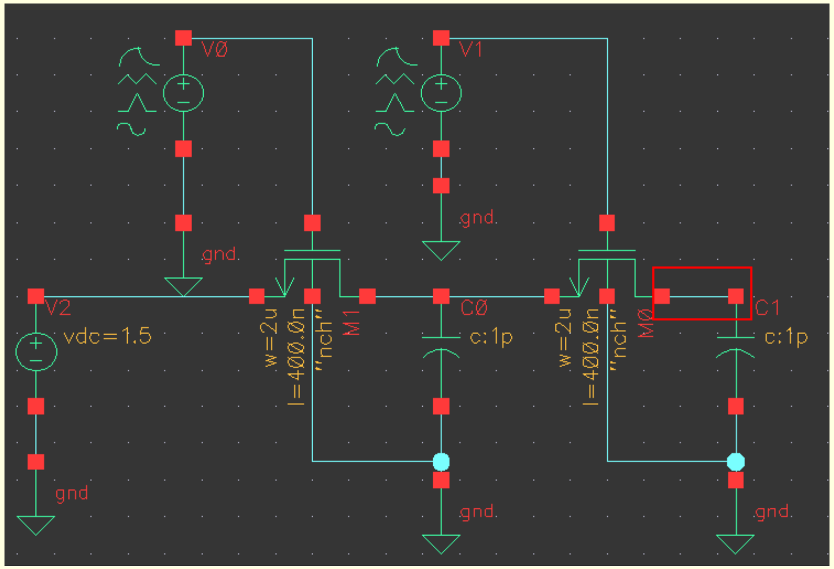
Click Results → Direct Plot → Main Form.Choose pac → spectrum →dB20.

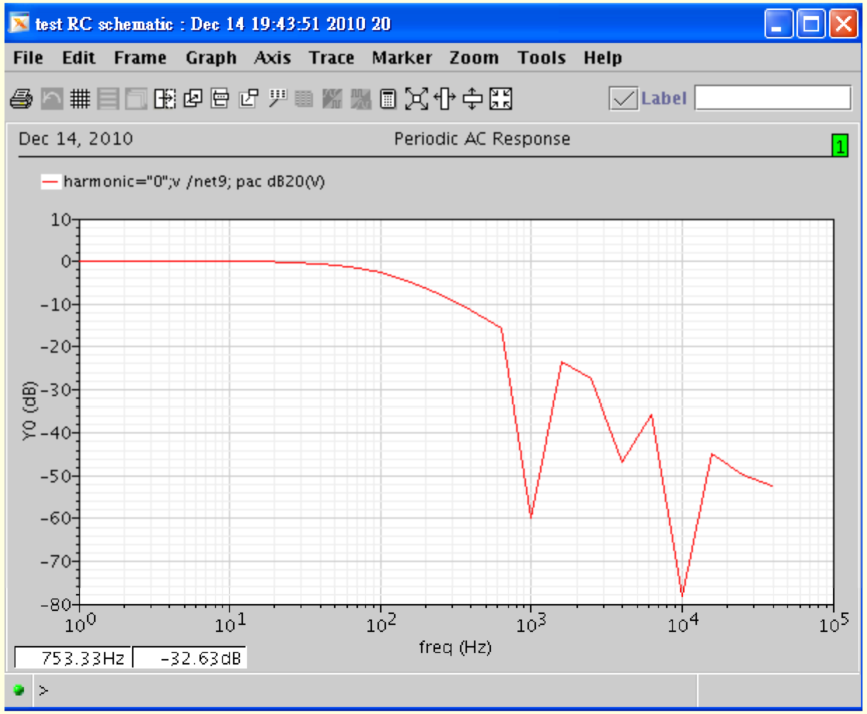




An example-SWC RC filter

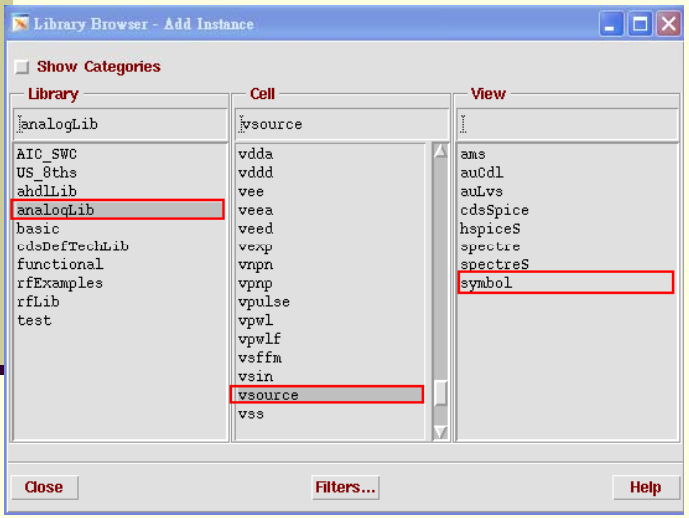
Click on the output node,and you'll see the frequency response.

