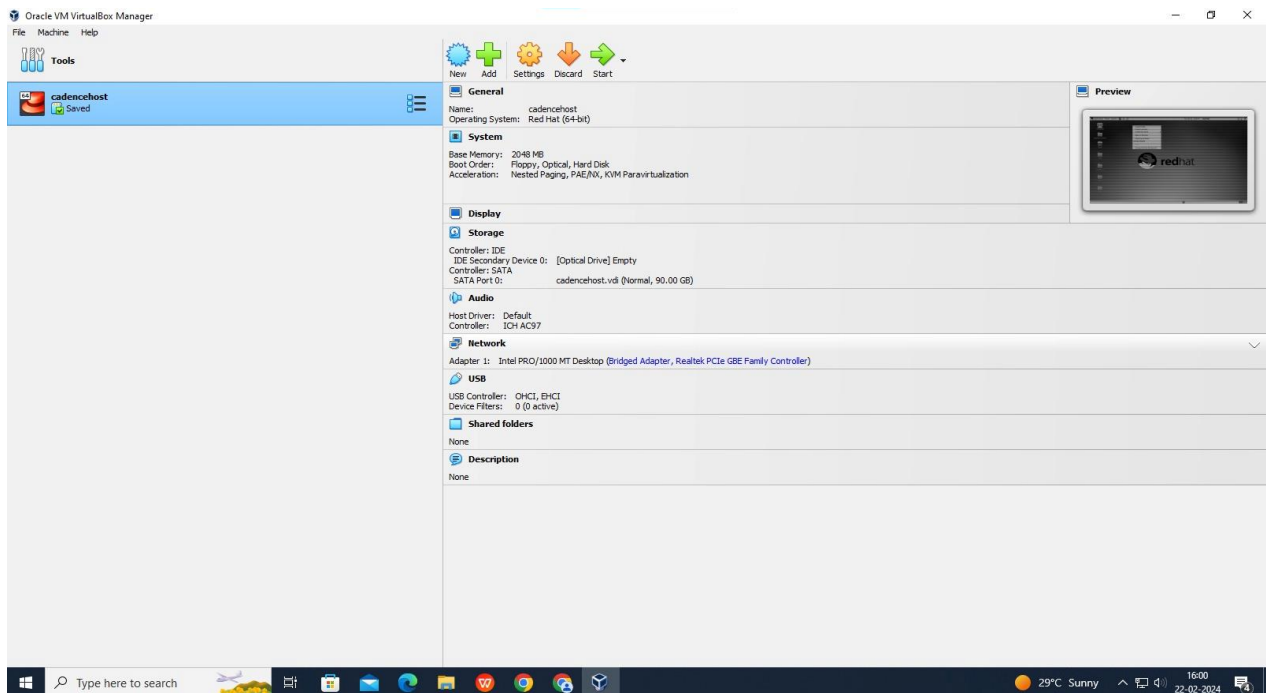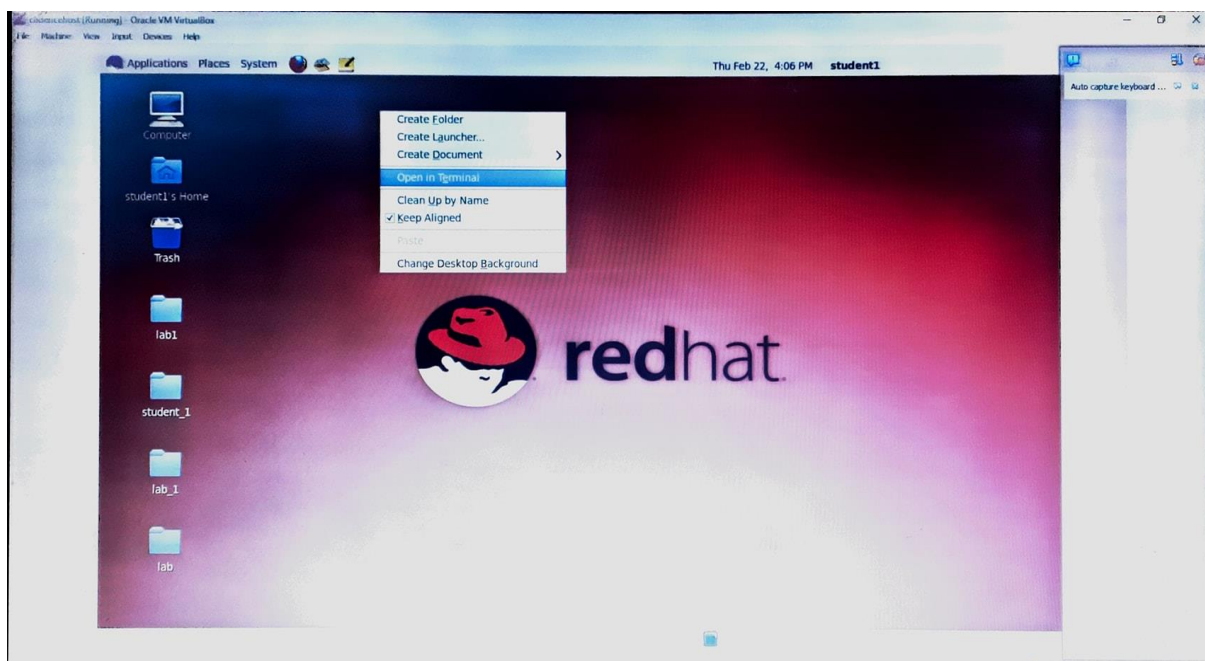# STEPS TO IMPLEMENT IV CHARACTERISTICS OF CMOS USING CADENCE TOOL

STEP-1:**LIBRARY CREATION**

1)Open oracle VM virtual box

2)Click on start

3)Right click on workspace, select **open in terminal**

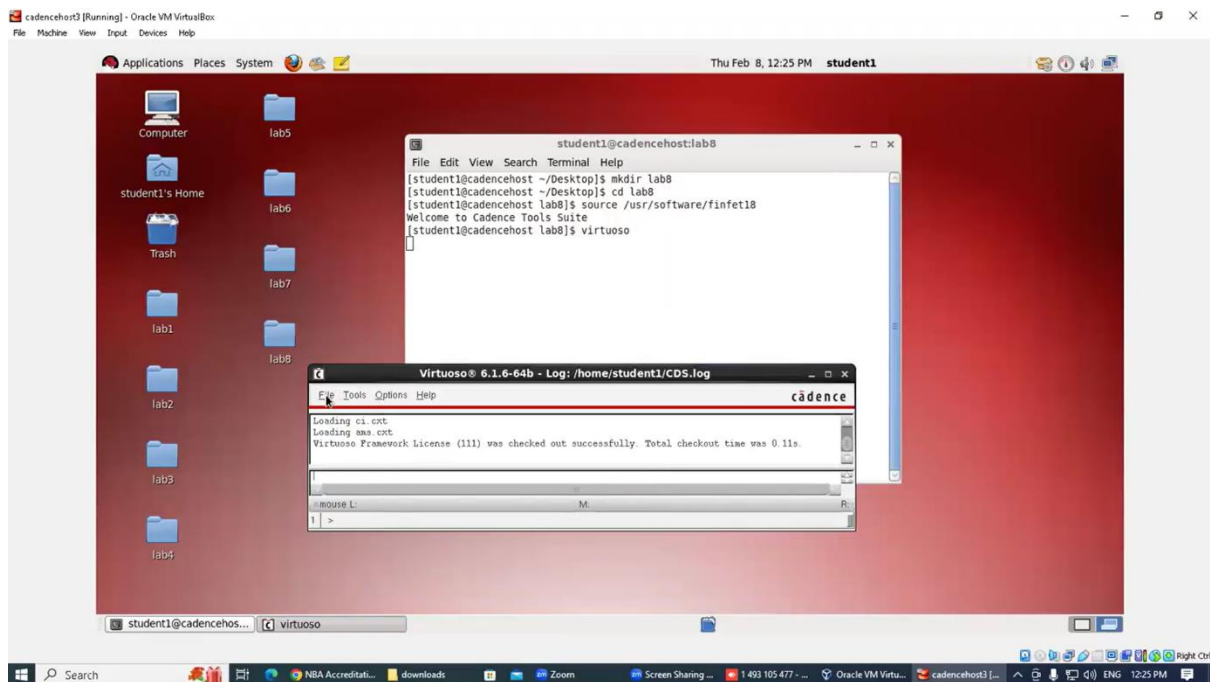4)Type the commands

mkdir <any name>   (ENTER)

cd <any name>   (ENTER)

source /usr/software/finfet18   (ENTER)

virtuoso &   (ENTER)

