المعهد الكوري الفلسطيني المتميز لتكنولوجيا المعلومات

PCB Design

Learning Outcomes
Upon completion of the PCB design course, the students should be able to carry out any PCB design necessary for their graduation projects
Students will also be able to create schematics from blue-prints, they will also be able to perform simple simulations
The course is intended to give the students the necessary knowledge and of PCB design steps, starting from a simple schematics, through creating new components, and all the way to down a final PCB layout ready for population
The course includes single-sided and double-sided printed circuit board design, emphasizing the drawings, standards, and the processes required to layout printed circuit board and manufacturing documentation
Recognize the technologies used in electronic industry through the practical experience gained in the course
Who Should Attend this Course
Electrical, communication, computer mechatronic, energy and industrial engineering students who want to include the PCB Design experience as part of their skill set
Those who want to peruse a career in the PCB industry by for example working as a online freelance engineer
Beginners and hobbyist who are interested in starting a career in electronics engineering

PCB Design Course Outline			
No. of Sessions	**Title**	**Course Content**	**Estimated Time**
Module-1	Introduction	Defining the main stages of PCB design from a simple idea to physical reality by having a PCB board ready for production	*4 Hours*
		Familiarize the students with EDA user interface and design environment	
		Defining the types of libraries and the function of each one	
		Getting started with a new project	
		Opening, storing and managing various schematic designs and projects	
		Understanding different patterns of differed components and their corresponding package styles and footprints	
		Description of IPC (industry standards) of every aspect of PCB design	
		Developing a comprehensive design strategy to produce a state of the art schematics and PCB layouts	
Module-2	Schematic Capture	This session walks the students through placing schematic component from various integrated libraries into Altium-designer schematics	*6 Hours*
		connection of components such as (Integrated-Circuits or Passive-components) by using Wire, Bus, Net-label, Harness-Connector or a Port	
		Compiling and checking the schematic design against warnings, errors and faults	
		Creating output reports such as BOM (Bill of Material)	
		Exporting and importing schematic data	
Module-3	PCB Layout	This session starts with PCB design process by defining the board shape	*10 Hours*
		Defining the PCB board profile and details	
		Specifying the number of signal-layers and power-planes	
		Placement of components using either: manual, interactive and automatic technique	
		Routing using either manual or interactive routing	
		Tricks and tips of best component placement and interactive routing strategies	
		Defining, setup and editing of power and ground planes (Polygons)	

Module-4	Component & Schematic Libraries	Introducing to the basic steps of Using, editing and creating new component and schematic libraries	*7 Hours*
		Creating schematic components with single and multiple parts	
		Checking the components using Schematic Library Editor reports	
		Creating PCB component footprints manually and using the PCB Component Wizard	
		Handling other special footprint requirements, including irregular pad shapes	
		Checking the component footprints using PCB Library Editor reports	
		Creating an integrated library of the new components and models	
Module-5	PCB Anatomy	Description of different types of PCB by discussing the difference between single & double sided PCB	*2 Hours*
		Understanding the difference between single-layer, double-layer and multiple-layer PCB	
		Understanding the main physical layers of a given PCB: Top-Overlay, Mechanical-Layer, Solder-Paste, Solder-Mask, Solder-Pad	
		Demonstrating the physical parts of a PCB such as: Via, Drill-hole, Copper-Tracks, Mounting-Holes and Legends	
Module-6	Fabrication Outputs	Generation of GERBER File (Gerber Setup in order to specify the accuracy, Layers and Drill-Drawings)	*2 Hours*
		Setup and generate an NC-Drill Files	
Module-7	Design Considerations	Optimizing the Copper-tracks width according to the required maximum carrying current capability	*3 Hours*
		Setup a Design rule check (DRC)	
		Description of mechanical constraints	
Module-8	Electronic Components	Showing different types of electronic components	*2 Hours*
		Categorizing electronic components according to their size, power-ratings, package style and placements	
		Understanding the difference between through-hole components and SMD components	
		Reading component's data-sheets and transferring their mechanical layout and dimensions to a new footprint library design	

Module-9	DIY PCB Prototyping	Prepare a copper clade double sided PCB and clean it properly with a piece of paper-sand	*4 Hours*
		Use the guillotine paper cutter to cut the copper clade according to the defined pcb dimensions	
		From the Gerber viewer software make a print out of i.e. actual top-layer, bottom-layer and the top-overlay layer, use a glossy paper and laser printer	
		Stick the papers of top and bottom layers on sides of copper clade PCB	
		Put in the trimmed Copper PCB inside laminating machine several times while the temperature at maximum	
		Use a small amount of water in a bowel to cool-down the PCB and slowly peal out the glossy papers	
		Merge the PCB inside a ferric chloride etchant for at least 2 hours at 50 degree Celsius	
		Use a milling machine with 1mm drill pit to drill the entire solder holes of the PCB	
		Note: A module may take more than a session to finish	