

CB-Distribution Training schedule 2012

CB-Distribution | Mosweg 42 | 7556 PG Hengelo
Tel. +31 0 74 256 1424 | Fax. +31 0 74 256 1434
<http://www.cb-distribution.nl> | The Netherlands

cādence[™]
CHANNEL PARTNER

Table of contents

Training overview 2012	3
OrCAD Capture CIS / Allegro Design Entry CIS	4
OrCAD/Allegro PCB-Editor Introduction	5
Orcad/Allegro PCB-Editor	6
OrCAD/Allegro Front-to-Back Flow	8
Analog simulation with Pspice AD / AMS Simulator	9
Advanced simulation with Pspice AD / AMS Simulator	10
Pspice / AMS Simulator advanced analysis	11

Training overview 2012

Name	Date
Orcad Capture CIS Allegro Design Entry CIS	6 February
	16 April
	18 June
	17 September
	5 November
Orcad PCB-Editor Introduction Allegro PCB-Editor Introduction	7 February
	17 April
	19 June
	18 September
Orcad Front-to-Back flow Allegro Front-to-Back flow	6 and 7 February
	16 and 17 April
	18 and 19 June
	17 and 18 September
	5 and 6 November
Orcad PCB-Editor Allegro PCB-editor	30 and 31 January
	19 and 20 March
	4 and 5 June
	1 and 2 October
	19 and 20 November
Analog Simulation with Pspice AD Analog Simulation with AMS Simulator	12 and 13 March
	8 and 9 October
	26 and 27 November
Advanced Simulation with Pspice AD Advanced Simulation with AMS Simulator	15 March
	7 June
	20 September
	22 November
Pspice Advanced Analysis AMS Simulator Advanced Analysis	19 April
	11 December

Registration can be done via the CB-Distribution web-site.

CB-Distribution | Mosweg 42 | 7556 PG Hengelo
Tel. +31 0 74 256 1424 | Fax. +31 0 74 256 1434
<http://www.cb-distribution.nl> | The Netherlands

cādence[™]
CHANNEL PARTNER

OrCAD Capture CIS / Allegro Design Entry CIS

Duration: 1 day
Time: 9:00 – 17:00

Description: This one-day, hands-on training course covers the full range of front-end design processes, from setting up design templates to creating a netlist for board layout, as well as part management. Class size is limited to facilitate maximum interaction between you and the instructor. You are encouraged to bring specific real-world design questions with you to class.

The highlights of this class include the following:
Learn how to use design templates, create a new design, build parts, draw schematics and validate them using DRC.

Audience: Engineers, designers and technicians engaged in schematic design who are seeking maximum productivity in a minimum amount of time.

Prerequisites: Proficiency with Windows and standard Windows applications.

Course agenda:

- Structure of Capture
- Capture settings and preferences
- Setting up a project
- Building a multi-sheet schematic
- Simple and Complex Hierarchies
- DRC check and Annotation
- Part properties
- Configuration of CIS
- Libraries management both symbols and footprints
- Variants
- Bill of Materials
- ActiveParts
- End discussion

OrCAD/Allegro PCB-Editor Introduction

Duration: 1 day
Time: 9:00 – 17:00

Description: This one day course will cover all the necessary steps for designing a printed circuit board, from loading logic/netlist data through producing manufacturing/NC output. Learn about the user interface for the Orcad/Allegro PCB Editor

Audience: This course is for printed circuit designers and design engineers who have an understanding of PCB layout and manufacturing considerations.

Prerequisites: You should have a working knowledge of PCB design and a basic understanding of PCB manufacturing processes. You must also be proficient with using Windows and standard Windows applications.

Course Agenda:

- User Interface and Configuration
- Rule Manager from Capture to PCB Editor
- Set up rules
- PCB templates
- Location of components
- Routing
- Copper Fields
- Postprocessing including reports, silkscreen, Gerber / ODB ++, drill, etc..

Orcad/Allegro PCB-Editor

Duration: 2 days
Time: 9:00 – 17:00

Description: PCB Editor is a central part of the OrCAD PCB Designer and Allegro PCB design solutions. This two day course will cover all the necessary steps for designing a printed circuit board, from loading logic/netlist data through producing manufacturing/NC output

Audience: This course is for printed circuit designers and design engineers who have an understanding of PCB layout and manufacturing considerations.

Prerequisites: You should have a working knowledge of PCB design and a basic understanding of PCB manufacturing processes. You must also be proficient with using Windows and standard Windows applications.

