

Hardware – Complex PCB Designer

5-days session

Title	Hardware - Complex PCB designer training
Overview	<p>This training will allow current students, engineers and hardware designers to have the required skills and know-how for designing complex PCB boards. Practical Labs will be held with Altium 18 Designer (10 PCs will be available for the training session).</p> <p><u>The course/training will mainly focus on the following items:</u></p> <ul style="list-style-type: none"> ▪ Schematic design methodology ▪ PCB Design process ▪ PCB assembly ▪ Layer Stackup best practices ▪ PCB design best practices for EMI reduction ▪ Altium Designer mastering ▪ Controlled Impedance calculation ▪ Power Supply design considerations ▪ High-Speed signals routing ▪ Clock routing ▪ DDR3/4 routing techniques ▪ Signal Integrity ▪ RF signal routing ▪ Digital signal routing ▪ ADC and DAC signals routing: analog-digital planes considerations ▪ FPGA critical signals routing ▪ Processors critical signals routing ▪ Matched lines routing ▪ MTBF considerations
Labs	Designing with Altium Designer an 8-14 Layers board given the system specifications that will be communicated in the training: Labs will be starting from schematics, stackup design, component selection and placement to complete PCB routing with output Bill-of-Material and Gerber files
Audience	Hardware designers and CAO PCB routers that intend to design complex PCBs
Prerequisite	Basic knowledge of electronics
Seats	[min = 8, max = 16]
Duration	5 days – 40 hours (50% courses, 50% Labs)