4) EDA Tools

Electronic Design automation (EDA), also referred to as electronic computer aided design (ECAD), is a category of software tools for designing electronic systems such as integrated circuits and printed circuit boards.

OrCAD EDA tools

OrCAD is a software tool for electronic design automation which is used to design electronic schematics and electronic prints for manufacturing printed circuit boards. It consists of a schematic editor (capture) a circuit simulator (pspice) and a PCB designer.

OrCAD capture :- It is a schematic capture application containing component information system to link the data of component package footprint (or simulation behaviour) with schematic circuit symbol. It is used to export netlist data to the simulator (orCADEE) and hardware description of the circuit schematic to verilog or VHDL. It also exports netlists to designs of circuit board.

OrCAD EE PSICE :- PSPICE abbreviates personal simulation program with integrated circuit emphasis. orCADEE pspice is a SPICE circuit simulator application used to simulate and verify analog and mixed signal circuits. It is used to run simulations for orCAD capture defined circuits

and also provides an interface to integrate with MATLAB.

PSPICE is a simulator which creates an output file to store simulation results, orCADEE interface display these results graphically. Hence, orCADEE PSPICE is also called as an improved version of pspice simulator. It includes automatic circuit optimization, viewing, recording, curve-fitting and post processing of waveform.

orCADEE consists of a library of models for physical components and mathematical functions. It also supports model editor, parameterized models, auto convergence ~~and~~, checkpoint restart and magnetic part editor.