Analog IC Design Homework 6 (1/5)

- 1. Use CIC 0.18um 1.8V Virtual Process SPICE Model (cic018.l)
- 2. Please express the steady-state output (V_{out}) of the multiplying-by-four gain amplifiers (as shown in Fig. 1 & Fig. 2) in terms of A, C_1, C_2, C_3, C_4, V_{in}, $V_{CM,}$ and V_{offset}, and explain the difference between Fig. 1 & Fig. 2.
- 3. Please design the multiplying-by-four gain amplifiers (as shown in Fig. 1 & Fig. 2) including two-stage OPAMP, bias circuit and non-overlapped clock circuit (as shown in Fig. 3). Then verify your design with Spectre/HSPICE. Assume C_1=C_2=C_3=C_4=1pF and C_L=1.5pF
  - (a) With V_{in}=V_{CM}+0.025V and V_{offset}=0V
  - (b) With V_{in}=V_{CM}+0.025V and V_{offset}=0.05V

- Note
  - OPAMP and bias circuit should be included in your homework
    - You can use the OPAMP and bias circuit designed in homework 2
  - Follow design rules of maximum and minimum transistor width and length
  - Please set the common mode voltage V_{cm} in a reasonable voltage
  - Please use Cadence Virtuoso to build schematic and export the .cir file for verification

Analog IC Design Homework 6 (2/5)

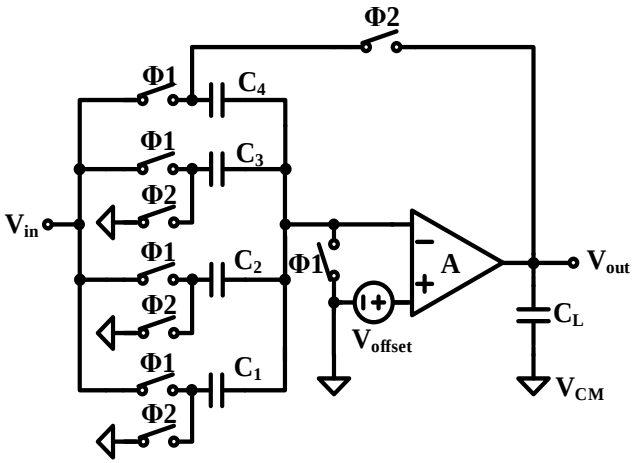


Fig. 1

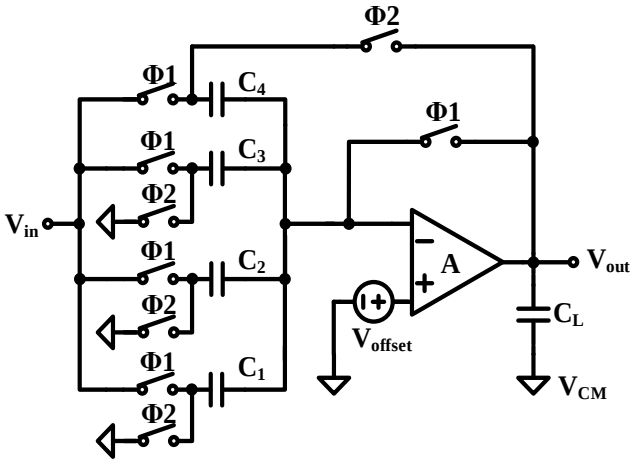


Fig. 2

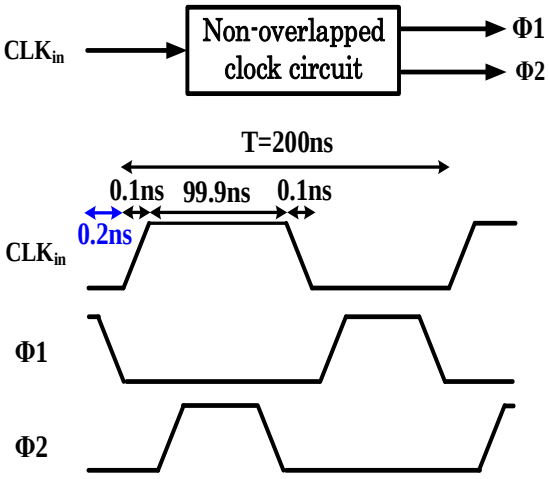


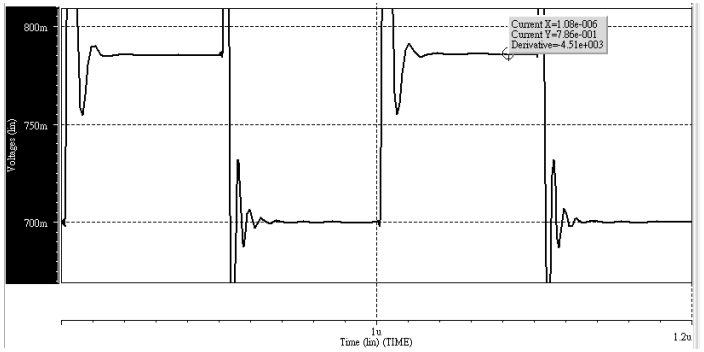
Fig. 3

- You should define the clock characteristics as the one shown in Fig. 3.

$$\text{Accuracy} = \left[1 - \frac{|v_{OUT} - (4v_{IN} - 3V_{CM})|}{4(v_{IN} - V_{CM})}\right] \times 100\% \,@\, t = 1.275\mu s$$

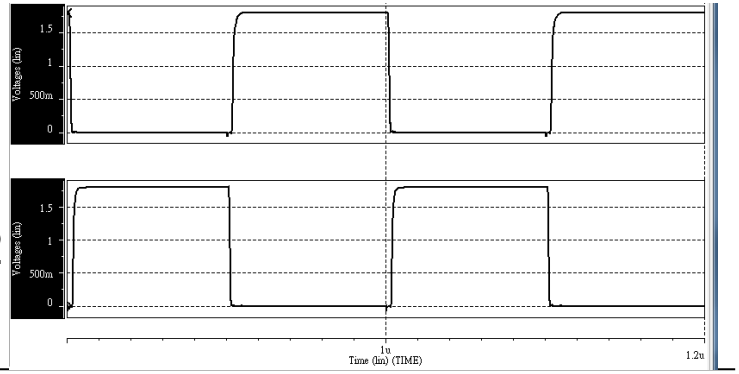
- The waveform of V_{out} , Φ1 and Φ2

V_{out}



Φ1

Φ2



Analog IC Design Homework 6 (3/5)

- Your report should include
  - Design flow (Example: The analysis of switches and non-overlap clock circuit)
  - Spectre/HSPICE verification results
    - The waveform of V_{out} , Φ1 and Φ2
    - Accuracy of the multiplying-by-four gain amplifiers (Please refer to testbench)
  - Virtuoso schematic (Include the .cir file and circuit diagram)
  - Total current and power consumption
  - Area
    - MOSFET: Please calculate the sum of the W*L for all the MOS used in the designed circuit $A_{MOS} = \sum W_i \times L_i$
    - Resistor / Capacitor: Just show the resistance / capacitance (if used)
  - Table of specifications

	Fig. 1		Fig. 2	
Area (μm^2)				
Power consumption (μW)				
Accuracy (%)	w/o offset	w/ offset	w/o offset	w/ offset

- Note: The total current, power consumption and area should include bias circuits, two-stage OPAMP and non-overlapped clock circuit

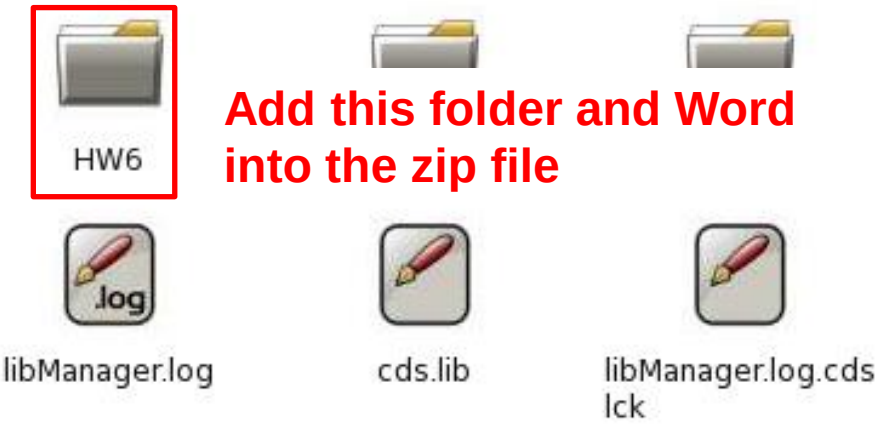
Analog IC Design Homework 6 (4/5)

