AIR UNIVERSITY	**Department of Electrical and Computer Engineering**
	Subject: Linear Integrated Circuit Design
	LAB # 1 Introduction to Cadence Virtuoso

Student Name: Muhammad Haseeb **Reg. No:** 200718 **Section:** BEEE-7A

Objective: Familiarizing students with Cadence Virtuoso

As directed by the lab instructor, create three random schematics.

LAB ASSESSMENT:

Attributes	Excellent (5)	Good (4)	Average (3)	Satisfactory (2)	Unsatisfactory (1)
Ability to Conduct Experiment					
Ability to assimilate the results					
Effective use of lab equipment and follows the lab safety rules					

Total Marks: 15 Obtained Marks : _____

LAB REPORT ASSESSMENT:

Attributes	Excellent (5)	Good (4)	Average (3)	Satisfactory (2)	Unsatisfactory (1)
Data presentation					
Experimental results					
Conclusion					

Total Marks: 15 Obtained Marks: _____

Date: _____ Signature: _____

Title:

- Introduction to Cadence Virtuoso

Equipment:

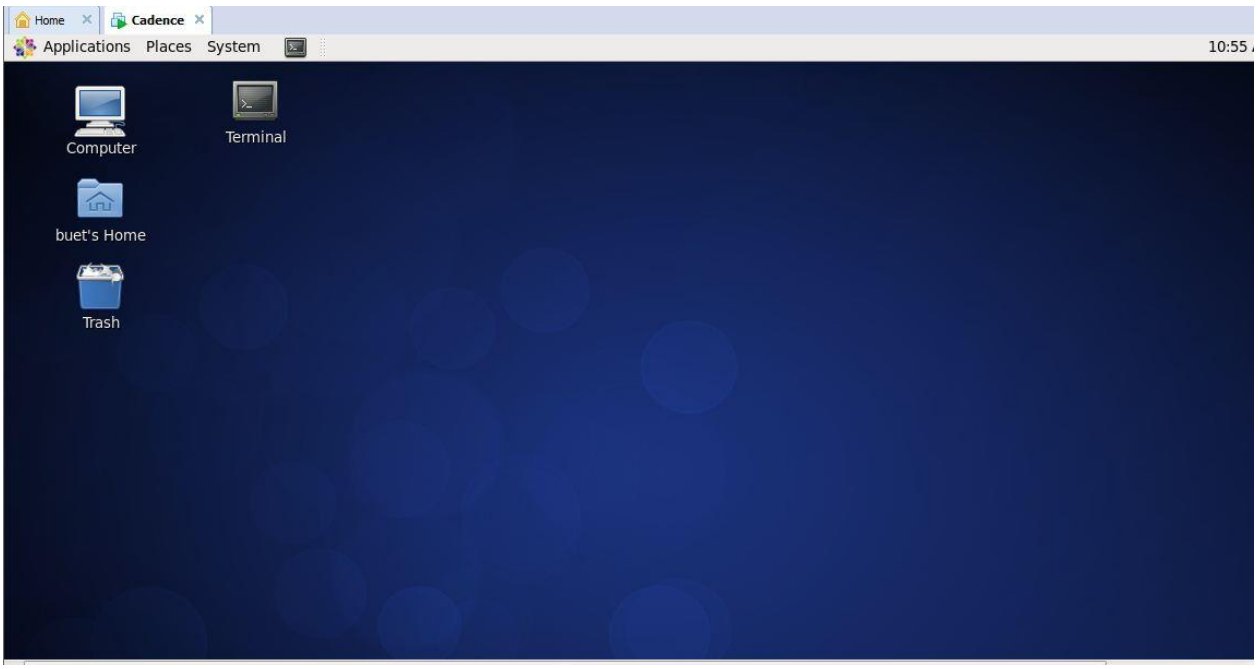
- Personal Computer
- VMware software
- Cadence Virtuoso

Introduction

Cadence virtuoso so is a software that is that is used for PCB designing and IC designing it is a software that is only compatible for Linux operating system so in order to use the software on Windows PC we need to create an environment that is compatible to Linux for this we use VMware which is the software that is used to to run Cadence on a Windows PC.

