Department of Electrical & Electronic Engineering

Dhaka University of Engineering & Technology (DUET)

EEE-4706 VLSI Circuit Sessional

LAB 3:

Parametric Analysis of a Full Adder Circuit

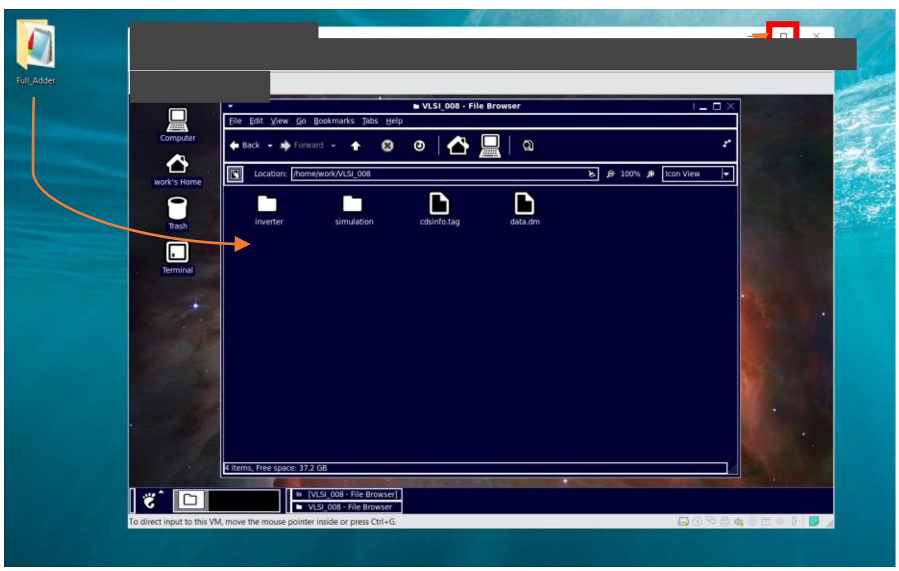
Objectives:

- Performing transient and DC analysis of a full adder circuit.
- Observing the effect of channel width of PMOS in terms of DC response and power consumption.
- Observing the effect of supply voltage on transient power consumption and delay.
- Determining the transient power consumption for all possible input pulses.
- Understanding the actual use of parametric analysis.

3.0 Setup File

You will be given a file, namely **Full_Adder**. You've to put the file in your work directory to work on it. Following that way of transferring files, you can work on any circuit that is designed with the same technology.

1. Go to **work's Home** and open the folder **VLSI_XXX**. Now, **restore** the VMware and drag the **Full_Adder** folder to the **VLSI_XXX** folder.



2. Launch **Virtuoso** and open **Library Manager** from the **CDS.log** window.
3. Click on VLSI_XXX in the library section and then double-click **Full_Adder** in the cell section.

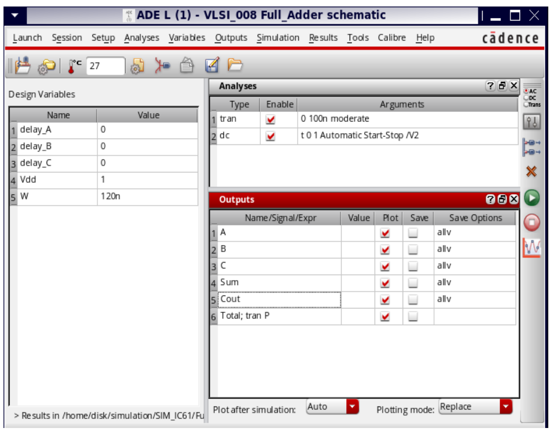
It will open the full adder schematic. Take time to understand the construction and the procedure of its signal flow through the circuit. (Hints: Low gate voltage activates PMOS and passes V_{dd}; while high gate voltage activates NMOS and passes V_{ss} in CMOS circuits). Check the properties of A, B, C, and Vdd input signals. For a clear view of the circuit, you can go to ***view>hide instance labels***.

