

Misr International University
Electronics and Communication Engineering
ECE 542 (VLSI Lab)
Spring 2023
Lab 1:
Inverter Gate

Dr. Ghazal Attia
Eng. Habiba Mohamed

Name	ID
Nour-Aldin Ibrahim Ahmed Azzab El-badawy	2018-13394

Contents

Table of Figures:.....	2
Aim:.....	3
Introduction:	3
Run Cadence	4
Inverter Gate Schematic Design.....	7
Inverter Gate Testbench	8
DC analysis	9
schematic:	9
simulation Output:	9
Transient Response analysis	11
schematic:	11
simulation Output:	12
Inverter Gate Layout:	13
Testing.....	14
Design Rule Check Simulation (DRC):.....	14
Layout Versus Schematic (LVS)	14
Parasitic Extraction (PEX)	15
Post Layout Simulation	16
Conclusion:.....	17
Reference:.....	17

Table of Figures:

Figure 1: Inverter Truth Table	3
Figure 2: Run The VMware	4
Figure 3: Log in.....	4
Figure 4: open Cadence	5
Figure 5: open virtuoso	5
Figure 6: Creating New Library and cell	6
Figure 7: Inverter Schematic	7
Figure 8: Inverter Symbol.....	8
Figure 9: Create New Cell View.....	8
Figure 10: files of inverter testbench.....	8
Figure 11: DC analysis Schematic.....	9
Figure 12: DC analysis Output.....	9
Figure 13: DC analysis output with variation in W.....	10
Figure 14: Transient Analysis Schematic.....	11
Figure 15: Transient Analysis Simulation Output.....	12
Figure 16: Inverter Layout.....	13
Figure 17: DRC.....	14
Figure 18: LVS.....	14
Figure 19	15
Figure 20	15
Figure 21: Parasitic Components	15
Figure 22: Post Layout Simulation Output.....	16

Aim:

The aim of this report is to present the design process of an inverter gate using the Cadence tool. The design process consists of the following steps:

- Defining the specifications of the inverter gate
- Developing a behavioral model of the inverter gate
- Creating a schematic diagram of the inverter gate
- Testing the functionality and performance of the schematic design
- Designing a layout of the inverter gate
- Testing the layout design for any errors or violations
- Performing a post-layout simulation and verification

Introduction:

An inverter gate is a basic logic gate that produces an output that is the opposite of its input. For example, if the input is 1, the output is 0, and vice versa. Inverter gates are widely used in digital circuits and systems, such as computers, calculators, and communication devices. In this report, we will describe how we designed an inverter gate using the Cadence tool, which is a software platform for electronic design automation. We will explain the steps we followed to create a schematic diagram, a layout design, and a post-layout simulation of the inverter gate. We will also present the results of our testing and verification of the design, and discuss the challenges and limitations we faced during the design process.

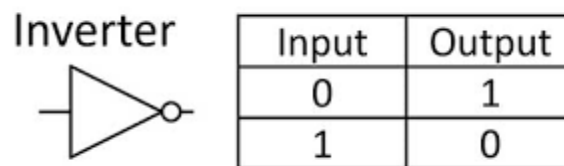


Figure 1: Inverter Truth Table

Run Cadence

- Launch VMware and access the CentOS virtual machine.

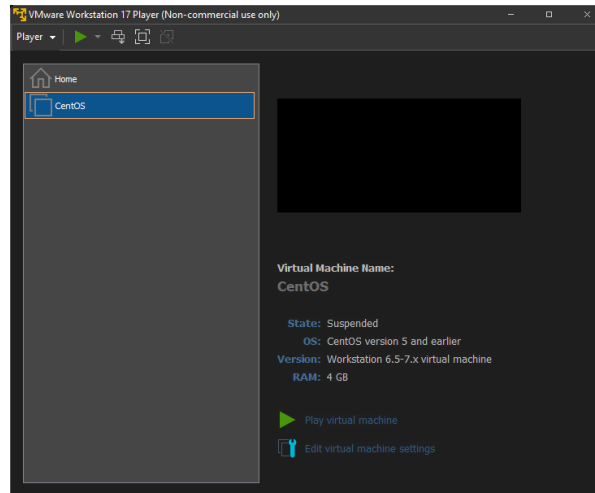


Figure 2: Run The VMware

- Log in as root user with the password elc411.

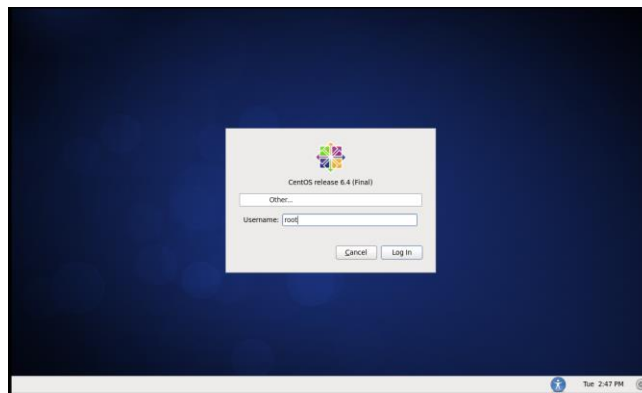


Figure 3: Log in

- Open a terminal window and enter “cadence 1”.



Figure 4: open Cadence

- Then enter “virtuoso &” and wait for cadence to open.

