

Concepts in VLSI Design Laboratory

IT126IU

Instructor: Nguyen Toan Van

Lab 2

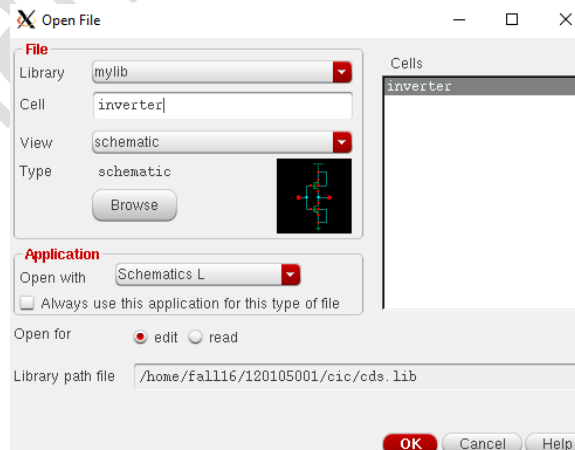
Parametric analysis of DC and transient simulation of an inverter and Symbol creation

Objectives:

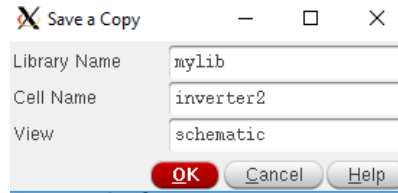
- To learn how to perform parametric simulation in ADE L
- To learn how to create symbol view from schematic view

Parametric Analysis in ADE L: DC simulation

1. To open the schematic of inverter, execute **File** → **Open** in **CIW**. In the 'Open File' window, select the inverter schematic from the list. Click **OK**.



2. Schematic editor window will open. Execute **File** → **Save a copy**. In the following window, change the name of the cell to *inverter2*. Click **OK**. Close the schematic editor window and open the *inverter2* cell from **CIW**.

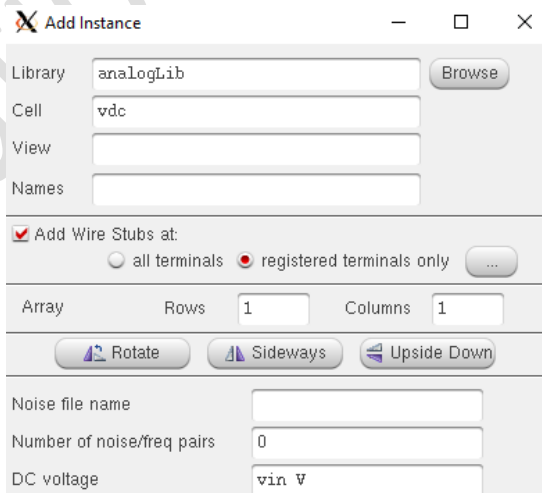


- This time we will perform both DC and parametric simulation at the same time on inverter schematic. We will obtain the transfer characteristics of inverter from DC simulation and by varying the width of the PMOS transistor; we will observe its effect on transfer characteristics.

Select the PMOS transistor in the schematic editor window, click 'q' and 'Edit object properties' window will open. Place **w** under 'Total Width' and press **tab** on keyboard. The 'Finger Width' field will be automatically changed as follows:

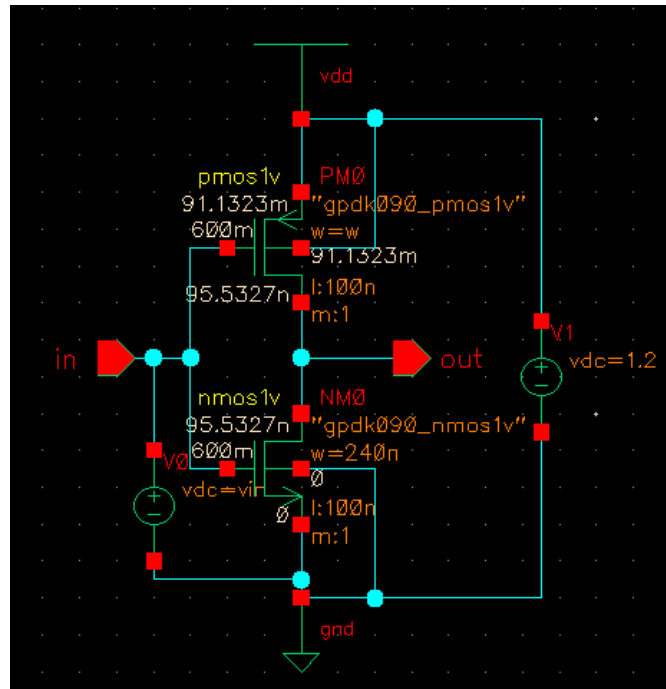
CDF Parameter	Value	Display
Model Name	gpdk090_pmos1v	
Multiplier	1	
Length	100n M	
Total Width	w M	
Finger Width	iPar("w") / iPar("fingers") M	