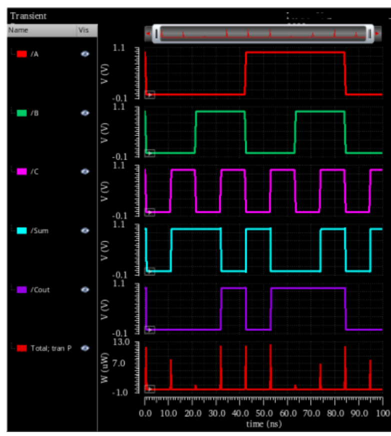
3.1 Transient and DC analysis

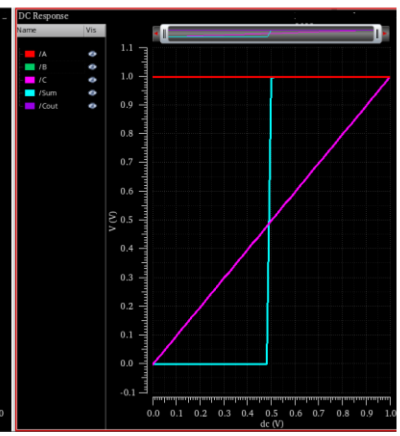
Before doing parametric analysis, we need to store all the necessary data in the ADE L window.

1. Click on **check and save** (blind-key **Shift+X**). Go to ***Launch>ADE L***. An ADE L window will appear.
2. Go to ***Setup>Model Libraries***. Select model file- **gpdk045_mos.scs** and select section- **tt.** Click OK.
3. Now, go to ***Analyses>Choose***. Click **trans** and set **stop time- 100n**. Mark **moderate** and click **Apply**.

1. Now, 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 **C** 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.
4. In the left blank portion of the ADE L 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_A →0, Delay_B →0, Delay_C →0, Vdd →1, W →120n.
5. Go to ***Output>To Be Plotted>Select On Design*** and from the left side **Navigator,** select **A, B, C, Sum, Cout**. Or, you can select them by clicking on the pin in the schematic.
6. 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, run the simulation.
7. Again, open the ADE L window and go to **Results>Direct Plot>Main Form**. A **Direct plot** form window will appear.
8. For transient power consumption, **Analysis →tran, Function →Power, Select →Total Power** and enable **Add to Output**. Click on the **plot**. In the transient waveform, split all strips.





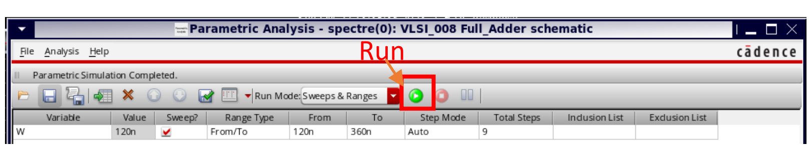


9. Click on the transient power waveform and right-click on the mouse. Now, ***Send to>calculator*** and from the functional panel select **average**. Send the buffer expression to the ADE L window.
10. Now, from the waveshape window, select the **C** input waveshape and send it to calculator. Similarly, send the waveshape of **Sum** input also. In the calculator window, select **delay** in the **functional Panel.** Set the data as- **Signal1 → v("/C" ?result "tran")** (using dropdown), **Signal2 → v("/Sum" ?result "tran")** (using dropdown), **Threshold Value 1 → 0.5*ymax(v("/C" ?result "tran"))** (use copy paste), **Threshold Value 2 → 0.5*ymax(v("/C" ?result "tran")), Edge Number 1 →1, Edge Number 2 →1, Edge Type →either**, **Edge Type →either**. Select apply and send the buffer expression to the ADE L window.
11. Return to the ADE L window and rename the newly added formula as **Avg. transient power** and **Delay_edge1** respectively. Now, go to ***tools>parametric analysis.***

3.2 Parametric Analysis

In this analysis, you will observe another importance of adding variables in the schematic.