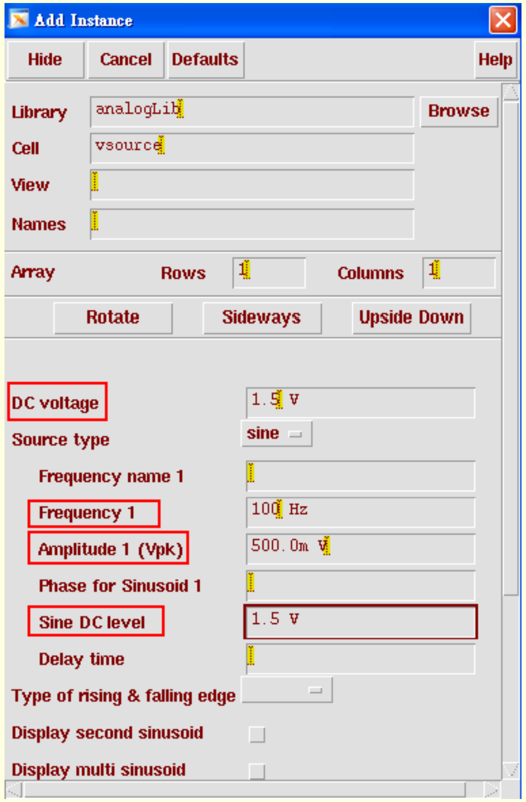


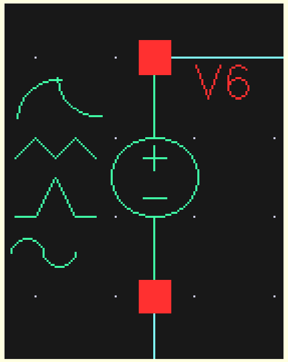


An example-SWC RC filter

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

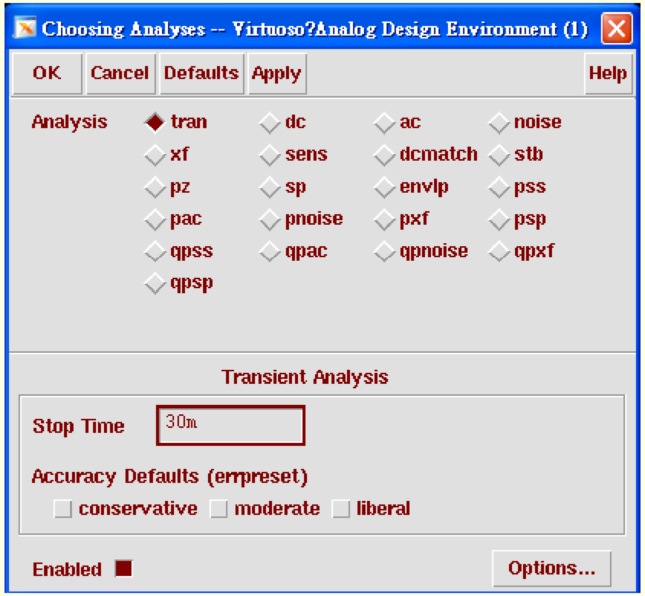


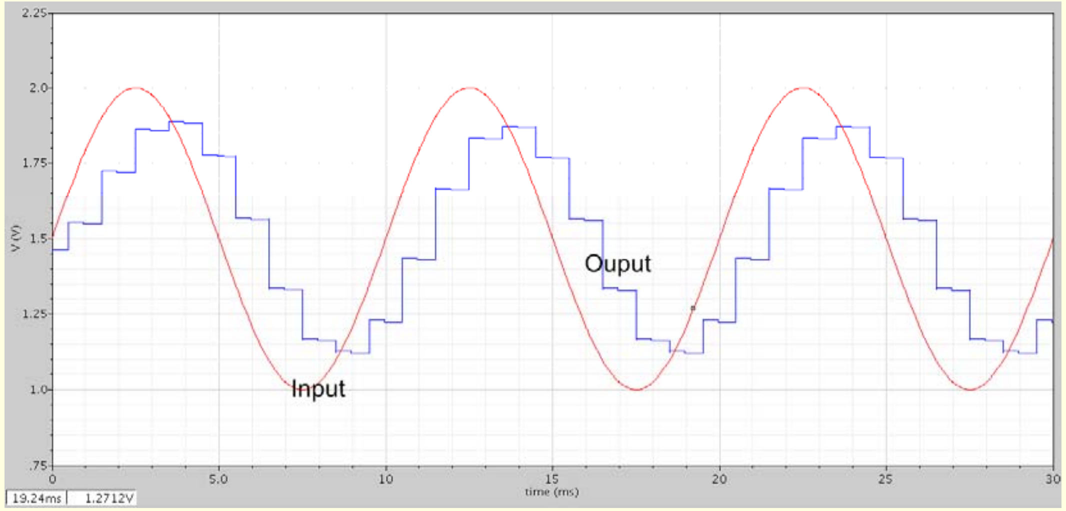




An example-SWC RC filter

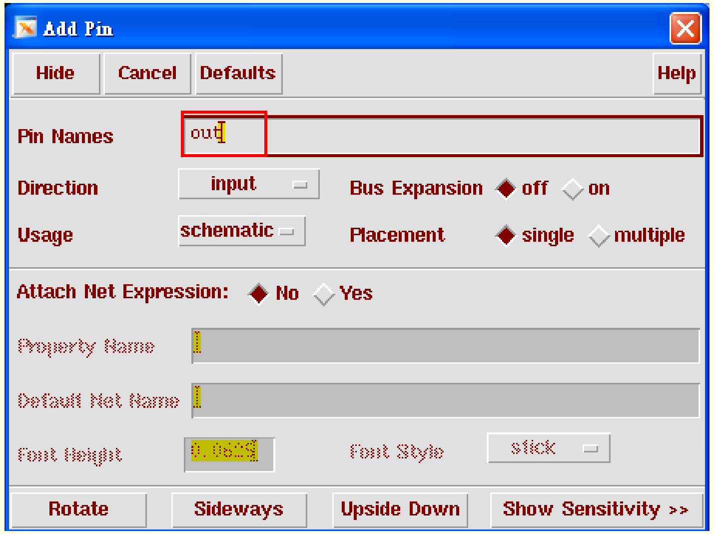
Simulation results when clock frequency=1KHZ and input is a 100HZ Sinusoidal wave.