EXPLANATION:

mkdir: This command is used to create a new directory (folder) within the current directory.

cd: Short for "**change directory**," this command is used to navigate between directories. For example, cd folder_name would move you into the directory named "folder_name."

virtuoso: Virtuoso is a widely-used tool within Cadence for electronic design automation (EDA). It's primarily used for designing and simulating integrated circuits (ICs) and electronic systems. It includes various modules for schematic capture, layout editing, simulation, and more.
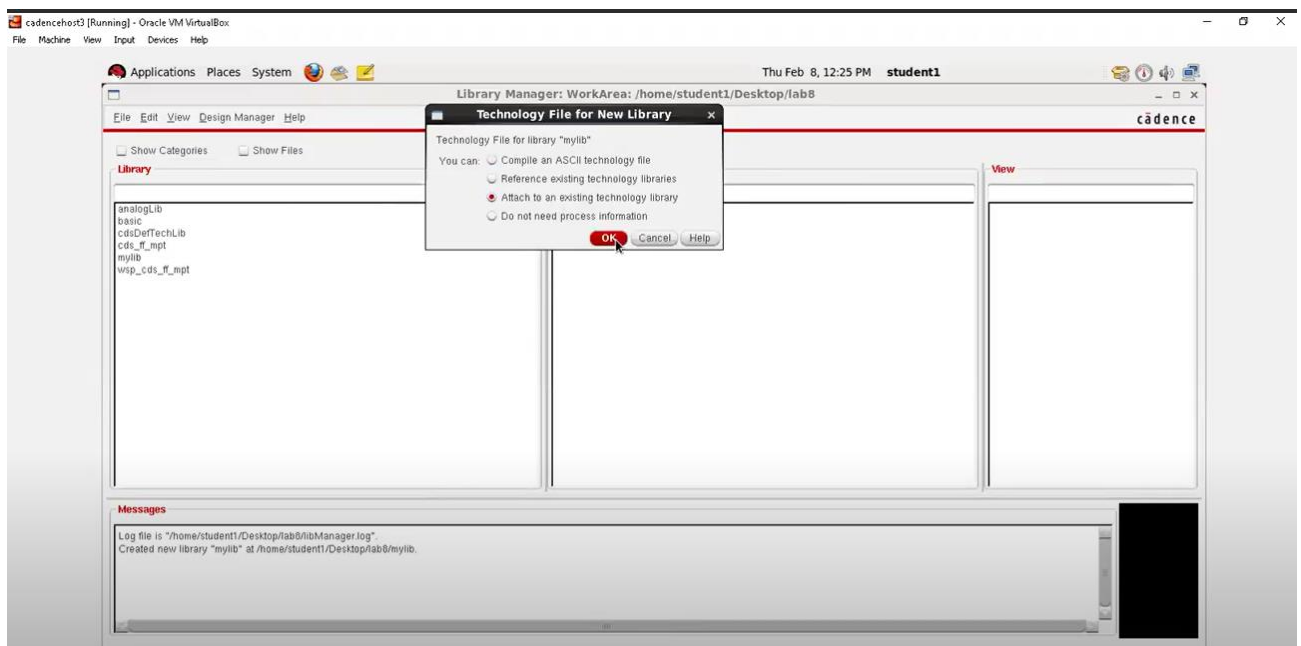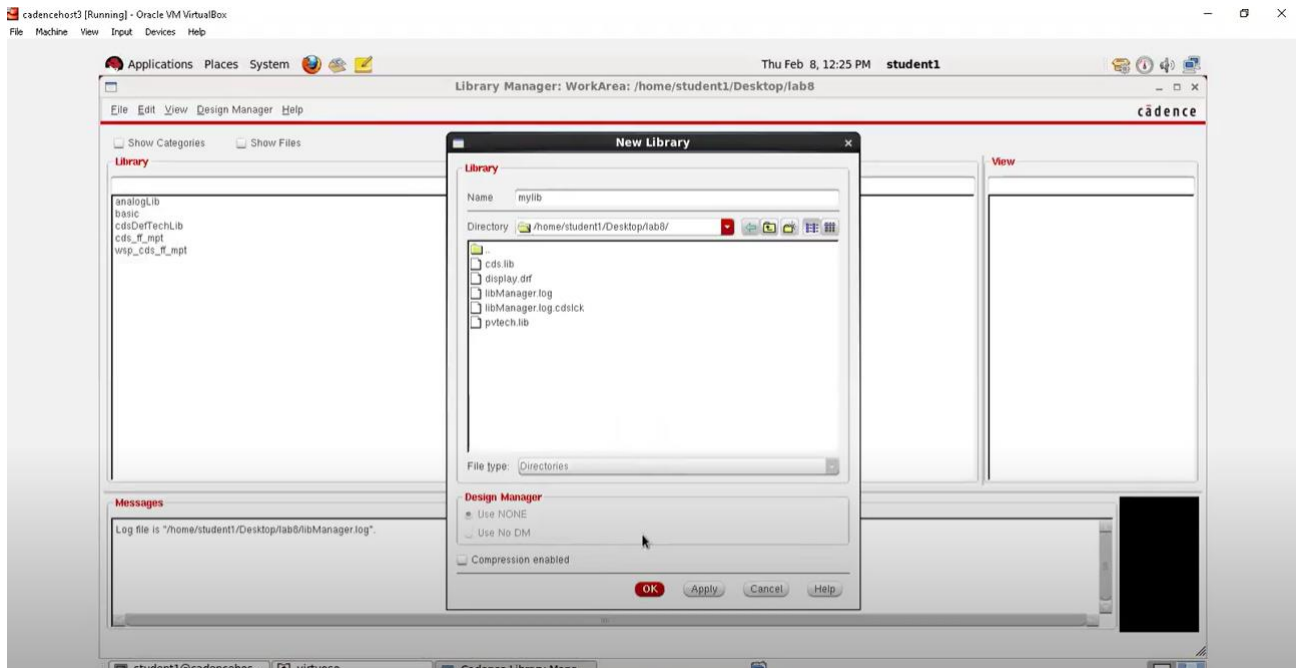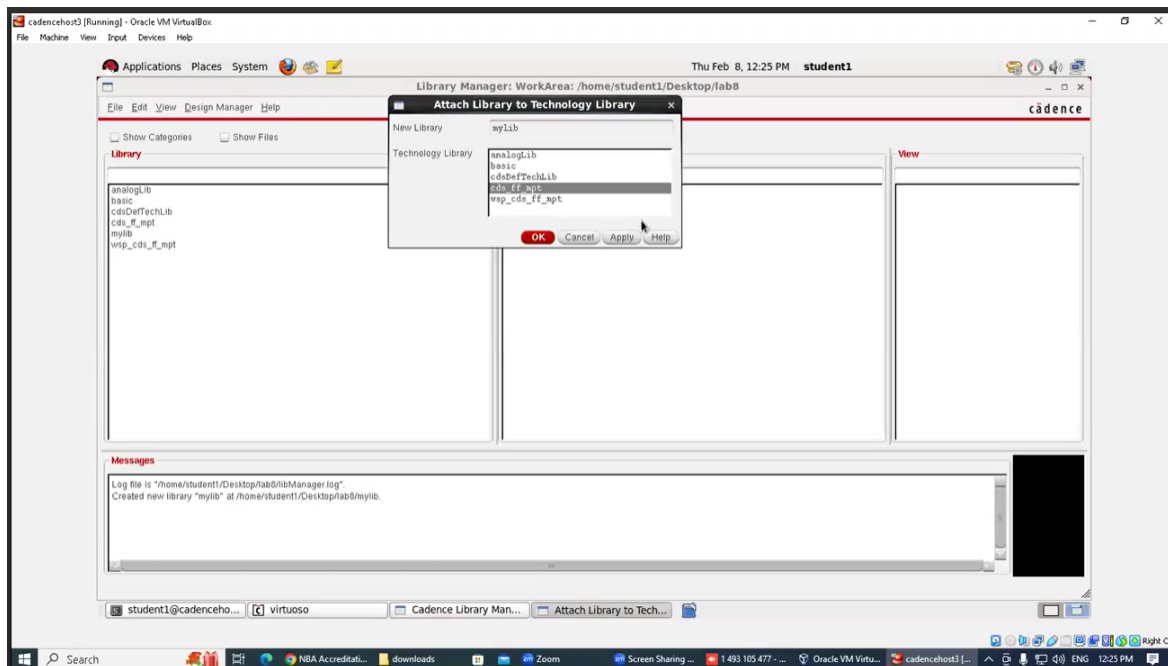
5)virtuso tab appears

