Full Spectrum – Altium Designer Course Agenda

The Full Spectrum Course is a four-day training that combines the core areas of our three-day Boot Camp course, with the principle subjects of our three-day Spec Ops course.

This training is intended for Altium Designer users with early to moderate experience, or for those with high proficiency in other EDA tools who wish to evaluate the abilities of Altium Designer through instruction and hands on exercises. This course is an effective alternative to spending a total of six days over two weeks attending the Boot Camp and Spec Ops courses. With Full Spectrum, it's done in as few as four days in the same week, while still covering a considerable amount of information at a somewhat faster pace.

The Nine Dot Connects Boot Camp course helps both new and more experienced users ramp up on the fundamentals of Altium Designer. The Full Spectrum course builds on this by quickly covering the fundamentals of each subject, then moving into more advanced techniques and features that would normally be taught in our Spec Ops course. Due to time constraints, the areas not covered from the Spec Ops course are the more specialized features of the tool.

Like our Boot Camp course, this is a project-based training that covers many functions and features of the software and how to use them most effectively in a standard project flow. Emphasis is on setup, navigation, and features that are key to becoming efficient and comfortable with the tool. Advanced routing, layout techniques, and high-speed design constructs, like differential pair routing, length matching, and tuning, will be discussed and demonstrated as well. After each module's instruction, highly applicable exercises reinforce the understanding of the content.

Areas covered by this training:

- Altium Designer Overview for newer users to the tool.
- Advanced tool settings, navigation, and use of panels.
- Project creation and design file management.
- Creating components using the Schematic and PCB Footprint library editors.
- Schematic capture utilizing component libraries and the extensive feature set of its editor.
- Advanced PCB setup (stack up, all via type definitions, back drill, rigid/flex, board outline).
- Extensive coverage and setup of Design Rules in both the schematic and PCB editors.
- Efficient component placement using special techniques and features.
- Basic Interactive routing as well as advanced routing features used for high speed designs.
- Planes and Polygon use.
- Generating drawings and Outjob manufacturing file sets.

Full Spectrum – 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
- In depth study of 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 Capture
- Schematic Preferences and Templates.
- Schematic capture and editing using primitives.
- Using Directives to assign design rules and classes.

Module 6 – Schematic Library and Editor
- Schematic Library editor including specific panels and functions.
- Adding and editing existing schematic symbols.
- Building new schematic symbols from scratch.

Module 7 – Schematic Tools and Reports
- Preparing the Schematic for the PCB Layout
- Generating reports such as the Bill of Materials.

Module 8 – PCB Setup
- Stack-up Manager and defining parameters in the stack-up.
- Defining Vias; Thru, Blind, Buried, and Micro Vias, along with the Back-Drill definitions and rules.
- Defining a Board Outline from a DXF file and creating a Rigid/Flex design.

Module 9 – Design Rules
- In depth look at all design rules and their use.
- Defining design rules in the Schematic and PCB editors.
- Design rule checking and resolution tools.

Module 10 – PCB Library Editor

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

Module 11 – Rooms and Footprint Placement

- Basic and advanced placement tools and features.
- Use of Rooms to aid in placement and define special rules.

Module 12 – Interactive Routing

- Interactive routing functions and techniques.

Module 13 – Advanced Routing

- Advanced routing techniques and tools.
- Differential Pair routing.
- Length tuning for differential and single ended signals.
- Multi-route and Active Route features.

Module 14 – Planes and Polygons

- Use of Planes and Polygons in a design.
- Special rules for Planes and Polygons.

Module 15 – PCB Back Annotation and Output Files

- Annotation of a design from both the Schematic and PCB editors.
- Creating Altium Outjobs for manufacturing design file packages.

These additional topics in the manual are included in the optional fifth training day:

Module 16 – Pin/Part Swapping
Module 17 – Multi-channel Designs
Module 18 – Multi-Board Design
Module 19 – X-Signals
Module 20 – Introduction to Draftsman