Digital Electronics Lab (CPE 462)

Lab 1

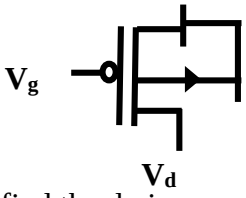
Transistor Analysis and TG

In this lab we will

- Analyze the I-V characteristics of the NMOS and PMOS transistors using LTspice.
- Design a transmission gate
- Test the ability of the NMOS and PMOS transistors to transfer voltage.

PART 1: Transistor Analysis

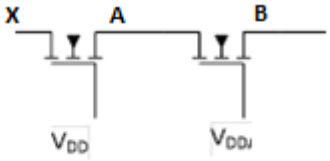
Build the circuit shown in the figure below using **PMOS** transistor and answer the following questions: $\mathbf{V_s = V_{DD} = 1.2V}$



1- For W=200n and L=50n find the drain current I_d and the mode of operation when the gate voltage is 0.3 V and the drain voltage is 0.2V.
2- Draw the I-V characteristics (V_{ds} vs I_{ds}) for the PMOS transistor for the following gate voltage values (0, 0.3, 0.6 and 0.8)
3- Draw the relation between the Length of the transistor and the current for the values used in Q1.
4- Draw the relation between the temperature and the current for the values used in Q1.
5- [**Post Lab**] Draw the relation between the body voltage and the current for the values used in Q1.

PART 2: Pass transistors and Transmission Gate

1- Using pass transistor design an experiment to measure the V_{th} voltage of an NMOS transistor.
Hint: Connect the gate of the transistor to Vdd and apply a pulse signal at node X.
2- Draw the circuit below to find the voltage at Node A and B in the figure below.



PART 3: Transmission Gate(Post Lab)

Design a transmission gate using NMOS and PMOS transistors and then design an experiment to test its ability to transfer strong 0 and strong 1.

Digital Electronics Lab (CPE 462)
Lab 2
Inverter Design and Analysis

In this lab we will

- Analyze the functionality of the NMOS resistive load inverter.
- Analyze the functionality of the CMOS inverter.
- Design the layout of the CMOS inverter

PART 1: NMOS Resistive Load Inverter (Schematic)

Design an NMOS resistive load inverter with the following parameters: RL=5K, W=128n , L=32n , Capacitance at output = 40fF and answer the following questions:

1- What are the values of V_{OL} and V_{OH} of the inverter?
2- Change the value of W and show its impact on the following inverter characteristics.

Parameter	V_{OH}	V_{OL}	$I_{C(Vin=0)}$	$I_{C(Vin=1)}$
W↑				

3- Measure the average power of one transition from 0 to 1 or 1 to 0. Assume the cycle time is 100n with 40% duty cycle

PART 2: CMOS Inverter (Schematic)

Design CMOS inverter with the following parameters: $W_n=64_n$, $L_n=32n$, $W_p=128n$, $L_p=32n$ Capacitance at output = 60fF and answer the following questions:

1- What are the values of V_{OL} and V_{OH} of the inverter?
2- What is the current of the NMOS and PMOS transistors when $V_{in=0}$ and $V_{in=1.}$
3- Measure the average power of one transition from 0 to 1 or 1 to 0.
4- Measure the rise time of the inverter.

PART 3: CMOS Inverter (Layout)

Design CMOS inverter with the following parameters: $W_n=64_n$, $L_n=32n$, $W_p=128n$, $L_p=32n$ Capacitance at output = 60fF and answer the following questions:

1- What are the values of V_{OL} and V_{OH} of the inverter?
2- What is the current of the NMOS and PMOS transistors when $V_{in=0}$ and $V_{in=1.}$
3- Measure the average power of one cycle.
4- Compare the rise time of the Layout and against the rise time of the schematic version measured in Part2-4.

Note: Please watch this video on how to use Electric to design the layout.
https://youtu.be/Jqj8VmS38fw

Instructions on Simulating Layout Using LTspice

Layout simulation using LTspice

1- In our class we will not use the **C5_models.txt** file that is used in the tutorial. Instead we will use the **32nm** technology models. You can find the file **Tech_models.txt** on MS teams under the Files tab. In your spice code use Tech_models.txt instead of C5_models.txt.