6)In virtuoso tab

- Tools>Library Manager>File>New>Library>select Attach library to technology>Ok
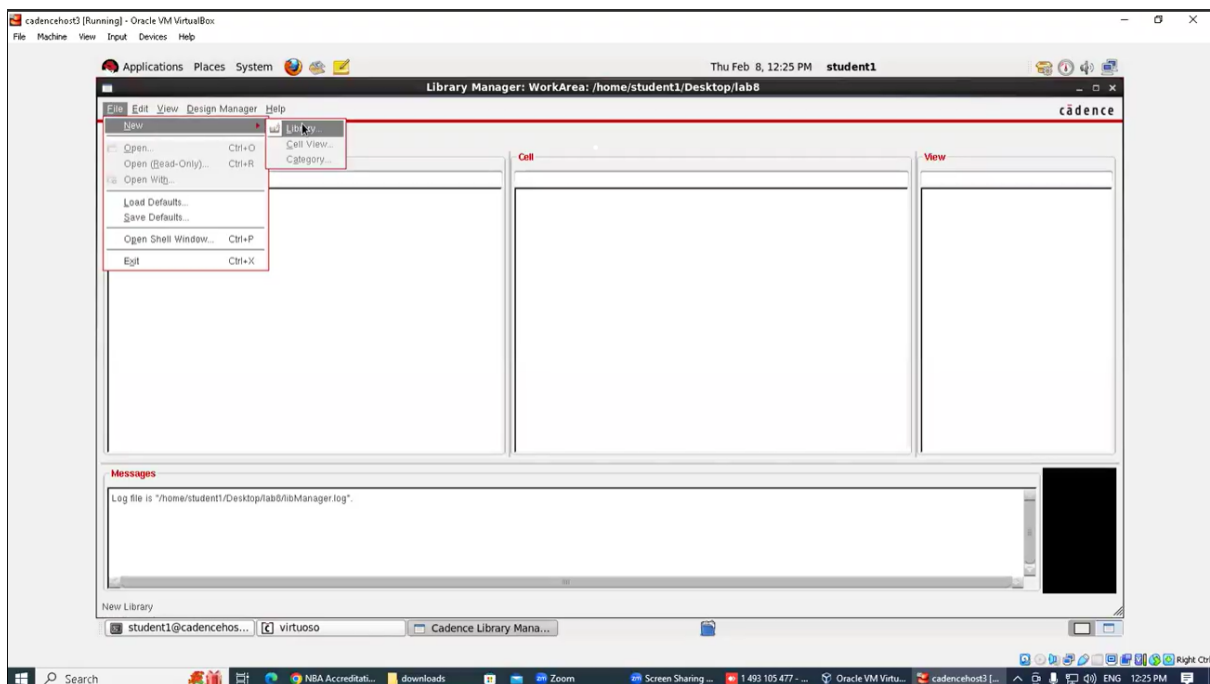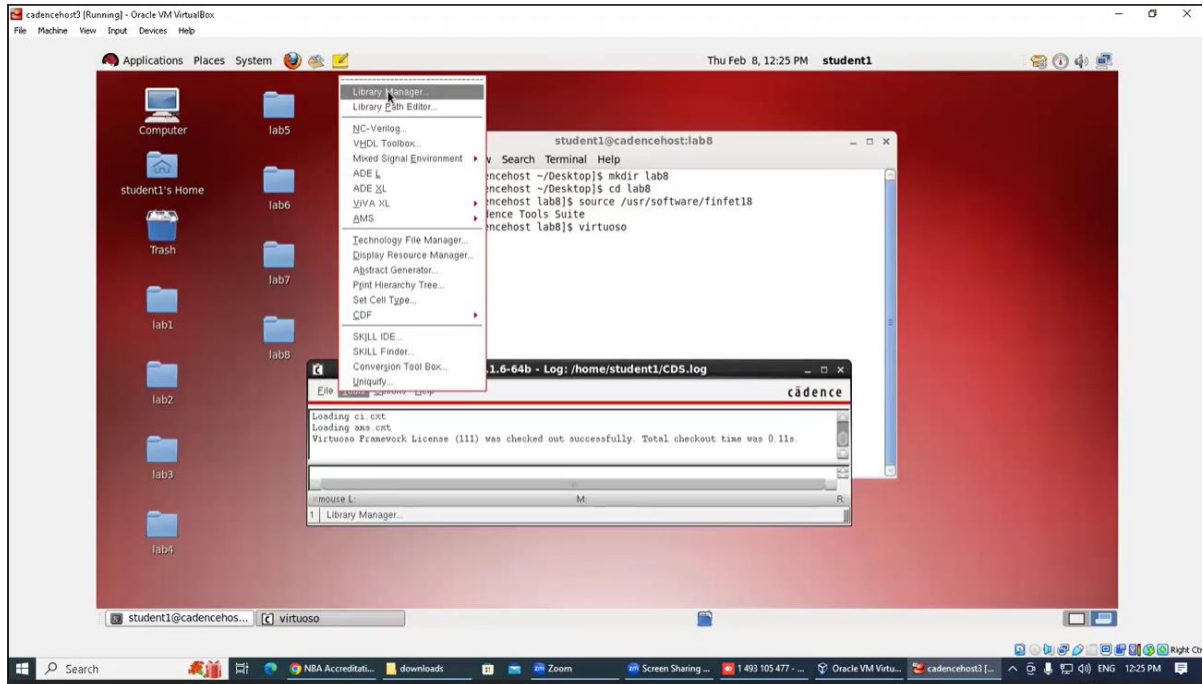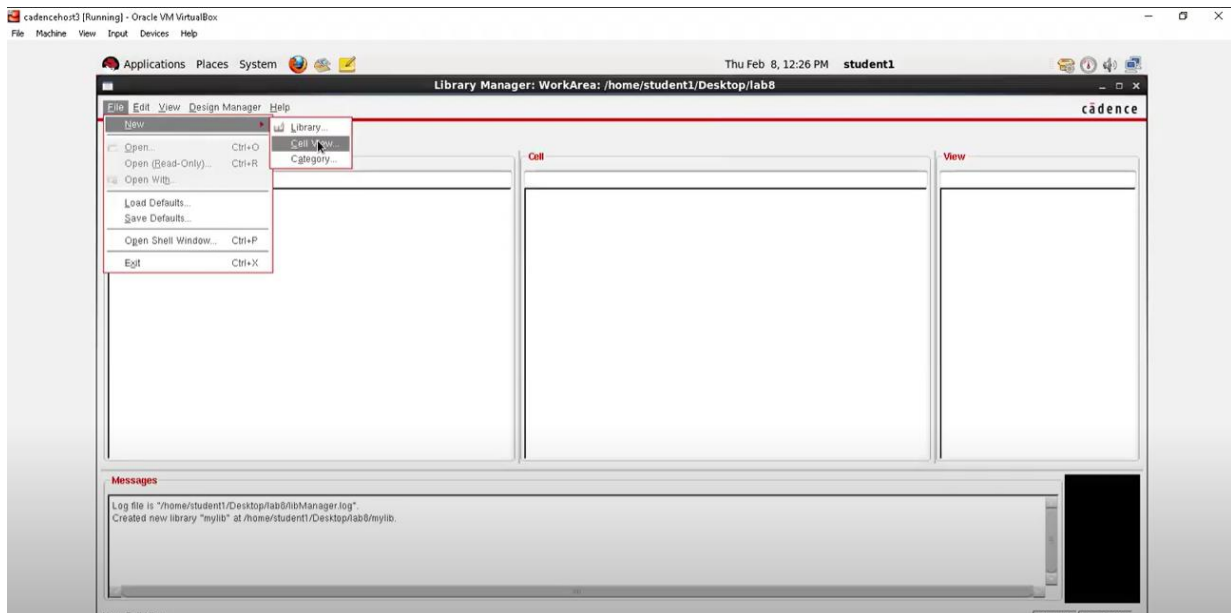
Give any name

