

Altium Designer Essentials

OEM: Altium • Code: ALTIUM_DESIGNER_ESSENTIALS

COURSE DESCRIPTION

Altium Designer Essentials Instructor-led and Private training provides comprehensive coverage of the entire design capture process—from User Interface and Project Creation to PCB Layout and Fabrication Output—seamlessly integrating workspace features. Create high-quality PCB designs in less time. Learn powerful best practices to help you save time and reduce costly errors. Access, create, and update projects and documents in the workspace. Earn a certificate to showcase your skills to potential employers.

AUDIENCE PROFILE

- Beginners who are new to PCB design and want to establish a solid foundation in the field.
- Experienced designers seeking to transition to or gain proficiency with Altium Designer and its advanced features.

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: Course Introduction: Navigating Altium Platform and Customizing Altium Designer

- Accessing Altium Platform: Learn how to access the designated training workspace and connect to the Altium Platform environment
- Workspace Folder Structure: Navigate the training workspace, understand the web interface, and utilize the Explorer panel
- Project Management: Access existing design projects and create new ones within Altium Platform for seamless project management and collaboration

- **Design Environment Customization:** Discover how to tailor the design environment to your needs, optimizing panels, toolbars, and menus for efficient project work
- **Schematic Preferences Setup:** Configure schematic environment preferences to streamline design capture and ensure consistency across projects
- **Navigation Fundamentals:** Master basic operations in the schematic editor, including preference settings, mouse actions, frequently used shortcut keys, and search actions

Module 2: Schematic Design and Capture Essentials

- **Project Creation and Storage:** Create training folders in the workspace and establish new projects using project templates
- **Component Placement and Libraries:** Access and place components efficiently using Altium Platform libraries, including reuse blocks, ensuring a streamlined design flow
- **Creating Connectivity:** Establish connections between components using wires, net labels, and bus structures to create accurate electrical pathways
- **Hierarchical Design Implementation:** Convert flat design structures to hierarchical designs using bottom-up methodology for better organization and manageability
- **Schematic Graphics and Annotation:** Manage graphic objects, text strings, formatting, and component annotation for clear documentation
- **Advanced Editing Capabilities:** Use alignment commands, net color highlighting, and global editing tools to make swift, consistent changes across multiple components
- **Design Rules Configuration:** (New Module 12B) Learn how to define design rules by checking project settings and placing directives, including Constraint Manager navigation, net classes definition, and width rules configuration
- **Electrical Rules and Validation:** Apply electrical rules, perform design verification, and conduct Electrical Rule Checks (ERC) to maintain design integrity
- **Multi-Sheet Design Management:** Control multi-sheet designs and manage hierarchical structures for larger projects
- **Design Transfer Preparation:** Transfer schematic rules and design data to PCB layout, ensuring continuity and consistency in the design process

Module 3: PCB Layout Essentials and Advanced Techniques

- **PCB Environment Setup:** Comprehensive walkthrough of the PCB interface, preferences, navigation techniques, and viewing tools
- **PCB Configuration Fundamentals:** Board shape definition, mechanical layer setup, layer stack management, and custom grid configuration
- **Layer Stack Management:** Configure layer stacks using the Layer Stack Manager and load predefined layer stacks from the workspace
- **Grid Setup:** Configuration of the Grid and its visibility / appearance for Component Placement and Routing
- **Design Rules and Constraints:** Overview of design rule creation, rule priority management, and the significance of proper rule configuration
- **Advanced Constraint Manager:** (New Module 20B) Hands-on experience with the Constraint Manager interface, including defining clearances and widths, using net classes, setting via sizes, establishing component clearances, and importing/exporting constraints and directives
- **Component and Net Class Management:** Organize designs using component classes and net classes for rule-driven design
- **Component Placement Strategies:** Effective strategies for placing components to balance performance and manufacturability, including precise positioning and mounting hole placement
- **Routing Techniques and Methods:** Explore various routing commands and options to create reliable connections while adhering to design constraints
- **Polygon Pour and Plane Management:** Configure power planes and polygons, manage polygon pour creation and modification, and understand connection styles for pads and vias
- **Global Editing and Analysis Tools:** Apply global edits efficiently using the PCB List panel and PCB Filter panel for design optimization

- Design Rule Checking (DRC): Configure and run design rule checks, identify and resolve violations using the PCB Rules and Violation Panel
- Design Reuse with Altium Platform: Utilize Altium Platform for seamless design reuse, speeding up repetitive tasks and ensuring consistency
- Component Creation for Altium Platform: Creating custom components within the Altium workspace, essential for ongoing design work and library management.

Module 4: Documentation and Final Output Generation

- Professional Documentation with Draftsman: Create assembly drawings and fabrication drawings using predefined Draftsman documents and templates
- Output Job File Configuration: Utilize Output Job files to streamline the generation of multiple outputs, including local fabrication files and documentation, ensuring consistency and accuracy
- Fabrication & Assembly Outputs: Generate comprehensive fabrication and assembly files in various formats to meet manufacturer requirements
- Bill of Materials (BOM): Create detailed BOMs that list all components and quantities for assembly
- Project Release to Altium Platform: Release projects to the Altium Workspace, enabling seamless collaboration and version control through the complete release process