- Grading
  - Customization of the .sp file will not be accepted
  - Please use Virtuoso Spectre or HSPICE to verify your circuit
  - Under the condition of all circuits are composed of MOS, R and C and meet the test conditions, the higher accuracy, area, and power consumption will receive higher scores
  - Please clearly describe the design flow in your report and attach your calculation process (Do not copy)
  - Report with simulation results only but no design flow will get deducted points according to the situation
  - Report with advanced discussion and analysis will receive higher scores

- Precautions
  - Deadline: 12/18/2022 (Sun.) 23:59:59 pm (不接受作業補交)
  - Personal work, please upload Word and Virtuoso library (or HSPICE code) to moodle
  - Please compress files into a .zip file and name it as HW#_student ID,
    ex: HW6_E24064088 Font size: 12pt (Chinese: 標楷體, English: Times New Roman)
  - Refer to the IEEE submission regulations, set the picture resolution to 300dpi.
  - Upload file size is recommended to be 2MB

Analog IC Design Homework 6 (5/5)

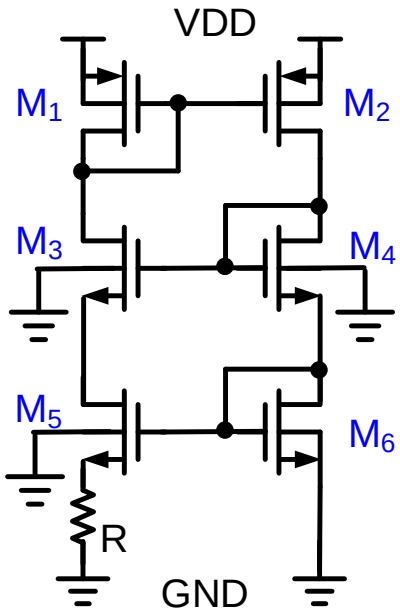
- Virtuoso library folder
  - Add virtuoso folder into the zip file (make sure your file can be opened by virtuoso)

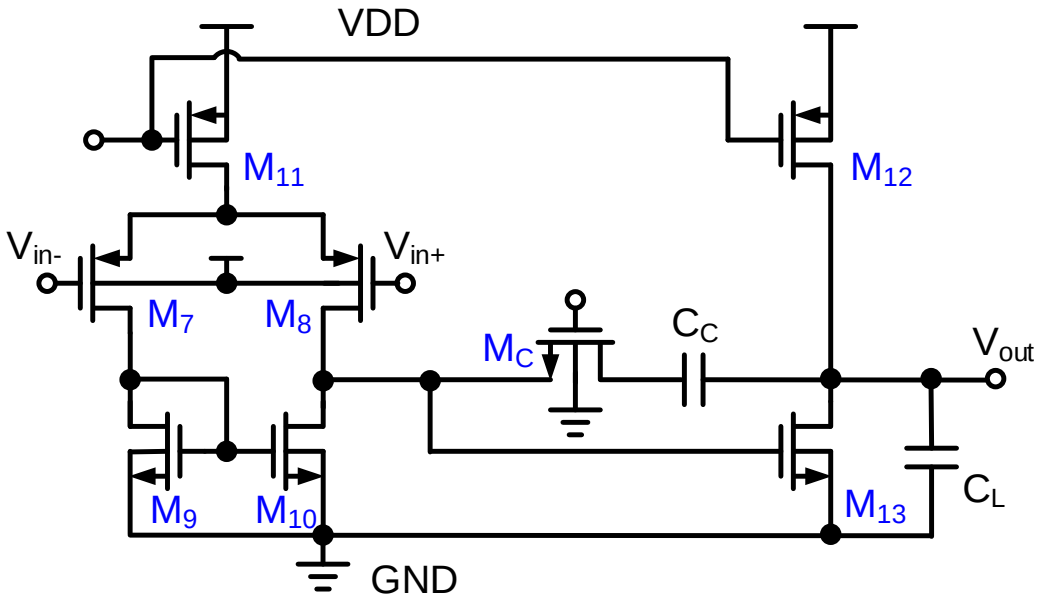


- Use the resistor and capacitor in analogLib library (res, cap)

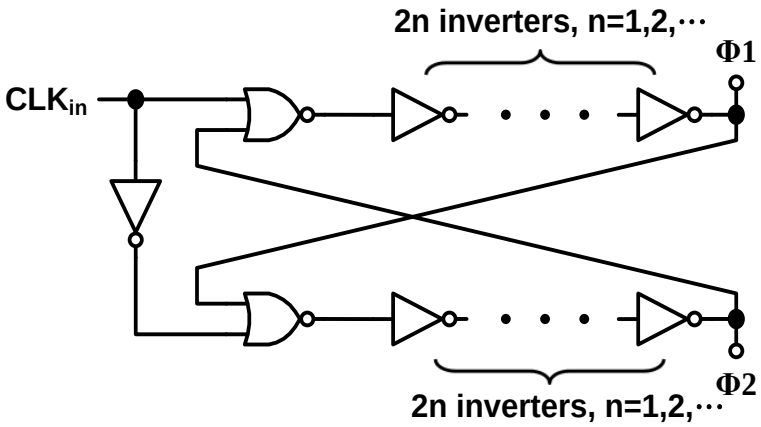
Appendix 1: Circuits for Reference (1/4)

- Bias circuit & Two-stage OPAMP





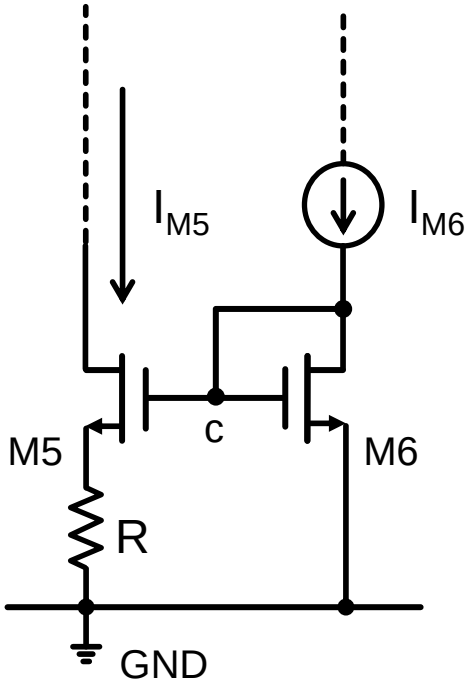
- Non-overlapped clock circuit



Appendix 1: Circuits for Reference (2/4)

Design Flow of Bias Circuit

- Step 1: 設定 I_{M6} 跟 V_{eff6} ➔ Find $\left(\frac{W}{L}\right)_6$
  - $I_{M6} = \frac{1}{2} Kn' \left(\frac{W}{L}\right)_6 (V_{gs6} - V_t)^{\alpha}(1 + \lambda V_{DS})$
    - $I_{M6} = \frac{1}{2} Kn' \left(\frac{W}{L}\right)_6 V_{eff6}{}^{\alpha}(1 + \lambda V_{DS})$
- Step 2: 設定R ➔ Find $\left(\frac{W}{L}\right)_5$
  - $V_{eff6} = V_{eff5} + I_{M5} * R$
  - $I_{M5} = \frac{1}{2} Kn' \left(\frac{W}{L}\right)_5 V_{eff5}{}^{\alpha}(1 + \lambda V_{DS})$
  - $I_{M5} = I_{M6}$
- Step 3: 設計M1~M4之size，使其操作在飽和區且Veff符合限制
- Step 4: 設計start up circuit



Appendix 1: Circuits for Reference (3/4)