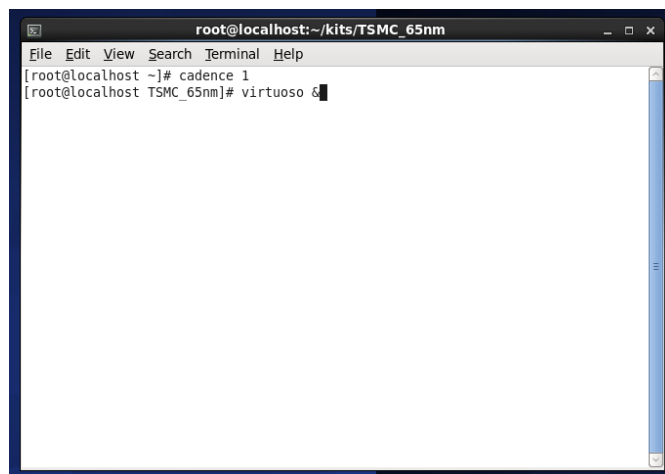


Figure 5: open virtuoso

- Make a new library with a name of your choice.
- Make a new cell named “Inverter”.

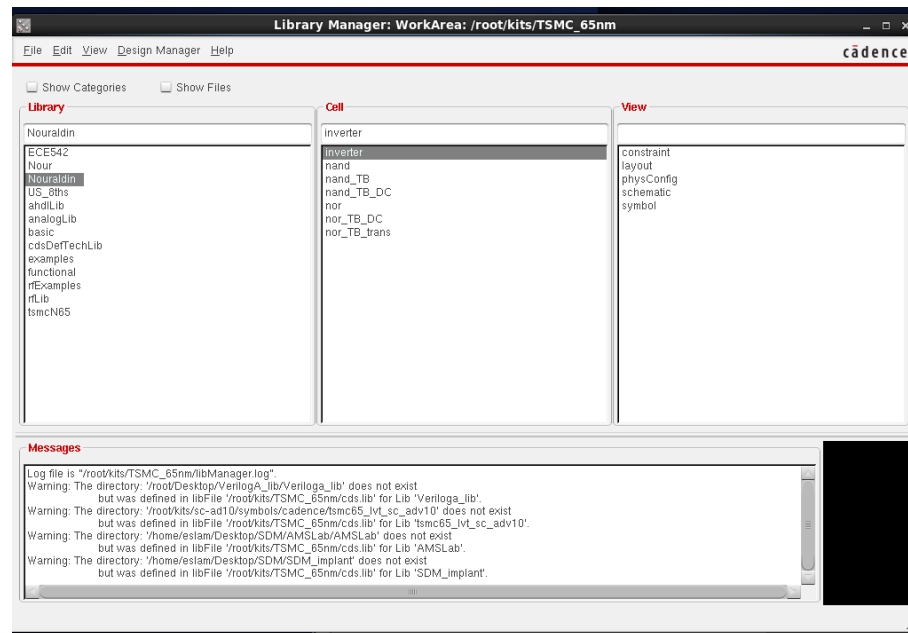


Figure 6: Creating New Library and cell

Inverter Gate Schematic Design

- Press 'I' to insert the components required for the schematic.
- From the "TSMC65" technology library, insert a pmos "pch" and a nmos "nch".
- Change the pch parameters to $W = 2\mu$.
- change the nch parameters to $W = 1\mu$.
- From the "analoglib" library, insert vdd and gnd.
- Make your Vin and Vout ports.
- Press 'W' to start connecting the components.

