DEPARTMENT OF ELECTRICAL & ELECTRONICS ENGINEERING
ACADEMIC YEAR 2021-22 EVEN
Internal Students Seminar – Report

Title of the seminar	: "Circuit Simulation"
Date	: 23.05.2022
Resource Person	: Mr.S.R.Karthikeyan, AP/EEE, KCE
Beneficiaries	: EEE Students - 52
Venue	: III –EEE – ICT Classroom – Hall No: 133

THE MAIN OBJECTIVE OF THE INTERNAL SEMINAR:
➢ The main objective of the internal seminar is to provide exposure to various research areas to our students.
➢ To provides a critical view into the behaviour of electronic circuits.
➢ To validate circuit behaviour and performance via circuit simulation prior to manufacturing.

THE FOLLOWING POINTS WERE DISCUSSED DURING THE SESSION:
➢ Circuit simulation is a process in which a model of an electronic circuit is created and analyzed using various software algorithms, which predict and verify the behaviour and performance of the circuit. Since fabrication of electronic circuits, especially integrated circuits (ICs), is expensive and time-consuming, it is faster and more cost-effective to verify the behaviour and performance of the circuit using a circuit simulator before fabrication.
➢ There are different types of circuit simulators catering to varied needs across the accuracy-performance/capacity spectrum. At one end of the spectrum are analog simulators that solve accurate representations of the electronic circuits.
➢ Digital simulators are commonly used to simulate very large circuits.

Types of Circuit Simulation:
There are three basic types of circuit simulation: analog, digital, and mixed-mode.
➢ **Analog circuit simulation** involves the use of highly accurate models (i.e., representations) of the electronic circuit to achieve high accuracy.
➢ **Digital circuit simulation** involves the use of simpler models of the electronic circuit.
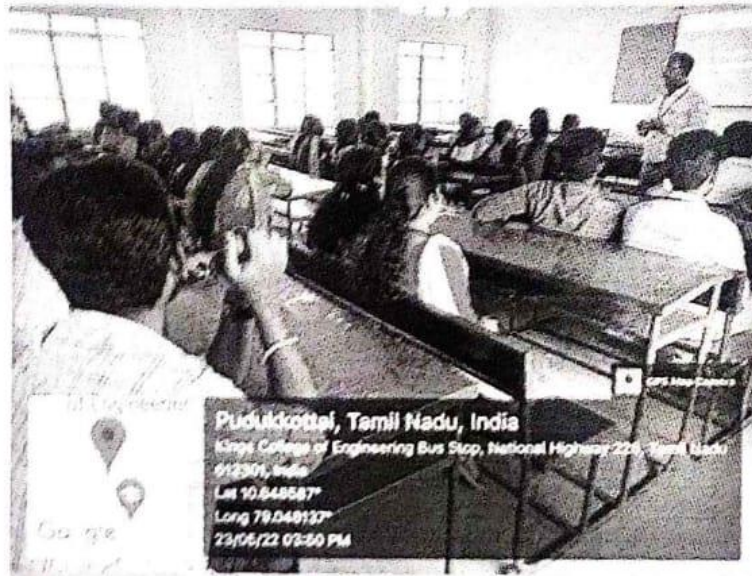➢ **Mixed-mode circuit simulation** combines the analog and digital simulation approaches.

Benefits of Circuit Simulation

- **Memory Performance.** The read and write access times and latency of memory devices are built from analog circuit simulation of the bit cells and read/write paths inside these memories.

- **Overall Digital Simulation Accuracy.** Digital circuit simulators model the propagation of voltage for logic level 1 and logic level 0. Analog circuit simulation is used to determine the time it takes for a circuit to transition between these voltage levels. This forms the basis for the overall accuracy of the digital circuit simulator.

- **Noise and Crosstalk.** Higher level models for noise and crosstalk are developed based on the detailed circuit level analysis of these parasitic effects from analog circuit simulations.

- **Optimization of High-Frequency and High-Power Circuits.** These types of circuits must undergo detailed continuous time analysis to determine their behavior and performance criteria. Analog circuit simulation delivers these important analyses.

- The overall performance and behavior of complex digital circuits (core processors and AI accelerators are examples) are verified with digital circuit simulation.

Outcome:

- Students will be able to highlight theoretical knowledge on Circuit Simulation.
- Students can be able to understand the different types of simulation.
- Students can be able to understand the benefits of Circuit Simulation.
- Students can able to use Circuit Simulation for Publication, Conference presentation and PCE activities.
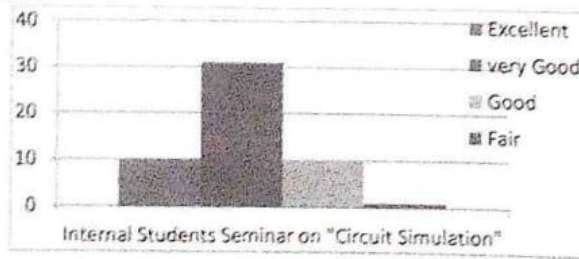

Pudukkottai, Tamil Nadu, India
Block 2, Manamadurai - Tanjavur Rd, Tamil Nadu 613301, India
Lat 10.647743°
Long 79.048245°
23/05/22 03:51 PM

Pudukkottai, Tamil Nadu, India
Kings College of Engineering Bus Stop, National Highway 226, Tamil Nadu
613301, India
Lat 10.648587°
Long 79.048127°
23/05/22 03:50 PM

Pudukkottai, Tamil Nadu, India
Block 2, Manamadurai - Tanjavur Rd, Tamil Nadu 613301, India
Lat 10.547807°
Long 79.048215°
23/05/22 03:57 PM

Snapshot from Seminar

Feedback Analysis:

Internal Students Seminar on "Circuit Simulation"

Legend: Excellent, very Good, Good, Fair

Faculty In-Charge 23/5/22   HOD/EEE 23/5/22   Principal 23/5/2022