Design Flow of Two Stage OPAMP

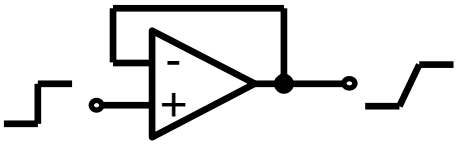
- Step 1: 決定C_C (C_C值會小於C_L，太大會增大整體面積，太小會受到雜散電容的影響)
- Step 2: 由 $\varphi_m = 90° - \tan^{-1}(f_t/f_{P2}) = 63°$ ➔ $f_{P2} = \frac{g_{m_{13}}}{2\pi C_L} \approx 2 f_t$ 且 $f_t = \frac{g_{m_8}}{2\pi C_C}$ ➔ 求得 $g_{m_{13}}$
- Step 3: 由 Slew rate(SR) $< \frac{I_{11}}{C_C}$ & $\frac{I_{12}}{C_C + C_L}$ ➔ C_C已知且SR須同時滿足兩公式➔ 求得 I_{11}、I_{12}
- Step 4: 由DC Gain $A_0 = \frac{\alpha I_{D_8}}{V_{eff8}} (r_{o_8} || r_{o_{10}}) g_{m_{13}} (r_{o_{12}} || r_{o_{13}})$ ➔ $I_{12} = I_{13}$ ➔求得 V_{eff8}
- Step 5: 求出W/L (由上述條件可算出除了M_9~M_{10}跟M_C的(W/L)值)
  - M_9跟M_{10}的size決定V_{gs13}的大小，調整$(W/L)_{9\sim10}$得到先前算出的V_{eff13}
  - M_C的size可由設計好的R_C值及公式 $R_c = \frac{1}{Kn'\left(\frac{W}{L}\right)_{Mc}(V_{GS_{Mc}} - V_t)}$ 求出

- Note: n3與V_{out}的偏壓點會決定M8、M10、M12跟M13的操作區域
  - 如果V_{out}的偏壓點過低，讓M13接近Linear region，會影響Gain的大小(ro變小)
  - 若往下拉力量太強，只要稍微降低V_{gs13}，就可以讓V_{out}的偏壓點上升
  - 若 V_{OUT} bias 偏低，可以 M9↑, M10 ↑ 或 M13 ↓

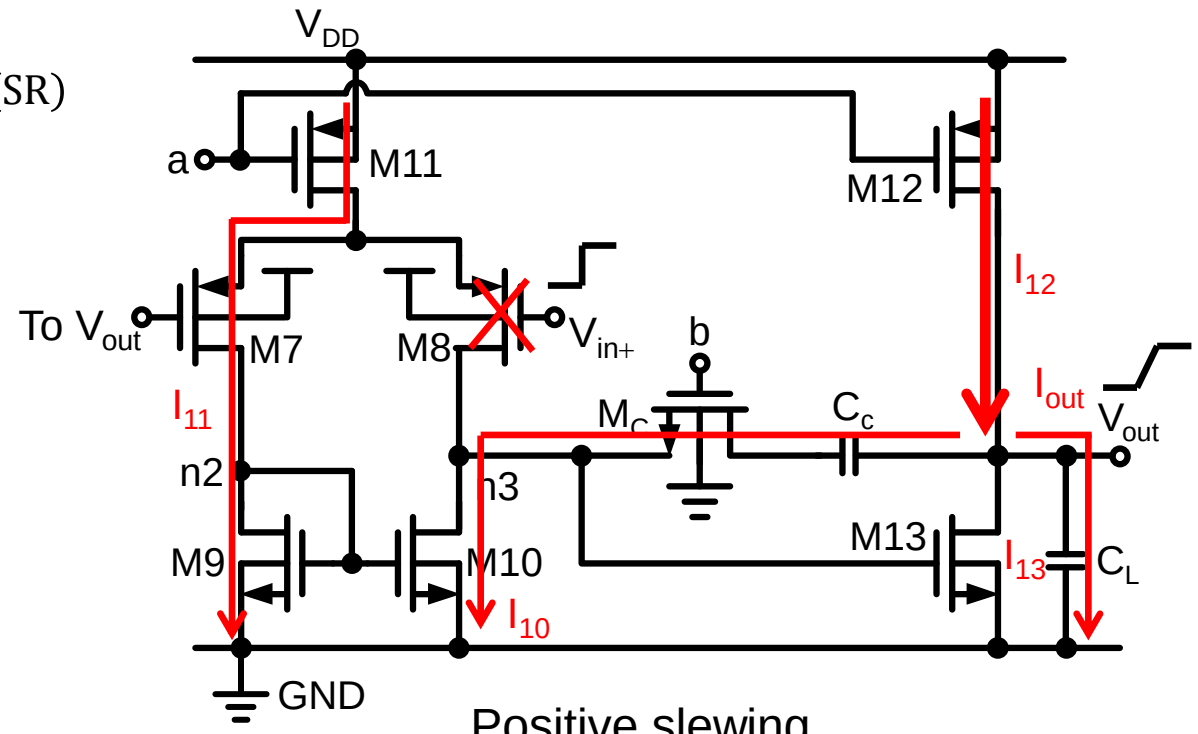
Appendix 1: Circuits for Reference (4/4)

Information of Slew Rate

- 講義6-29, 6-30所推得結果: Slew rate(SR) $= \frac{I_{11}}{C_C}$ ➔ 不完全適用於這次作業
  - 因為講義上之範例，第二級為ideal OP且無C_L
  - 當I_{12}足夠大 ➔ Slew rate(SR) $= \frac{I_{11}}{C_C}$
  - 當I_{12}不夠大 ➔ Slew rate(SR) $< \frac{I_{11}}{C_C}$ & $\frac{I_{12}}{C_C + C_L}$



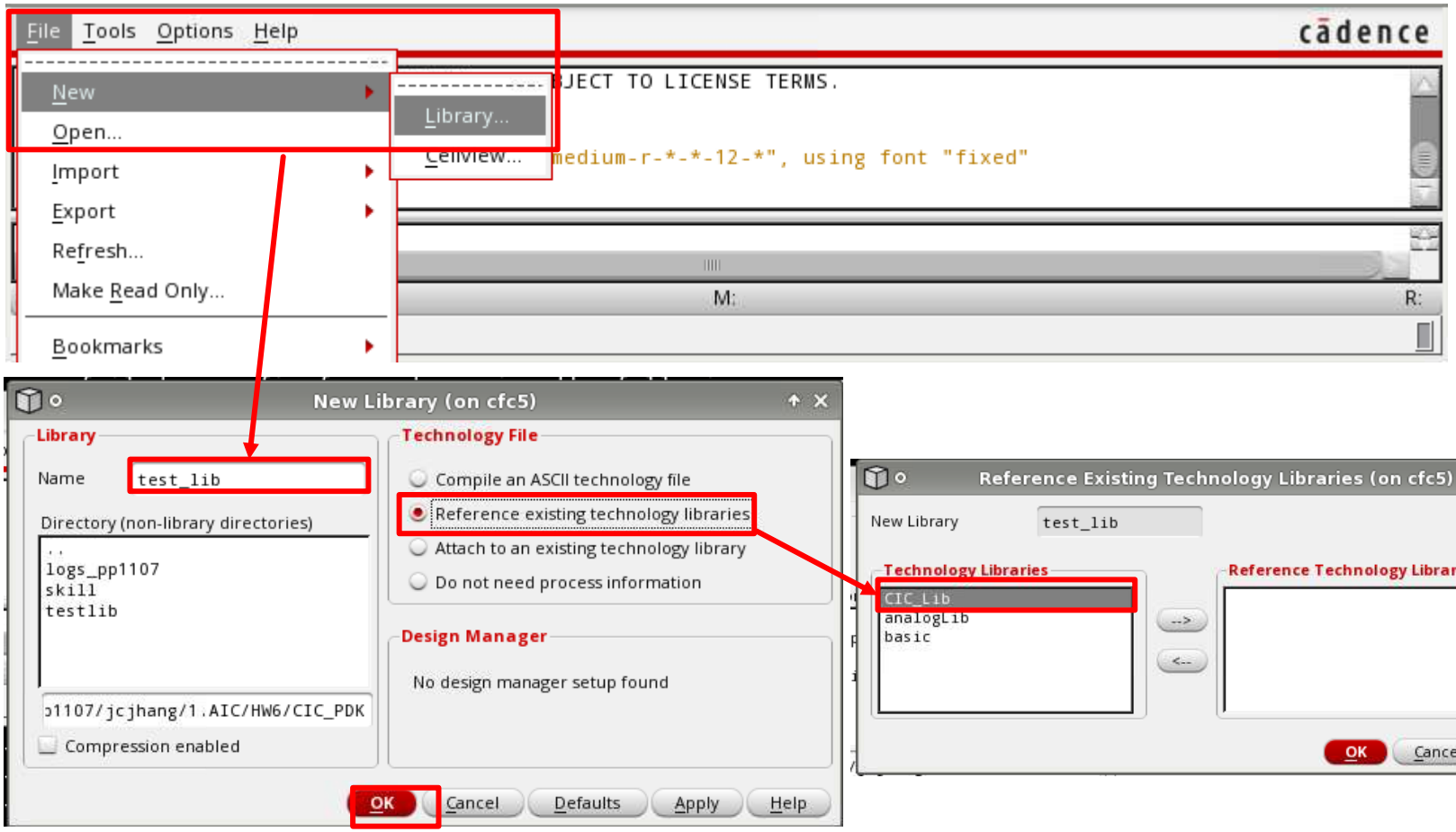
- Rising & Falling 之 Slew rate(SR)



