

VLSI Workshop Using LTSpice 2-Hour Syllabus

Objective

Introduce participants to VLSI design concepts using LTSpice for circuit simulation and analysis, focusing on practical hands-on experience to keep engagement high.

Workshop Structure

Session 1: Introduction to LTSpice (20 minutes)

1. Basics of LTSpice

- Overview of LTSpice as a simulation tool.
- Installing and setting up the software.
- Introduction to the LTSpice interface and key features.

2. Fundamentals of Circuit Design

- Understanding basic components: resistors, capacitors, transistors (MOSFETs).
- Overview of DC, AC, and transient analysis.

Session 2: Hands-On Experience Part 1 (30 minutes)

1. Simple Circuit Simulation

- Create and simulate a basic RC circuit.
- Perform DC analysis to calculate voltages and currents.

2. Transistor Basics in VLSI

- Simulate a MOSFET-based inverter.
- Understand the transfer characteristics and threshold voltage.

Session 3: Advanced Concepts and Practical Use Cases (20 minutes)

1. Introduction to VLSI Design Principles

- Concept of CMOS logic gates (AND, OR, NOT).
- Power dissipation and delay in circuits.

2. Hands-On: CMOS Logic Gate

- Build and simulate a CMOS NAND gate.
- Perform transient analysis to observe switching behavior.

Session 4: Hands-On Experience Part 2 (30 minutes)

1. Designing a 2-Bit Adder

- Create and simulate a half-adder and full-adder.
- Analyze functionality using transient simulation.

2. Debugging and Optimization

- Techniques for troubleshooting circuit errors.
- Tips for improving circuit performance in LTSpice.

Session 5: Q&A and Wrap-Up (20 minutes)

1. Interactive Problem-Solving

- Answer participants' questions based on their simulations.
- Discuss real-world applications of VLSI circuits.

2. Takeaway Projects

- Provide small assignments: design and simulate a 4:1 multiplexer or a ring oscillator.