



שם הקורס:

Altium Schematics

משך ההדרכה: 18 שעות

Altium Designer Environment	Environment, file types, project Structure and more
It Starts with a Project	Create, package and add files to a project
Navigating Schematics	Mouse actions, frequently used shortcut keys and search actions in the schematic environment
Schematic Preferences	Customize the schematic environment and ensure document clarity and standardization
Schematic Capture	Add components and objects into the schematic editor
Making Connections - Schematic	Create the connection between all objects within the schematics
Components Annotation	Number the Objects automatically in the schematics
Design Verification	Compile the Schematics with ERC Checker (Electrical Rule Check)
Variants	Learn how to create different versions of the same PCB design — all within a single project (components control)
Manufacturer Part Search	Built-in tool that allows sophisticated components search feature based on categories and parametric filtering (includes Symbol, Footprint, 3D, price, alternatives and more)
Design Reuse Blocks	Create and share ready to use Schematics\PCB blocks (managed)
Spice/PSpice Simulations	Powerful Spice Simulation on top the Schematics - no need to export and import files
Variants	Fitted - on Not fitted components variations, in addition - alternate parts
Multi-Channel Design	Repeated channels design - automation of Schematics and Layout
Board Level Annotation	Annotation options for multi-channel design
Adding Hierarchy	Structural relationships between the sheets, how it works, and the tools and techniques available to create a multi-sheet design
Export Schematics Options	BOM, Variants, Netlist, Smarted and more
Grids	Grid options ~ Schematics and components
Schematics Constrains	Apply Rules and Constrains that affect the Layout directly and Transferring Design Rules to PCB

**Libraries**

Symbols and Footprints	Detailed explanation on those libraries
Library types	File-Based, Data Base Library, Integrated and Cloud Library
Managed Libraries	Component history and revision update
Symbol Creation	Create Symbols, from small to large scale
Footprint Creation	Create Footprint by hand and with automation
Import Libraries	Import to Altium, and upload to Cloud