

Course Type	Course Code	Name of Course	L	T	P	Credit
DP	NECC528	VLSI Design and Project Lab-I	0	0	3	1.5

Course Objective

The objective of this lab is to get familiarization with PSPICE and to design, analyze, and simulate complex analog, digital, and mixed-signal circuits. The course will also impart knowledge about Layout Design, useful for designing Integrated Circuits.

Learning Outcomes

By the end of the course, the student will be able to:

- Model various circuits with SPICE and mathematically predict their behavior.
- With this course, students will be able to analyse the performance of MOS-based circuits in particular.
- Independently carry out a small-scale project related to circuit/device modelling.

Unit No.	Topics to be Covered	Lecture Hours	Learning Outcome
1	Introduction to SPICE and overview of various MOS models. SPICE simulation of simple circuits consisting of active/passive devices. Development of SPICE code to simulate the output dc characteristics of MOSFETs, transfer characteristics of a CMOS Inverter and obtain its transient response, simulation of a 2-i/p CMOS NAND/NOR and other MOS-based circuits. Manual Layout of MOS-based circuits based on Layout Design Rules.	21	To learn the role of SPICE and Layout in integrated circuit and board-level design to check the integrity of circuit designs and to predict circuit behavior.
2	Small-scale Project	21	Students will be asked to carry out a small-scale project, based on their understanding of circuit/device modeling.
Total		42	

Reference Books:

1. R. Jacob Baker, CMOS Circuit Design, Layout and Simulation, Wiley IEEE Press, 3rd Edition, 2010
2. Sung-Mo Kang & Yusuf Leblebici, "CMOS Digital Integrated Circuits, Analysis & Design", TMH 4th Edition.