2-To setup the simulator to use LTspice follow these steps

1. In Electric Go to **File->Preferences.**
2. In the open window expand the **Tools** directory that appears on the right side of the window
3. click on Spice/CDL
4. In the **Running Spice** section fill the **Run Program** part with the path to the **XVIIx64.exe** file on your PC. It is the file that we execute when we launch LTspice.. You can find it inside **program files\LYC\LTspiceXVII**
5. Fill the **With Args** textbox with the following text **(COPY THE TEXT AS IT IS including the double quotation mark)**

"-i ${FILENAME} -r ${FILENAME_NO_EXT}.raw -o ${FILENAME_NO_EXT}.out"

To avoid any errors use the following spice code to simulate the 2 input NAND gate

vdd VDD 0 DC 1.2 vin **A** 0 PULSE(0 1.2 0 1n 1n 20n 40n 5)
vin2 **B** 0 PULSE(0 1.2 0 1n 1n 40n 80n 5)
cload **OUT** 0 50fF
.tran 0 40n .include C:\\Tech_models.txt (**Fill the right path to the Tech_models.txt file)**

Please let me know if you need any help

Digital Electronics Lab (CPE 462)
Lab 3
Combinational Circuit Design

In this lab we will

- Design the schematic for the basic logic gates
- Analyze the performance and power of the schematic of the basic gates
- Design the layout for the basic logic gates
- Analyze the performance and power of the layout of the basic gates
- **Note: In all questions assume that load capacitance is 60fF.**

PART 1A: NAND gate (Schematic)

Design the schematic of the 4-input NAND gate and answer the following questions

1- Test all input combinations for the gate
2- Size the transistors such that the worst case rise and the fall time are almost the same.
3- Measure the average power of the NAND gate 1ns before the transition and 1ns after the transition of the input signal.

PART 1B: NOR gate (Schematic)

Design the schematic of the 4-input NOR gate and answer the following questions

1- Test all input combinations for the gate
2- Size the transistors such that the worst case rise and the fall time are almost the same.
3- Measure the average power of the NOR gate 1ns before the transition and 1ns after the transition of the input signal.

PART 2A: NAND gate (Layout)

Draw the stick diagram of the 3-input NAND gate and then Design the layout of the gate before answering the following questions

1- Test all input combinations for the gate
2- Size the transistors such that the worst case rise and the fall time are almost the same.
3- Measure the average power of the NAND gate 1ns before the transition and 1ns after the transition of the input signal.

PART 2B: NOR gate (Layout)

Draw the stick diagram of the 3-input NOR gate and then Design the layout of the gate before answering the following questions

1- Test all input combinations for the gate
2- Size the transistors such that the worst case rise and the fall time are almost the same.
3- Measure the average power of the NOR gate 1ns before the transition and 1ns after the transition of the input signal.

PART 3: NOR gate using Transmission gates and inverters (schematic)(Post Lab)

Design 2-input NOR gate using transmission gates and inverters only and answer the following questions:

1- Test all input combinations for the gate
2- Size the transistors such that the worst case rise and the fall time are almost the same.
3- Measure the average power of the NOR gate 1ns before the transition and 1ns after the transition of the input signal.

Note: Please watch this video on how to use Electric to design the layout.
https://youtu.be/Jqj8VmS38fw

Digital Electronics Lab (CPE 462)

Lab 4

Combinational Circuit Design

In this lab we will

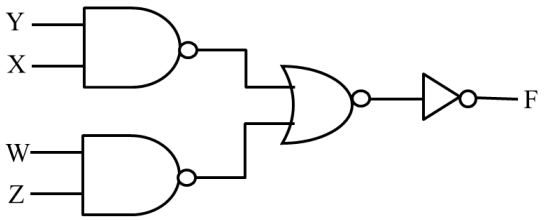
- Design and Analyze the functionality of the Combinational circuits layout built as complex function and using standard cells.

PART 1: Standard Cell Design (Layout)

Given the NAND, NOR and Inverter gates layouts build the layout for the following functions.

1- Test the functionality of the design using all input combinations

2- What is the rise time of the design when the input changes from WXYZ = 1111 to WXYZ = 1100



PART 2: Design the layout for the following expression as a complex function(Layout).

