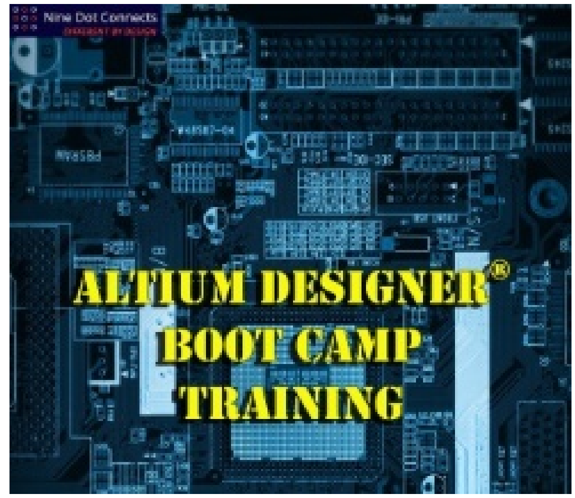
Boot Camp – Altium Designer Course Agenda

This training helps students quickly ramp up on the fundamentals of Altium Designer. In this training, students will create components using the symbol and footprint libraries, and then use these components in both a schematic design and PCB layout. The PCB layout instruction includes: preliminary groundwork necessary such as layer stack up and rules; placement of footprints; and routing of connections. Students will also get hands-on practice generating manufacturing files from schematics (BOM) and PCB (Gerber and drill files).



Students will progress through a standard project flow, modifying an existing design to include additional circuitry.

- Day 1 – Schematic and PCB Libraries
- Day 2 – Schematics
- Day 3 – PCB Layout

The example project used in this course (Developer tool – DT01) is available from the 'Examples' projects folder included with the Altium Designer installation.

Students will add a new schematic to an existing project, and then pass along those components to the PCB to be placed and routed. After we ensure our PCB adheres to all our design rules, the PCB manufacturing output files can be created. All of the major subordinate tasks along the way will be examined, as well.

At the conclusion of the course, students should be well acquainted with all of the steps necessary to successfully implement Altium Designer into their PCB production workflow.

Day 1 – Schematic and PCB Libraries

Mod 1 – Workspace Panels
- Exercise 1: Workspace Panels

Mod 2 – SCH Library Options / Preferences
- Exercise 2: SCH Library Options / Preferences

Mod 3 – Creating New SCH Library Components
- Exercise 3: Creating New SCH Library Components

Mod 4 – PCB Library Panels

- Exercise 4: PCB Library Panels

Mod 5 - PCB Library Properties / Preferences

- Exercise 5: PCB Library Properties / Preferences

Mod 6 – Creating New PCB Library Components

- Exercise 6: Creating New PCB Library Components

Mod 7 – Footprint Wizards

- Exercise 7A: The IPC Compliant Footprint Wizard
- Exercise 7B: The Component Wizard

Mod 8 – Integrated Libraries

- The Library Project
- Exercise 8: Integrated Libraries

<u>Day 2 – Schematics</u>

Mod 1 – Altium Designer PCB Project Flow

- Exercise 1: AD Project Flow

Mod 2 - Schematic Editor Panels

- Exercise 2: Schematic Editor Panels

Mod 3 – System Preferences / Project Options / Schematic Properties

- Exercise 3: System Preferences / Project Options / Schematic Properties

Mod 4 – Placing Schematic Components

- Exercise 4: Placing Components from the Libraries panel

Mod 5 - Placing Wires and Other Objects

- Exercise 5: Placing Wires and Other Objects

Mod 6 – Compiling the PCB Project

- Messages and Compiler Errors
- Exercise 6: Compiling the PCB Project

Mod 7 – Parameter Manager/Footprint Manager

- Exercise 7: Parameter Manager/Footprint Manager

Mod 8 – Schematic Output Files

- Exercise 8: BOM / PDF Output Files

<u>Day 3 – PCB Layout</u>

Mod 1 – PCB Editor Panels

- Exercise 1: PCB Editor panels

Mod 2 – PCB Properties and Preferences

- Exercise 2: PCB Properties and Preferences

Mod 3 – PCB Layers

- Exercise 3: PCB Layers

Mod 4 – SCH / PCB Comparator

- Exercise 4: SCH / PCB Comparator

Mod 5 – Placing PCB Components

- Exercise 5: Placing PCB Components

Mod 6 – Object Classes / Design Rules / Design Rule Checking

- Exercise 6: Object Classes / Design Rules / Design Rule Checking

Mod 7 – Routing

- Exercise 7: Routing the PCB

Mod 8 – Polygons

- Exercise 8: Polygons

Mod 9 – PCB Output Files

- Bare PCB files
- Assembled PCB files
- Mechanical PCB files
- Exercise 9: PCB Output Files