Department of Electrical & Electronic Engineering

Dhaka University of Engineering & Technology (DUET)

EEE-4706 VLSI Circuit Sessional

LAB 2:

DC Analysis along with Power, Delay and Energy Analysis of CMOS Inverter Circuit

---

Objectives:

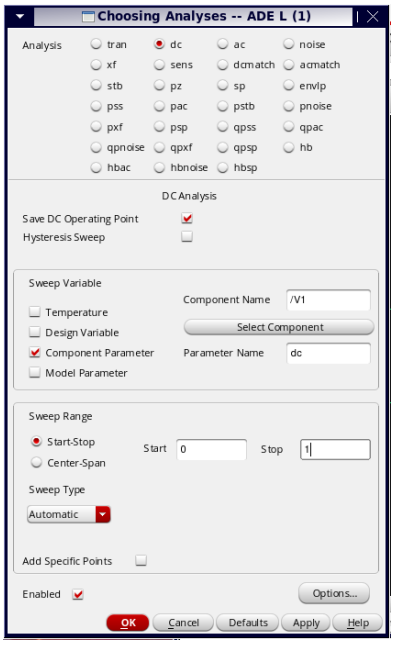
- Perform DC analysis of CMOS inverter circuit.
- Plotting transient and DC power consumption curves.
- To get familiar with the calculator tools of Cadence.
- Determining average transient power, static power and energy consumption.
- To determine propagation delay in different ways.

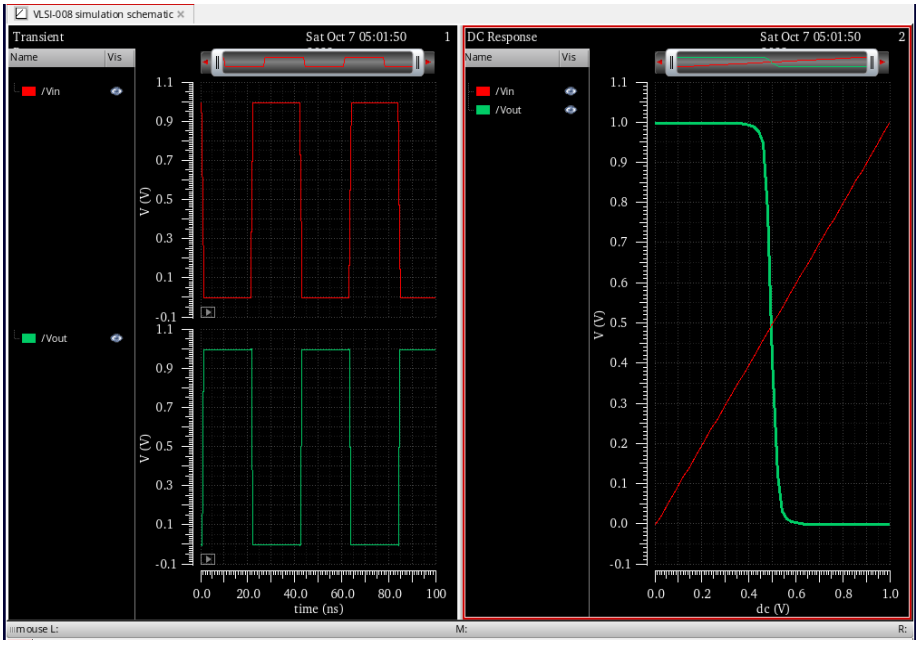
2.0 Open Saved File

1. Launch **virtuoso** and open **Library Manager** from the **CDS.log** window.
2. Click on **VLSI_xxx** in the library section and then double-click on **Simulation** in the Cell section.
3. Check and save. Go to ***Launch>ADE L>Session>Load State*** and select **state_inverter**, which you saved in the last class. Click OK.

2.1 DC Analysis

1. Go to ***Analyses>Choose*** and click on **DC**. Enable **Save DC Operating Point** and then tick on **Component Parameter.** Click on **Select Component** and it will open the simulation schematic window.
2. Select the **Input voltage source (vpulse)** and a **Select Parameter** window will appear. Select **DC Voltage** and set Sweep Range value, **Start→0** and **Stop→1.** Press ok.



