

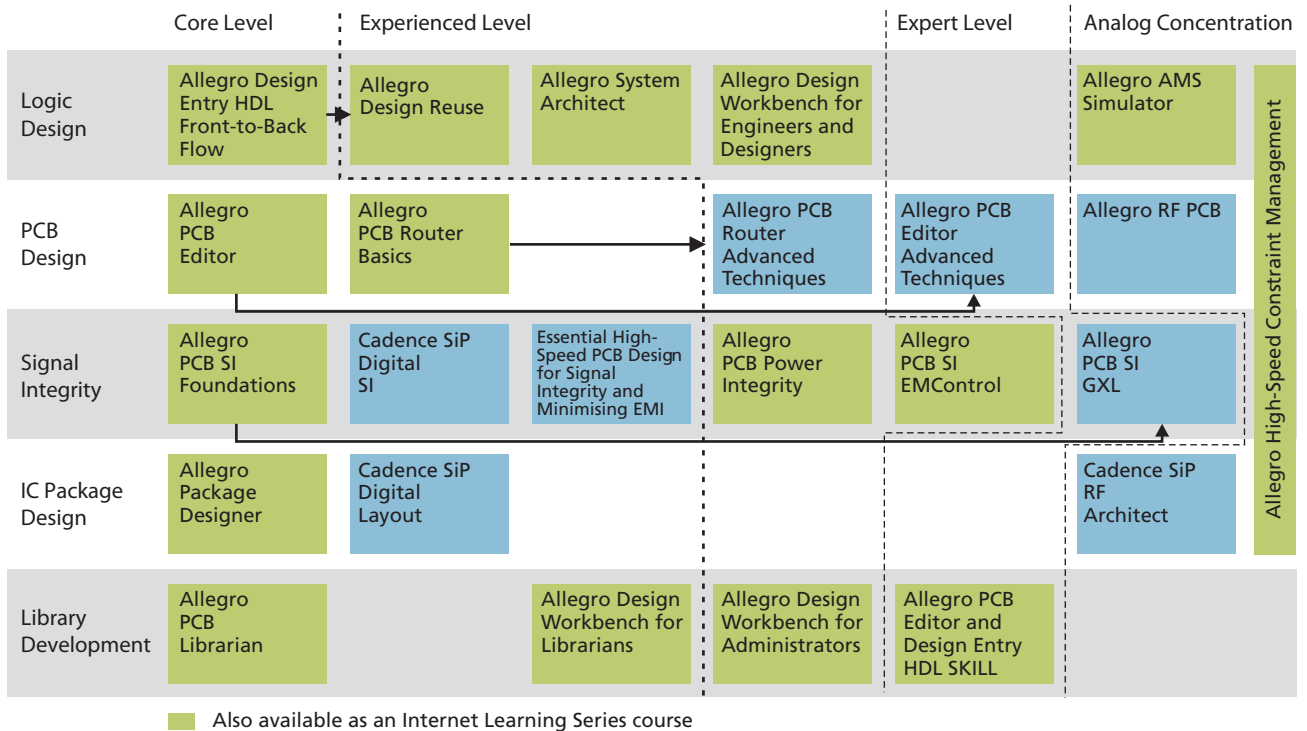


EMEA TRAINING

A direct path to increased productivity

Cadence® Education Services can help you get the most out of your investment. With its innovative and hands-on course content, state-of-the-art training centers, 200+ certified instructors, and a variety of learning modes (instructor-led, self-paced, and virtual), Cadence Education Services helps you become proficient with Cadence software, EDA languages, and design methodologies—quickly and cost-effectively.

PCB, PACKAGE DESIGN, AND ALLEGRO TECHNOLOGY



ALLEGRO DESIGN ENTRY HDL FRONT-TO-BACK FLOW

The main goal of this course is to teach you how to use Allegro® Design Entry HDL to create board-level schematic designs. The class teaches the integration between Design Entry HDL and other tools in the design flow, including Allegro PCB Editor. You will follow the actual design flow by creating a schematic and taking it all the way through board layout.

ALLEGRO SYSTEM ARCHITECT

In this course you will learn to use Allegro System Architect GXL in a front-to-back board design flow. The course introduces you to a new paradigm in design entry, making use of a spreadsheet or the table-based System Connectivity Manager.

ALLEGRO PCB EDITOR ADVANCED TECHNIQUES

The objective of this Engineer Explorer advanced class is to provide the information, hands-on experience, and methodology you require to augment your knowledge of Allegro PCB Editor. After completing the course, you will have a better understanding of the intricacies of the PCB Editor software and will be able to optimize your PCB design environment to improve productivity and reduce time to market.