- Place symbol of an instance **vdc** from **analogLib** to the schematic. In the 'DC voltage' field, type **vin** and press **tab** on keyboard.



- Connect the voltage source between 'in' and 'gnd!'. Now place another instance of **vdc** on the schematic, in the 'DC voltage' field, put **1.2** and connect it between **vdd!** and **gnd!**

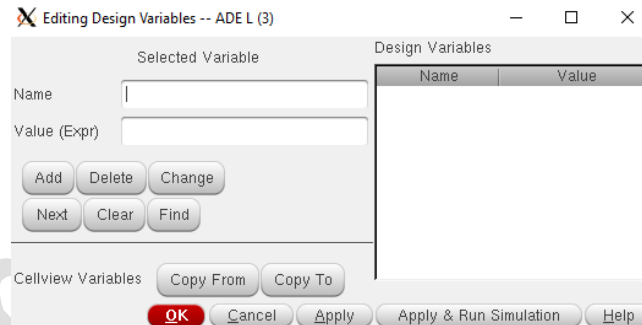
The final schematic should look like the following:



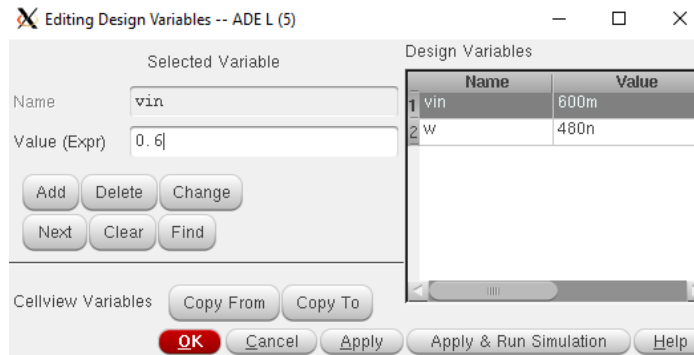
Click **Check and Save**.

6. Execute **Launch** -> **ADE L** and setup **Model Library** to **gpd090_mos.scs** and section to **TT_s1v**

7. Execute **Variables** -> **Edit** and 'Edit Design Variables' window will open.



8. Select '**Copy From**'. You will see '**w**' and '**vin**' appear in the '**Design Variables**' window. Click on '**w**' and in '**Value (Expr)**' field, put a default value of **480n**. Click **Apply**. Similarly click on '**vin**' and in '**Value (Expr)**' field, put a default value of **0.6**.



Then click **OK**.

9. Execute **Analyses** -> **Choose** and in the '**Choosing Analyses**' form, select **dc**. Click on '**save dc operating point**'.

