

EEE-4706 VLSI Circuit Sessional

LAB 1:

**Design of Basic Logic Gates in Cadence Virtuoso Schematic Editor
and Performing Transient Simulation with Spectre**

Objectives:

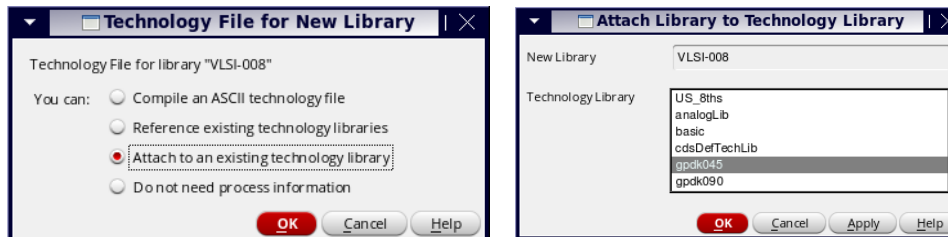
- Getting started with Cadence Virtuoso software and creating a library.
- Draw the schematic of basic logic gates in Cadence Virtuoso Schematic Editor.
- Perform Transient Simulation of logic gates using MMSIM spectre.
- To learn the effect of different process corners.
- Introducing variables and their use in design.

1.0 Getting Started

1.1 Creating a Library

First of all, we will add the generalized 45nm process design kit from cadence known as **gpdk045** to our library path as follows.

1. In the CDS.log file go to **Tools>Library Manager>File>New>Library**.
2. Put the name as **VLSI_xxx**, where xxx is the last 3 digits of your student ID and press ok.



3. Select **Attach to an existing technology library** and click OK.
4. Select **gpdk045** and click OK.
5. In the Library Manager window verify that **VLSI_xxx** library is listed.
6. Click on that library (**VLSI_xxx**) and again go to **File>New>Cell View**.

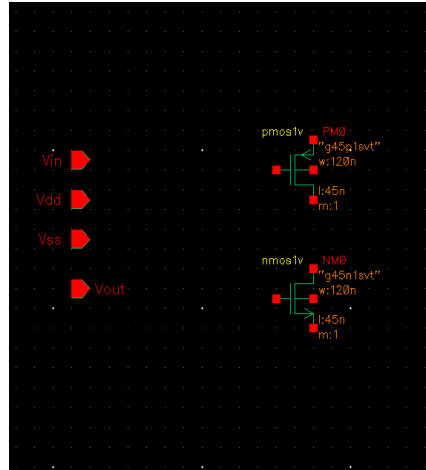
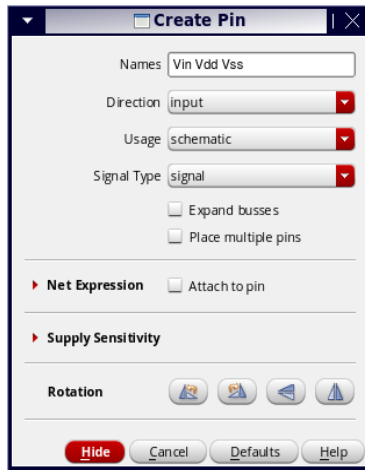
We are going to name the cell as the logic gate's name, e.g., inverter, NAND, NOR and so on. For the first time, type **inverter** and press OK.

1.2 Drawing Schematic

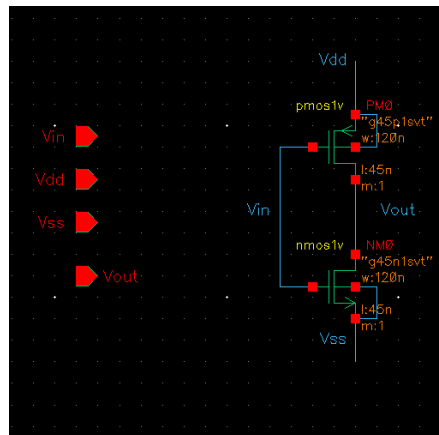
A blank schematic window for the inverter will be open. Go to **Create>Instance** (or use blind-key "I") to add PMOS and NMOS to the schematic. We recommend you learn the necessary blind-keys for Cadence; Lists of necessary blind-keys are provided at the end of this supplement.