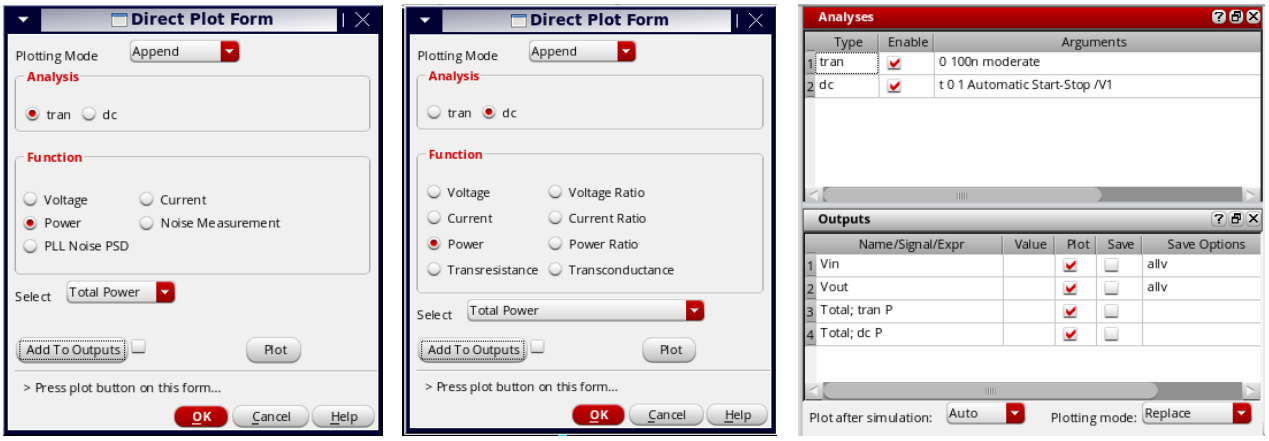


3. Now, Run the simulation and you will see both the Transient and DC response results.

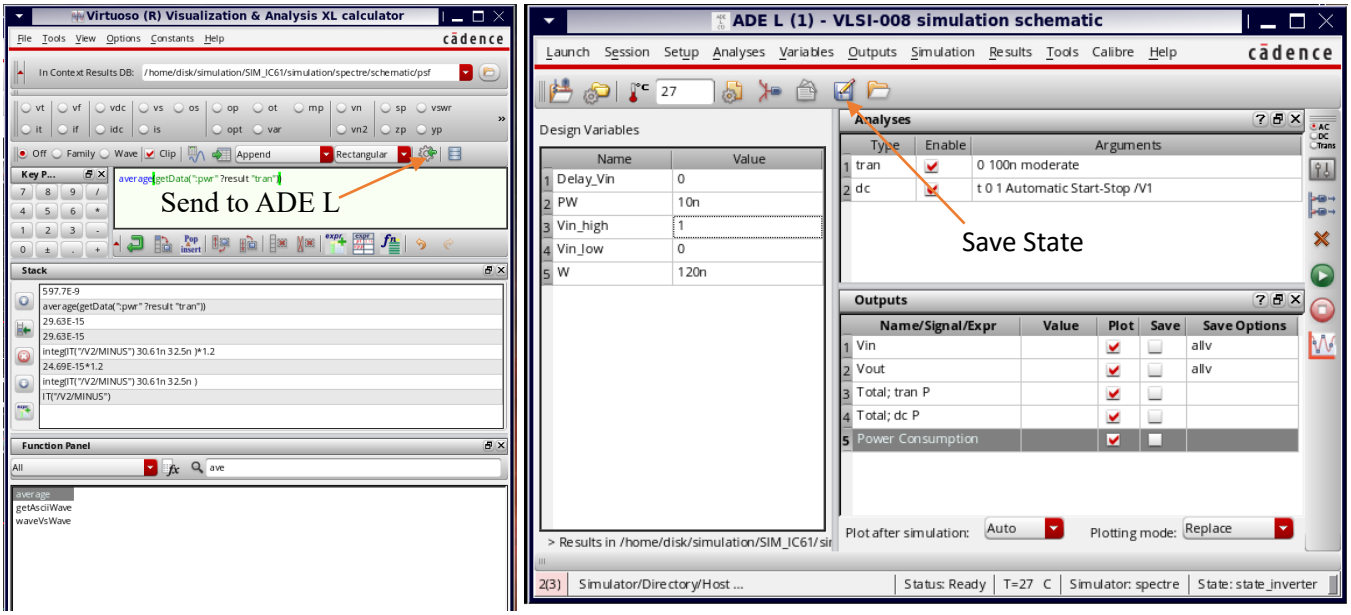
2.2 Power Analysis

Every time we simulate in Cadence, it saves some specific data for the simulation results. In case of power consumption of a circuit, this has to be saved before running the simulation.