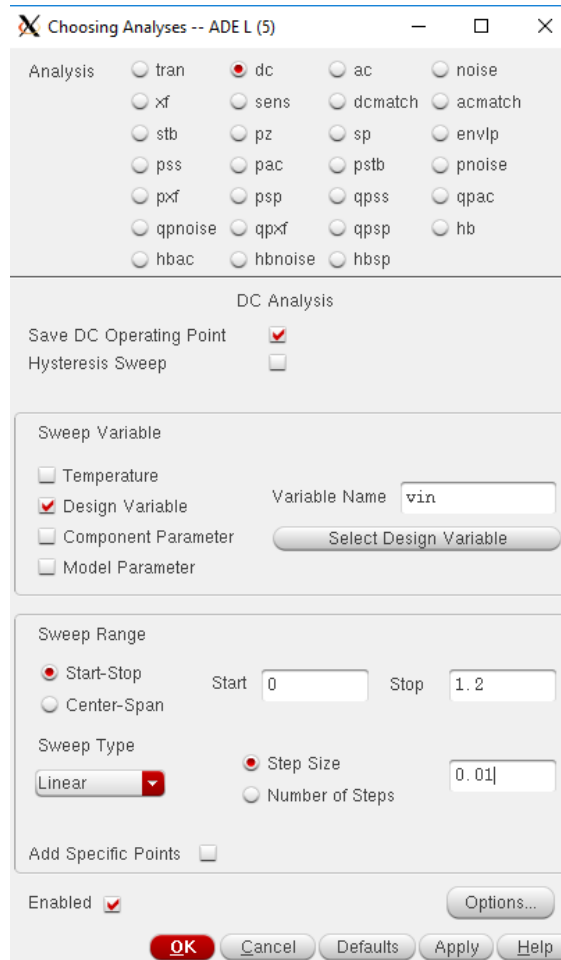
10. Under '**Sweep Variable**', select '**Design Variable**' and click on '**Select design variable**'.



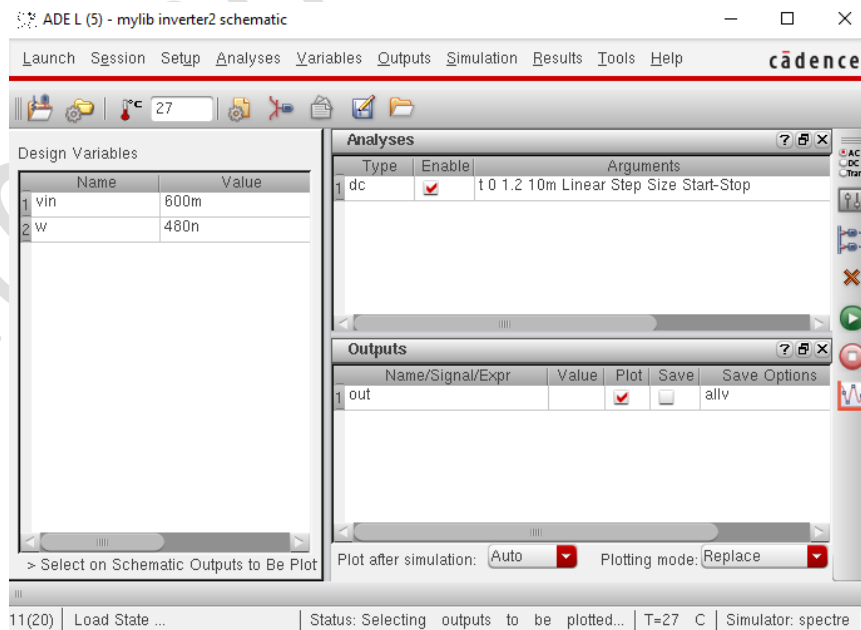
Select '**vin**' and click **OK**.

11. Under '**sweep range**', select '**start-stop**' and put a **start value** of **0** and a **stop value** of **1.2**.

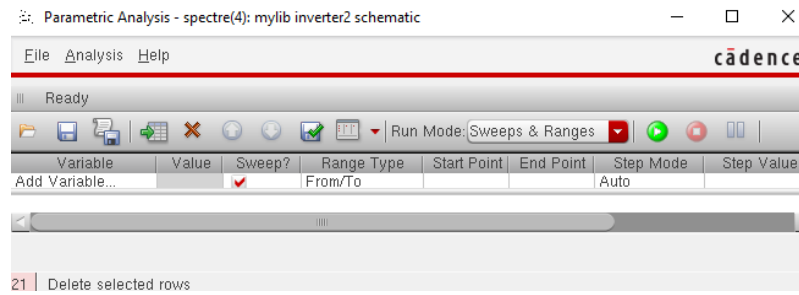
Select '**Sweep type**' to be '**linear**' and '**Step size**' to be 0.01. Click **OK**.



