<u>Synopsys waveview analyzer</u>

Once the SPICE netlist (*.sp) file is successfully run in HSPICE (job concluded), it creates out files (both text and graph data).

Let us assume, the HSPICE input file name is test-inverter.sp and both DC and TRANSIENT characteristics are required to be performed.

Then in command prompt, we need to write

>: hspice test-inverter.sp>test-inverter.lis

Once job is concluded, write in command prompt,

>: wv

This is for waveview analyzer.

Once the waveview window comes, from top menu

File> Import waveform files

<u>**Select waveform files (*.sw0 for DC, *.tr0 for transient etc)**</u>

Then click on + icon left to the transient output file (example, + D0:test-inverter.tr0); After that click on **top level** ; a widow should come up below with all the possible node voltages and currents.

Finally, you can double click on the desired output voltage or current to view the waveform in the right panel.

There are many functionalities in the wave view; surf through it to learn more.