Positive slewing

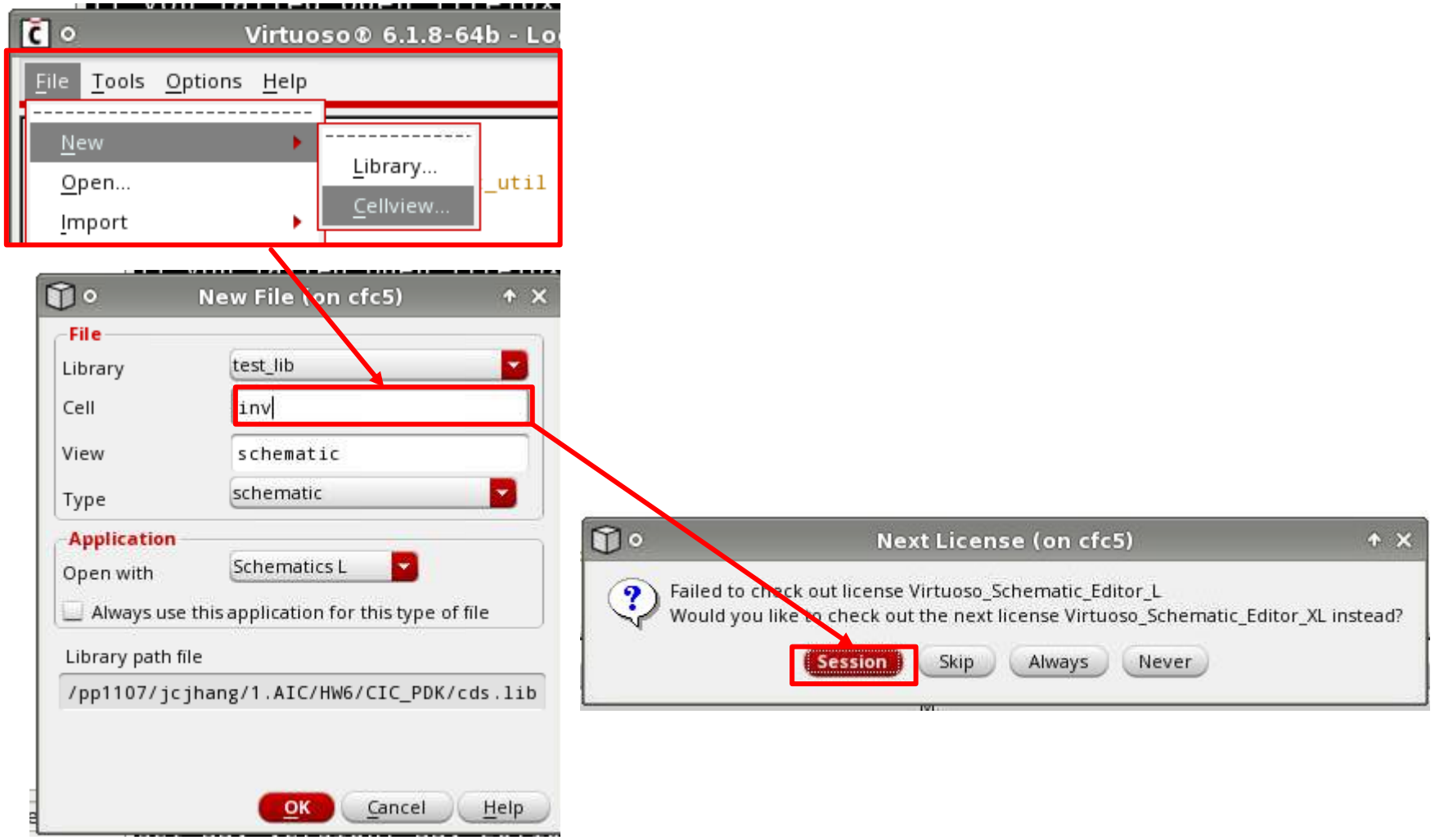
Appendix 2: Virtuoso Spectre Simulator (1/12)

- Terminal: cd CIC_PDK (0.18um virtual process virtuoso library)
- Terminal: source sourceMe
- Terminal: virtuoso & → Create a Library for CIC018 Model



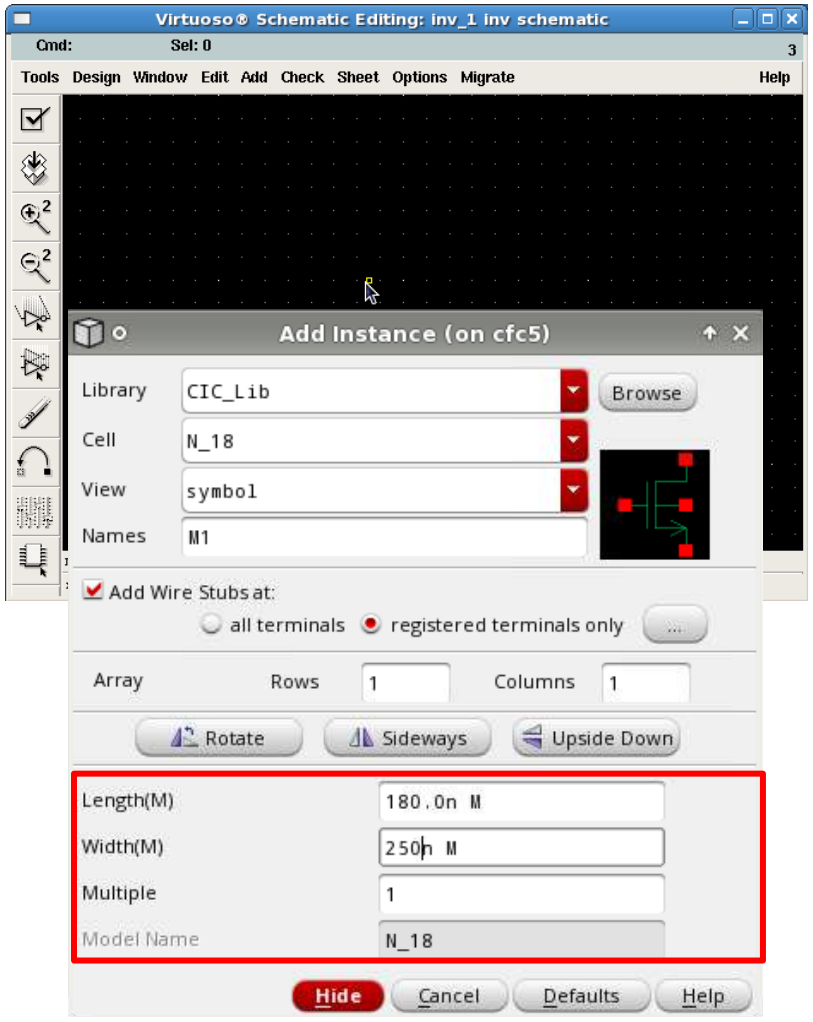
Appendix 2: Virtuoso Spectre Simulator (2/12)