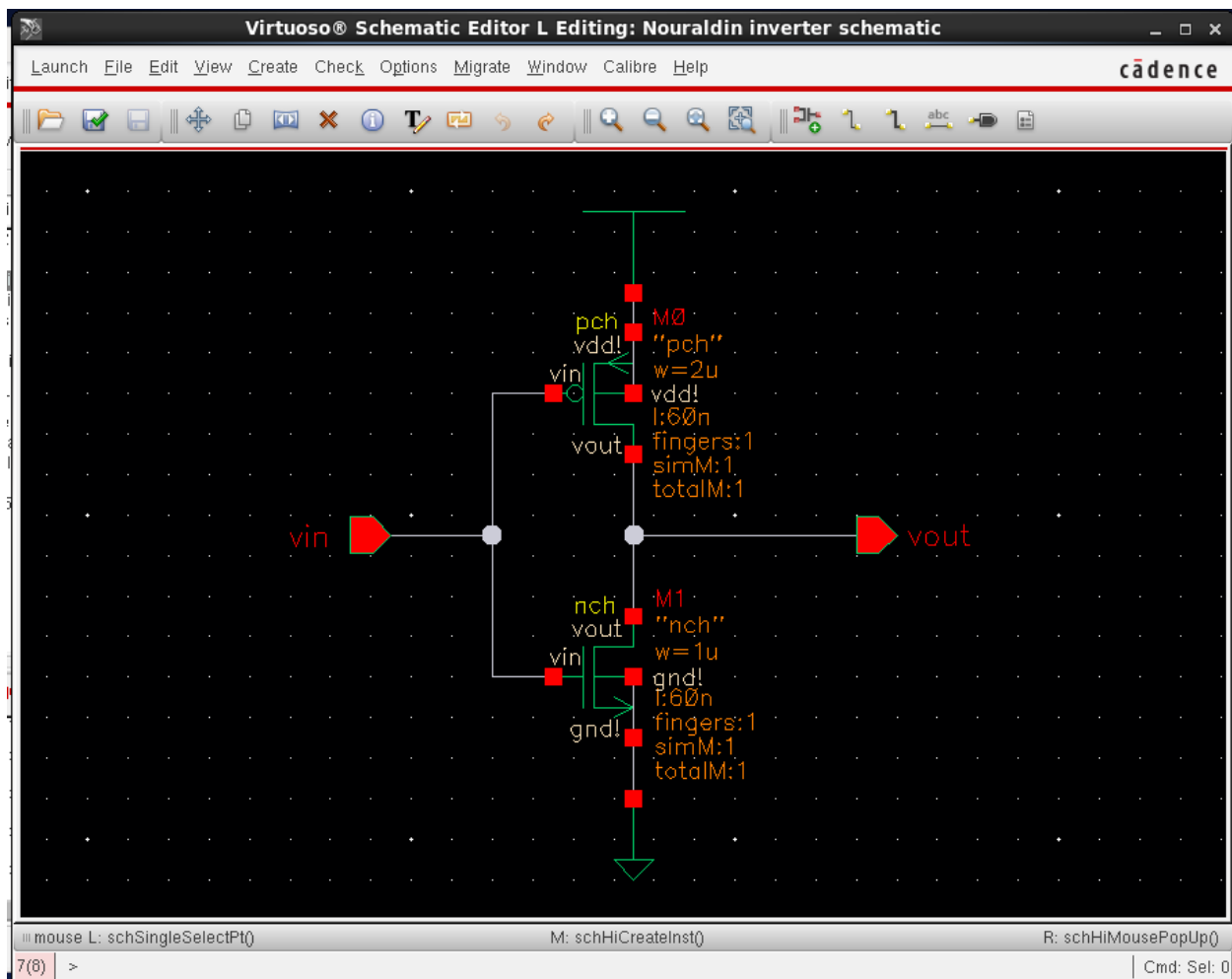


Figure 7: Inverter Schematic

Inverter Gate Testbench

- We need to create a cell view

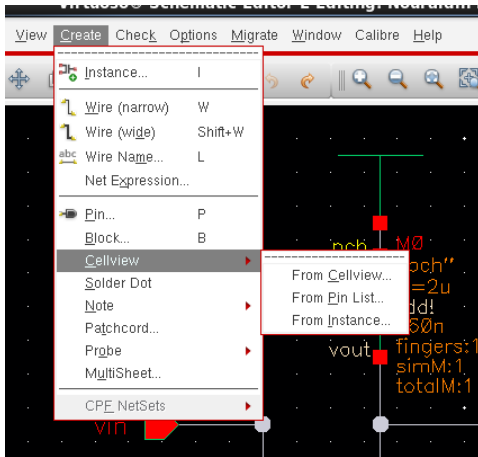


Figure 9: Create New Cell View

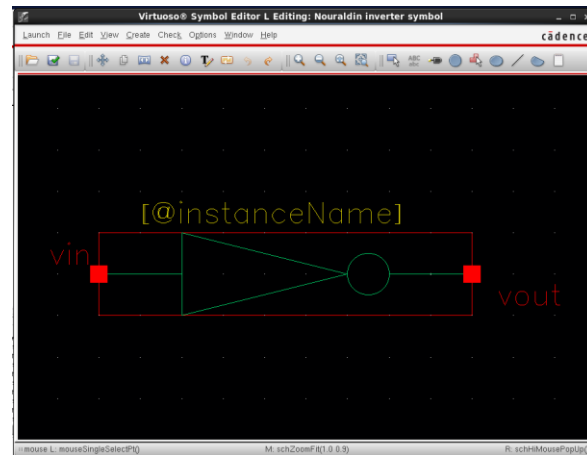


Figure 8: Inverter Symbol

- Then we create a new cell called "Inverter_TB_DC" for DC analysis and "Inverter_TB_trans" for transient analysis.



Figure 10: files of inverter testbench

DC analysis

schematic:

We used a DC voltage source with value = vb as an input.

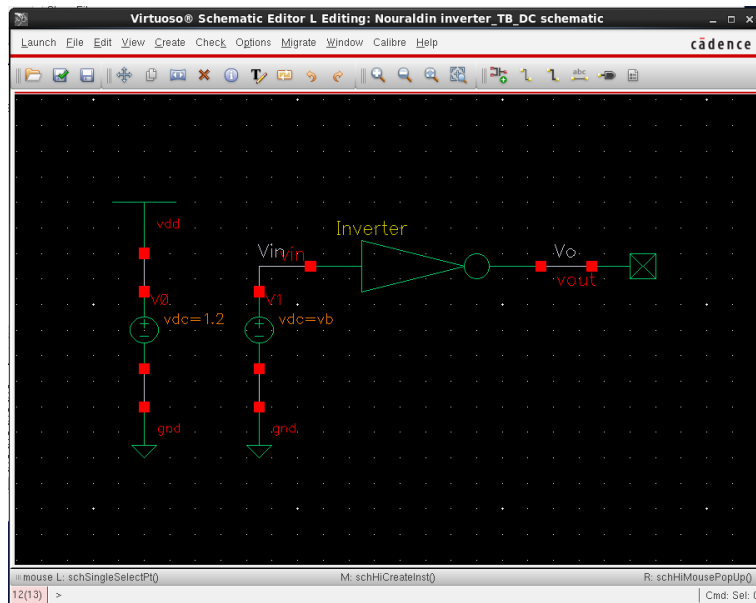


Figure 11: DC analysis Schematic

simulation Output:

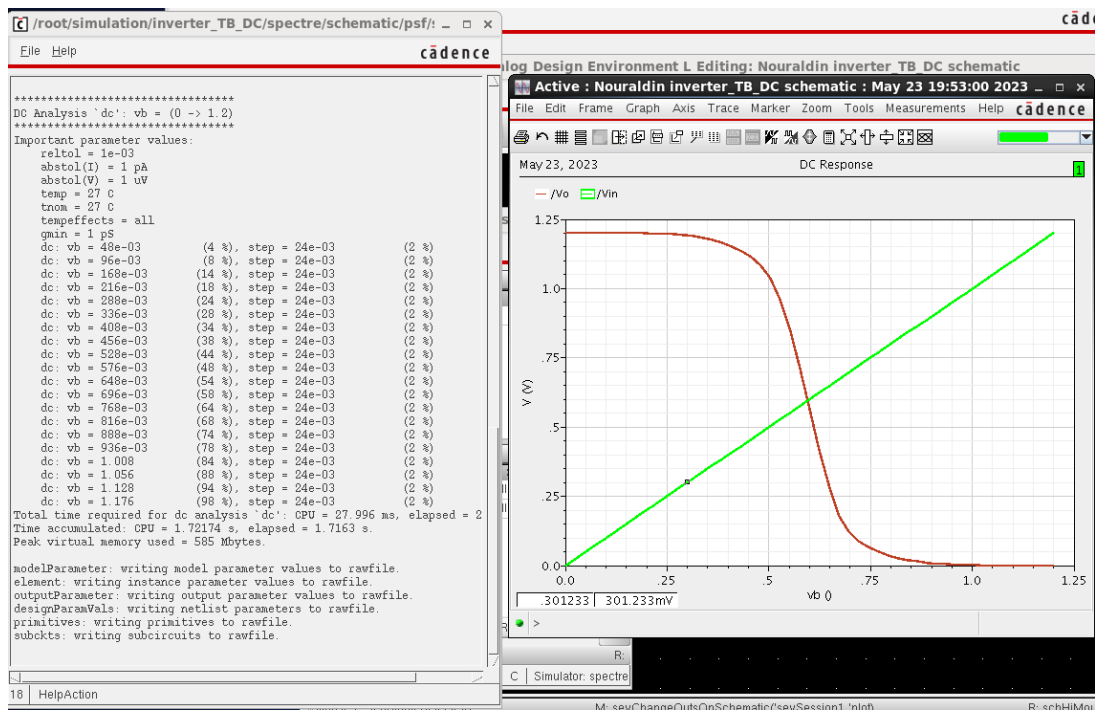


Figure 12: DC analysis Output

The parametric analysis investigates how the circuit performance varies with the width of the pmos transistor

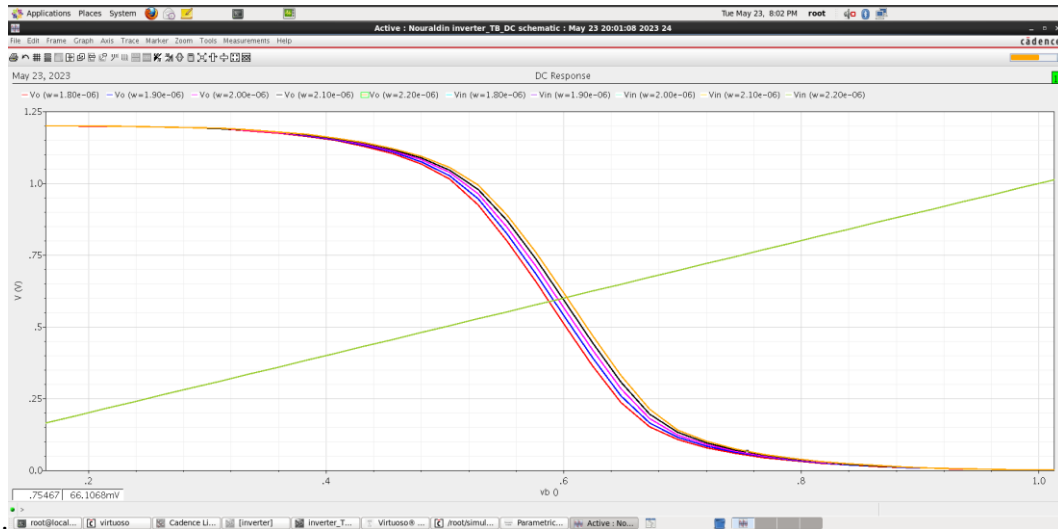


Figure 13: DC analysis output with variation in W

Transient Response analysis

schematic:

