Altium Training

COURSE CONTENT

About Multisoft

Train yourself with the best and develop valuable in-demand skills with Multisoft Systems. A leading certification training provider, Multisoft collaborates with top technologies to bring world-class one-on-one and certification trainings. With the goal to empower professionals and business across the globe, we offer more than 1500 training courses, which are delivered by Multisoft's global subject matter experts. We offer tailored corporate training; project Based Training, comprehensive learning solution with lifetime e-learning access, after training support and globally recognized training certificates.

About Course

The Altium Training course by Multisoft Systems is designed for professionals and enthusiasts aiming to excel in Printed Circuit Board (PCB) design. This comprehensive training program equips participants with in-depth knowledge and practical skills required to create efficient and error-free PCB designs using Altium Designer, a leading software in the electronics industry.

Module 1: Introduction to Circuit Designing

- ✓ Need of Circuit Designing
- ✓ Introduction to Basic Components like Resistor, Capacitor, Inductor etc.
- ✓ How to select components
- ✓ Basic Circuit Designing Process

Module 2: Introduction to PCB Designing

- ✓ Types of PCBs
- ✓ Machines used for designing
- ✓ The Designing Process

Module 3: Introduction to Schematic Symbol

- ✓ Types of Components
- ✓ Designing of schematic symbol
- ✓ Auto and Manual Designing

Module 4: Introduction to PCB Footprint

- ✓ Types of PCBs footprints
- ✓ Designing of PCBs Footprint
- ✓ Auto and Manual designing

Module 5: Introduction of Schematic Sheet

- ✓ Beginning a New Schematic
- ✓ Placing Items in the Schematic
- ✓ Placing Symbols and Ports
- ✓ Labeling components
- ✓ Editing the Schematic
- ✓ Working with Sheets and Ports
- ✓ Checking the Schematic for Errors

Module 6: Introduction of PCB Layout

- ✓ Beginning a New Layout
- ✓ Placing Items in the Layout
- ✓ Editing the Layout
- ✓ Placing Power and Ground Planes
- ✓ Changing the Board's Perimeter
- ✓ Creating New Components
- ✓ Linking the Schematic and PCB
- ✓ Keyboard Shortcuts
- ✓ Troubleshooting errors
- ✓ Single Layer and Multilayer Routing methods

Module 7: BOM and Fabrication Output

- ✓ Exporting of BOM from design
- ✓ Part Selection and Filtration
- ✓ Introduction of Gerber
- ✓ Layer Selection in Gerber
- ✓ Generation of NC Drill data

Module 8: Pick and Place Data (X-Y Data)

- ✓ Introduction on assembly machine
- ✓ Generation of Pick and Place data