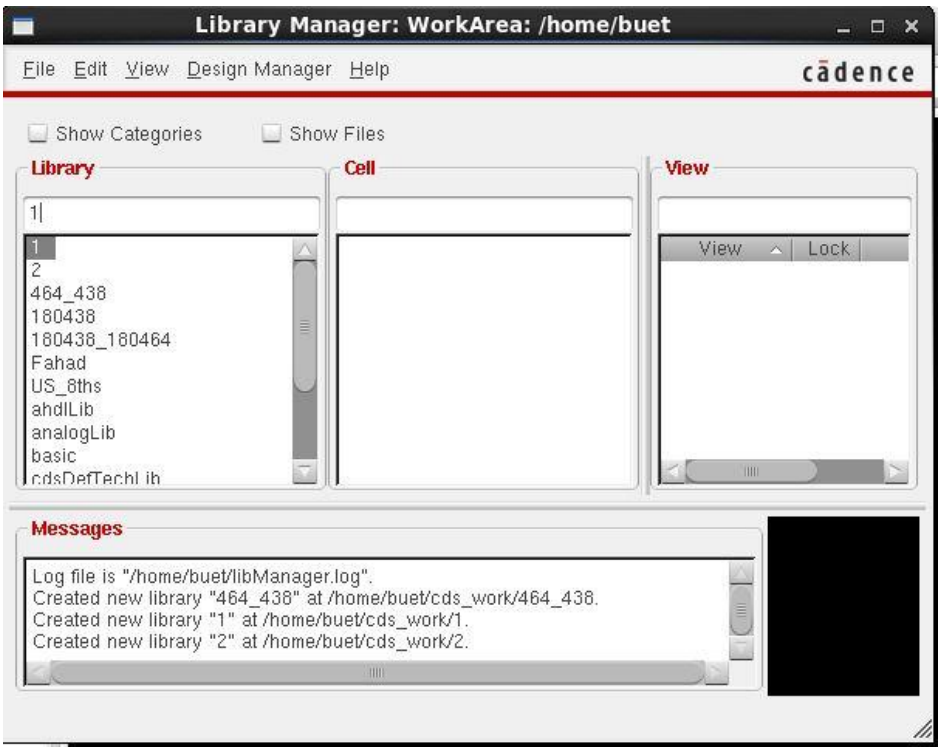
Procedure

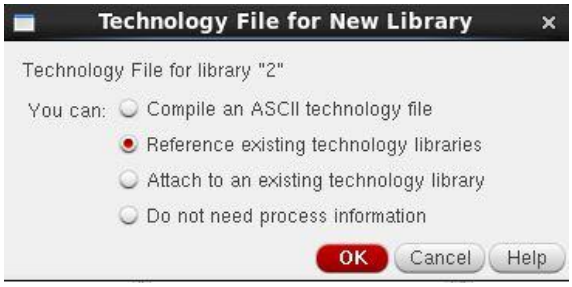
- First of all, go to the website and install VMware software.
- Copy the VM_RF_Liscenced folder from any workstation in the lab.
- Open the VMware workstation, click on open virtual machine, go to the folder VM_RF_Liscenced. Now find the file CentOS and double click that.
- After this we will start the virtual machine and wait for the screen of the Linux window to appear. The desktop background may appear different from the following one.



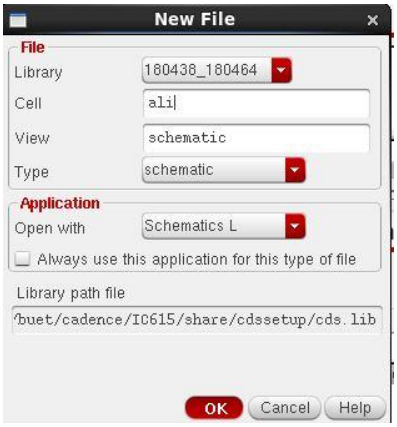
- Now in the linux window we will open the terminal and type “su” and press enter.
- Now enter password , elc411 and press enter
- Write command cadenc and press enter
- Write command virtuoso and press enter
- A window of the software Virtuoso will appear
- Now we will go to the tools section and select “Library manager”.