12. Execute **Outputs** -> **To be plotted** and select 'out' pin on the schematic.
ADE L window will now look like the following:



13. Execute **Tools -> Parametric Analysis**. 'Parametric Analysis' window will open.



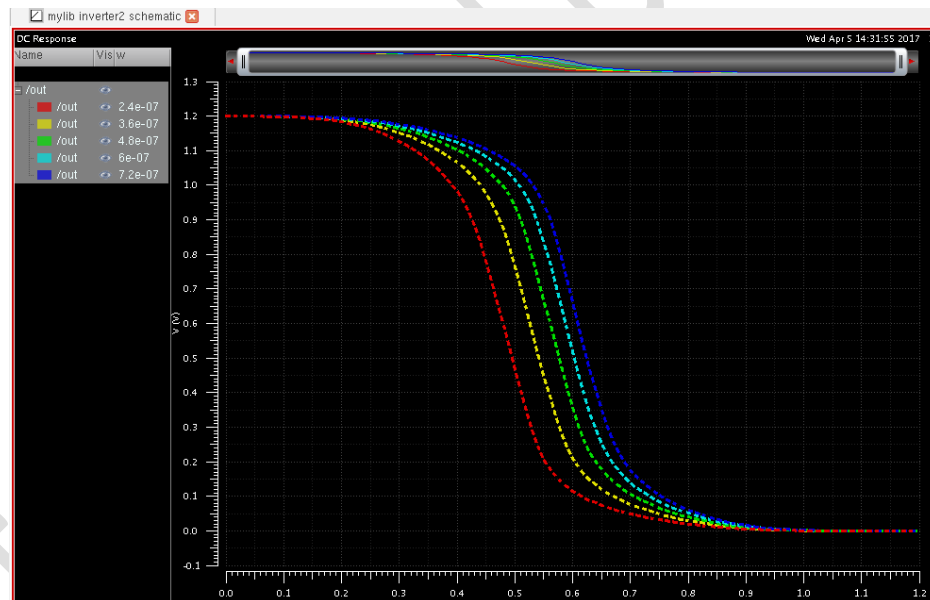
14. Click on 'Add Variable'. From the drop down menu, select 'w'. Put **From:** 240n , **To:** 720n , **Step Mode:** linear steps and **Step Size:** 120n.

Variable	Value	Sweep?	Range Type	From	To	Step Mode	Step Size
w	480n	<input checked="" type="checkbox"/>	From/To	240n	720n	Linear Steps	120n

Click on 'Run selected sweeps' icon to start simulation.



The simulated waveforms will be displayed.



These transfer characteristics can be further explored to find Noise margin, and inversion voltage for different widths of PMOS.

15. This parametric analysis option can be saved for later use by clicking on the save icon.



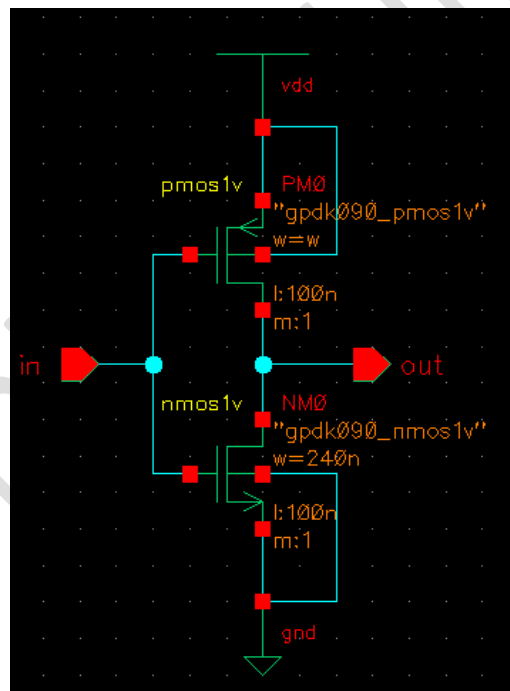
In the following window, give a name to the configuration file and click **Save**.