Transient analysis and AC analysis are the same as HSPICE.Just keep in mind that use Vdc for AC and Vsource for transient.

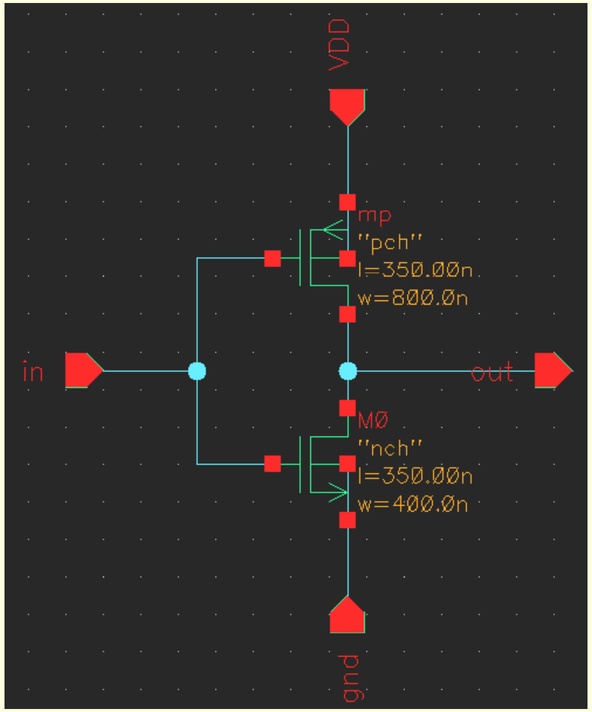




Create symbol

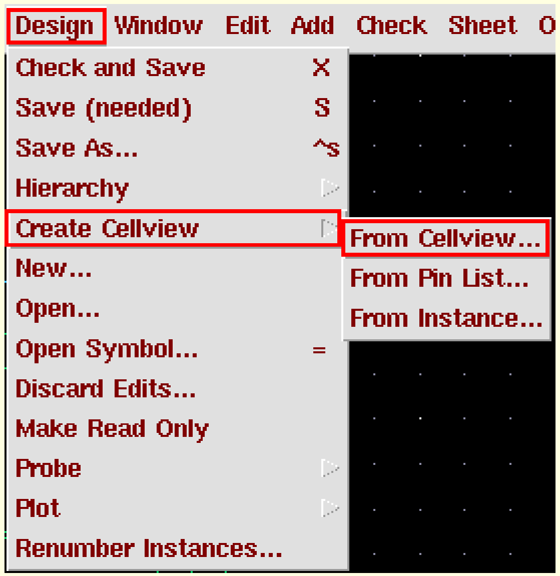
Use an inverter as example. Press “p” to create pin. Define your pin names. Connect them to the corresponding nodes