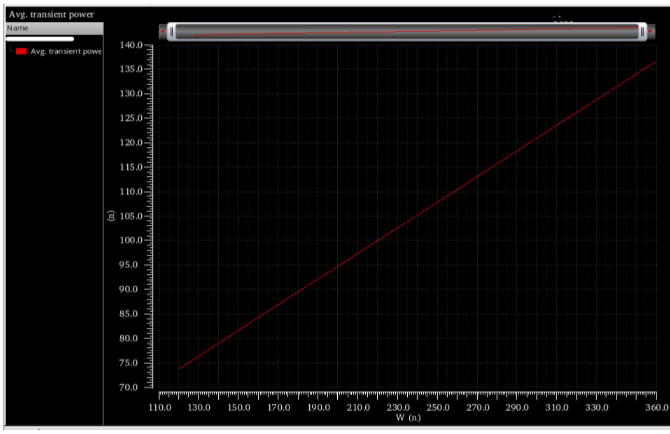
1. Click on the **variable** column and select **W**. Set the range **From→ 120n, To→ 360n, Total steps→ 9**. Now, run the parametric simulation.

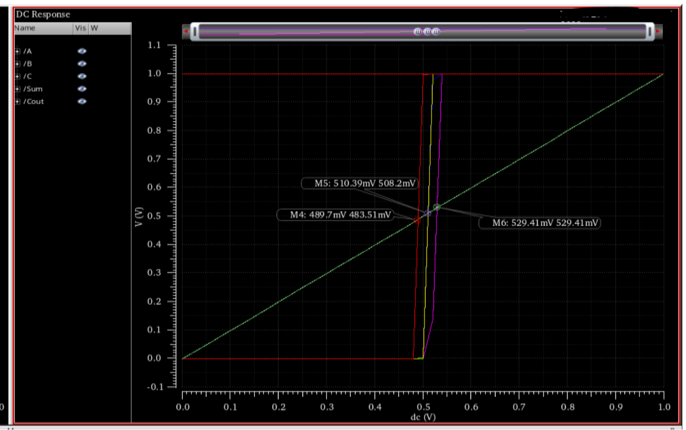


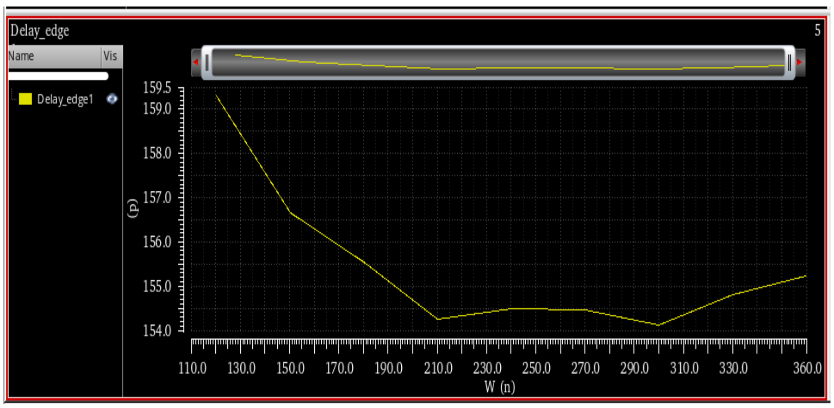
2. You will see 4 windows, delete the **Transient** and **Total; tran p** windows. Select any empty place of those waves and click on the **delete** button on the keyboard to delete them. Right click on **Delay_edge1** and move it to **new subwindow**.

3. Observe the variation of **average transient power consumption** and **inversion voltage** variation by the change of width value of PMOS.

(After observation, it is evident that power consumption increases with the widening of the PMOS width. So, why do we typically increase the width of PMOS transistors compared to NMOS? This is primarily done to account for the variation in the mobility of charge carriers. Increasing the width of PMOS transistors can reduce circuit delay, which is advantageous. Therefore, when selecting the **effective width** of PMOS, it is essential to consider **power consumption, delay**, and the physical circuit's **size/area**. Remember that, the closer the inversion voltage value is to ½ Vdd, the better the average delay of the circuit will be.