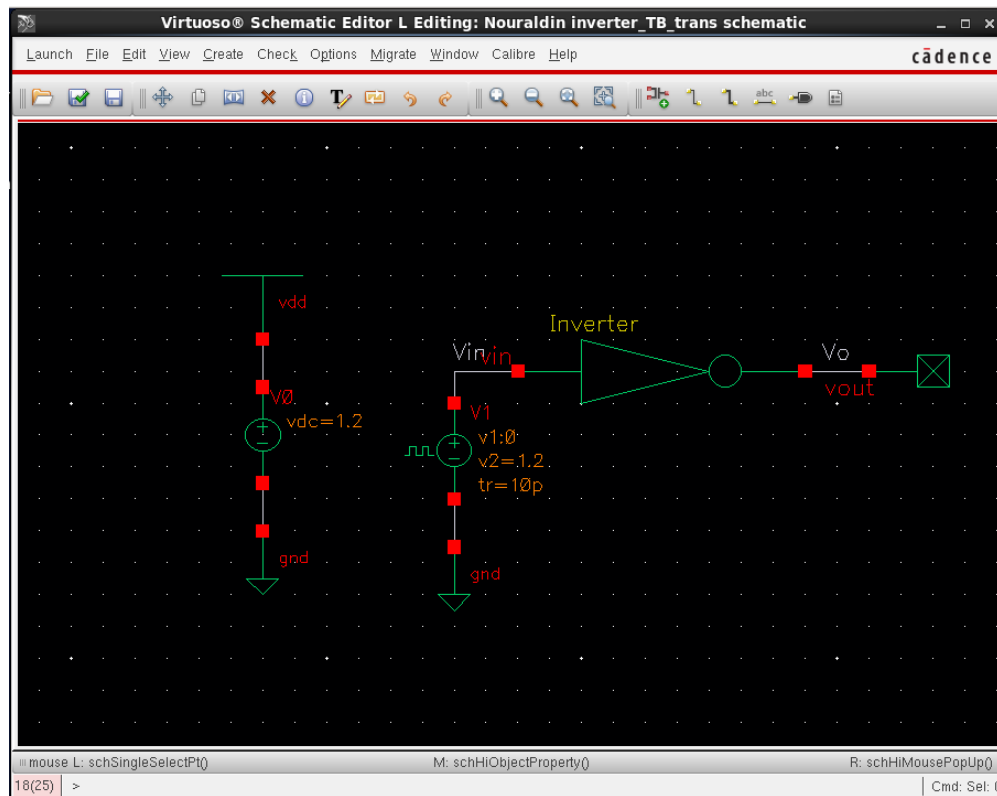


Figure 14: Transient Analysis Schematic

Vpulse voltage source was used to provide the input signal.

The image shows the "Edit Object Properties" dialog box for a VPULSE source. The dialog has tabs for User Property, Master Value, Local Value, and Display. The Master Value tab is selected. The dialog contains a table of parameters and their values:

CDF Parameter	Value	Display
Frequency name for 1/period		off
Noise file name		off
Number of noise/freq pairs	0	off
DC voltage		off
AC magnitude		off
AC phase		off
XF magnitude		off
PAC magnitude		off
PAC phase		off
Voltage 1	0 V	off
Voltage 2	1.2 V	off
Period	1u s	off
Delay time	0 s	off
Rise time	10p s	off
Fall time	10p s	off
Pulse width	500.0n s	off
Temperature coefficient 1		off
Temperature coefficient 2		off
Nominal temperature		off
Type of rising && falling edge		off

At the bottom of the dialog are buttons for OK, Cancel, Apply, Defaults, Previous, Next, and Help.

simulation Output:

The simulation shows that the output is the opposite of the input, which confirms the correct functioning of the inverter.