1. For PMOS: Library→gpdk045, Cell→pmos1v, Total width→120n (Don't change anything else).
2. Click on **Hide** or press the **Esc** button on the keyboard and place it on the schematic.
3. For NMOS: Library→gpdk045, Cell→nmos1v, Total width→120n (Don't change anything else).

- Repeat task 2 and place it a little down to the PMOS.
- Now, we are going to enter input and output pins. To do that, go to **Create>Pin** (blind-key “P”).
- All 3 input pins can be added together by using a <space> between them. In the name option, type **Vin Vdd Vss** and **Direction** → **input**. Click Hide or press the **Esc** button. Place them on the left side of the schematic.
- Again, go to the pin option and type **Vout** in the name option. This time, **Direction** → **output**.



- Now, we are going to make a wire connection. Go to **Create>Wire (narrow)** (blind-key “W”) and connect the necessary diagram.
- To keep the schematic neat and clear view we will not use wire to connect the pin and circuit, while we are going to use **Label** to connect them virtually.
- Go to **Create>Wire Name** (blind-key “L”) and type all necessary labels using <space> between them e.g., **Vin Vdd Vss Vout** and place them in the schematic. Click on **check and save** (blind-key **Shift+X**).



1.3 Creating Symbol

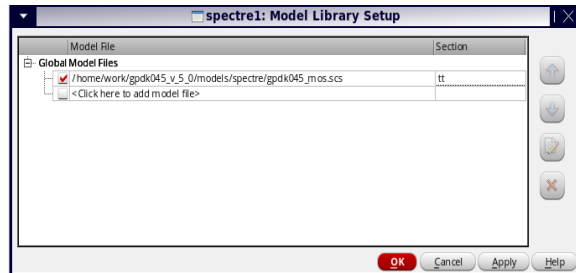
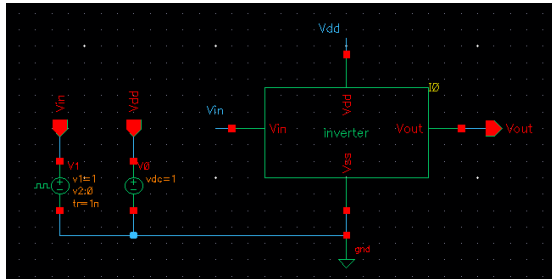
- Go to **Create>Cellview>From Cellview**. **Library Name** → **VLSI_xxx**, **Cell Name** → **inverter**. Click OK.
- Place **Left pins** → **Vin**, **Right pins** → **Vout**, **Top pins** → **Vdd**, **Bottom pins** → **Vss**. Click OK.
- Now check and save the symbol. (The design of the symbol can be customized, in this course, we're not going to do that)

1.4 Transient Simulation

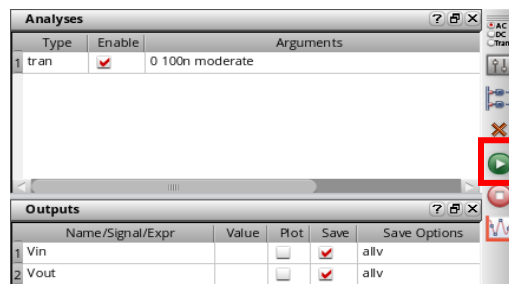
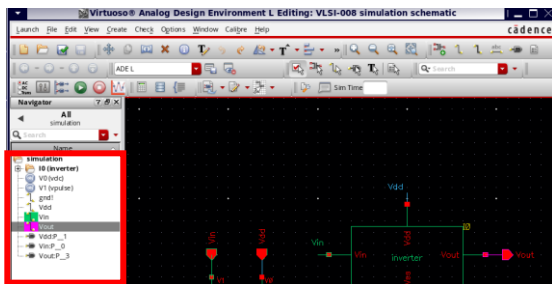
We are going to use another cell, named **simulation** and we're going to use that cell to simulate all the schematic by using symbols.