Select Attach to an existing technology library> Select cds_ff_mpt

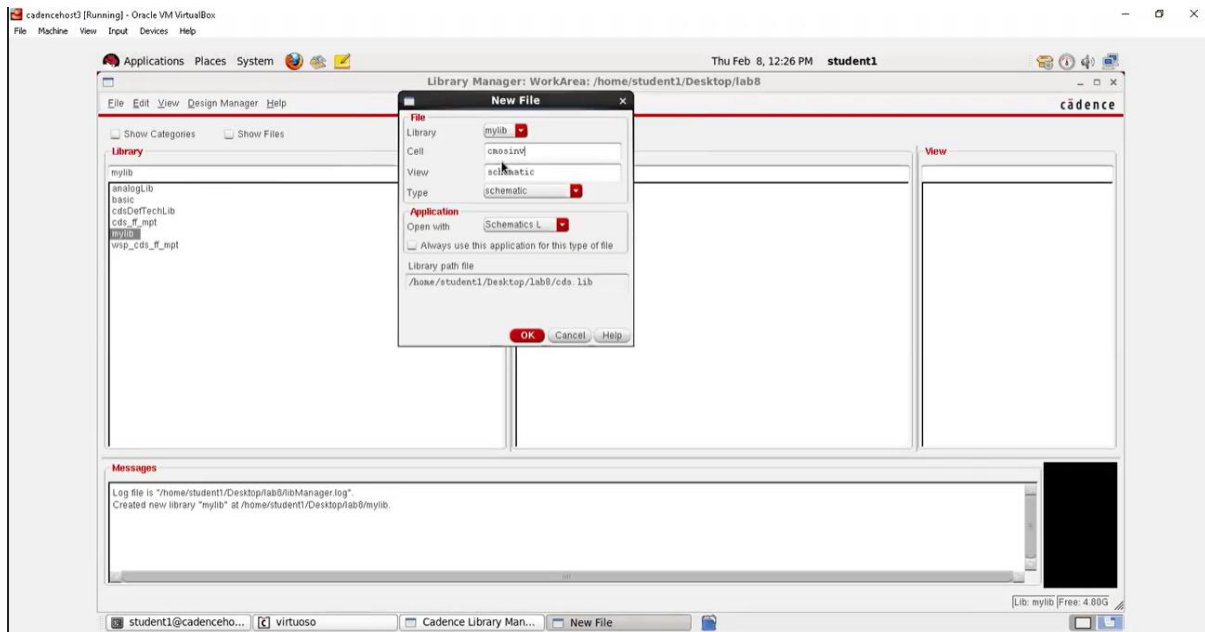## STEP-2:**LIBRARY MANAGING(SET UP CONNECTIONS AND ADD VALUES)**

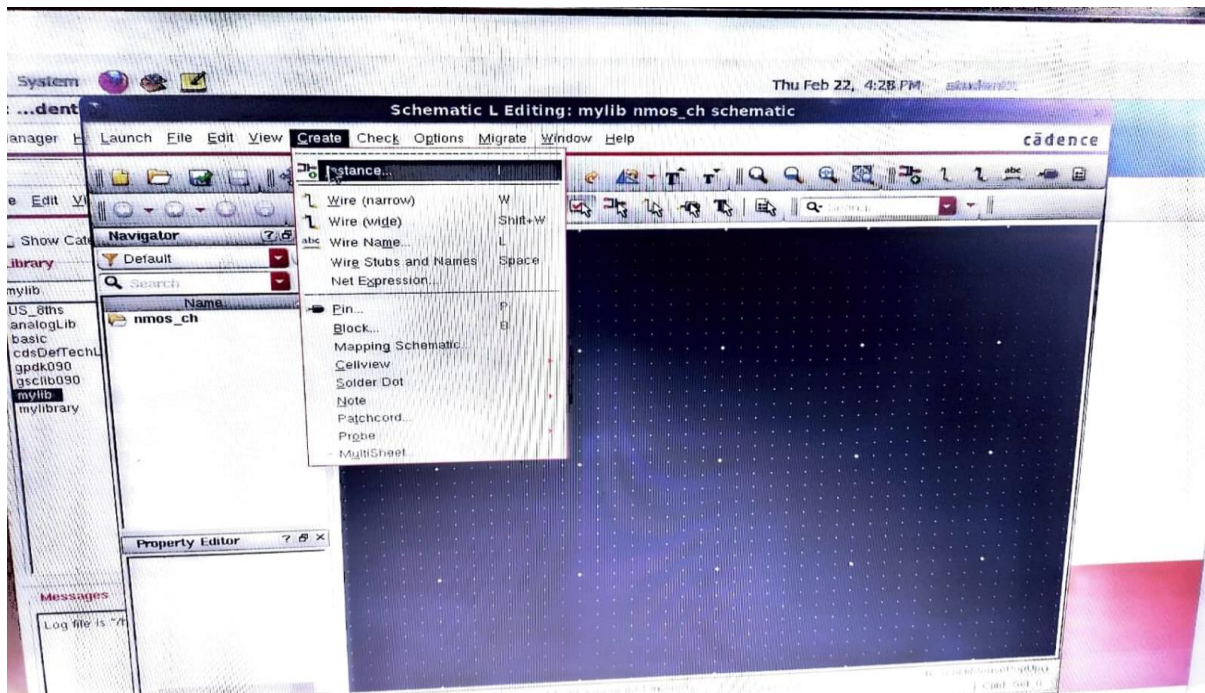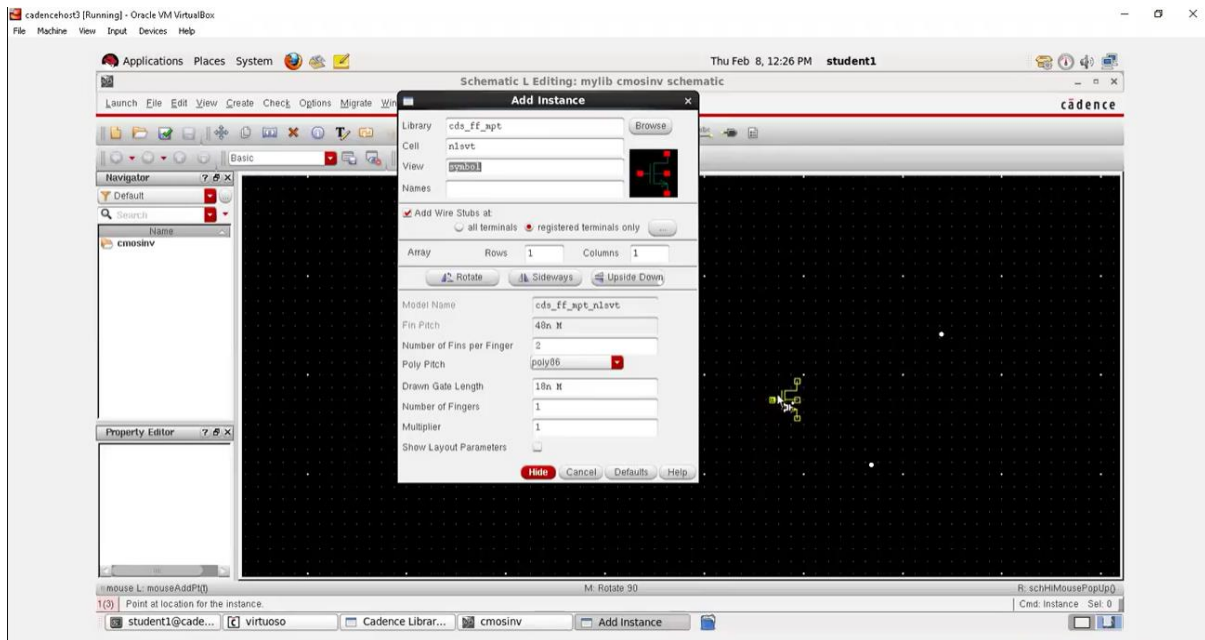7)In mylib

