STEPS TO IMPLEMENT STUDY OF OPERATING REGIONS OF A MOSFET 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/gpdk090	(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

- File>New>Library>mylib(give any name)>select Attach library to technology>Ok
- Select **gpdk090>Ok**

Again in Virtuoso tab

- Tools>Library Manager>mylib

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

7)In mylib

- File>New>cell view
- Enter cell view: **mynmos**
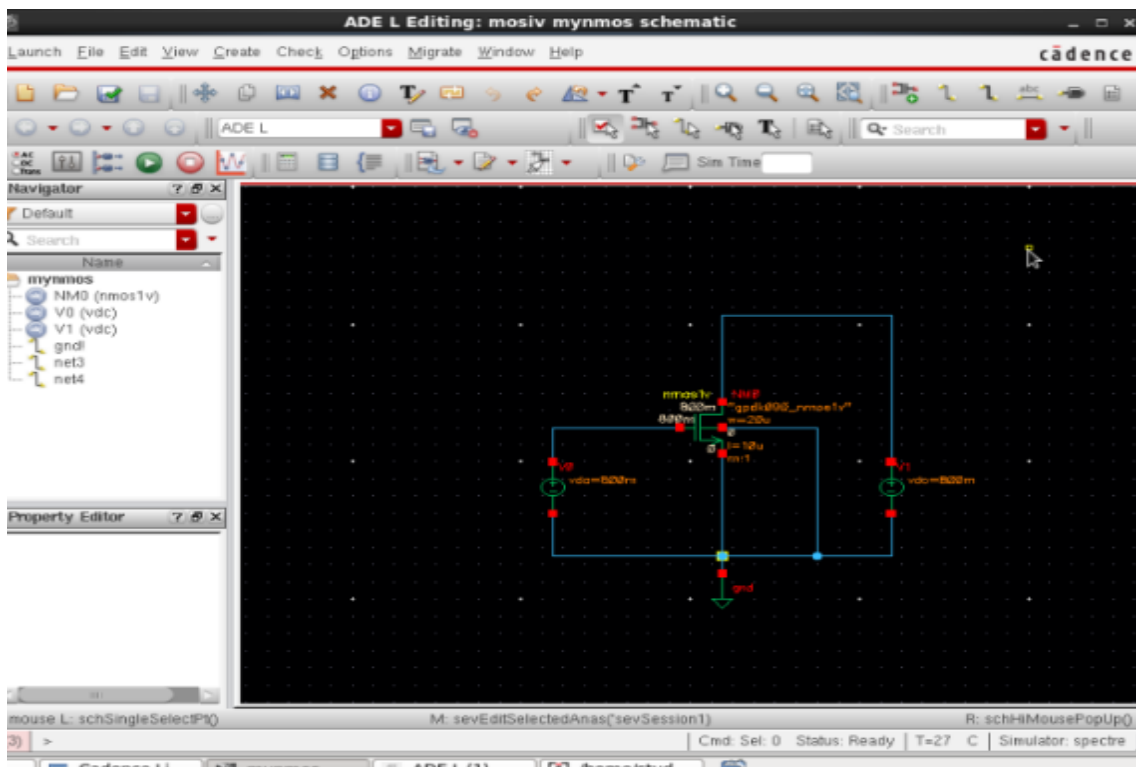- Select OK> A schematic tab appears

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

- Select the following and place it on the schematic Editing window each time.
- Select vdc twice.

Library	Cell	View
gpdk090	nmos1v (Length- 10u, total width-20u)	symbol
analogLib	vdc(2)	symbol
analogLib	gnd	symbol

9)Set up the connections as shown

Press **"W"** for wire to connect the circuit

10)Save (on the top left corner)

