



Altium Designer Advanced with Altium 365

Custom Instructor-Led Training • Private • 3 Days

Course Overview

This comprehensive Custom [Altium Designer Advanced ILT training](#) provides experienced users with advanced techniques and workflows tailored to their specific design challenges. This instructor-led course combines 20 essential core modules with 8 customizable modules selected from our extensive library of specialized topics, ensuring maximum relevance and value for your team's unique requirements.

Delivered over three intensive days, this course features expert-led demonstrations, hands-on exercises, and real-world case studies that allow participants to apply advanced concepts immediately. The customizable approach ensures your team focuses on the most relevant advanced features while building upon their existing Altium Designer knowledge.

Course Highlights

- **Customized Learning Path:** 20 core advanced modules plus 8 selectable modules from our library of modules to address design challenges and industry requirements
- **Advanced PCB Design Techniques:** Master complex routing strategies, high-speed design principles, and advanced constraint management
- **Hands-On Learning:** Practical exercises using realistic design challenges and reference projects relevant to industry.
- **Altium 365 Advanced Features:** Leverage advanced collaboration, release management, and enterprise-level capabilities
- **Team-Focused Content:** Standardize advanced practices across your engineering team with consistent methodologies
- **Industry-Specific Applications:** Choose optional modules that align with your specific industry needs
- **Real-Time Problem Solving:** Address specific design challenges your team faces with expert guidance and solutions

Requirements:

- **Completion of Altium Designer Essentials training** or equivalent experience (minimum 6 months of regular use)
- **Active Altium Designer license** (Professional or Enterprise recommended)
- **Access to Altium 365 workspace** for collaboration exercises

This intensive, instructor-led course ensures your team gains advanced skills while addressing specific design challenges relevant to your projects and industry requirements.

License Type: Professional / Enterprise license recommended for full feature access



Agenda

Core Modules • Days 1 & 2 • 16 Hours

Comprehensive coverage of all 20 modules listed below, including advanced topics like multi-board design, BOM control, high-speed routing, rule checks, and collaborative workflows.

✓ Multi-Board Design

Master advanced multi-board system design and integration strategies for complex projects.

- Create hierarchical multi-board assemblies with proper connectivity
- Manage inter-board connections and cable definitions
- Establish design synchronization across multiple PCBs
- Implement proper project structure for system-level designs

✓ Active BOM

Implement dynamic Bill of Materials management with real-time component lifecycle control.

- Configure BOM templates and advanced filtering options
- Manage component variants and substitutions effectively
- Track component lifecycle states and supplier data integration

✓ Active Route and Re-Routing

Master interactive auto routing and automatic Length matching techniques and re-routing strategies for optimal design outcomes.

- Utilize advanced routing modes and push-and-shove algorithms
- Optimize design changes through various re-routing techniques
- Maintain signal integrity during routing modifications

✓ Creating and Using Design Reuse Block in a Workspace

Build and implement reusable design blocks for maximum efficiency across multiple projects.

- Create snippet libraries with proper naming conventions
- Define block parameters and variants for flexibility
- Implement version control for reusable assets
- Integrate with Altium 365 workspace for team access

✓ Design Rule Queries

Develop advanced query language skills for complex design rule creation and precise validation control.

- Master query syntax and logical operators
- Create conditional rules based on object properties
- Implement scope-based rule application strategies



✓ **Differential Pair Routing with Impedance Profile**

Implement high-speed differential pair routing with precise impedance control for signal integrity optimization.

- Define impedance profiles and via compensation
- Manage length matching within differential pairs
- Handle layer transitions and coupling requirements
- Validate signal integrity for high-speed applications

✓ **Doing Design Review**

Establish systematic design review processes and collaborative validation methods for professional design environments.

- Configure design review workflows and approval processes
- Utilize commenting and markup tools for effective communication
- Implement automated design checks and validation reports
- Manage review cycles and track resolution of design issues

✓ **Length Tuning Routing**

Optimize signal timing through advanced length matching and tuning strategies for high-speed designs.

- Configure length tuning rules and target specifications
- Implement serpentine and accordion tuning patterns
- Manage group delay matching for critical signal groups
- Validate timing requirements and generate length reports

✓ **Multi-Channel Design 1 - Schematic Entry**

Create multi-channel schematics with synchronized channel management and efficient replication techniques.

- Set up multi-channel projects and channel templates
- Configure channel synchronization and parameter mapping
- Implement global changes across multiple channels
- Manage channel variants and configuration differences

✓ **Multi-Channel Design 2 - Rooms in the PCB**

Organize PCB layout using room-based design methodology for multi-channel routing and component management.

- Create and configure rooms for channel organization
- Implement room-based routing strategies and constraints
- Manage component placement within room boundaries
- Coordinate inter-room connections and routing

✓ **Multi-Channel Design 3 - Board Level Annotation**

Implement advanced annotation strategies for complex multi-channel board designs with proper component referencing.

- Configure annotation schemes for multi-channel designs
- Manage component reference designators across channels
- Implement automated annotation updates and synchronization
- Handle Back annotation and Forward Annotation conflicts and resolution strategies



✓ PCB Layout Replication

Execute automated layout replication techniques for consistent multi-channel implementations and design efficiency.

- Set up layout replication templates and constraints
- Configure replication rules and component mapping
- Implement automated placement and routing replication
- Manage design changes and replication updates

✓ Pin Swapping

Optimize pin assignments dynamically using Powerful Automatic Pin/Net Optimizer during routing for improved layout efficiency, Layer count reduction and signal integrity optimization.

- Configure pin swapping rules and gate equivalency
- Implement automatic pin optimization during routing
- Manage pin swapping in high-speed and differential designs
- Validate pin assignments and generate swap reports

✓ Releasing a Project

Implement professional project release management and version control workflows for design integrity.

- Configure release processes and approval workflows
- Manage design documentation and manufacturing outputs
- Implement version control and change tracking
- Coordinate release notifications and stakeholder communication

✓ Schematic Design Rules (CM and DR)

Implement advanced schematic design / PCB rules using Constraint Manager and Design Rules integration.

- Configure schematic-based constraint definitions
- Implement electrical / physical rules and signal integrity constraints
- Manage rule inheritance from schematic to PCB
- Validate schematic constraints and resolve conflicts

✓ Using Design Variants

Manage multi-variant design configurations for different product options and manufacturing requirements.

- Create and configure design variants and parameters
- Implement component fitting and DNP strategies
- Implement Alternative component fitting strategies
- Manage variant-specific constraints and rules
- Generate variant-specific outputs and documentation

✓ Using xSignals in a High-Speed Design

Implement advanced signal management techniques for high-speed design applications and optimization.

- Configure xSignals for complex signal topologies
- Manage signal splitting and merging strategies
- Implement advanced timing and length constraints
- Validate signal integrity across complex topologies



Custom Modules · Day 3 · 8 Hours

Select 8 of the 20 specialized modules below to address your team's unique challenges and enhance your expertise.

PCB Structure & Fundamentals ☐ Clearance Checking in 3D Validate 3D designs and mechanical clearances for manufacturing reliability. ☐ Defining the Board Shape from a 3D Model Extract precise board outlines from 3D CAD data for accurate fit. ☐ Embedded Board Array Design efficient panelization and embedded board arrays for manufacturing. ☐ Rigid-Flex 1 - Creating Multiple Layer Stacks Manage multi-layer stacks for complex rigid-flex PCB designs. ☐ Rigid-Flex 2 - PCB Design Optimize layout and bend areas for flexible circuits. ☐ PCB CoDesign Establish collaborative workflows for complex PCB projects.	Components, Pads & Vias ☐ Creating Custom Pads Create customized pad shapes and parameters for specific components. ☐ Pad & Via Templates Standardize pad/via structures for efficiency and consistency. ☐ Teardrops Automate teardrop generation for improved reliability and signal integrity. ☐ Via Shielding & Stitching Use advanced via techniques for EMI control and signal integrity. ☐ Back Drills Use back drilling to reduce signal stubs and improve integrity. ☐ HDI Vias for High-Speed and Multilayer Designs Apply high-density interconnect vias for miniaturized, high-performance designs.	Documentation & Manufacturing Prep ☐ CAMtastic Edit CAM files and prepare manufacturing data for optimized production. ☐ Template 1 – Draftsman Templates Create custom templates for consistent and automated design documentation. ☐ Template 2 - Project Templates Develop project templates for standardized workflows and team consistency.
Routing & Connectivity Techniques ☐ Fanout and Escape Routes Optimize BGA fanout and escape routing for dense designs. ☐ Using Signal Harnesses Integrate cable and harness designs with PCB layout for full system coordination.	Design Rules & Verification ☐ Clearance Checking in 3D Verify 3D design and mechanical clearances for manufacturing. ☐ Applying Creepage Constraints in PCB Layout Ensure safety compliance with creepage and clearance constraints. ☐ Design Comparisons Validate design accuracy and consistency across iterations.	

