

Boot Camp – Altium Designer Course Agenda

This training helps both new and more experienced users ramp up on the fundamentals of Altium Designer version 19 and beyond. These versions represent the largest upgrade to the software since the release of Altium Designer version 10. The look, feel, and functionality have all been updated to match modern tool displays and interfaces.

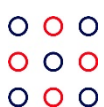
This is a project-based training that covers many functions and features of the software and how they are effectively used in a standard project flow. Emphasis is placed on setup, navigation, and features that are key to becoming efficient and comfortable with the tool. After each module's instruction, highly applicable exercises are used to reinforce the understanding of the content.



Areas covered by this training:

- Setting up Preferences, Project Options, and Document Options that make the personal and corporate design environments more efficient and customized for the type of work being performed.
- Navigation of Altium Designer's panels and editors.
- Project creation and design file management.
- Creating components using the symbol and footprint library editors.
- Schematic capture utilizing component libraries and the extensive feature set of its editor.
- Establishing the layer stack up, design rules, and board outline in the PCB Layout editor.
- Component placement and routing in the PCB layout environment.
- Generating manufacturing files from schematics (such as the Bill of Materials) and PCB (such as Gerber and drill files), using the Outjobs editor.

In summary, this training provides the attendee with the fundamentals necessary to effectively use Altium Designer in any stage of a project, and is an excellent basis for further learning of the advanced features of the tool. It also provides exposure to many of the settings and features that more advanced users can utilize to become more efficient and comfortable with the updated look, feel, and functionality of Altium Designer version 19 and beyond.



Boot Camp – Altium Designer Course Agenda

Course Outline:

Module 1 – Altium Designer Environment Overview

- Overview of the Schematic, PCB, and Library editors for new users.

Module 2 – Configuration Management

- Detailed instruction on setting System Preferences, Project Options, and Document Options.

Module 3 – Navigation

- Navigating the tool with heavy emphasis on Panel use and configuration.

Module 4 – Creating a Project

- Creating a project by adding new and existing files to a project.
- Introduction to the Projects panel

Module 5 – Schematic Library and Editor

- Schematic Library editor details including specific panels and functions.
- Adding and editing existing schematic symbols.
- Building new schematic symbols from scratch.

Module 6 – PCB Library Editor

- PCB Library Editor details including specific panels and functions.
- PCB Layers setup
- Creating a footprint from scratch using the editor and the IPC Footprint Wizard
- 3D Body placement

Module 7 – Schematic Capture

- Schematic Preferences and Templates.
- Schematic capture and editing using primitives.
- Using Directives to assign design rules and classes.

Module 8 – Schematic Tools and Reports

- Update from Libraries, Annotation, Compilation, and Bill of Materials.

Module 9 – PCB Setup

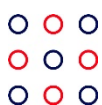
- Details regarding PCB setup for placement and routing preparation including PCB Stack-up, Board Outline, Grids, Classes, and Design Rules.

Module 10 – PCB Place and Route

- Placement and Routing functions and techniques.

Module 11 – Output Files

- Creating output files to send to Fabricators and Assemblers for project build.



Leader in EDA and PCB training and service

info@ninedotconnects.com (214) 699-7719

ninedotconnects.com/training