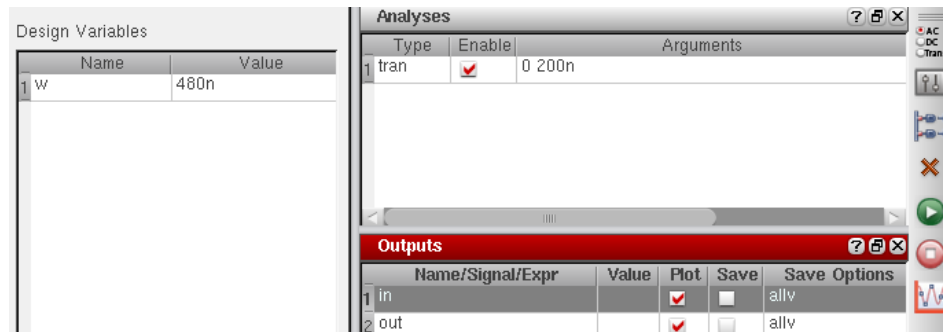
16. Also save the state of the ADE L window.

Parametric Analysis in ADE L: Transient simulation

1. For parametric analysis of transient simulation, open the schematic of the cell *inverter* from library *mylib* and save it as *inverter3*. Open the schematic of the cell named *inverter3*.
2. Change the 'Total Width' of PMOS to "w" as you have done before. Click **Check and save**.



3. Now, launch ADE L, and **Setup Model Library, Analysis type, Stimuli and Outputs to be plotted** in the same way as you have done in Lab 1.
4. Then execute **Variables -> Edit**. Select 'Copy From'. "w" will appear under 'Design Variables', select it and enter a default value of 480n under 'Value (Expr)' and click **OK**.

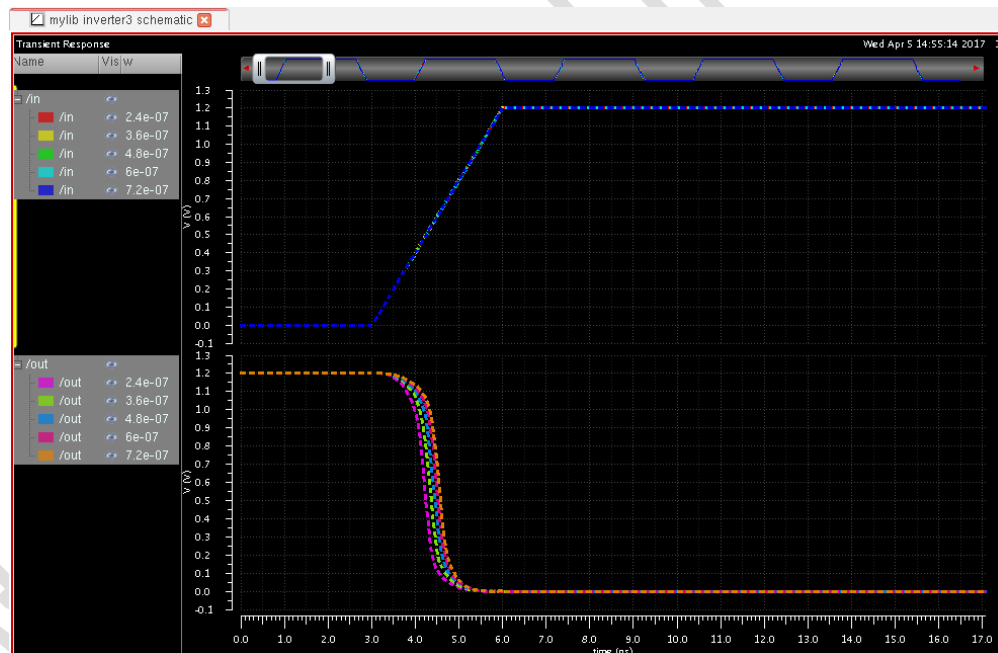


5. Now, execute **Tools -> Parametric Analysis**. Select 'w' from the 'Add variable' list, and fill in the rest as follows:

Variable	Value	Sweep?	Range Type	From	To	Step Mode	Step Size
w	480n	<input checked="" type="checkbox"/>	From/To	240n	720n	Linear Steps	120n

Click on 'Run selected sweeps'.

6. From the waveform display window, change the time scale and observe a specific transition from high to low or low to high.



You can clearly see the effect of changing width on the inverter performance.