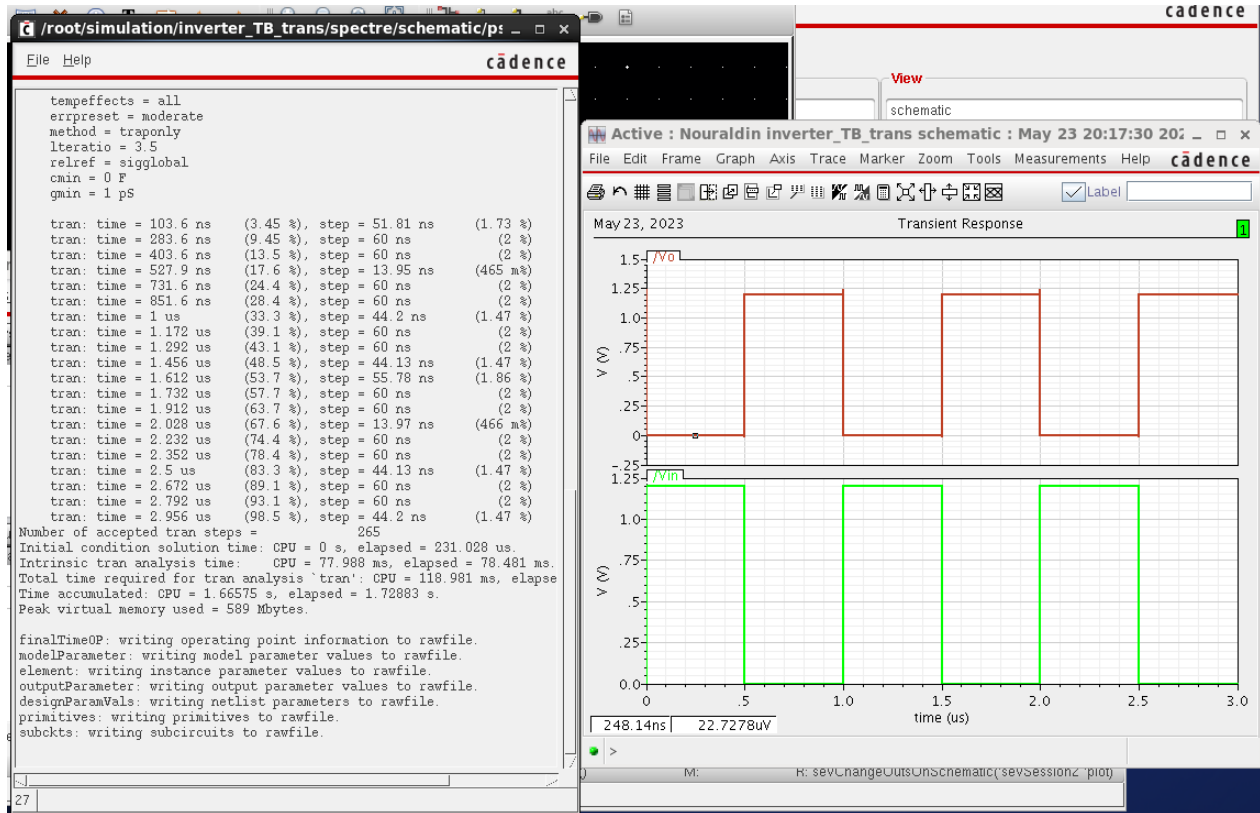


Figure 15: Transient Analysis Simulation Output

Inverter Gate Layout:

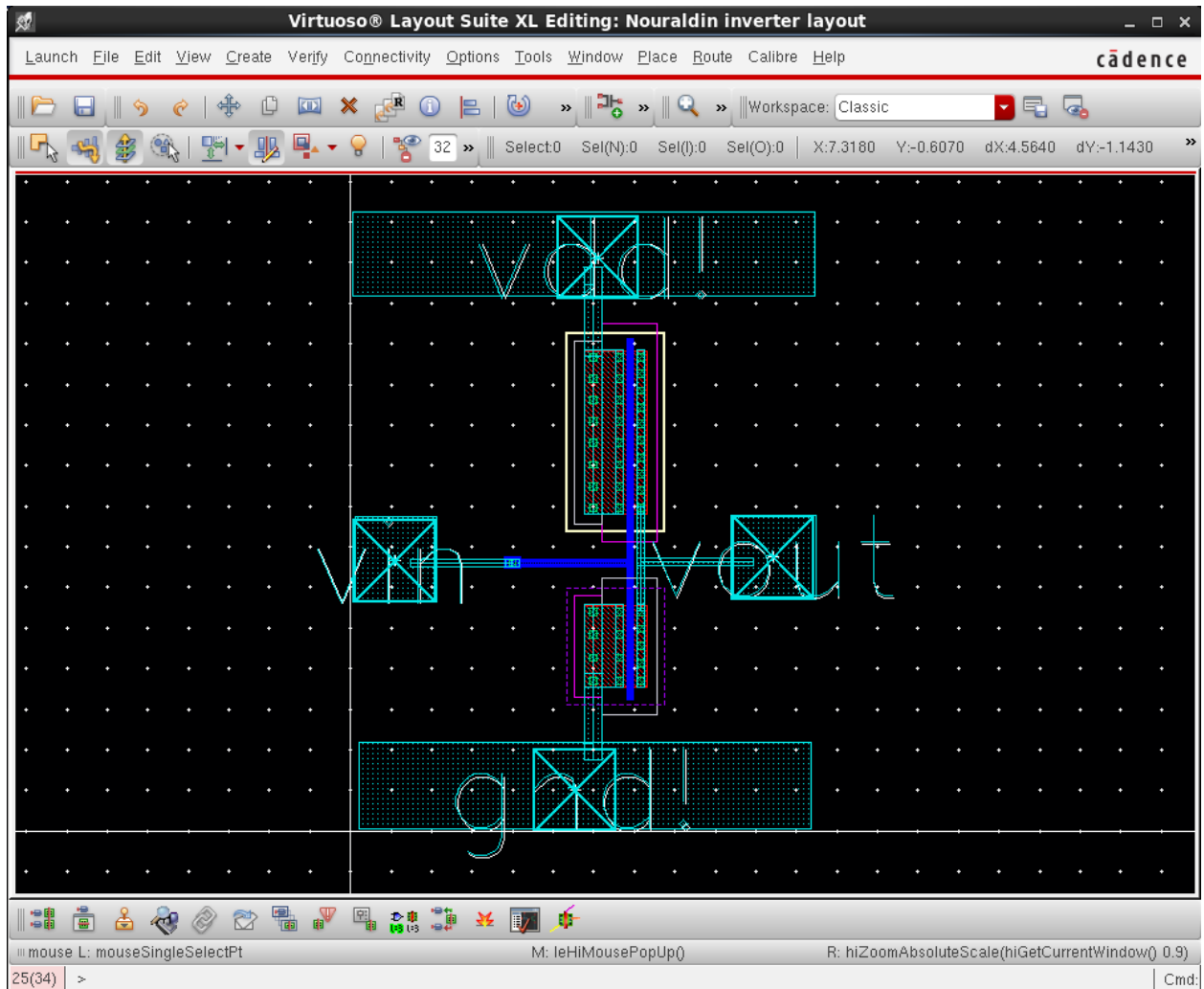


Figure 16: Inverter Layout

Design Rule Check Simulation (DRC):

There is No Mx.S or A or W.