- In the Library manager we will select file and create a new library with the name of our choice and press enter. A dialog box will appear asking technology of file. Select attach to an existing technology in the dialog box. Another dialog box will appear that will ask for the existing technologies. Select “tsmc13rf”





- After creating the library we will now create a cell with the name of our choice and select "OK"



- Finally, the systematic window opens and we can design our circuit here.

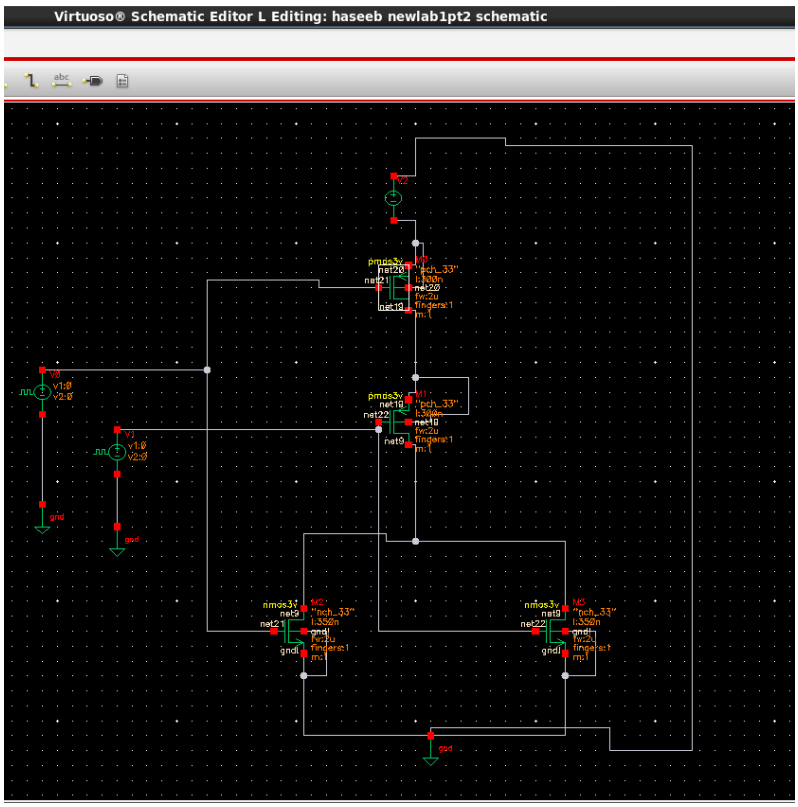
Lab Task

Adding Components on schematic:

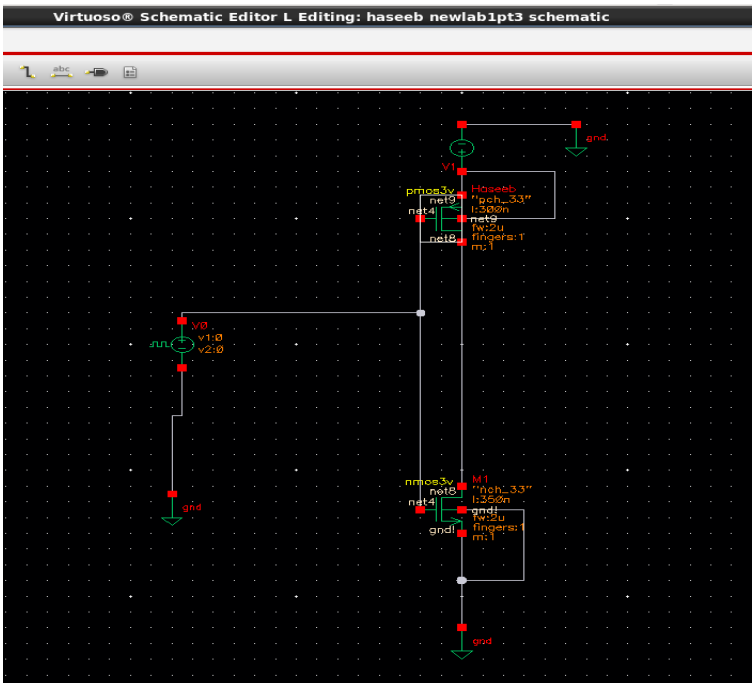
- In order to add a component on the schematic click on **"create instance"** or press **"I"** on the keyboard.
- To add a MOSFET click on browser then choose library **"tsmc13rf"** and under cell search **"nmos3v"** or **"pmos3v".**
- Then click on symbol and place it anywhere on the schematic.
- To add resistors and voltage supply use "**analogLib".**
- Once all the components are placed to connect them with wires, click on create narrow wire and patch all the components together.