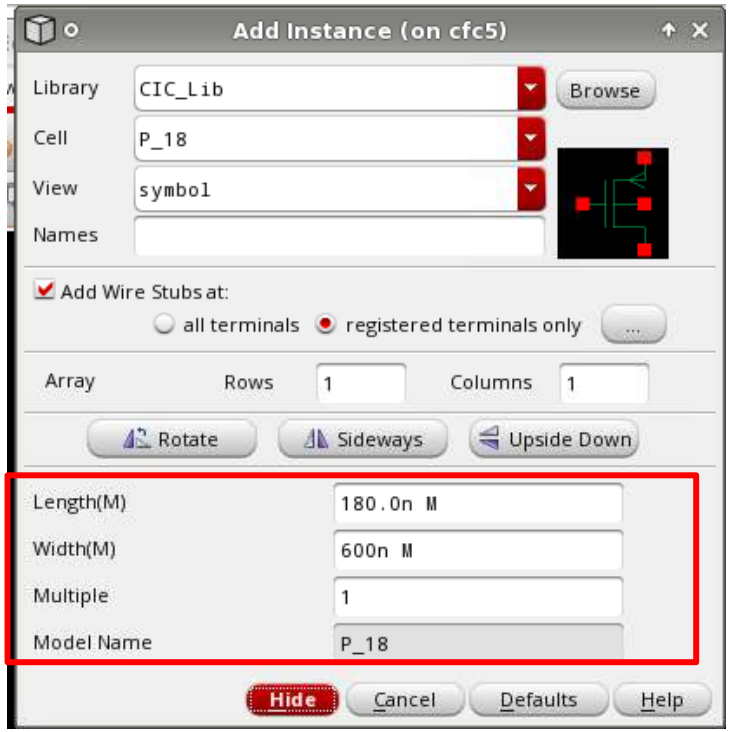
- Create Cell View



Appendix 2: Virtuoso Spectre Simulator (4/12)

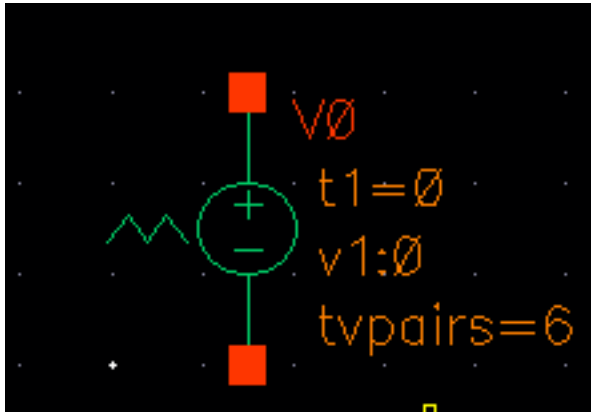
- Editing Interface → Add an Instance (Press 'i' to add instance)
- Press 'q' to edit object properties → Please show properties in your schematic circuit

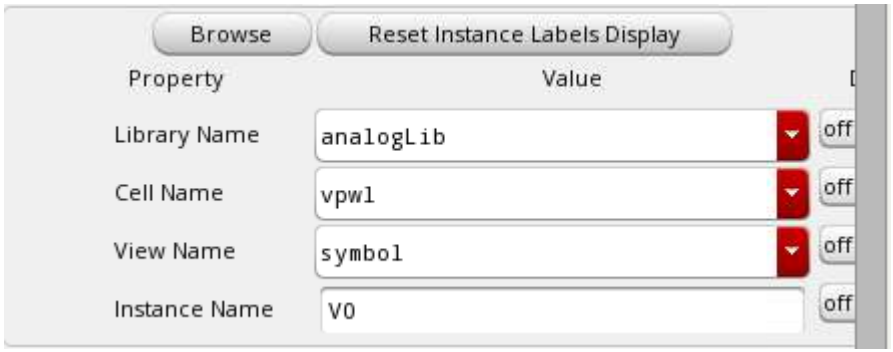




Appendix 2: Virtuoso Spectre Simulator (5/12)

- Piece-Wise Linear Voltage Source (Cell name: vpwl)
  - In analogLib



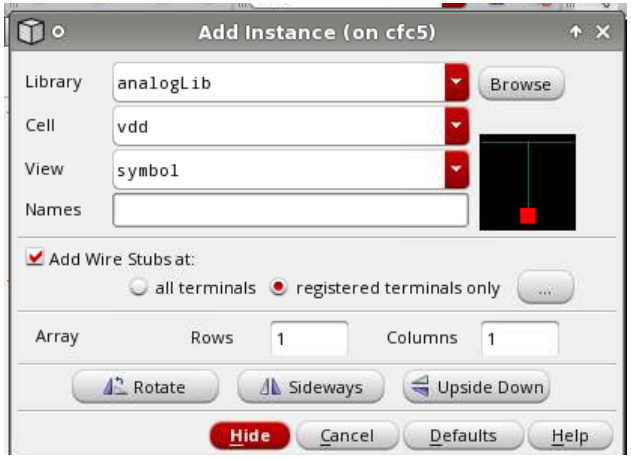


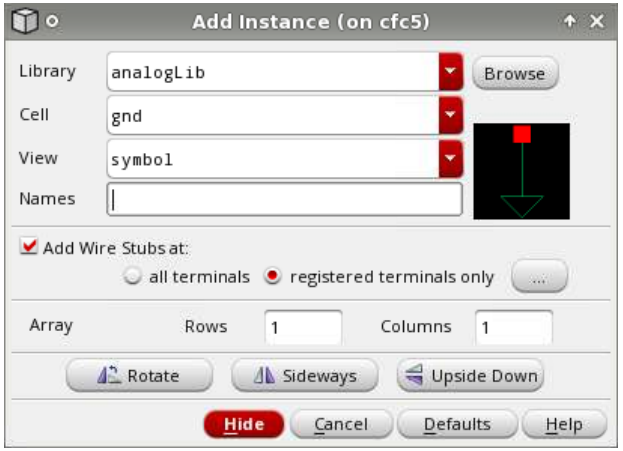
CDF Parameter	Value
Frequency name for 1/period	
Number of pairs of points	6
Time 1	0 s
Voltage 1	0 V
Time 2	1u s
Voltage 2	0 V
Time 3	1.5u s
Voltage 3	1.8 V
Time 4	2.5u s
Voltage 4	1.8 V
Time 5	3u s
Voltage 5	0 V
Time 6	4u s
Voltage 6	0 V

Period of the PWL	4u s

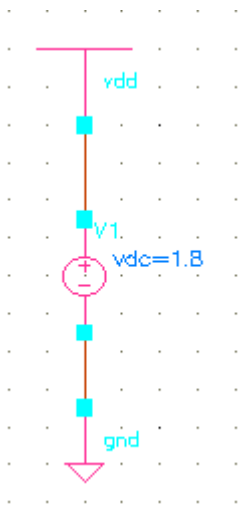
Appendix 2: Virtuoso Spectre Simulator (6/12)

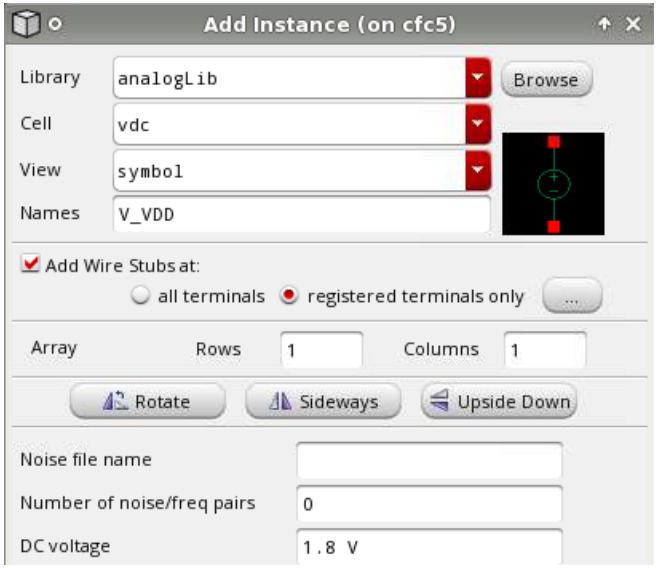
- Global VDD and GND (Cell name: VDD and GND)





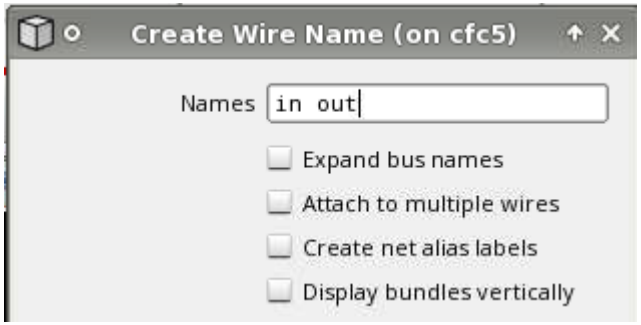
- Set VDD as 1.8V → Add a 1.8-V DC voltage source (Cell name: VDC)