11)Launch> ADE L

12)In ADE L

Setup > Simulation/Directory> Host > And verify the simulator is spectre > click Ok
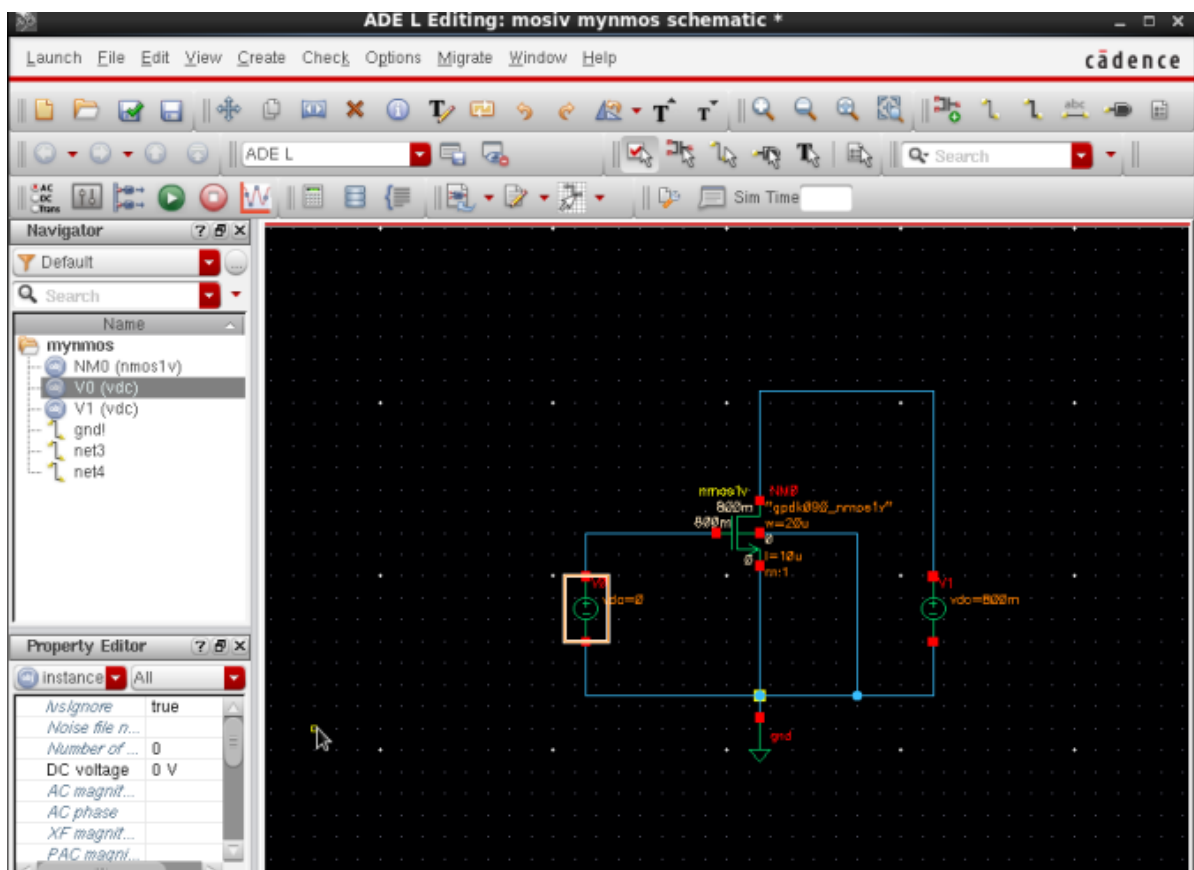
Setup > Model libraries > check if the model file is correct

Analyses > Choose > Choose **DC;** In DC analysis (same tab) choose Save DC Operating point and click OK

13) On ADE L window select Simulation> Netlist and Run (A tab appears indicating the simulation is successful)

On Schematic tab > select V0(vdc) >press Q(shortcut for edit object properties)
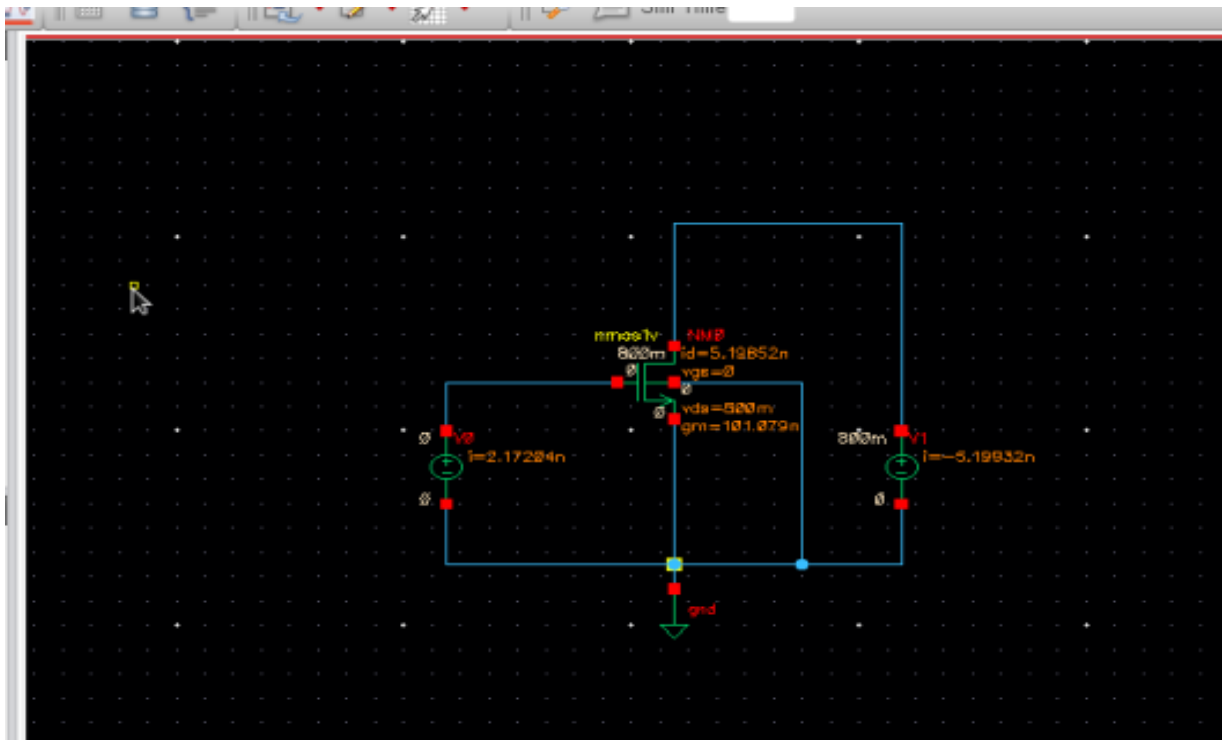
Give DC voltage as 0

14) Run the simulation by clicking on run in Schematic tab

15)Again, in ADE L tab select Results> Annotate> DC node voltage

Then again Annotate> DC Operating points

(You can see all the values are visible on schematic)

16) ADE L tab > Results > Print >DC Operating Points (you can select the component for which you want to see operating points - A result display window appears and sows the values)