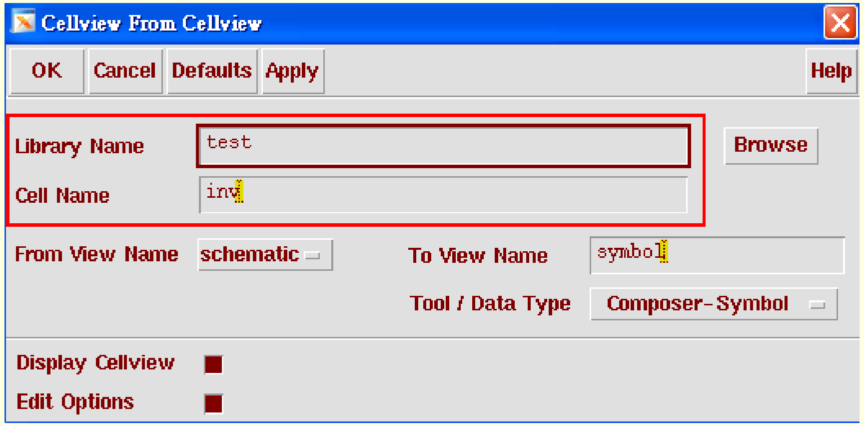




Create symbol

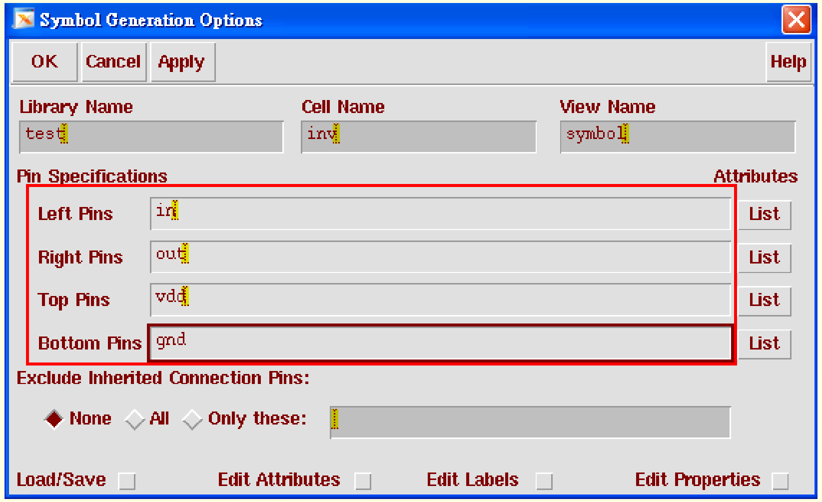
Click Design → Create Cellview → From Cellview→ key in your library and cell name

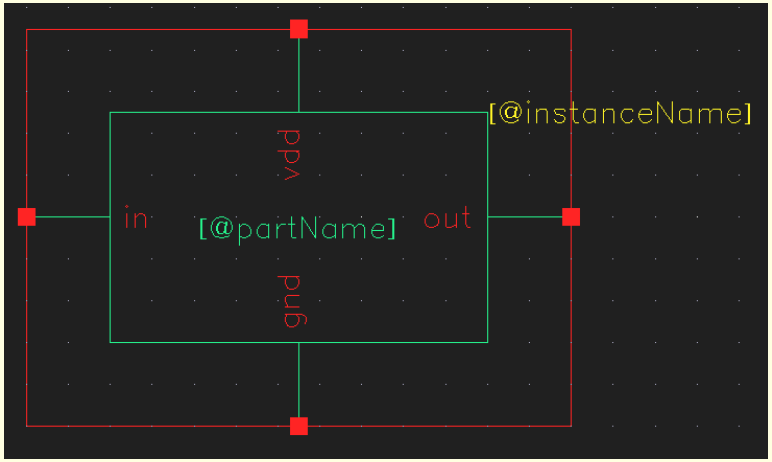




Create symbol

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