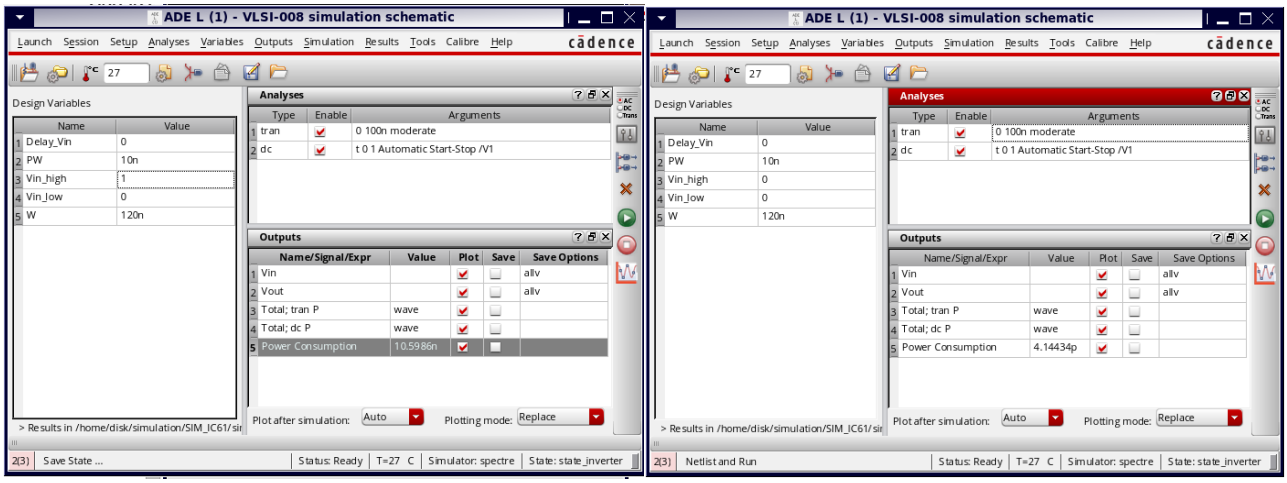
1. Open the ADE L window and go to ***Outputs>Save All*** and in the section of **Select power signal to outputs (pwr)**, enable **all** and press ok. Now rerun the simulation.
2. Again, open the ADE L window and go to ***Results>Direct Plot>Main Form***. A Direct plot form window will appear.
3. For transient power consumption, **Analysis →tran, Function →Power, Select →Total Power** and Click on **Add to Output.** (Don't click OK or don't close the window)
4. For DC power consumption, in the same window, click on **Analysis →DC, Function →Power, Select →Total Power** and Click on **Add to Output.** Now, press OK.



5. The ADE L window will add **Total; tran P** and **Total; dc p** in the output section. Now, Run the simulation.
6. Transient and DC power consumption waveshape will appear. Click on the waveshape of transient power consumption. Right-click on the mouse and go to ***Send to>Calculator***. A calculator window will appear.
7. In the **Functional Panel** search box, type "average". Click on "average" from the below box and then send it to the ADE L window.
8. Get back to the ADE L window and rename the formula as **Power Consumption** as shown figure. Save the state and overwrite it.



9. Run the simulation and the power consumption that is displayed in the ADE L window, is the **Average Transient power consumption.**
10. Now, change the **Vin_high>0** and again run the simulation. The power consumption is now showing the **Static power consumption** of the circuit.

