ELECTRONIC DESIGN AUTOMATION TOOLS
(ELECTIVE – II)

Course Code: 15EC2213  L  P  C
                       3  0  3

Pre requisites: PSPICE, VERILOG, VHDL

Course Outcomes: At the end of the course the student will be able to
CO1: Illustrate different simulations and delay models which are available for HDL.
CO2: Classify the different synthesis using CAD tools.
CO3: Design and Analyze Analog and Digital Circuits Using PSPICE model of Transistor.
CO4: Describe about Analog, Digital & Mixed Signal Simulators.
CO5: Illustrate PCB Design and also describe the tools used for PCB design.

UNIT I (10-Lectures)
SIMULATION USING HDLS:

UNIT II (10-Lectures)
SYNTHESIS USING HDLS:
CAD Tools for Simulation and Synthesis: Modelsim and Leonardo Spectrum

UNIT III (10-Lectures)
CIRCUIT DESIGN AND SIMULATION USING PSPICE:
Pspice Models For Transistors, A/D & D/A Sample And Hold Circuits etc., And Digital System Building Blocks, Design And Analysis Of Analog And Digital Circuits Using PSPICE.
UNIT IV
(10-Lectures)
AN OVERVIEW OF MIXED SIGNAL VLSI DESIGN:
Fundamentals Of Analog And Digital Simulation, Mixed Signal
Simulator Configurations, Understanding Modeling, Integration To
CAD Environments.

UNIT V
(10-Lectures)
TOOLS FOR PCB DESIGN AND LAYOUT:
An Overview of High Speed PCB Design, Design Entry, Simulation
and Layout Tools for PCB. Introduction to Orcad PCB Design Tools.

TEXTBOOKS:
3. M.H.RASHID, “SPICE FOR Circuits and Electronics Using

REFERENCE BOOKS:
   Addison Wesley