7. Let's find the delays for this transition. Open **Waveform calculator** and fill in the form as follows for calculating the delay for this transition.

delay

Signal1: VT("/in")

Signal2: VT("/out")

Threshold Value 1: 0.6

Threshold Value 2: 0.6

Edge Number 1: 1

Edge Number 2: 1

Edge Type 1: rising

Edge Type 2: falling

Click **OK** and click on 'Evaluate the buffer and display the results in a table' icon.



The results will be displayed in a table:

	w	delay(?wf1 VT("/in"), ?value1...)
1	240.0E-9	-292.1E-12
2	360.0E-9	-174.5E-12
3	480.0E-9	-87.91E-12
4	600.0E-9	-19.98E-12
5	720.0E-9	35.04E-12

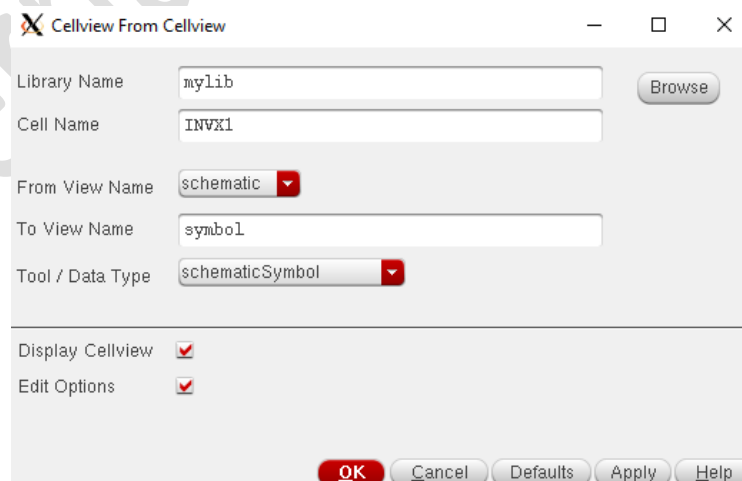
You can see that some of the propagation delays are negative. Can you explain why? Also notice that propagation delay increases with the width of PMOS transistor for a falling transition in inverter output. Can you explain why? What will happen during a rising transition?

Symbol Creation

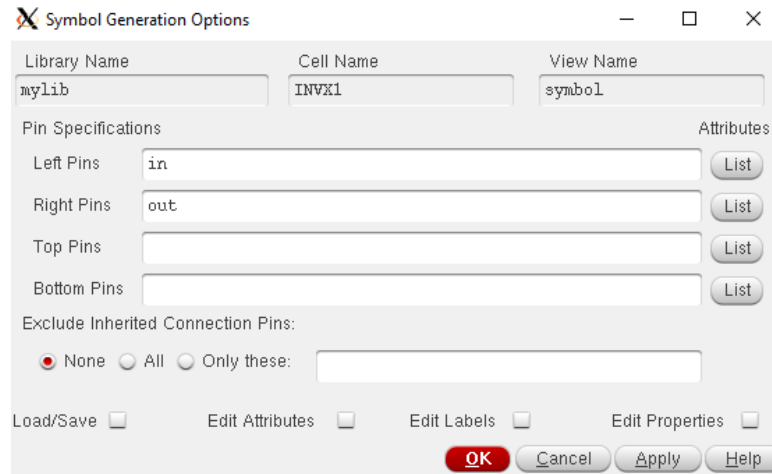
In this section, you will create a symbol for your inverter design so that you can use this symbol view for the schematic in a hierarchical design. In addition, the symbol has attached properties (cdsParam) that facilitate the simulation and the design of the circuit.

1. Open the schematic of the cell *inverter* from library *mylib*. Save a copy named **INVX1**. Change the **width** of the PMOS transistor to 240n.
2. In the schematic editor window for **INVX1**, execute **Create -> Cellview -> From Cellview**.

