GERMAN-MALAYSIAN INSTITUTE

Training for Advanced Technology

COURSE TITLE : ORCAD PSPICE CIRCUIT SIMULATION AND ANALYSIS

COURSE OVERVIEW

This course provides comprehensive content on OrCAD PSpice, native analog, mixed-signal, and analysis engines to deliver a complete circuit simulation and verification solution. Whether you're prototyping simple circuits, designing complex systems, or validating component yield and reliability, OrCAD PSpice technology provides the best, high-performance circuit simulation to analyze and refine your circuits, components, and parameters before committing to layout and fabrication.

COURSE OBJECTIVES

At the completion of this course, participants will be able to :

- Analyse circuit problems from electrical schematics and analyse output response in the time and frequency domain for various type of input signals
- Develop precision with computer skills using simulation tools such as ORCAD & PSpice for analysis, design and performance evaluations of electronic circuits.
- Select the characteristics of semiconductor devices and integrated circuits to the design and construction of basic analog electronic functions such as amplifier, filters and regulator.
-

THE UNIQUENESS OF THIS COURSE

- Training approach will be 60% practical based
- Project based examples and exercises

WHO SHOULD ATTEND

This course is designed to those who want to simulate and verify simple or complex circuit using OrCad PSpice technology.

TARGET GROUP

Industry Professionaks, Electrical & Electronic Engineers, technicians and technical teachers

KEY TOPICS

- OrCad Capture
- PSpice netlist creation
- DC Sweep
- AC Sweep

METHODOLOGY

Lectures, discussions, Exercises & Practical, Lab work

COURSE DURATION

4 Days

PRE-REQUISITE

Electronics

CERTIFICATION

Certificate of attendance will be issued to those who successfully completed the course.

Minimum participants: 3

CONTACT PERSON: Mr. Ravdarn Raman, 03-8921 9046 or 012-2656041 or email ravdarn@gmi.edu.my