ALLEGRO HIGH-SPEED CONSTRAINT MANAGEMENT

This Engineer Explorer advanced course uses a series of lectures, examples, and hands-on experience to deliver the information you need to apply and verify high-speed constraints across your design process. Examples will show you how to schedule nets, control impedance on nets, control the propagation delay from drivers to receivers, match the propagation delay of driver and receiver pairs, and more.

ALLEGRO PCB EDITOR

The first three days of the Allegro PCB Editor course covers all the necessary steps for designing a printed circuit board, from loading logic/netlist data through producing manufacturing/NC output. The last two days introduce ways to build on the basics of the tool and address board design for high-speed purposes, while touching on ways to increase productivity. The task-oriented labs show you the combined use of interactive and automatic tools.

ALLEGRO RF PCB

In this course, you learn about various RF PCB front-to-back flows. We also show you how to create an RF schematic in DE-HDL, how to merge RF blocks from ADS, and how to back-annotate the DE-HDL schematic with part, property, and connectivity changes made in the RF layout.

ALLEGRO PCB SI FOUNDATIONS

This course teaches you how to use Allegro PCB SI to successfully develop and drive design rules for high-speed designs. You will learn to develop high-speed design rules using Allegro PCB SI XL and then add the resulting physical and electrical constraints to the design through topology templates. These constraints are then used to drive the routing of nets on the PCB. You will perform pre-route and post-route signal simulations to analyze the PCB for reflection, simultaneous switching, crosstalk, and other high-speed design factors.

NEW COURSE: ESSENTIAL HIGH-SPEED PCB DESIGN FOR SIGNAL INTEGRITY AND MINIMISING EMI

In the first part, you will learn to apply basic physical principles to develop an understanding of the key issues of high-speed design, to ensure a successful design for signal integrity. The basic techniques can be applied immediately to improve PCB design, without the use of EDA signal integrity tools, but the course also provides a much-needed foundation for how to benefit from the use of such tools.

The second part extends the principles established in part one to develop techniques for design and test at frequencies above 1GHz for Gb/s serial transmission and for controlling the generation and propagation of EMI at the PCB level. Key topics include signal quality, material effects, and EMC from components to backplanes.

ALLEGRO PCB POWER INTEGRITY

Allegro PCB PI Option XL software provides a coherent methodology for the design and fabrication of power distribution systems (PDS) in high-speed PCBs. In this one-day course, you will learn to set up a board database for analysis, define a target impedance, perform single-node analysis to validate and refine your capacitor selection, perform multi-node analysis to refine your placement, create and edit capacitor models, add vendor-supplied capacitor models to a capacitor library, and assign capacitor models to an existing PCB.

ALLEGRO PCB SI GXL

This is an Engineer Explorer class that is designed around more advanced topics and exploration of the tool. Recent advances in silicon technology and high-speed communications have introduced buffers with data rates in the multi-Gigabit range. Simulation models need to be modeled for buffers and vias to enable accurate simulation results in this frequency range. Allegro PCB SI GXL introduces an approach to modeling and simulating these very high-speed structures. Using MacroModels, you can build buffer models that accurately operate in the multi-Gigabit range.

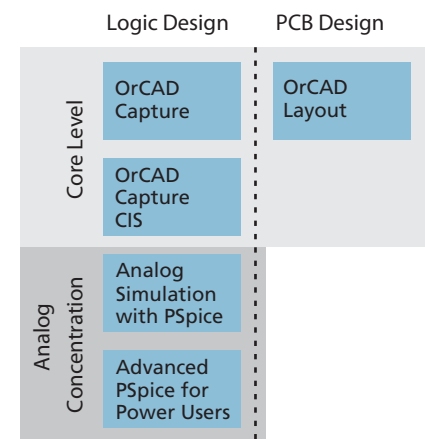
CADENCE SIP RF ARCHITECT

This class shows how to use a design flow between the Virtuoso® design environment and the SiP Layout environment. It enables designers to create a single, system-level, circuit simulation-ready schematic for an RF/analog die, SiP substrate, and packaged and embedded discrete devices.

ALLEGRO PCB LIBRARIAN

In this course, you will learn to create schematic libraries for use with Allegro Design Entry HDL and footprint libraries for use with Allegro PCB Editor. About 75% of the time is focused on front-end schematic library development; the remaining 25% of the time is spent on back-end footprint creation.

OrCAD PCB DESIGN



ANALOG SIMULATION WITH PSpice

PSpice training is a three-day course that starts with the basics of entering a design for simulation and builds a solid foundation in the overall use of the tool. This class is designed for engineers new to analog and mixed-signal simulation who are seeking maximum productivity in minimum time.

If you have any questions or would like assistance in selecting a program tailored to your needs and interests, please contact us at eur_training@cadence.com
Education Services EMEA