- File>New>cell view

Enter cell view:**cmosinv**

- Select OK

8)Create>Instance(shortcut-press **"I"**)>Browse

- Select the following and place it on the schematic Editing window each time.
- Select instance and give the following names.

| Library	Cell	View
cds_ff_mpt	nslvt	symbol
cds_ff_mpt	pslvt	symbol
analogLib	vdd(take 2 vdd's)	symbol
analogLib	gnd(take 2 grounds)	symbol
analogLib	Vdc(give DC voltage as 0.8)(take 2 vdc's)	symbol

9)Set up the connections as shown

Press **"W"** for wire to connect the circuit
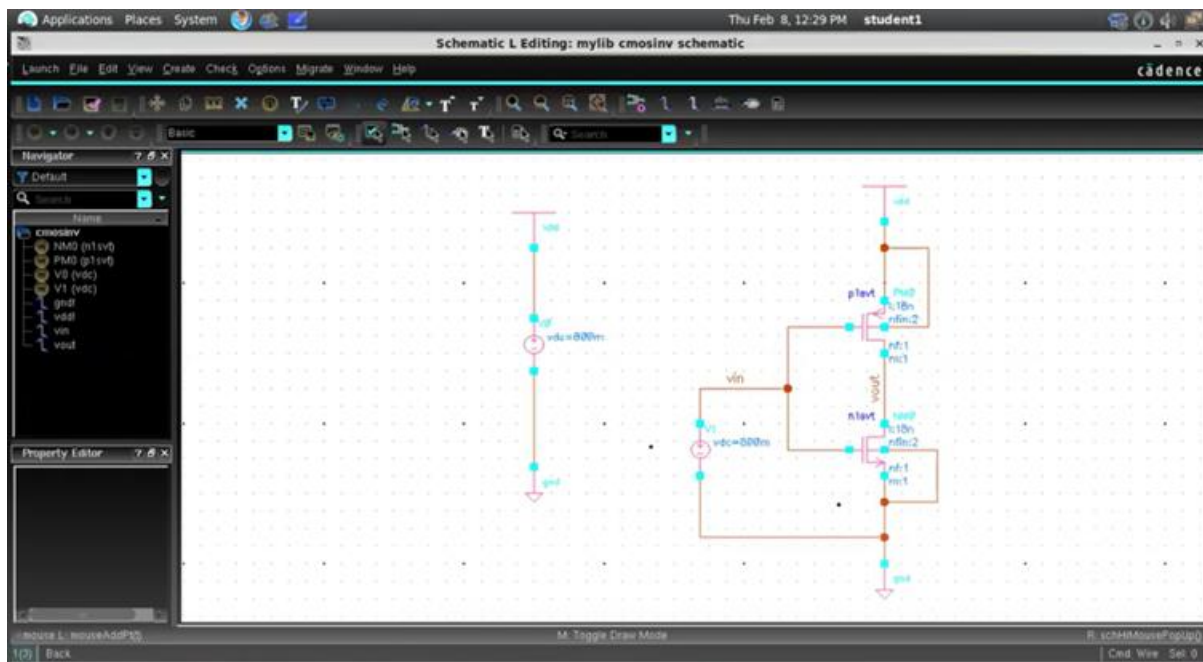
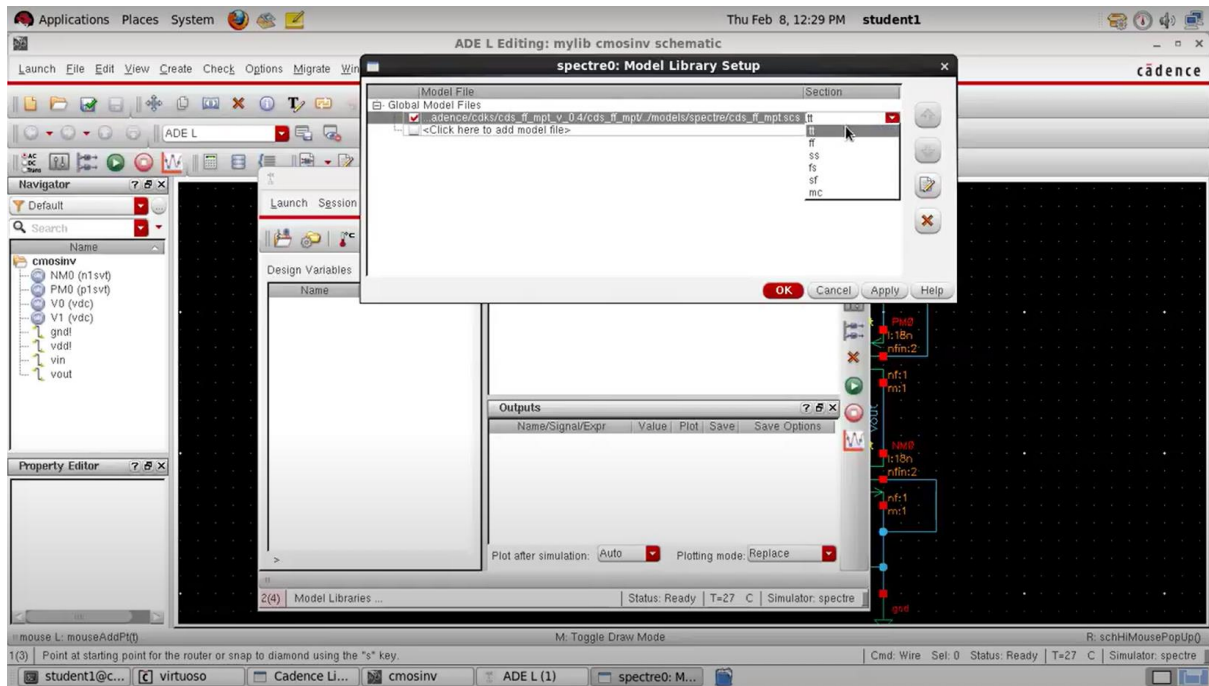## STEP-3: To observe output and characteristics