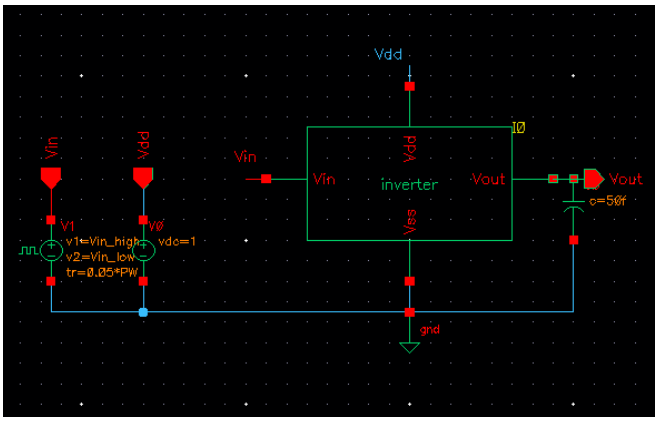


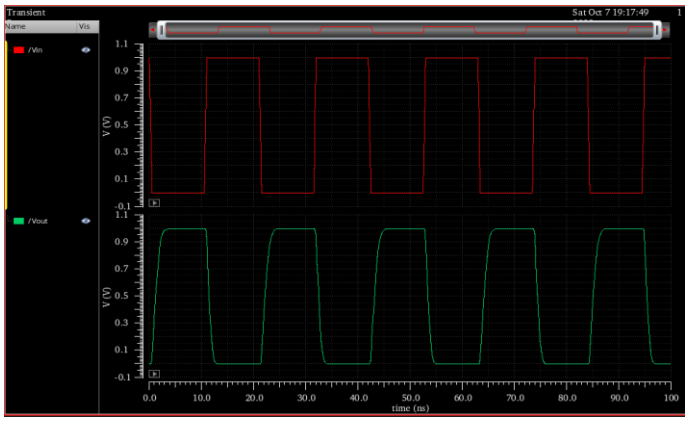
Remember, Transient power consumption is always higher than Static power consumption. On the other hand, Static power consumption and DC power consumption are not the same. Static power consumption happens due to the leakage current when the input pulses are in a static state or mostly in a zero state.

2.3 Delay Analysis

A small circuit like an inverter, generally has insignificant delay or sometimes negative delay for default capacitance in Cadence schematic design. After the actual implementation of the layout, the real delay can be found. But, in this experiment, we're going to learn how to determine delay. Hence, we're going to add a capacitor to the output of the simulation schematic, to create significant delay and to observe it clearly.

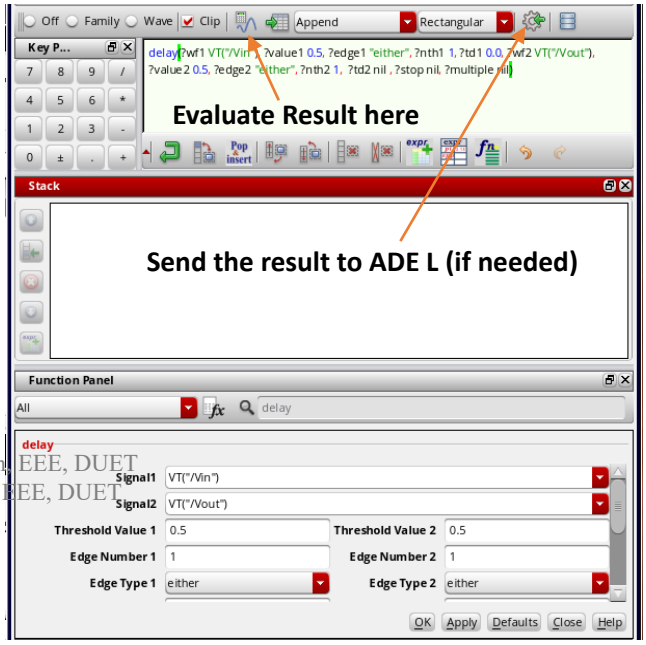
1. Go to the **simulation** schematic and add a "**cap**" instance to the output. Set the value of the **Capacitance>50f**. Connect wire as shown in the figure. Now, check and save. Go to the ADE L window, unmark **dc** analyses, **Total; tran P** and **Total; dc P.** Run the simulation. (**Vin_high should be "1")