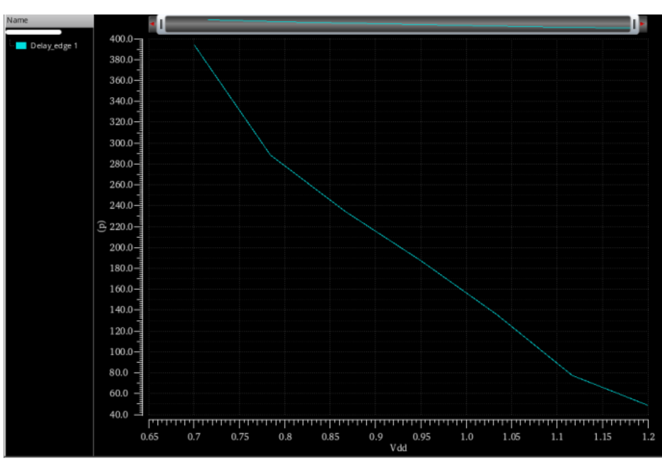


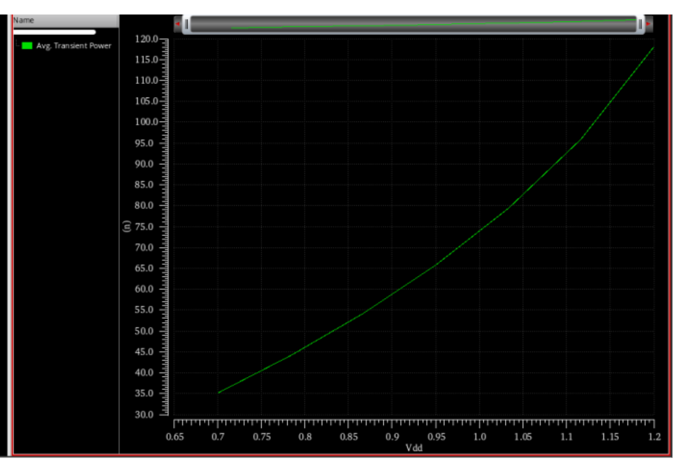




4. Now, return to the parametric analysis window. Replace the **W** with **Vdd** and set the value as **From→ 0.7, To→1.2, Total steps→ 7**. Hence, we will get results for 0.7V, 0.8V, 0.9V, 1V, 1.1V, and 1.2V in the graph. Run the simulation. Delete all the waveform windows except the average transient power and delay.

(Transient power consumption increases with the increase of supply voltage; it happens vice versa in case of delay. Determining a suitable supply voltage for an IC is called **voltage scaling**)



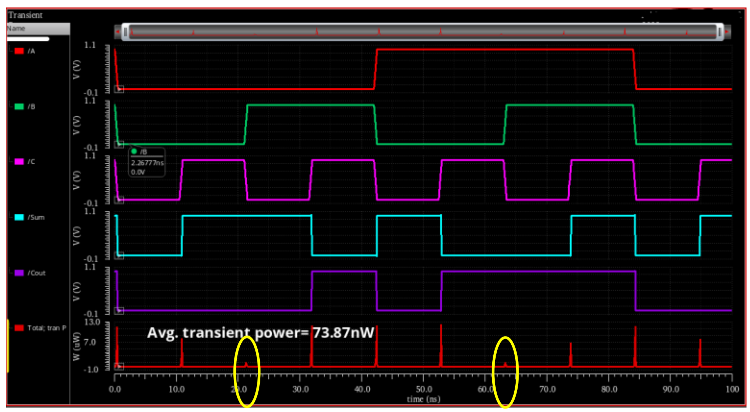


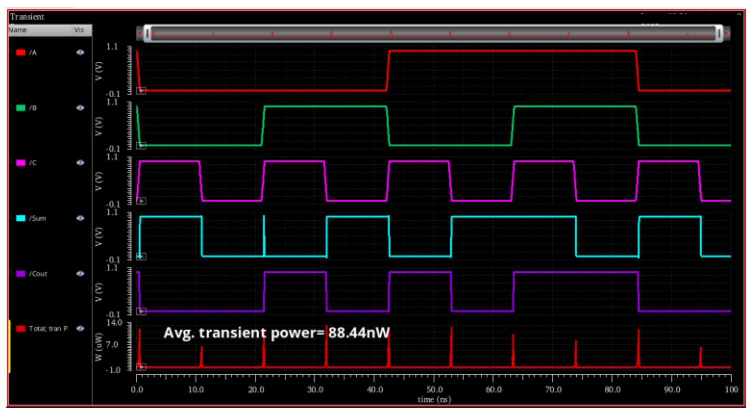
3.2.1 Actual Average Transient Power Consumption