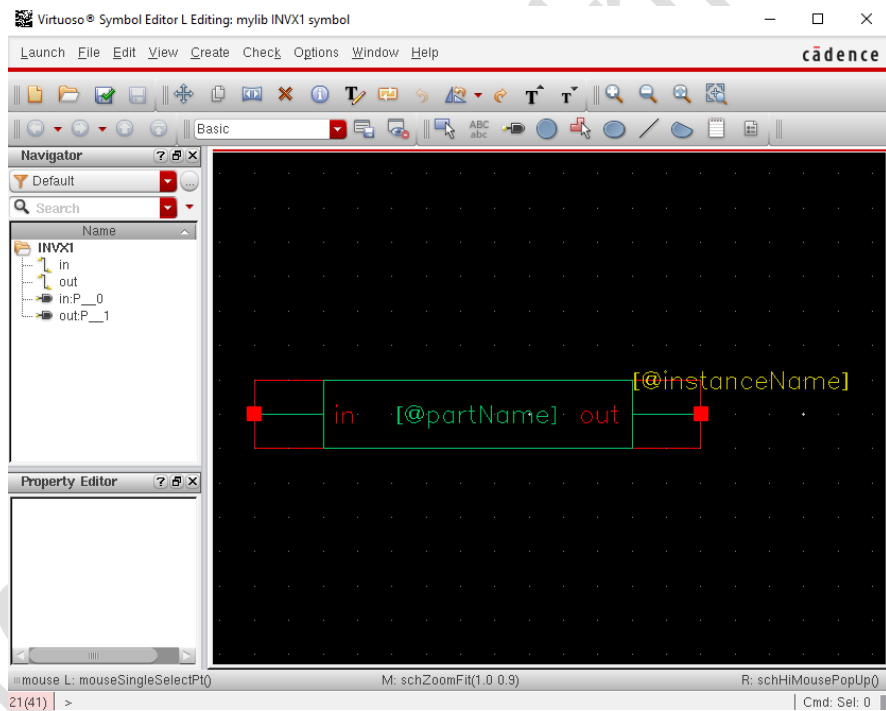
'Cellview from cellview' window appears. Click **OK**.



In the 'Symbol Generation Options' window, you can choose the location of the pins.



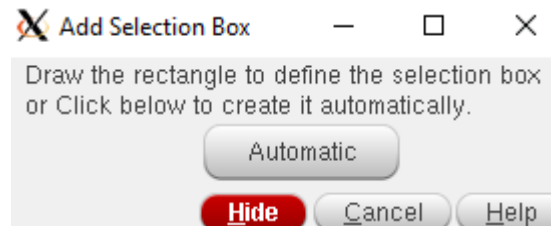
Click **OK**. Symbol Editor window will open.



3. Click **Delete** icon in the symbol window, delete the outer red rectangle, green rectangle and *instanceName* property.

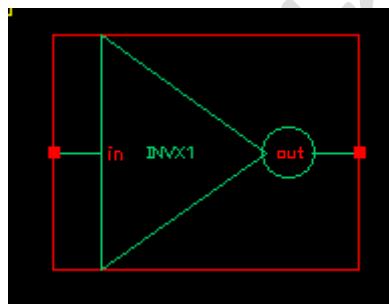


4. Execute **Create** -> **Shape** -> **polygon**, and draw a shape similar to triangle. After creating the triangle, press **Esc** key.
5. Execute **Create** -> **Shape** -> **Circle** to make a circle at the end of the triangle. You can move the pin names according to the location.
6. Execute **Create** -> **Selection Box**. In the 'Add Selection Box' form, click '**Automatic**'.



A new red selection box is automatically added.

7. After creating symbol, click on the save icon in the symbol editor window to save the symbol. In the symbol editor window, execute **File** -> **Check and Save**. Then close the symbol editor window.



QUESTIONS:

What is the effect of load capacitance on delay and power consumption? What will happen to delay and power consumption if sizes of transistors are increased? What is the effect of transistor sizes on noise margin level? What is the significance of inversion voltage? What is the effect of process corners on delay and power consumption?

Shortcut key	Tasks performed
w	Add a wire
i	Add an instance
p	Add a pin
l	Add label to a wire
e	Display options
q	Select an object and press q to open 'Edit Object Property' dialogue box
[Zoom out
]	Zoom in
c	Copy
m	Move
u	Undo
Shift+u	Redo
f	Fit the entire schematic in the window

Cadence Virtuoso® Schematic Editor L Shortcuts