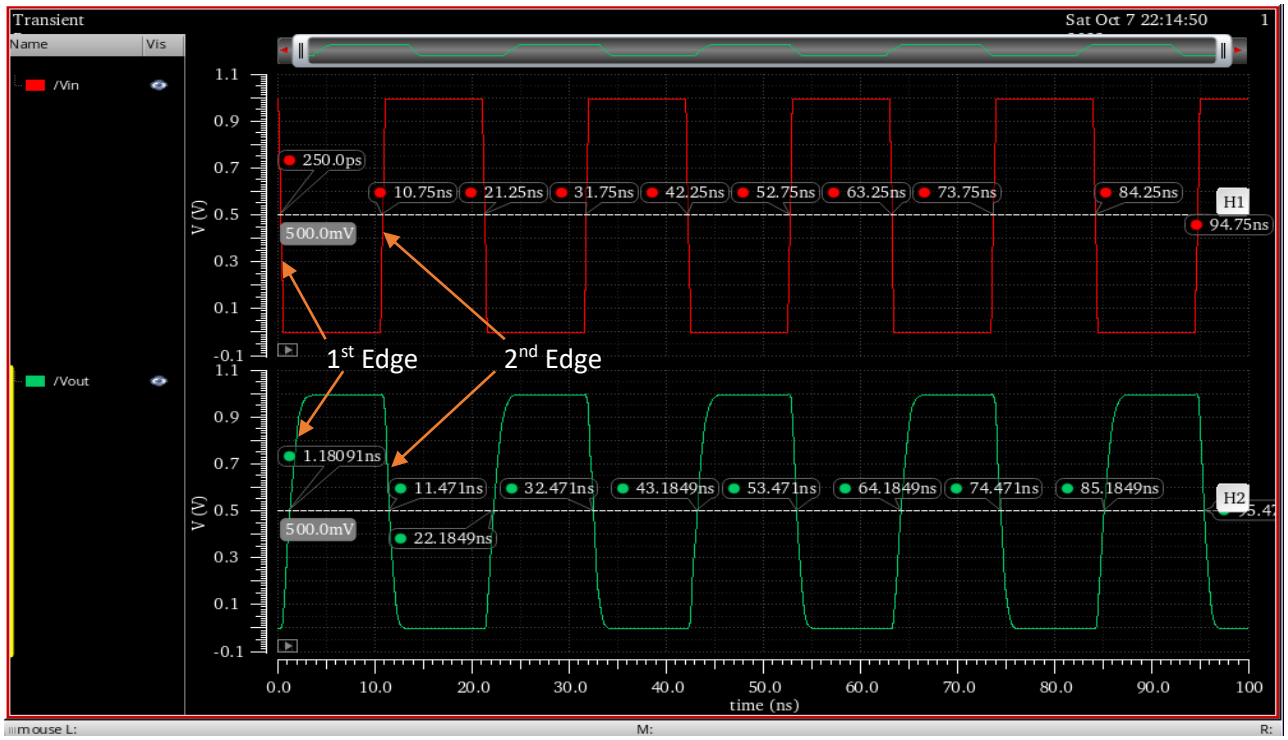




2. In the visualization window, go to ***tools>calculator***. In the functional panel, search for **delay** and click on it.
3. Provide the value as **Signal1 →VT("/Vin"), Signal2 →VT("/Vout"), Threshold Value 1 →0.5, Threshold Value 2 →0.5, Edge Number 1 →1, Edge Number 2 →1, Edge Type →either, Edge Type →either.** (Every value has significance here). Click **Apply**. Now, **Evaluate** the result.
4. In the same manner, change the **Edge Number** of each to **2** and evaluate the result again.
5. Delay found for edge 1 is **930.9e-12** (which is t_{pLH}) and for edge 2 is **721.0e-12** (which is t_{pHL}).

6. In that case, we have used **Edge Type** as "either", which counts both "rising" and "falling" edges. It can be done by using rising or falling edge types also.
7. Delay can be determined manually using a **Horizontal Marker** in the waveform window.
8. In the visualization window, type **H** on both Vin and Vout waveform (keep cursor on the waveform, then type on keyboard). Select the marker line and type **Q** to get the properties. Change the **Y Position>0.5, Intercepts>on**. Press OK. Do it for both lines.
9. Now, applying (Vout – Vin) timing in 0.5V, we get-
   For 1st edge, t_{pLH} = (1180.91 – 250) ps = **930.9** ps
   For 2nd edge, t_{pHL} = (11471 – 1075) ps = **721.0** ps
   Which are same results as found using calculator.