1- Draw the schematic of the expression

2- Draw the stick diagram

3- Based on the stick diagram design the layout of the expression.

$$F = \overline{WZ} + \overline{XY}$$

Digital Electronics Lab (CPE 462)
Lab 5
Combinational Circuit Design

In this lab we will

- Design the schematic for the basic logic gates using Pseudo-NMOS, Dynamic and Domino Logic
- Compare the power of the dynamic logic against the CMOS Circuits.
- **Note: In all questions assume that load capacitance is 60fF.**
- **In all experiments measure the power for the**

PART 1A: Pseudo-NMOS 2-Input NAND gate (Schematic)

Design the schematic of the 2-Input NAND gate using Pseudo-NMOS transistor

1- Test the functionality of the NAND gate.
2- What is the relation between the width of the PMOS transistor and V_{OL}

PART 1B: Dynamic 2-Input NAND gate (Schematic)

Design the schematic of the CMOS Inverter and the Dynamic Inverter and answer the following questions:

1- Test the functionality of the NAND gate
2- Create a symbol for the NAND gate
3- Measure the FO4 delay of the NAND gate

PART 2: NAND gate (Schematic) (Post Lab)

Design the schematic of the CMOS 2-Input NOR and the Dynamic 2-Input NOR and answer the following questions:

1- Test the functionality of the NOR gate.
2- Create a symbol for each NOR gate
3- Measure the FO4 delay of the NOR gate

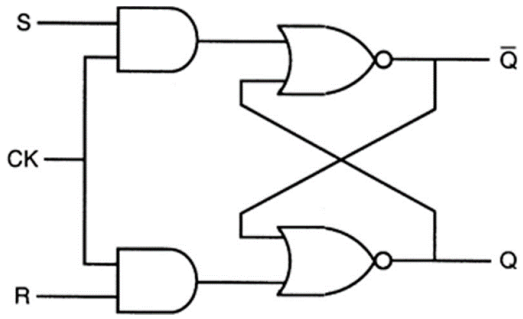
Digital Electronics Lab (CPE 462)

Lab 6

Sequential Circuit Design (Part 1)

In this lab we will

- Design the schematic for the Clocked SR latch and analyze its functionality
- Design the layout for the Clocked SR latch and analyze its functionality



PART 1: Clocked SR latch (Schematic)

Design the schematic for Clocked SR latch and verify its functionality

1- Measure the D-to-Q delay propagation delay of the latch
2- Measure the Clk-to-Q delay of the latch

PART 2: Clocked SR latch (Layout) (Post lab)

Design the layout for Clocked SR latch and verify its functionality

1- Measure the D-to-Q delay propagation delay of the latch
2- Measure the Clk-to-Q delay of the latch

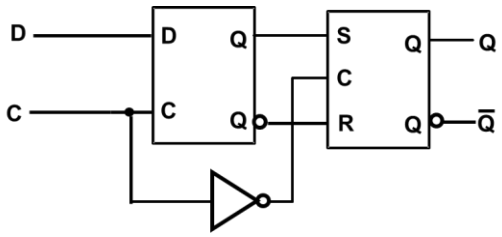
Digital Electronics Lab (CPE 462)

Lab 7

Sequential Circuit Design Part2 (Flip flops)

In this lab we will

- Design the schematic for the Master-Slave D Flip Flop and analyze its functionality
- Design the schematic for the Master-Slave D Flip Flop and analyze its functionality



PART 1: D-Master-Slave FF (Schematic)

Design the schematic for the D-Master-Slave FF and verify its functionality:

1- Measure the Clk-to-Q delay of the FF
2- Measure the setup time for the FF

PART 2: D-Master-Slave FF (layout)(Post lab)

Design the layout for the D-Master-Slave FF and verify its functionality:

1- Measure the Clk-to-Q delay of the FF
2- Measure the setup time for the FF

Digital Electronics Lab (CPE 462)
Lab 8
Memory Cells (6T SRAM Cell)

In this lab we will

- Design the schematic for the SRAM cell and test its functionality.

For more details about the SRAM operation please watch the video that I uploaded on the class page on MS teams.

Also it is recommended to watch this YouTube to get better idea about the design and testing
https://www.youtube.com/watch?v=lPnQVrzgC-M&ab_channel=SanjayVidhyadharan

PART 1: 6T SRAM Cell (Schematic)

- Design the schematic for 6T SRAM Cell
- Design the schematic for the sense amplifier
- Test the read operation
- Test the write operation