STEPS TO IMPLEMENT SCHEMATIC AND DC ANALYSIS OF A RESISTIVE VOLTAGE DIVIDER 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: **rvdivider**
- Select OK> A schematic tab appears

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

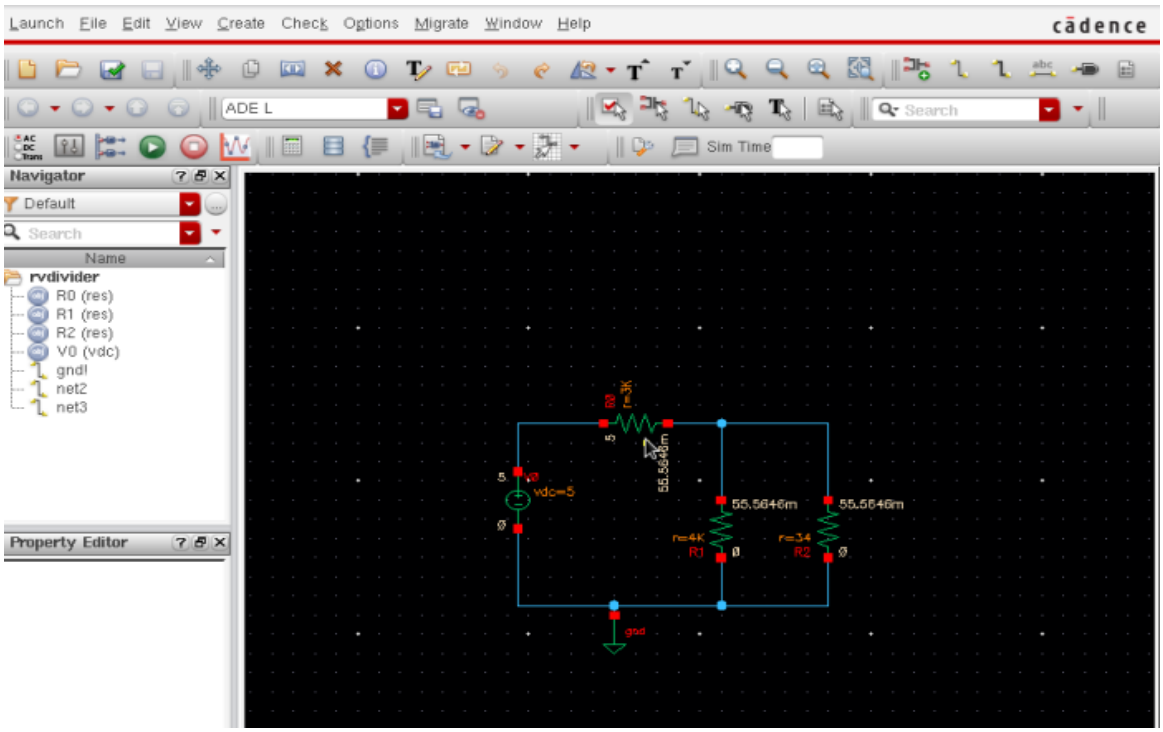
- Select the following and place it on the schematic Editing window each time.
- Select vdc twice.

Library	Cell	View
analogLib	vdc	symbol
analogLib	res(3 resistors)	symbol
analogLib	gnd	symbol

Resistors values: 3K,4K,4K

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

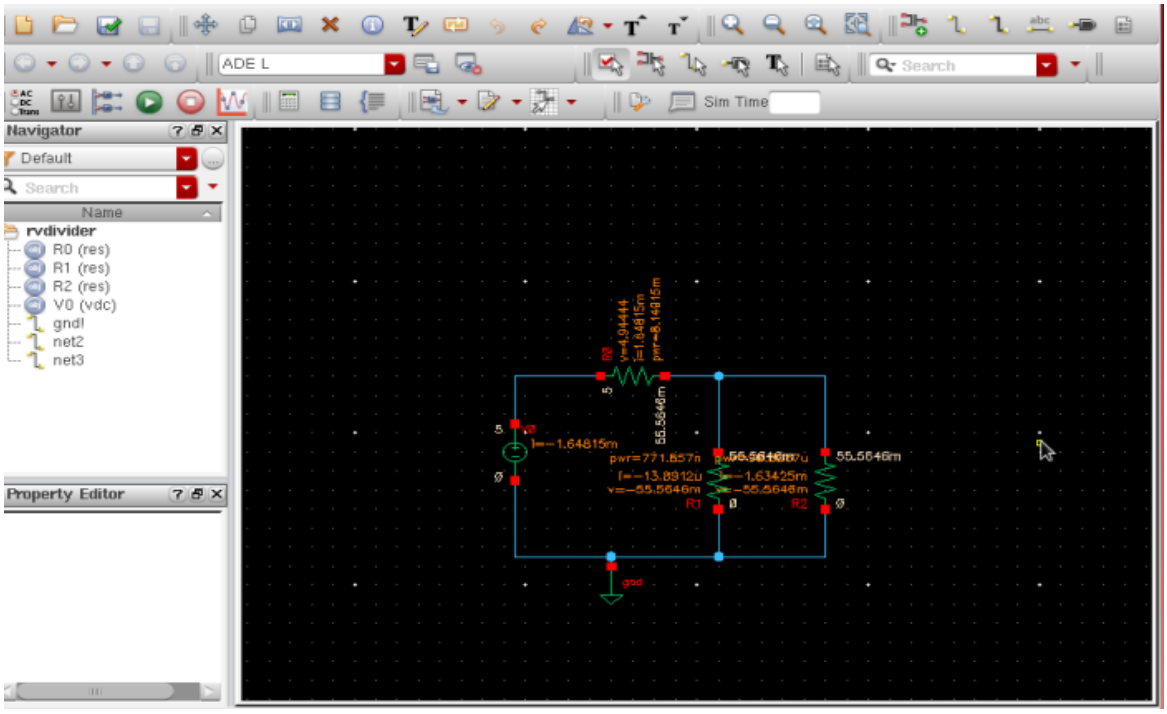
12)In ADE L right corner

Analyses > Right corner 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)

14)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)

15)Go to ADE L tab > Analyses > Choose **DC;** In DC analysis (same tab) choose Save DC Operating point and then in same tab click on Component parameter > Select Component

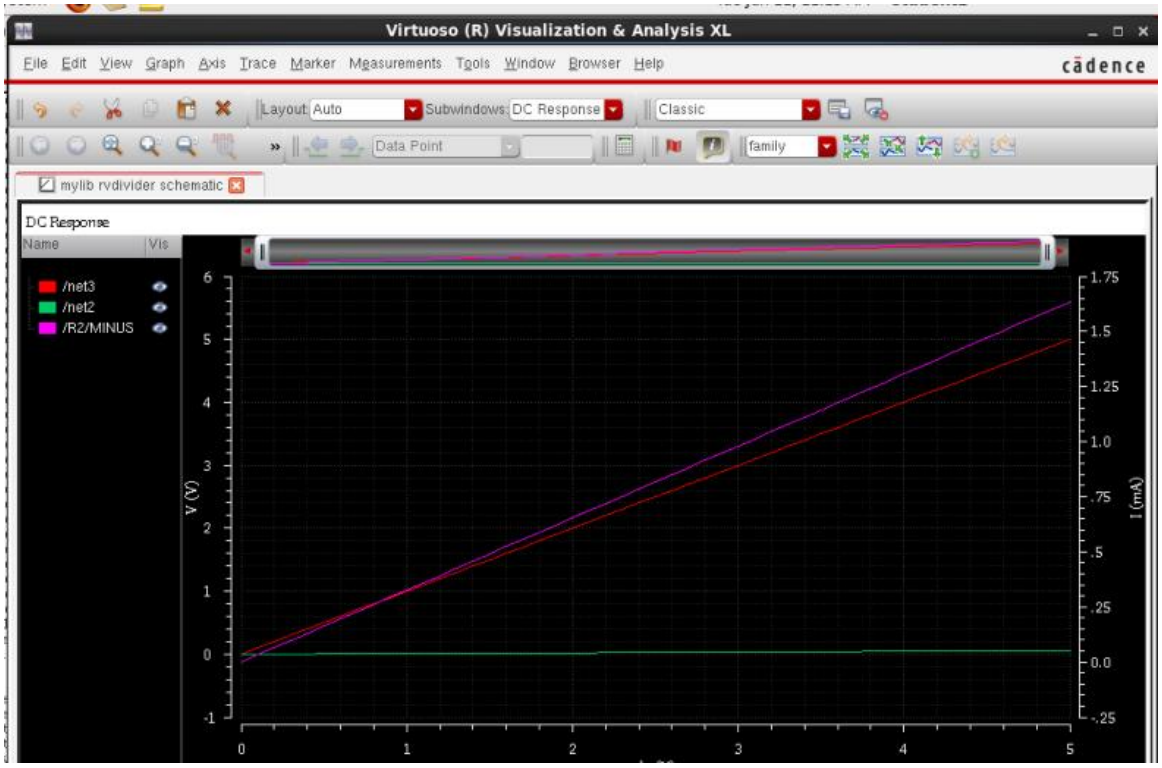
In schematic click on V0- select dc> OK

Start at 0 and stop at 5> OK

Select the nodes that needs to be plotted by opening ADE L tab> Output> To be plotted> Select on schematic nodes(net 2 and net 3 and R2)

Select **Netlist and Run**(Right Corner side in ADE L Tab)

- Graph displays



On ADE L window > Session > Save state(to save the schematic)