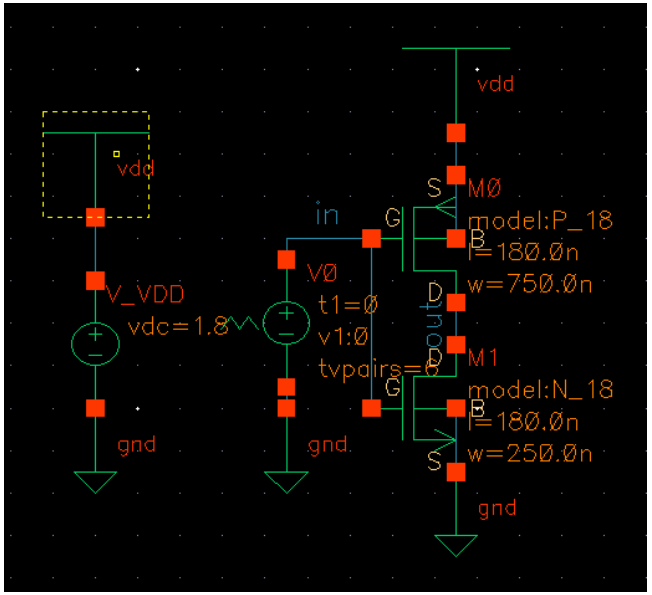




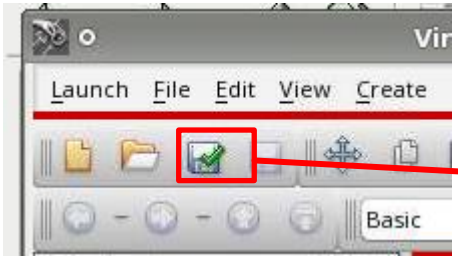
Appendix 2: Virtuoso Spectre Simulator (7/12)

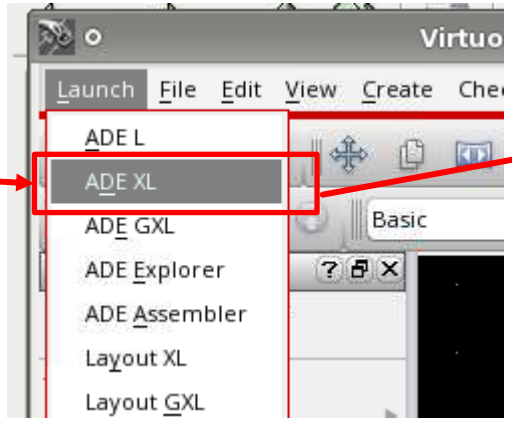
- Wiring and Wire Name setting
  - ‘w’ for wiring
  - ‘l’ (lower case of L) for adding wire name
    - Attach the auxiliary line to the wire to be named





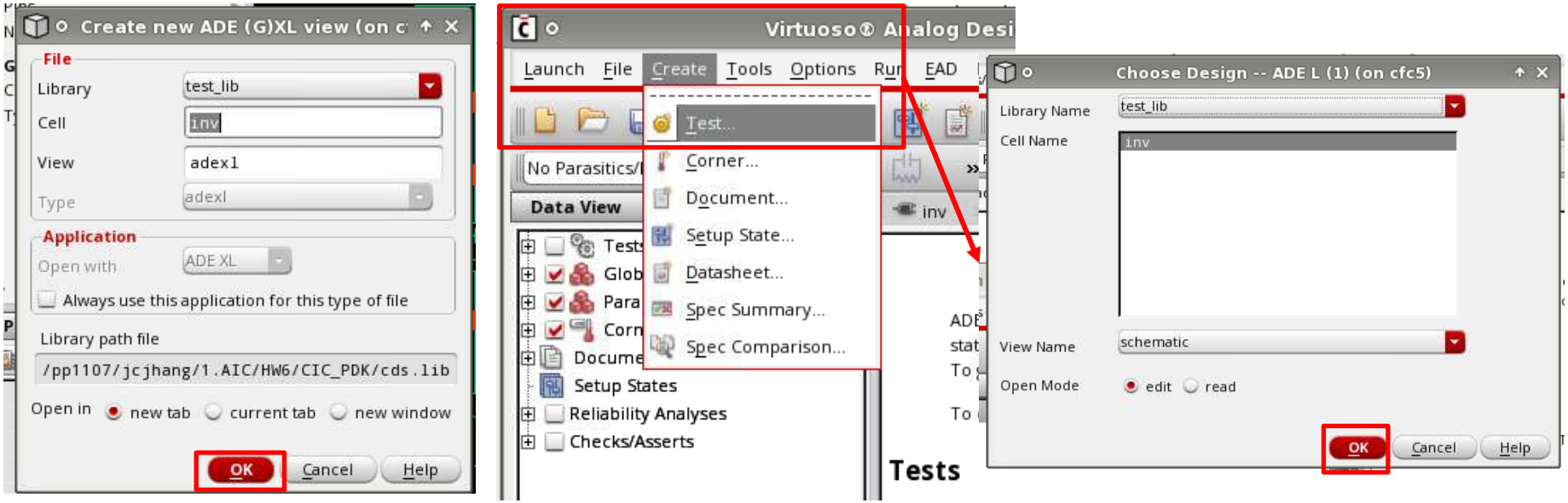
- Simulation by Spectre
  - Check and save it before using spectre for simulation → Launch → ADE XL



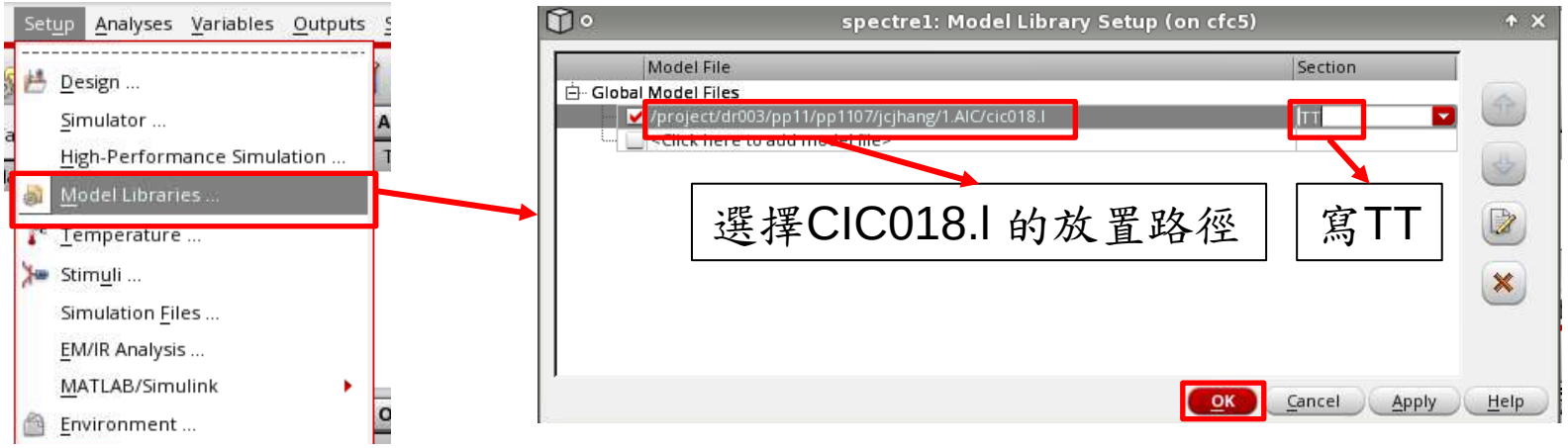


Appendix 2: Virtuoso Spectre Simulator (8/12)

