Department of Electrical Engineering
Indian Institute of Technology Kanpur

Proposal for a New Course

- **Course no.:** EE698AC
- **Course title:** Numerical Modeling and SPICE Simulations of Semiconductor Devices
- **Pre-requisite:** There is no official pre-requisite for the course.
- **Course Credits:** 2-0-3-0 [9]
- **Course Duration:** Full Semester
- **Proposing Instructor:** Avinash Lahgere, alahgere@iitk.ac.in
- **Other Faculty members who may be interested in teaching the proposed course:** Dr. Mohit Dineshkumar Ganeriwala, Dr. Shubham Sahay
- **Other Departments may be Interested:** MSE, PHY
- **Course Type:** PG, 4th year UG, Open Elective
- **Course description:**

The Technology Computer-Aided Design (TCAD) and the Simulation Program with Integrated Circuit Emphasis (SPICE) are vital tools in the VLSI industry. TCAD is valuable for optimizing the semiconductor manufacturing process and provides a deep understanding of various device-related characteristics. Meanwhile, SPICE is beneficial for simulating complex circuits quickly and efficiently. Given the growing demand for VLSI engineers with hands-on experience in TCAD and SPICE, acquiring these skills will greatly enhance student's chances of securing positions in the VLSI industry.

The main objective of this course is to provide hands-on experience with TCAD, SPICE model parameter extraction, and SPICE simulation related to semiconductor devices. Throughout the course, we will explore industry-level methodologies for designing semiconductor devices. We will begin with process-level simulations using TCAD to generate device-level results. These results will then be used to extract SPICE models, which will be integrated with SPICE tools for circuit-level simulations.

- **Course contents:**

Topics	Contents	Tentative Lectures	Tentative Labs
Part – I: TCAD Simulations	Fundamentals of Devices Simulations, and TCAD Pros and Cons, Industry Best Practice	2	1
	Hands on Design and Simulation of 2D/3D 28 nm Bulk Device	2	1
	28 nm Bulk Device Desing Industry Semiconductor Process	6	3
	Application of Mixed Mode Simulation for Circuit Design	2	1
Part – II: SPICE Model Extraction	SPICE History, PDK Components and their Integration	2	1
	28 nm Bulk Device DC Parameters Extraction	2	1

	28 nm Bulk Device CV Parameters Extraction	2	1
	28 nm Bulk Device As-fit – Center Model Extraction and Benchmarking using Industry RO	2	1
Part – III: SPICE Simulations	HSPICE/SPECTRE Simulations, Operating Point Simulations,	2	1
	DC, AC, and Transient Simulations,	2	1
	Monte Carlo Simulations	2	1
Total:		39	

Reading material:

The initial material will be covered from the following reference books:

[1] Sentaurus Device User Manual, Synop., Mountain View, CA, USA, 2014.

[2] Synopsys, Inc., HSPICE Reference Manual: Commands and Control Options, Version B-2008.09, Mountain View, CA, 2008

[3] Y. Cheng and C. Hu, MOSFET Modeling & BSIM3 User's Guide. Boston, MA: Springer US, 1999.

[4] Y. C. Wu and Y. R. Jhan, 3D TCAD Simulation for CMOS Nanoeletronic Devices. Singapore: Springer Singapore, 2018.

The rest of the material will mostly be taken from several journal articles and conference proceedings available online.

Signature of the Proposer

Convenor, DPGC

Chairperson, SPGC

Department of Electrical Engineering