



PHoEnix Association presents LTSPICE WORKSHOP

SUMMARY

1. **UNIT OF STUDY:** LTspice
2. **OBJECTIVE:** Teach the basic concepts of LTspice to prepare students for 2nd year courses and enable them to make a project based on it.
3. **TIME:** 5-6 hrs divided into 3 interactive live sessions.

WHAT IS LTSPICE?

LTspice is a powerful, fast and free simulation software, schematic capture and waveform viewer with enhancements and models for improving the simulation of analog circuits.

WHAT CAN YOU DO WITH LTSPICE?

LTspice is majorly used in testing and simulation of analog circuits, this powerful tool can be handy while building your very own project. LTspice is also used in PHoEnix CDCs, and learning it will give you an edge compared to other peers. It also helps in a better understanding of the theoretical concepts through hands on simulation and also ease the learning process whenever these concepts are taught in class. It is used in courses like

- **DIGITAL DESIGN:** CDC for ECE, EEE, ENI and CS.



COURSE PLAN

Class No.	Topics to be covered	Learning Objectives
1	Introduction to LTspice	<ul style="list-style-type: none">● Introduction to LTspice.● Implementation of majority circuits with digital ICs.
2	Combinational Circuits Part 1	<ul style="list-style-type: none">● Implementation of Full Adder.● Parity Generator with Digital ICs.● 4 bit Adder and Subtractor.● BCD Adder with Digital ICs.
3	Combinational Circuits Part 2	<ul style="list-style-type: none">● Decoders.● Multiplexers.● Demultiplexers.

SOFTWARE REQUIREMENTS

LTspice needs to be downloaded. The links will be shared in the server.

MATERIAL AND RESOURCES

Resources corresponding to the topics covered in each class would be provided as the course goes on for better understanding and further reading.

ASSESSMENT

Assignment needs to be completed to be eligible for the certificate.