10) Go to launch(top left corner) and select ADE L
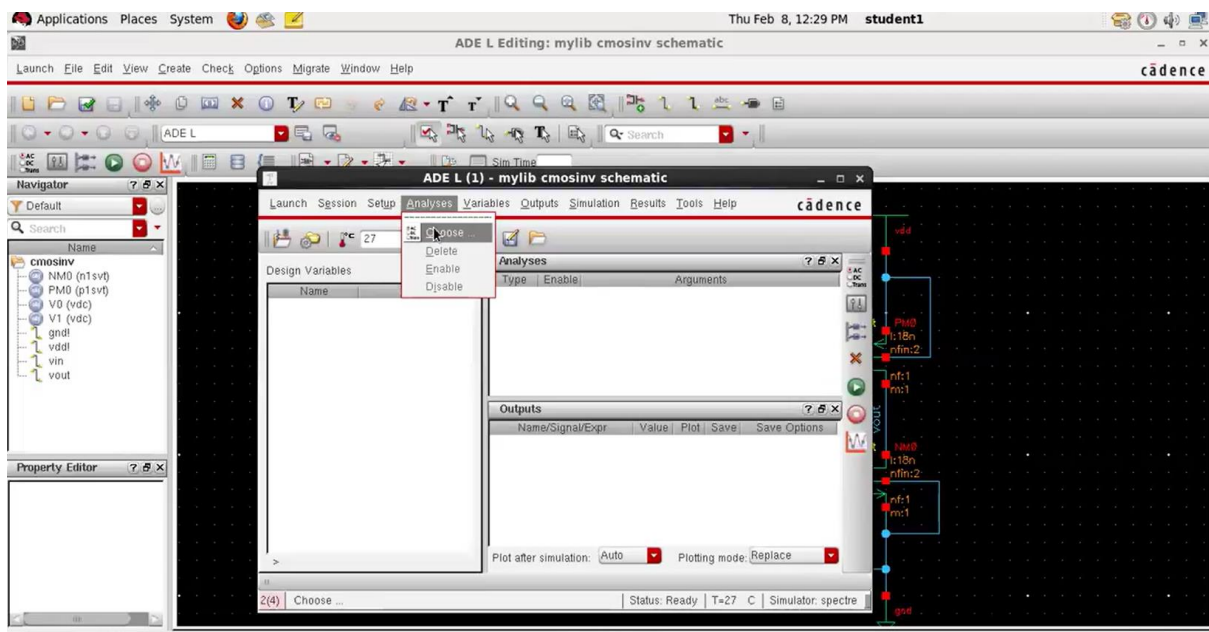
In ADE L window go to setup>model libraries
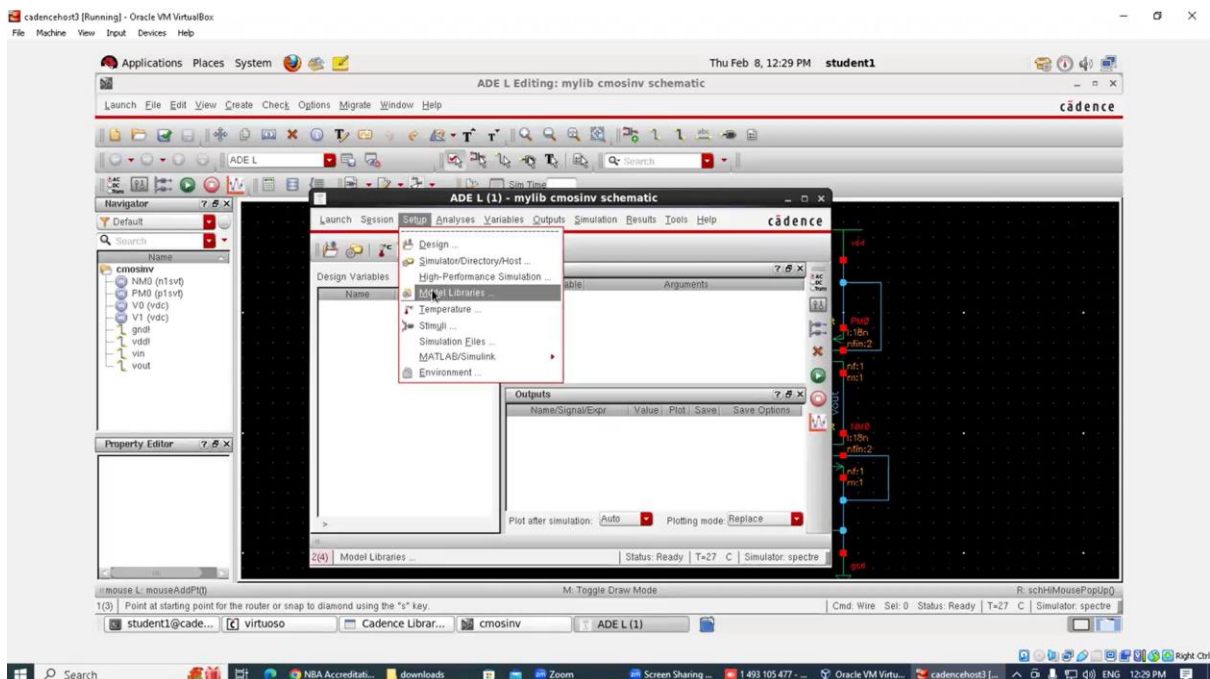
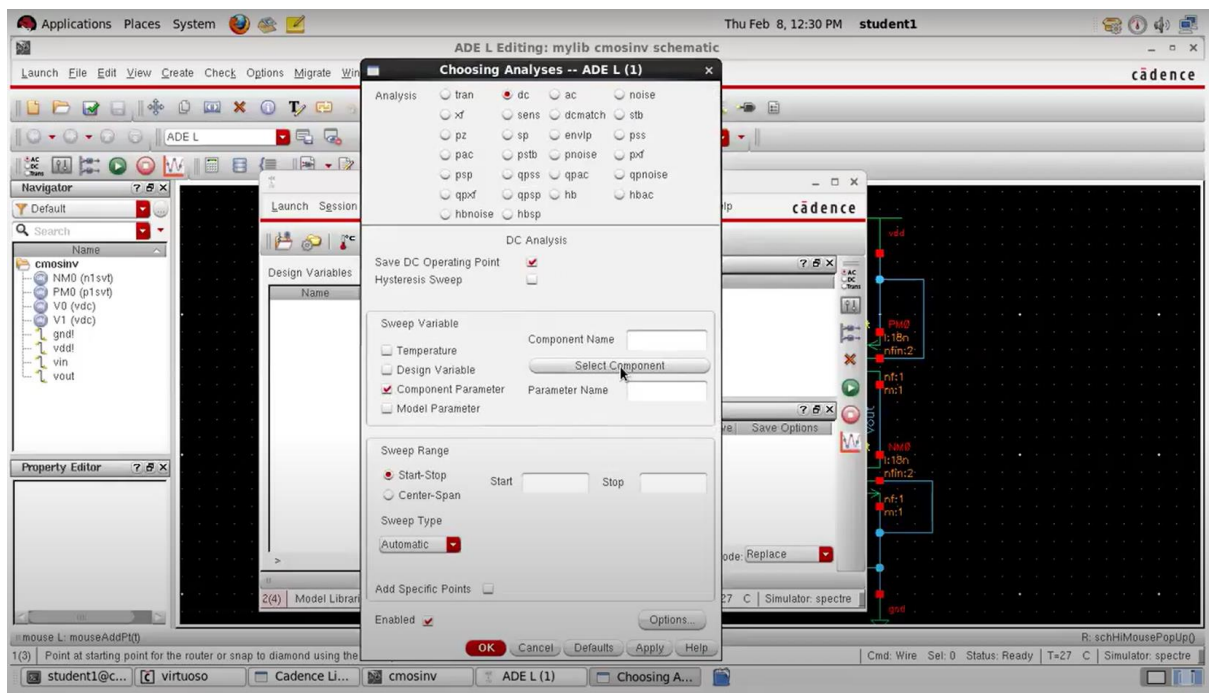**11)** In model library setup> check if the model file is "tt"
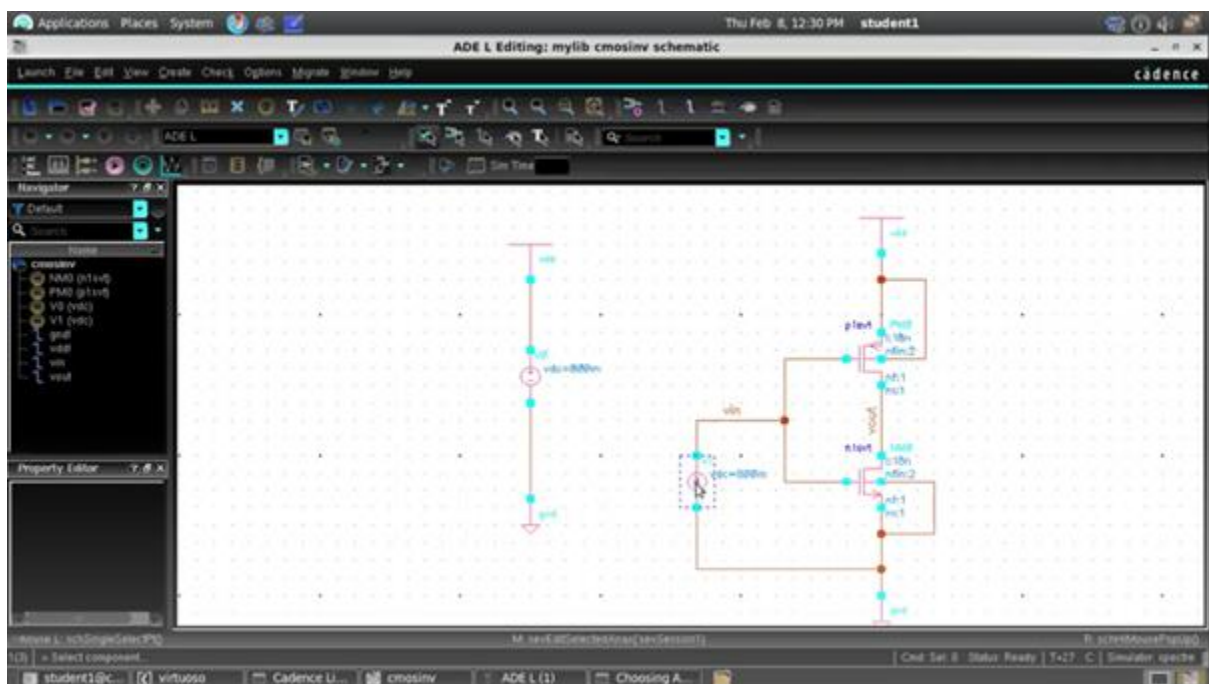

