

Library – Altium Designer Course Agenda

Course Overview

This 1-day 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 class 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.