- Go to **Library Manager** and create another cell named **simulation** under VLSI-xxx library.
- Go to **Create>Instance** (blind-key I), **Library** → **VLSI_xxx**, **Cell** → **inverter**. Place it in the schematic.

3. Add input pins- Vin and Vdd (don't add Vss); add output pin- Vout.
4. Now, go to **Instance** and add **Library** → **analogLib**, **Cell** → **vdc**, **DC Voltage** → **1**. Place it below the Vdd pin.
5. Again, add **Instance** as **Library** → **analogLib**, **Cell** → **vpulse**, **Voltage 1** → **1**, **Voltage 2** → **0**, **Period** → **42n**, **Rise time** → **1n**, **Fall time** → **1n**, **Pulse width** → **20n**. Press the Esc button and place it below Vin's pin.
6. Again, add **Instance** as **Library** → **analogLib**, **Cell** → **gnd**. Place it below the Vss node of the inverter.
7. Now, take the wire (blind-key **w**), connect pins to the below instances and connect the output to the output pin. The diagram should be like below.

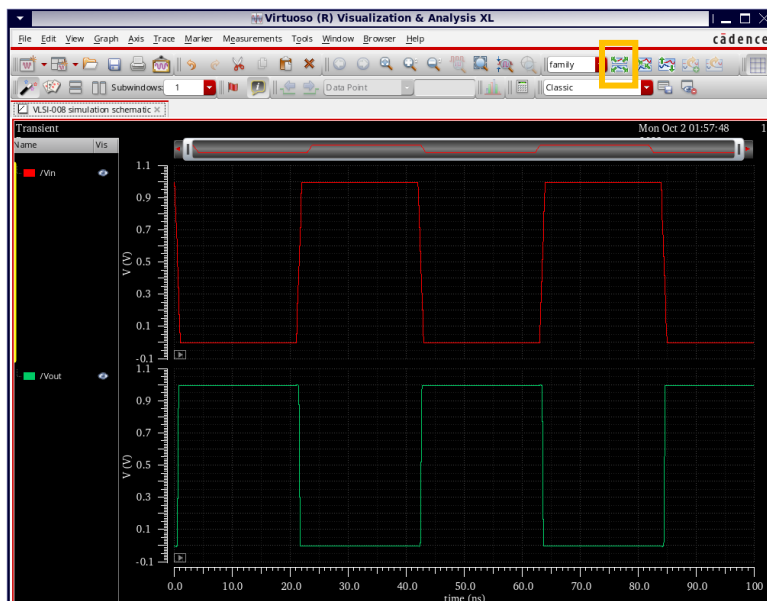


8. Click on **check and save** (blind-key **Shift+X**). Go to **Launch>ADE L**. An ADE L window will appear.
9. Go to **Setup>Model Libraries**. Select model file- **gpdk045_mos.scs** and select section- **tt**. Click OK.
10. Now, go to **Analyses>Choose**. Click **trans** and set **stop time- 100n**. Mark **moderate** and click OK.
11. Go to **Output>To Be Plotted>Select On Design** and from the left side **Navigator**, select **Vin** and **Vout**.



Run

12. Go back to ADE L using the taskbar and **Run** the simulation.
13. A timing diagram will appear. Use **Split All Strip** to separate each timing diagram.



- Now repeat task 9, change **tt** and replace it with **ff** or **ss**. Simulate again to check the best-case and worst-case scenarios. This is known as the process corner.

