

**שם הקורס:****Altium PCB Layout**

משך ההדרכה: 18 שעות.

PCB Introduction	Introduces the PCB editor, where electrical connections become physical copper traces and components are placed for manufacturing
Transfer to PCB	This process links the schematic components and nets to the PCB editor, creating a physical layout environment for board design
Updating the PCB	Ensures any modifications in the schematic (like added components or net changes) are reflected in the PCB to maintain design integrity
PCB Navigation	Involves zooming, panning, layer visibility, and selection techniques to easily inspect and edit PCB layouts.
PCB Layers	Layers define where signals, planes, silkscreen, and mechanical outlines exist—vital for clear design and fabrication
Shaping the PCB	Create Board Shape from Mechanical files like - Step, X t, DXF and more
PCB Grids	The invisible structure that keeps design elements aligned, create lined and polar grids combined
PCB Rules and Panels	Rules define clearances, widths, and constraints; panels manage multi-PCB designs or manufacturing panels efficiently
PCB Placement	Arranging parts to optimize signal paths, minimize noise, and support heat dissipation and assembly processes
PCB Routing	Route the PCB and learn routing methods like, single, differential, multilayer route
PCB Global Editing	Lets you apply edits (like changing track widths or text styles) across multiple objects or layers at once
DRCs on PCBs	DRCs identify spacing, clearance, and connectivity errors to ensure manufacturability and reliability
Polygons and Power Planes	Control polygons and planes, create them and create rules for them
PCB Fabrication Files	Includes Gerber, drill, and pick-and-place files that communicate board details to PCB fabricators and assemblers
Draftsman	Draftsman automates the generation of detailed assembly and fabrication drawings directly linked to your PCB project
Queries with Design Rules	Uses Altium's query language to define where and how specific rules apply — essential for advanced, precise design constraint management
Clearance Checking in 3D	Verifies that components and mechanical parts don't collide, guaranteeing manufacturable and enclosure-



	ready PCB designs
ActiveRoute	Altium's ActiveRoute automatically lays out multiple traces based on user-defined constraints, balancing speed and design precision
Rigid-Flex Design	Enables advanced PCB designs that bend or fold, ideal for compact electronics like wearables and foldable devices
Multi-Channel Design	Layout repeated channels with single layout through all same channels
Length Tuning Routing	Critical in high-speed designs, it fine-tunes track lengths to ensure equal propagation delays across signal paths
High Speed Design with xSignals	xSignals tracks signal flow across components and layers, ensuring timing, impedance, and skew control in fast digital circuits

**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