Results

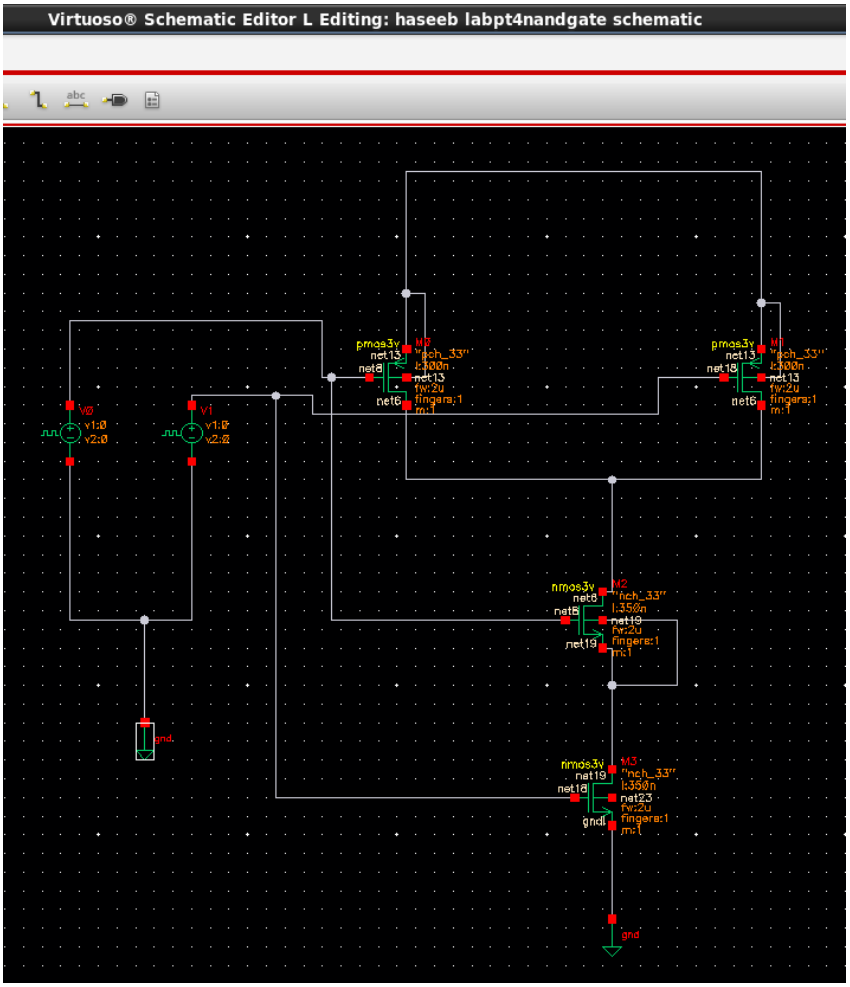
Nor Gate:



Inverter:



Nand Gate



CONCLUSION

In this Lab we learned how to install and run cadence virutoso on windows and how to create a schematic and select different libraries containing different components and place them.