- Create a test for the schematic

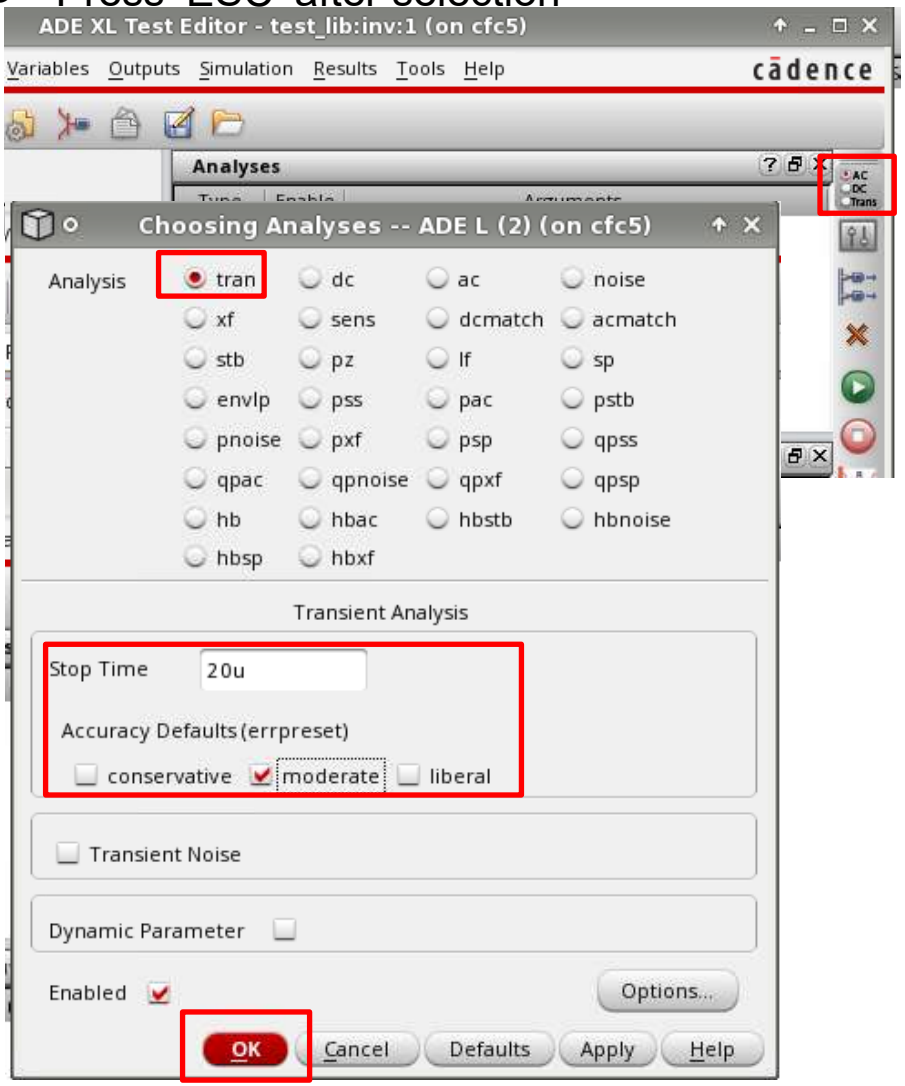


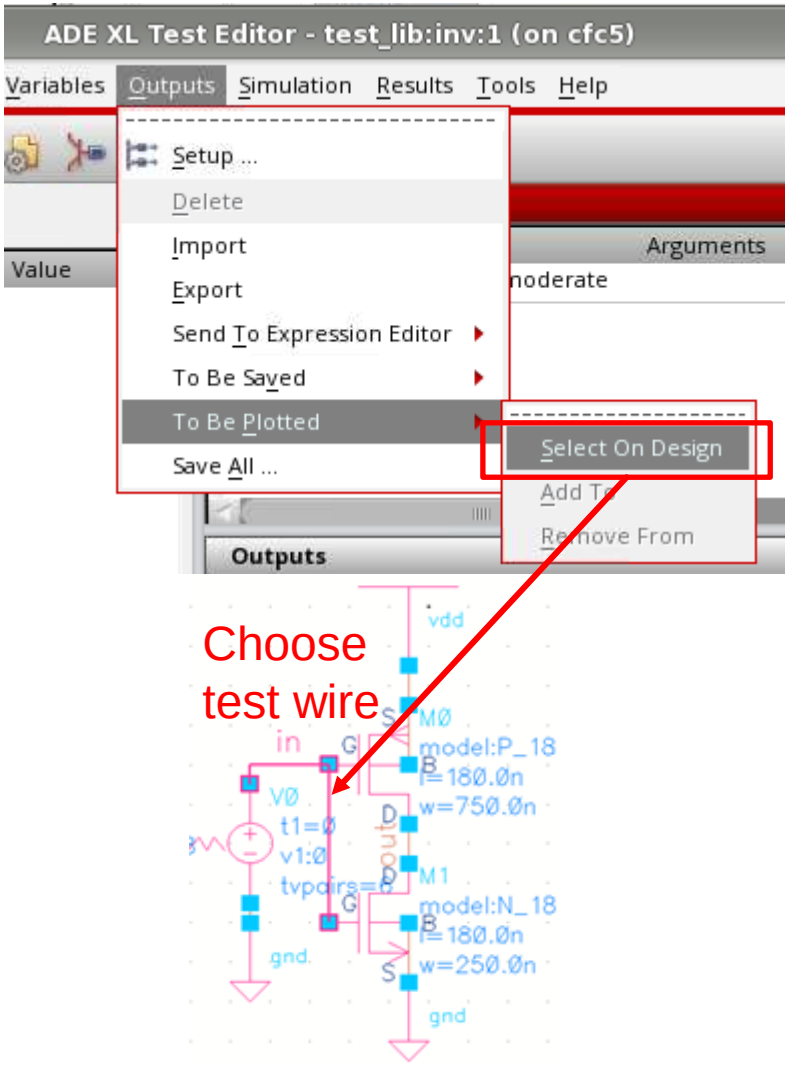
- Select CIC018 Model



Appendix 2: Virtuoso Spectre Simulator (9/12)

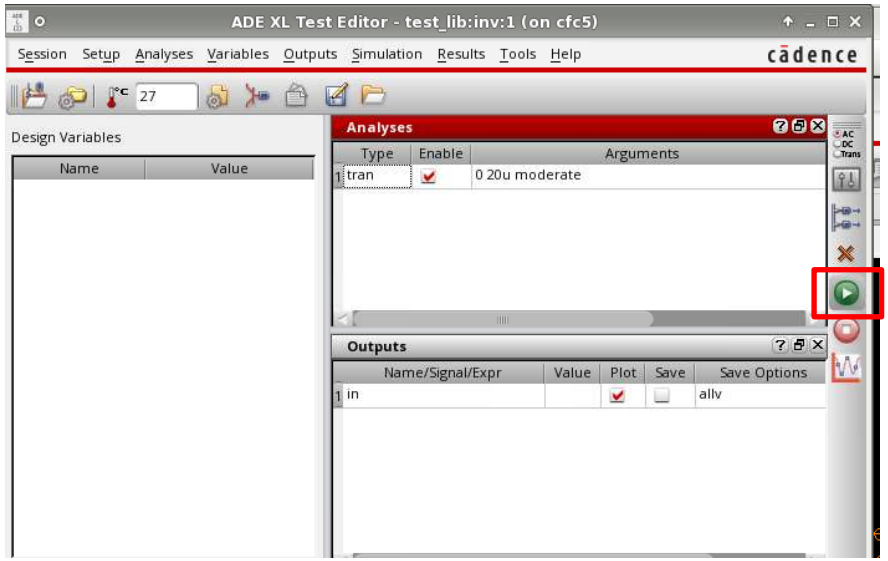
- Choose Analysis: .tran (Transient) → Plot Result → Click the wire to be plotted
- Press 'ESC' after selection

