

Altium Designer PCB Fundamentals

OEM: Altium

COURSE DESCRIPTION

PCB design is at the core of bringing hardware concepts to life, requiring precision and attention to detail. This guided hands-on course in Altium Designer and the workspace teaches you how to create manufacturable designs that integrate seamlessly with team workflows, ensuring professional results and efficient production processes. Instructor-led training helps you create production-ready PCB designs. Collaborate and work more effectively with the latest workspace capabilities. Minimize manufacturing issues with proper design rule implementation and industry best practices. Showcase your skills with a certificate of completion.

AUDIENCE PROFILE

- New users starting with PCB layout.
- Schematic designers expanding into PCB design.
- Engineering teams standardizing workflows.
- Educators and students learning industry practices.
- Hardware engineers enhancing design skills.

PREREQUISITES

- A stable internet connection
- Access to an Altium Account
- Latest Altium Designer installation

COURSE OBJECTIVES

By the end of this course, participants will be able to:

- This section doesn't contain any content.

COURSE OUTLINE

Module 1: Introduction: Navigating the Workspace and Customizing Altium Designer

- **Workspace Navigation:** Learn how to access the workspace, navigate folder structures, and utilize the Explorer panel to manage projects effectively.

- Design Environment Setup: Discover how to tailor the design environment to your needs, optimizing panels, toolbars, and menus for efficient project work.
- Project Management: Access existing design projects or create new ones within the workspace for seamless project management and collaboration.
- PCB Preferences Setup: Configure the PCB environment preferences to streamline design capture and ensure consistency across projects.

Module 2: PCB Design Essentials and Advanced Techniques

- PCB User Interface & Preferences: A guided walkthrough of the PCB interface, highlighting customization options to optimize your workflow.
- Efficient PCB Navigation: Tips and techniques to navigate complex layouts with ease.
- Net Classes: Learn how to manage schematic net classes for organized, rule-driven design.
- Layer Stack Management: Techniques for configuring and managing layer stacks to support multi-layer designs.
- Board Shape Definition: Define the board outline and constraints according to project specifications.
- Custom Grids Setup: Set up custom grids to facilitate precise component placement and routing.
- Design Rules & Constraints: An overview of design rules that ensure compliance and functionality in complex layouts.
- Component Placement Strategies: Effective strategies for placing components to balance performance and manufacturability.
- Design Reuse: Utilize the workspace for seamless design reuse, speeding up repetitive tasks and ensuring consistency.
- Routing Techniques: Explore routing methods to create reliable connections while adhering to design constraints.
- Global Editing Tools: Apply global edits efficiently to multiple design elements and components.
- Planes & Polygons Configuration: Configure power planes and polygons to manage signal integrity and grounding.
- Resolving Design Violations: Identify and resolve design violations to ensure a smooth transition to manufacturing.

Module 3: Documentation and Final Output Generation

- Fabrication & Assembly Outputs: Generate comprehensive fabrication and assembly files in various formats to meet manufacturer requirements.
- Bill of Materials (BOM): Create a detailed BOM that lists all components and quantities for assembly.
- Output Job Files: Utilize Output Job files to streamline the generation of multiple outputs, ensuring consistency and accuracy.
- Project Release: Release your project to the workspace, enabling seamless collaboration and version control.
- Design Review: Conduct project reviews using workspace Commenting and Task features to facilitate team feedback and collaborative problem-solving.