12) In ADE L window
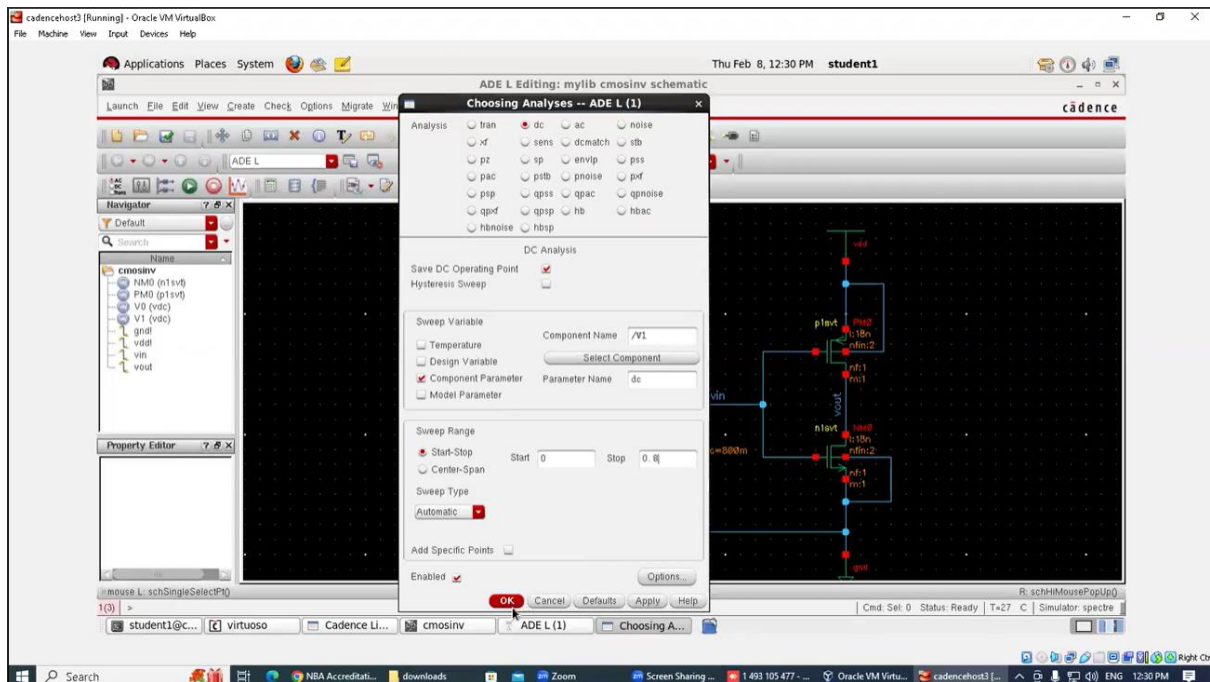
Go to analyses>choose

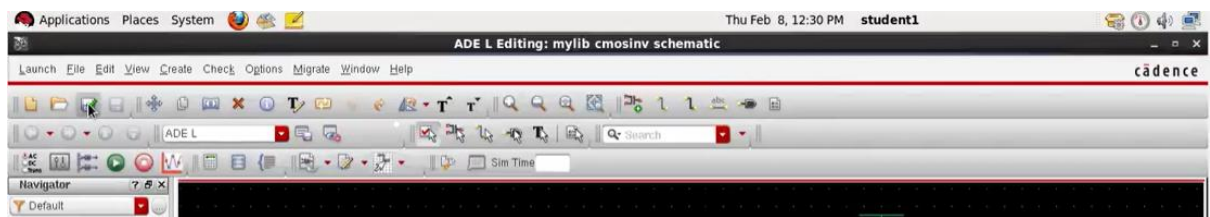## 13)Choose dc and component parameter>select component
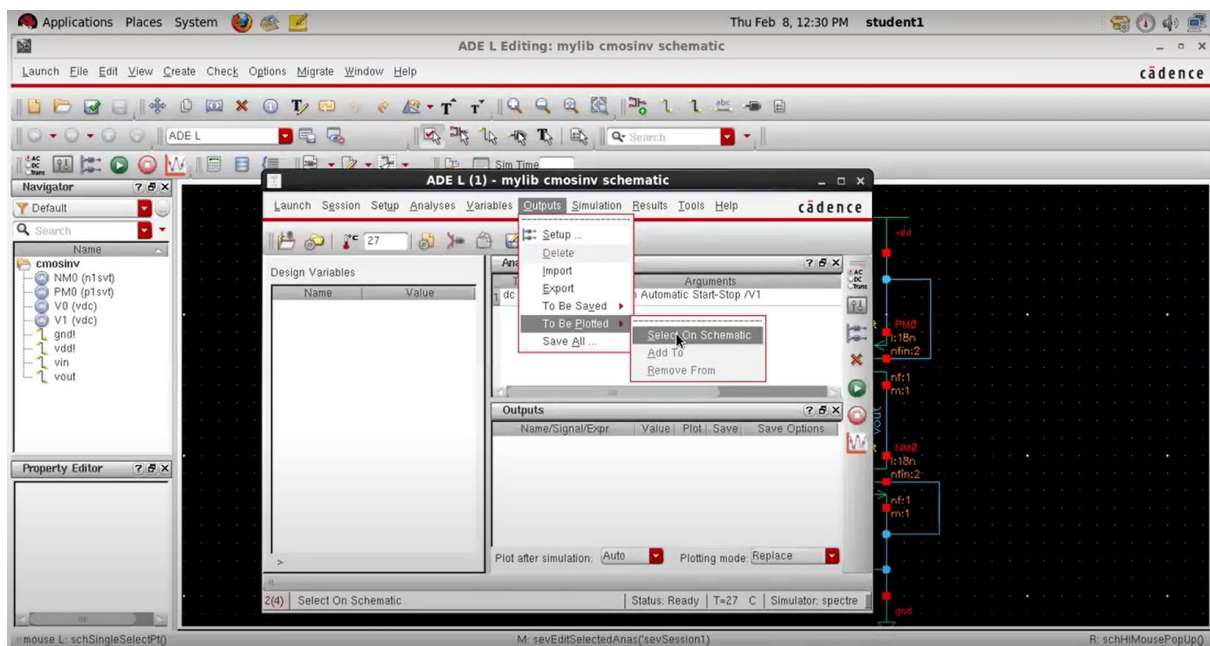