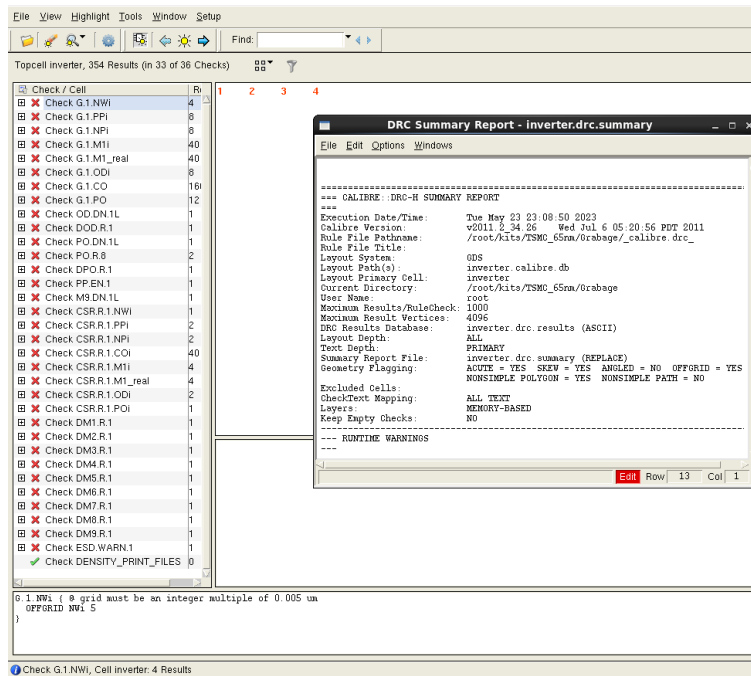


Figure 17: DRC

Layout Versus Schematic (LVS)

We have the smiley face 😊

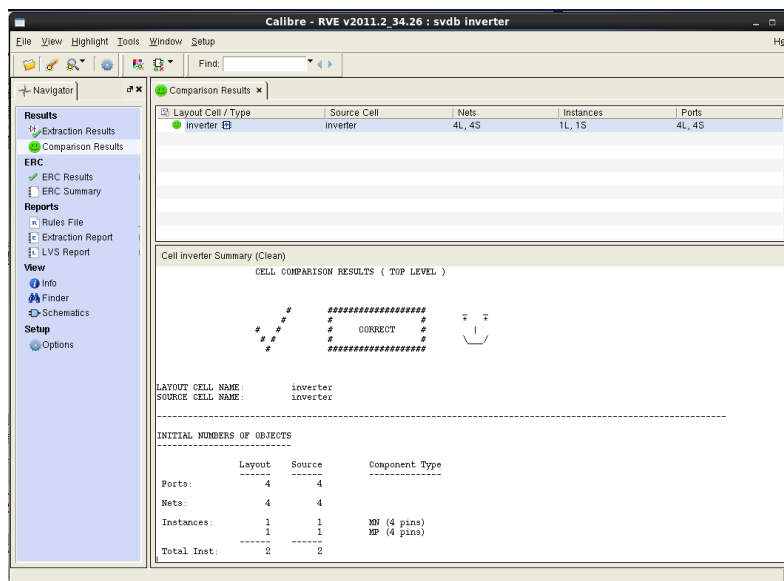


Figure 18: LVS

Parasitic Extraction (PEX)

Parasitic Extraction (PEX) is a process of calculating the unwanted effects of parasitic components, such as resistances, capacitances, and inductances, in an electronic circuit. PEX is used to create an accurate model of the circuit for simulation and verification purposes.