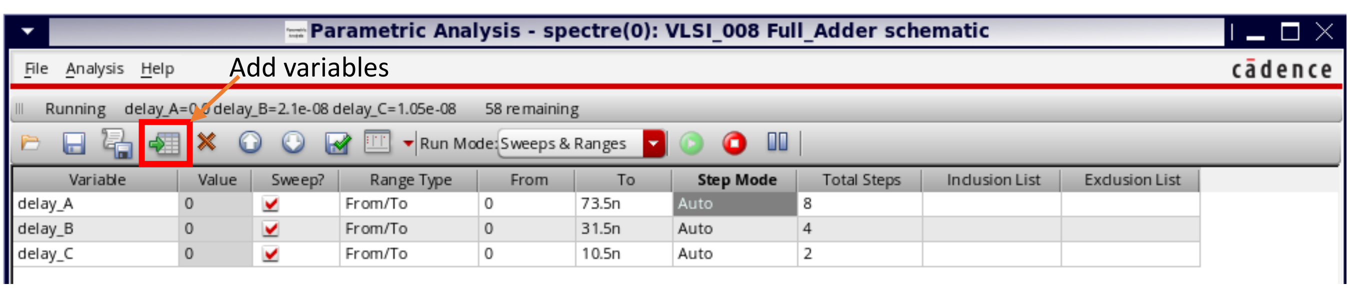
What is transient? We've heard that word very frequently in case of switching. When a switch changes its state from ON to OFF or vice-versa, within a very short period of time, the switch conducts both ON and OFF state. For electrical circuits, it means both wires got connected together. Which creates the momentary transient effect. But, what about the electronic components or specifically, MOSFET? In a CMOS circuit, PMOS transistors pass V_{dd} and NMOS transistors pass V_{ss}. Now, if for a certain voltage, both the PMOS and NMOS gate are triggered, what will happen then? The metal wire between PMOS and NMOS will carry both the V_{dd} and V_{ss}, which will lead to the transient condition in the circuit. At that moment, an unusually high amount of power dissipate through the circuit. This is called **transient power** and that certain voltage when both the PMOS and NMOS get triggered is called the **inversion voltage**.

Now, the input signals we applied in previous simulation, it maintains the order of 000→ 001→ 010→ 011→ 100→ 101→ 110→ 111 for A, B, C input pulses. During that period, the state of **Sum** has changed **5 times** and **carry** has changed states for **4 times**. From the observation of the transient power waveform, it is clear that the transient power dissipation happens when there is any change of states happens. What if the change of states increases or decreases for different order of input pulses (e.g., 001 before 000 input signal). Then, the power consumption will be changed also, and in reality, the input signal will never maintain any presumed order, but random signal order is realistic.

1. Go to the **ADE L** window and check the average transient power consumption. We found it 73.87nW.
2. In the left side of ADE L window, change the **delay_C → -10.5n** and then run the simulation.
3. From the waveform view, it is observed that the **C input** pulse train has changed and the number of changes of state for Sum and Carry has also changed.
4. Go to the ADE L window, you will see that the average transient power consumption has also changed.
5. There are **64 possible order** of input pulses that can be simulate at a time using parametric analysis.
6. Go to parametric analysis window and **add variable**. The value and range of variables are shown in the figure below. Run the simulation. Observe the average transient power waveform.







All data of the waveform can be **export** as an excel file (**.csv**) to find the actual average transient power consumption of the circuit. So, the fundamental use of parametric analysis is observing the variation of results for possible range of different variables or parameters.

Report: