Lib & Sch – Altium Designer Course Agenda

Course Overview

This course is combination of 1-day Libraries and 1-day Schematics course. Each day can stand by itself without any dependency of each other from the exercise perspective.

The 1st day libraries course teaches you how to make optimal use of the library features found in Altium Designer. In addition to learning how to create basic schematic component libraries (.SchLib) and associated printed circuit board footprint libraries (.PcbLib), you will also learn how to create Library Projects (.LibPkg), Integrated Libraries (.IntLib) and Database Libraries (.DbLib).

In this hands-on training, you will experiment with the available workspace panels available in the schematic library editor along with the different preferences used to configure the components. The creation of symbols will be demonstrated and practiced. The same process will be followed to acquaint you with the PCB footprint editor.

The standard component creation flow will become evident as available workspace panels and dialog boxes to facilitate component creation are introduced.

- Modules 1 – 4: Creating schematic symbols
- Modules 5 – 8: Creating associated printed circuit board footprints
- Modules 9 – 10: Creating integrated libraries and database libraries, respectively

Module	Description	Module	Description
1	Pertinent workspace panels	6	Footprint editor options and preferences
2	Symbol library panel features	7	Footprint creation
3	Symbol editor options and preferences	8	Footprint wizards for rapid creation
4	Symbol creation	9	Integrated libraries
5	PCB footprint basics	10	Database libraries

- The Developer Tool – DT01 available from the example projects folder included with the Altium Designer installation will be used to demonstrate library concepts.

- Many tips and tricks will be presented by our experienced instructor throughout the course to give you the information you need that can only be gained through years of experience with the tool.

- At the conclusion of the course, you will have a good understanding the types of libraries offered in Altium Designer and their configurations, in addition to component creation.

- This 1st day can also be used as Day 1 of a 3-day Altium Designer Boot camp course for PCB development. Alternatively, it can also be used as Day 1 of a 2-day Library Training class, where day 2 introduces more advanced features.

Lib & Sch – Altium Designer Course Agenda

The 2nd -day schematics course teaches you how to make optimal use of the schematic editor found in Altium Designer.

In this hands-on training, you will learn:

- The process of placing schematic components from various library sources into the schematic editor.
- The methods of component connectivity on and between schematic sheets.
- Managing the logistical needs of the schematics such as electrical rule checking for errors, linking PCB footprints to schematic components, and the use of parameters to create the Bill of Materials.
- Generating the output files from the schematics (BOM and .PDF) and the Outjob file.

This course concludes with a study of how repeated circuitry is implemented in Altium Designer (also known as multi-channel design) and introduces you to the self-study resources available for simulation and signal integrity tools in Altium Designer.

Module	Description	Module	Description
1	Altium Designer PCB project flow	7	Compiling the PCB project
2	Schematic editor panels	8	Parameter and footprint managers
3	Schematic editor preferences & options	9	Generating schematic output files
4	Placing symbols from the Content Vault	10	Multi-channel design concepts
5	Libraries panel features and function	11	Simulation and signal integrity analyses
6	Schematic Editor and features		

- Many tips and tricks will be presented by our experienced instructor throughout the course to give you the information you need that can only be gained through years of experience with the tool.

- At the conclusion of the course, you will have a good understanding of the schematic editor and why many Altium Designer users use the editor as an "off label" drawing tool for cabling and systems-level drawings.

- This 2nd day class can also be used as Day 2 of a 3-day Altium Designer Boot camp course for PCB development. Alternatively, it can also be used as Day 1 of a 2-day Library Training class, where day 2 introduces more advanced features.