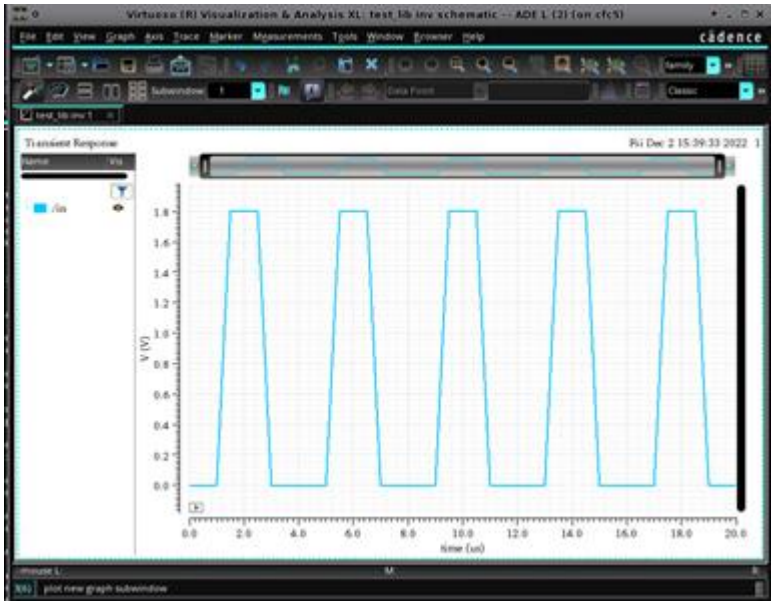




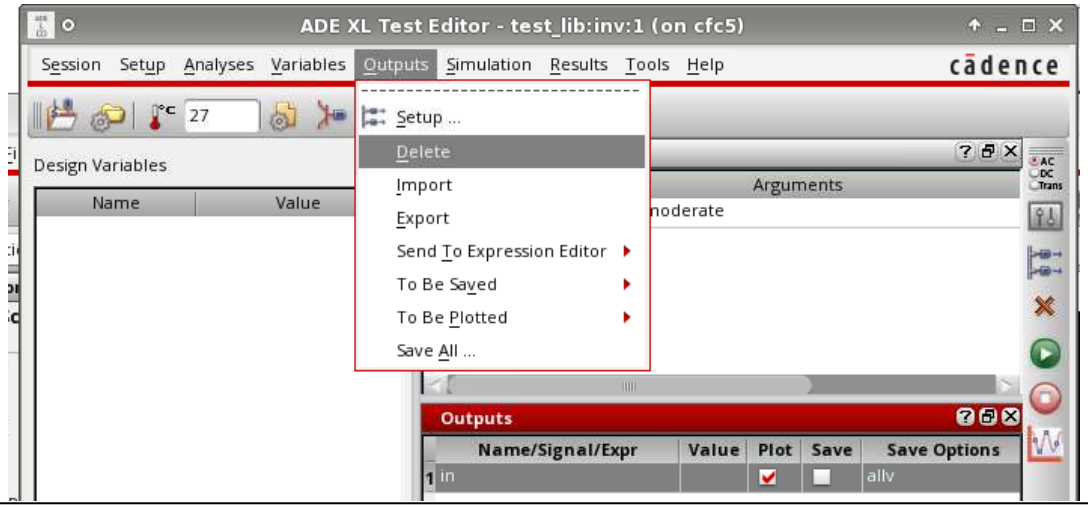
Appendix 2: Virtuoso Spectre Simulator (10/12)

- Run Simulation → Waveform Window



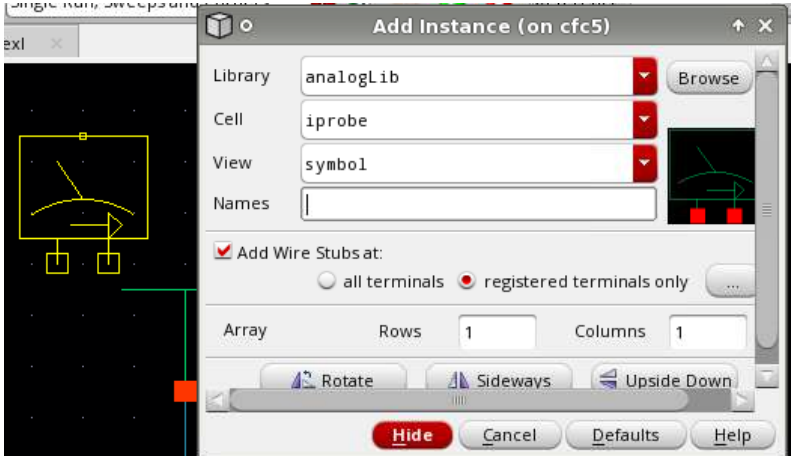


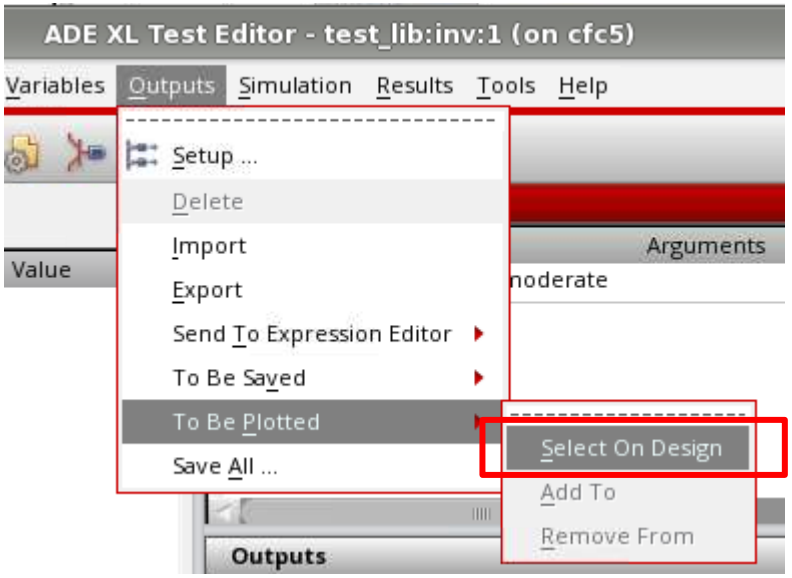
- Delete → Delete output variable



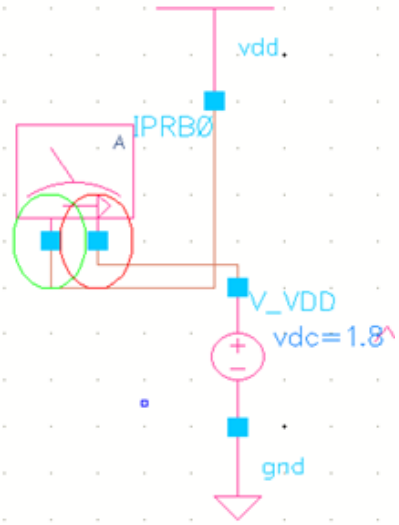
Appendix 2: Virtuoso Spectre Simulator (11/12)

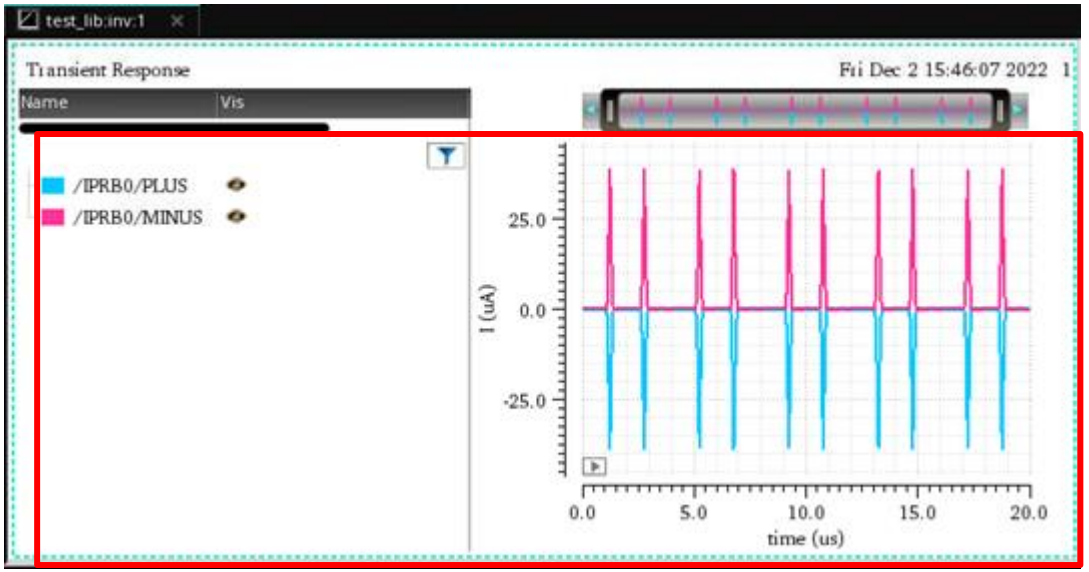
- Way 1 to get power: iprobe in analogLib





- 點iprobe component





Appendix 2: Virtuoso Spectre Simulator (12/12)

- Way 2 to get power: DC analysis and Annotate