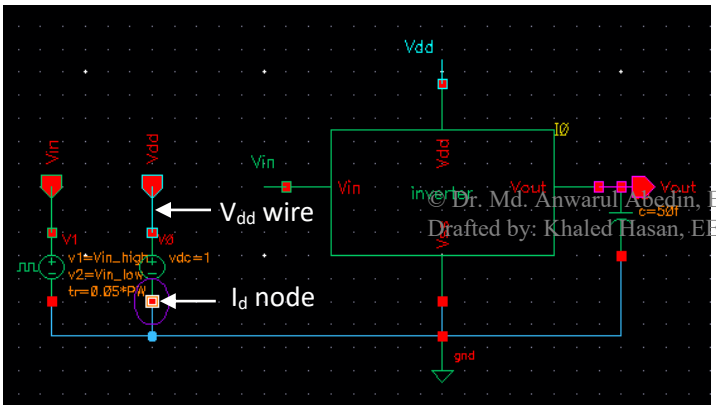


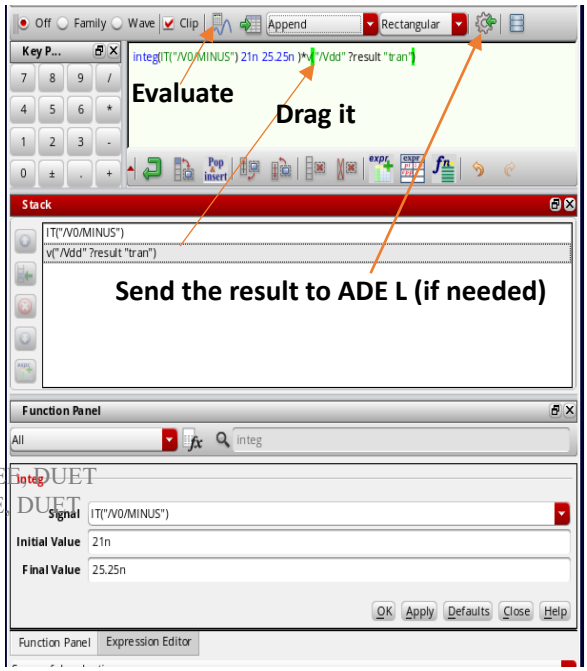
2.4 Energy Analysis

Energy consumption from power supply for a certain time can be calculated by integrating the transient current during that time and multiplying it with supply voltage. In order to include things from output waveform in the calculation, we need to add these two waveforms.

1. Go to **ADE L** window, then ***Output>To be plotted>Select on design*** and select the **wire** and **node** of V_{dd} and I_D respectively as shown in the figure below. Open ADE L window and **run** the simulation (Keep **Total; tran P** and **Total; dc P** unchecked). Split all strips.
2. A **/Vdd** and **/V0/MINUS** waveform is added that represents the voltage and current. Click on the Vdd waveshape and right click on mouse. Go to ***Send to>Calculator*** and press **Enter**. Check if it's saved in the **Stack** terminal below.
3. Again, go to waveform window, click on the current waveshape and send it to the calculator.

4. Now, in the **Function Panel**, search for **integ** and click on it (not the iinteg). Set the range of initial value and final value of a single transient current wave. Click on **apply**. Then fulfill the formula as shown in the figure or given below.
integ(IT("/V0/MINUS") 21n 25.25n)*v("/Vdd" ?result "tran")





5. Now, **Evaluate** the equation and it will create a waveshape. In the waveshape window, **Split the strips** and you will see a straight line representing the value of energy. Now, it's just a straight line! And we have to find the value by putting cursor on the waveshape or using the marker. So, we're going to plot the direct value using the calculator.
6. Go to calculator again. The previous formula should be in place. In the **Functional Panel** Select **All** from the dropdown menu and then search for **ymax** or **average** (as it is a straight line) and select it. Now the formula should be like - ymax(integ(IT("/V0/MINUS") 21n 25.25n)*v("/Vdd" ?result "tran"))
Evaluate the result and in that manner, you can directly obtain the value. (50.2fJ)
7. Before closing the software, open CDS.log window then go to ***File>close data*** and save all.

Report: