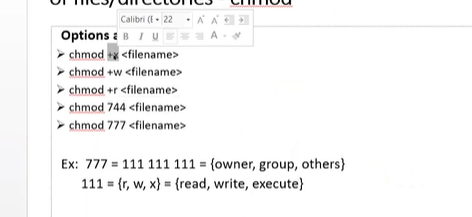
**Linux Basics:**

* Command to view all the folders and files in a specific location: “ ls ”
* Command to view all the last state or changes in the OS: “ ll ”
* Command to view all the last state or changes in the OS: “ ls -ltr ”
* Command to enter into a file or folder location: “ cd **space** file name ”
* Command to go back to the previous file or folder location: “ cd **space** .. ”
* To give permissions: 
* Command to open a specific file: “ gvim **space** **filename** ”
* In GVIM, when you have opened up a file, there are 2 modes:

(a) Command mode (Thick blinking bar over the character)

(b) Typing mode (regular line blinking)

This change can be made by clicking the insert button on the keyboard

* To save the file in the GVIM, use “ :w ” in the command mode
* To exit the file in the GVIM, use “ :q ” in the command mode
* To search for any character/word in the GVIM, use “ /**searchword** ” in the command mode
* To delete a line in the GVIM, use “ dd ” in the command mode when the blinker is set on that line that has to be deleted.
* Command to copy a file: “ cp **space filename** ”
* Command to rename a file: “ mv **space existing\_filename space renamed\_filename** ”
* Command to copy a directory: “ cp **space -r space existing\_filename space to\_filename**”
* Command to create a directory: “ mkdir **space folder\_name** ”
* Command to view the present working directory: “ pwd”
* Command to delete a file: “ rm **space filename** ”
* Command to delete a folder: “ rm **space -r space folder\_name** ”
* Command to view the first line in a file: “ head **space filename**”
* Command to view the last line in a file: “ tail **space filename**”
* Command to view the last few line in a file: “ tail **space -no\_of\_lines space filename**”
* Command to open a file or run a command and then do something more in the terminal: “ **command space** &”

**Cadence Virtuoso Tool Startup:**

* Go to /project/amsworkspace/amslab68\_workspace/virtuoso\_wa
* Command: “ source bashrc ”
* Command: “ virtuoso & ”
* Open the Command Interface and create a new library
* Give a name to the library (eg. ACD\_June23\_batch)
* Select the “attach to an existing technology” node option
* Select the gpdk 90nm technology node
* Select the cell
* Add a new cell by going to the files/new/cell option
* Give a name to the cell (eg. test\_schematic) and click “OK” keeping all other options as default.
* After clicking on the “OK” button, the schematic will open
* To get an instance, click the “I” button on the keyboard which will open the instance tab
* Search the required component in the cell option and click the “symbol” option.
* Now there will be a component floating in the matrix
* Press “ESC” to deselect the component
* Select multiple by holding left click on the mouse and delete using the delete option
* Press F to fit (zoom) the component
* Press W for wire
* To copy the component click “C” and then tap the component that has to be copied.
* To move the components click “M” on the keyboard and then tap on the component
* Tap on the component and Press “Q” to change the parameters
* To open the simulation tab, go to the “Launch” option on the top left and select the “ADE L” option
* Right-click on the “design variables window” and select the “copy from cell view” option
* Click on the AC/DC/Trans analyses option on the top right
* To save the parameters or state, go to the “Sessions” option and then click the “save state” option
* Save the file in the required directory (eg. ACD\_June23\_batch/test\_schematic\_/spectre/saved\_state1
* To change the simulator, go to the setup option and then select the simulator option
* For high-performance simulations go to high-performance simulations in the setup option then APS (uses multiple licenses and multi-threading)
* Temperature can also be changed using the temp C option
* To save the outputs, go to outputs and select the specific saving only the required outputs. All can also be saved but takes more time
* To perform a simulation, go to the “Simulation” option and then click the “Netlist and Run” option
* The simulation will begin and then it will print the outputs in another tab
* To set the initial conditions, select the net and set net with a initial condition and then assign the value.