Figure 20

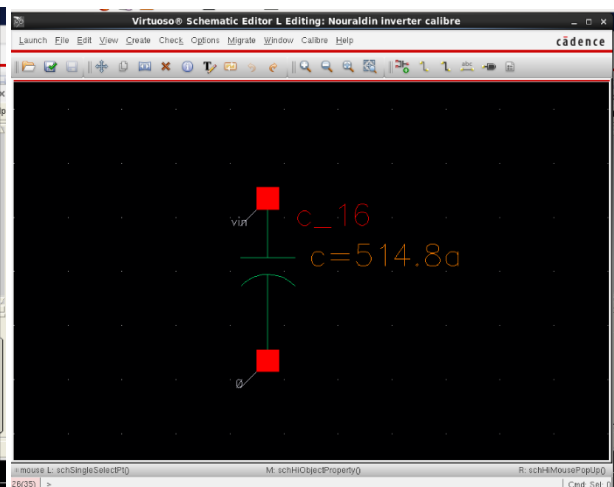


Figure 19

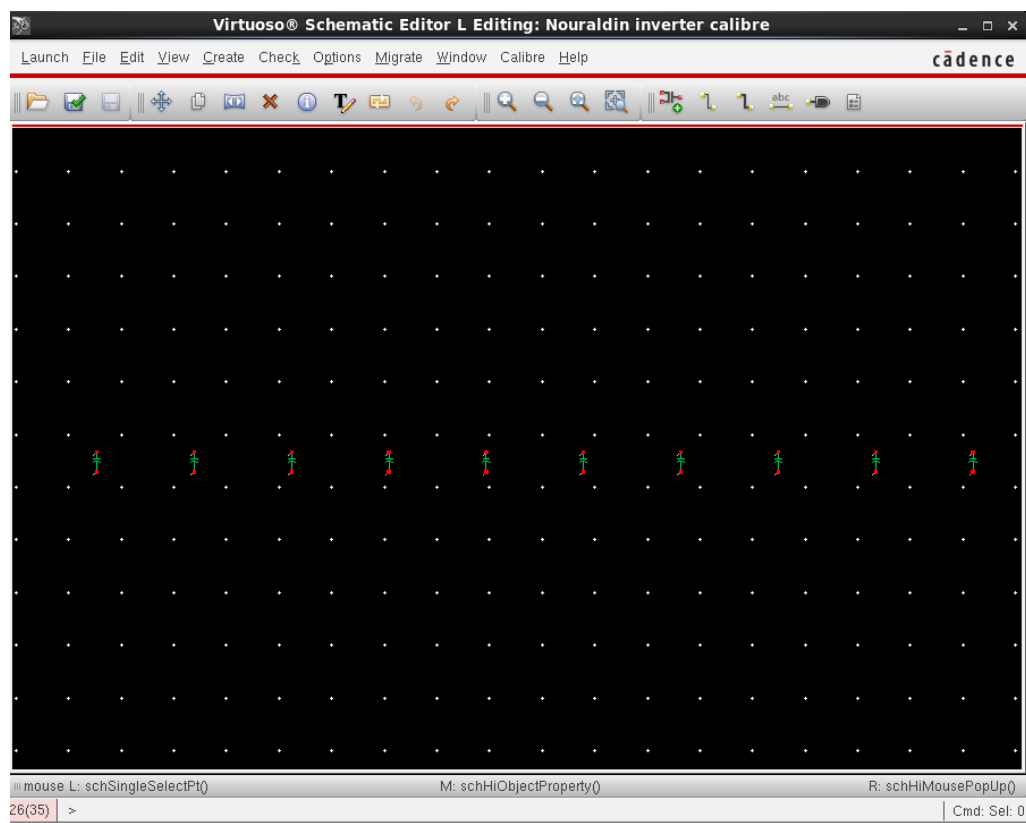


Figure 21: Parasitic Components

Post Layout Simulation

The delay, which is 1.59ps as seen in the following figure, is within the acceptable range.

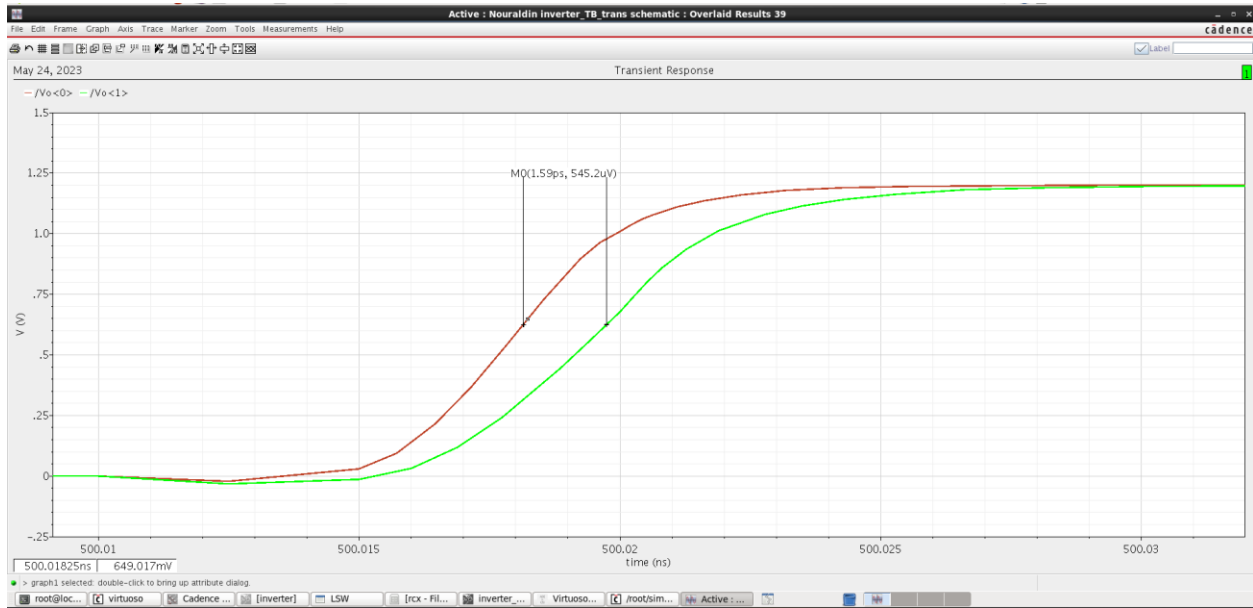


Figure 22: Post Layout Simulation Output

Conclusion:

In this report, we have presented the design process of an inverter gate using the Cadence tool. We have successfully completed the following steps:

- Defined the specifications of the inverter gate
- Created a schematic diagram of the inverter gate
- Tested the functionality and performance of the schematic design
- Designed a layout of the inverter gate
- Tested the layout design for any errors or violations
- Performed a post-layout simulation and verification

We have demonstrated that our design meets the requirements and specifications of an inverter gate. We have also learned how to use the Cadence tool for electronic design automation, and gained valuable experience and skills in digital circuit design. We have encountered some challenges and limitations during the design process, such as choosing the appropriate parameters, optimizing the layout area, and ensuring the reliability and robustness of the design. We have overcome these challenges by applying the relevant concepts, methods, and tools that we have learned in this course. We hope that this report has provided a clear and comprehensive overview of our design process and results.

Reference:

All the references are from the lab manual and from the Lecture by Dr. Ghazal Attia and Eng. Habiba Mohamed