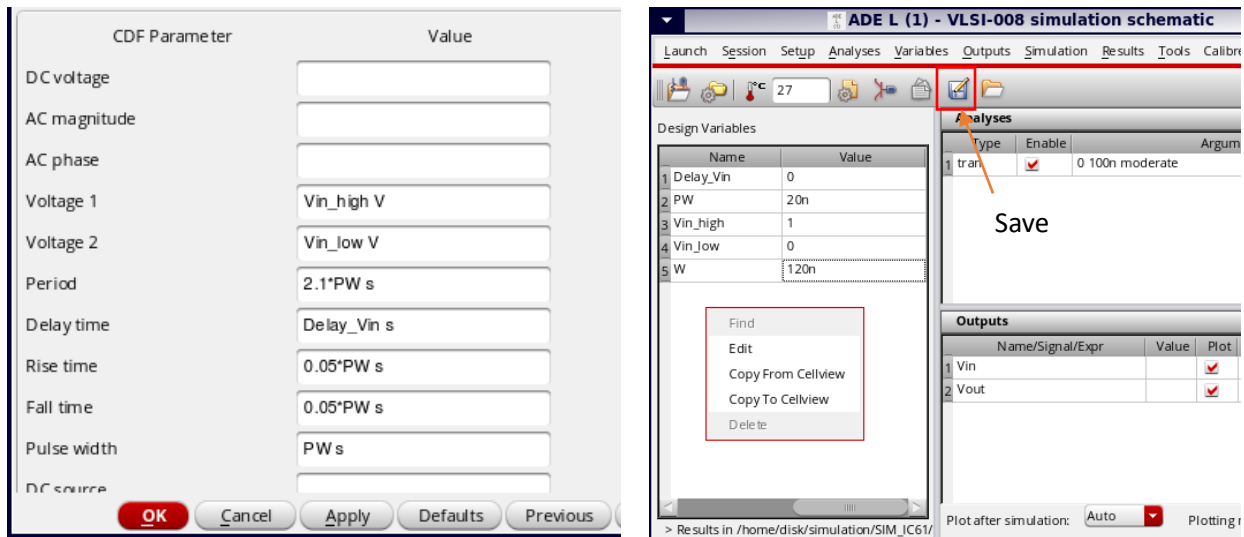
1.5 Use of Variables

Variables are generally used when a parameter needs to be changed frequently to observe different results. Variables in Cadence design can be controlled in the ADE L window; hence we don't need to change the value in schematic design all the time, which saves a lot of time in simulating complex projects.

- Go to the **Library Manager** window again using the taskbar. Open the inverter schematic and click on the PMOS. Right-click on it and go to the property (blind-key **Q**). Change the **Total Width** from 120n to any variable (the suggested variable name is- **W**, as it represents width). Now click OK. **Check and Save** the file.
- Now, the schematic has changed, but the information carried by the symbol has not been updated. Hence, go to **Create>Cellview>From Cellview** and click OK. Then, click on **Modify**.
- Again, we're going to add variables in the **simulation** schematic. Open the schematic of the simulation from the taskbar or Library Manager. Click on the **vpulse** below the Vin pin and open the property (blind-key **Q**). Now add variables as follows:

Voltage 1 → **Vin_high**, **Voltage 2** → **Vin_low**, **Period** → **2.1*PW**, **Delay time** → **Delay_Vin**, **Rise time** → **0.05*PW**, **Fall time** → **0.05*PW**, **Pulse width** → **PW**. Click OK, then **Check and Save**.

(Here, PW represents Pulse width and generally the rise time and fall time are 5% of pulse width, which makes the total period 2.1 times of the pulse width)



- Now go to the **ADE L** window, in the left blank portion of the window click the right button of your mouse and click on **Copy From Cellview**. All the variables will be enlisted here. Now, provide value as below- **Delay_Vin** → **0**, **PW** → **20n**, **Vin_high** → **1**, **Vin_low** → **0**, **W** → **120n**.
- Run the simulation and the result will be the same as we implemented the same value. In this way, you can add and control the variable of any value from the ADE L window.
- Before closing the software, open CDS.log window then go to **File>close data** and save all.

1.6 Necessary Blind Keys (for schematic)

C → Copy	I → Instance	L → Label	O → Display option	P → Pin
Q → Property	R → Rotate	U → Undo	W → Wire	Z → Zoom in
Ctrl+A → Select all	Ctrl+E → Return	Ctrl+Z → Zoom out	Shift+M → Move	Shift+S → Save
Shift+X → Check&Save		Shift+delete → Inactive instance		delete → delete

1.7 Report