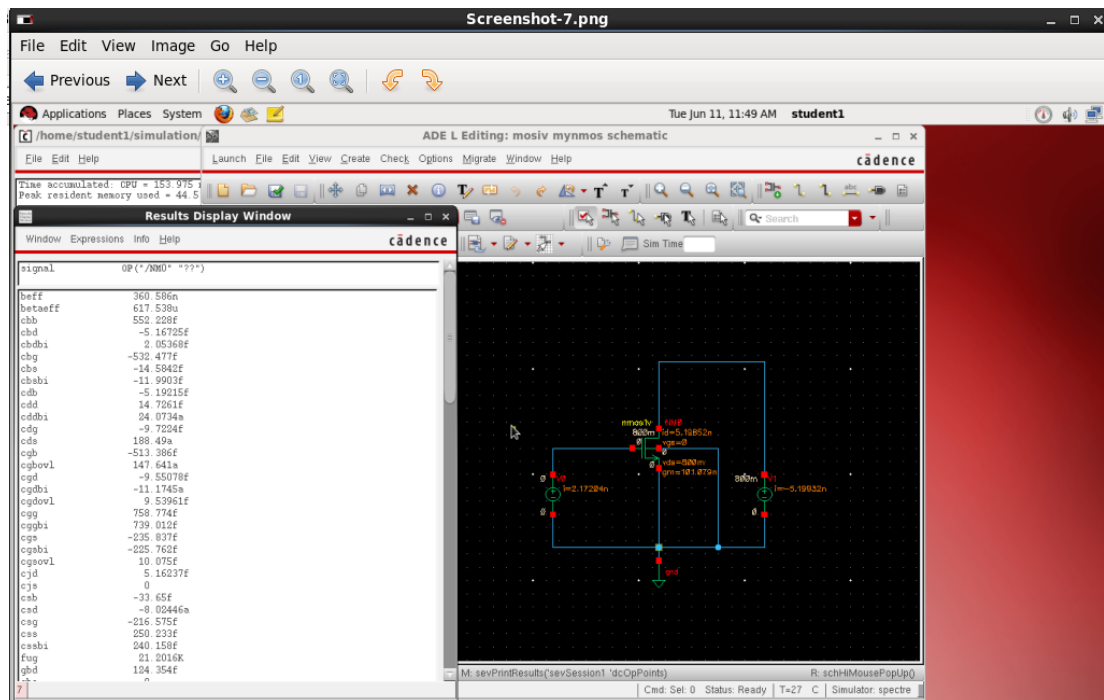17) Select the below shown component > press Q > AC magnitude as 0.8

(Similarly the other vdc apply 0.08) and SAVE > (Netlist and run)
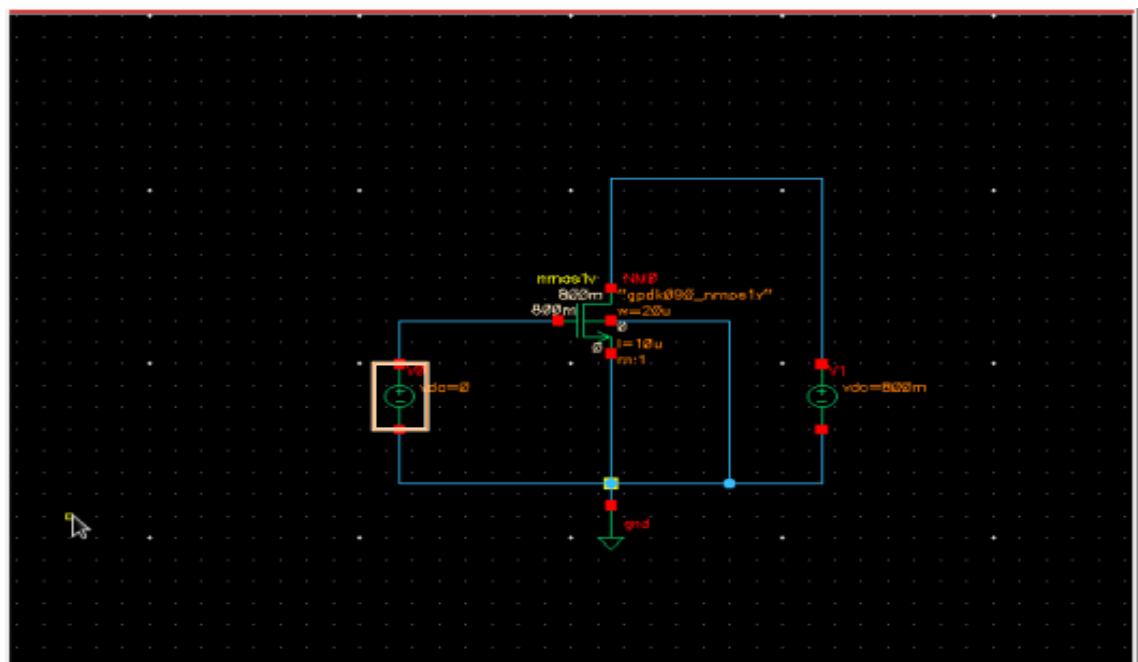
18) ADE L tab > Results > Print >DC Operating Points > select mosfet

19) This time select the other vdc and give AC magnitude as 0.8 V

ADE L tab > Results > Print >DC Operating Points

The result window is displayed