Select the vdc as shown below

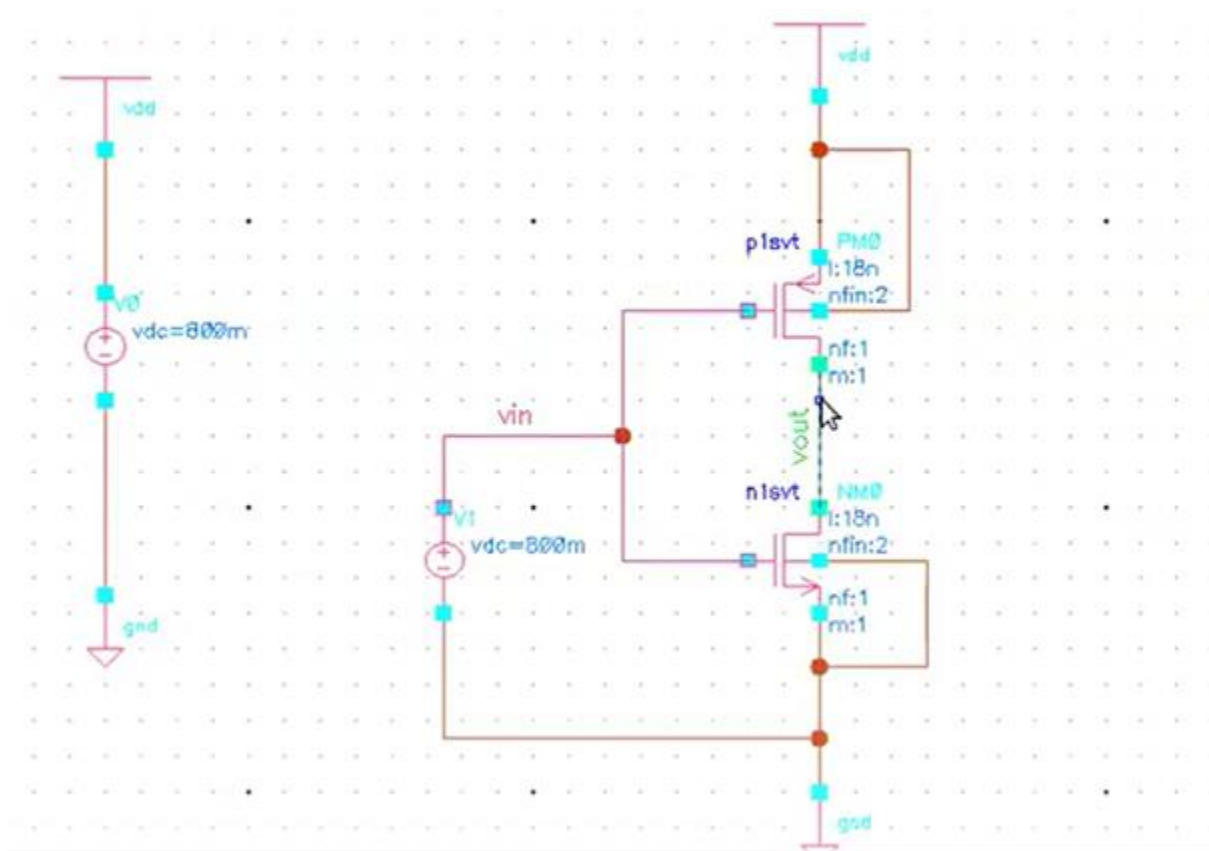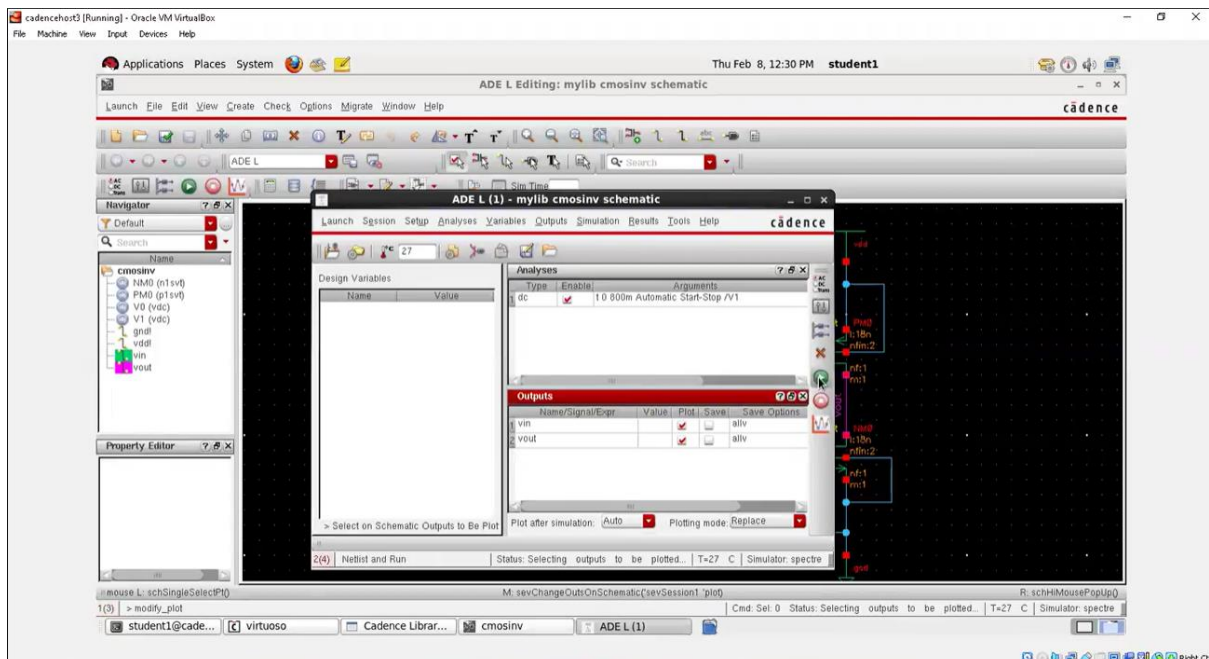In the same choosing analyses tab give start-stop values as 0 ; 0.8

Save

14)Go back to ADE L tab select output>To be plotted> Select on schematic

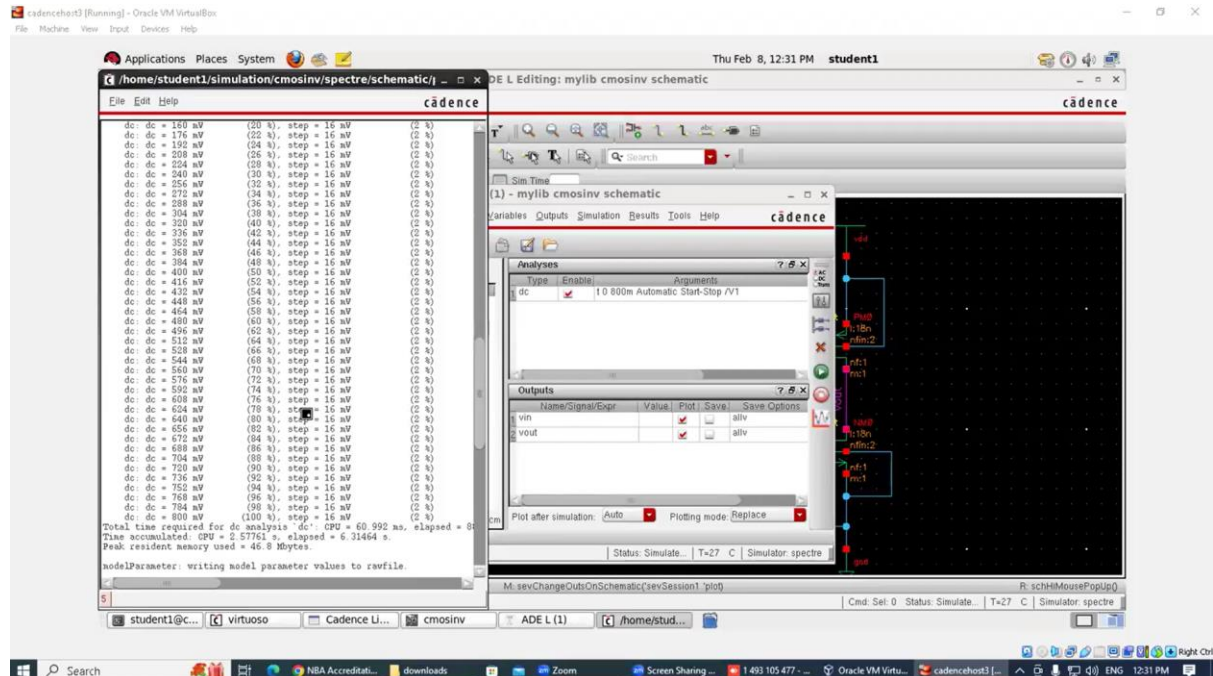Select from the schematic> select the line shown below

15)In ADE L tab > select run from right corner panel

16)After running the following output tab will appear

And a graph tab also appears >where output values can be observed by selecting lines of graph