Course Agenda:

- PCB Editor User Interface
 - Identify the user interface components
 - Navigate within the PCB Editor window and access UI features to tailor the tool for individual needs
- Managing the PCB Editor Work Environment
 - Create and use scripts
 - Use the Control Panel to locate board database objects and report information about them
- Padstacks
 - Create a flash symbol user for thermal relief
 - Use the Pad Designer to create padstacks for a number of typical pins, such as through-hole and surface-mount pads
- Component Symbols
 - Use the Package Symbol Wizard to create a Package Symbol
 - Use the Symbol Editor to create a surface-mount package
- Board Design Files
 - Use the Mechanical Symbol Editor to create Board Mechanical Symbols
 - Use the PCB Design Editor to create a Master Board Design
- Importing Logic Information into PCB Editor
 - The Logic information can come from Concept, Capture, Third party
- Setting Design Constraints
 - Assign Standard & Extended Design Rules for Spacing & Physical Dimensions
 - Add, Change & Delete Properties of Components & Nets

- Component Placements
 - Use Floorplanning to organize the placement of components with the same ROOM property
 - Assign Reference Designators for preplaced parts
 - Interactively place components
- Advanced Placement
 - Turn rastnets on and off to selectively place components
 - Use interactive and auto swapping for pins and gates
 - Perform cross placement between PCB Editor and DE HDL or DE CI
- Routing and Glossing
 - Define and Display etch grids used for Routing
 - Add and Delete connect lines and vias
 - Prepare for Autorouting by creating preliminary embedded planes
 - Route net connections with PCB Router
 - Use Slide and Replace Etch to improve routing
 - Use the Cut option in conjunction with other etch editing commands
 - Use Gloss to automatically clean up the routed etch in the Design
- Copper Areas and Positive or Negative Plans
 - Learn how to generate Positive and Negative Plans
- Preparing for Post Processing
 - Rename Reference Designators and backannotate changes made in the PCB Editor to the Schematic Environment
- Preparing the Board Design for Manufacturing
 - Generate and Edit Silkscreen layers
 - Use Reports available in PCB Editor
 - Setup the Design File for Artwork - Preview before plotting
 - Generate Drill Symbols and legend for fabrication drawing
 - Create check plots
 - Output a Drill file used for drilling the board holes in manufacturing

OrCAD/Allegro Front-to-Back Flow

Duration: 2 days
Time: 9:00 – 17:00

Description: This two day course will cover all the necessary steps for designing a printed circuit board, from schematic capture using Orcad Capture CIS to PCB-editor through producing manufacturing/NC output

Audience: Engineers, designers and technicians engaged in both schematic design and PCB layout. Who are seeking maximum productivity in a minimum amount of time.

Prerequisites: You should have a working knowledge of PCB design and a basic understanding of PCB manufacturing processes. You must also be proficient with using Windows and standard Windows applications.

Course Agenda Capture Part:

- Structure of Capture
- Capture settings and preferences
- Setting up a project
- Building a multi-sheet schematic
- Simple and Complex Hierarchies
- DRC check and Annotation
- Part properties
- Configuration of CIS
- Libraries management both symbols and footprints
- Variants
- Bill of Materials
- ActiveParts

Course Agenda PCB-Editor Part:

- User Interface and Configuration
- Rule Manager from Capture to PCB Editor
- Set up rules
- PCB templates
- Location of components
- Routing
- Copper Fields
- Postprocessing including reports, silkscreen, Gerber / ODB ++, drill, etc.
- End discussion

CB-Distribution | Mosweg 42 | 7556 PG Hengelo
Tel. +31 0 74 256 1424 | Fax. +31 0 74 256 1434
<http://www.cb-distribution.nl> | The Netherlands

cādence[™]
CHANNEL PARTNER

Analog simulation with Pspice AD / AMS Simulator

Duration: 2 days
Time: 9:00 – 17:00

Description: The PSpice training is a two-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

Audience: Engineers seeking maximum productivity in minimum time
Engineers new to analog and mixed-signal simulation

Prerequisites: Proficiency with Windows and standard Windows applications.

Course Agenda:

- Building a design for simulation
- Setting up and running DC bias point analyses
- Setting up and running DC and AC sweep analyses
- Viewing simulation results in the Probe window
- Setting up sources and using markers
- Creating and simulating a text netlist
- Accessing the stimulus editor using VSTIM, ISTIM, and DIGSTIM
- Running transient analysis
- Working with local and global libraries
- Examining common simulation errors
- Creating linear and non-linear transformers
- Setting up and running parametric analysis
- Creating a subcircuit
- Creating parts for simulation models
- Performing temperature analysis
- Configuring and running Monte Carlo analysis
- Simulating with hierarchical blocks and symbols
- Simulating using Analog behavioral modeling
- Using digital components in a design
- Combining analog and digital components in designs
- Using performance analysis and creating goal functions
- Setting up and running worst-case analysis
- Setting up and running noise analysis

Advanced simulation with Pspice AD / AMS Simulator

Duration: 1 day
Time: 9:00 – 17:00

Description: This one day training is for existing users who like to know more on the advanced features of Pspice AD

Audience: Engineers seeking maximum productivity in minimum time

Prerequisites: Know the basic analysis of Pspice AD. Proficiency with Windows and standard Windows applications.

Course Agenda:

- Building a design for simulation
- Accessing the stimulus editor using VSTIM, ISTIM, and DIGSTIM
- Working with local and global libraries
- Examining common simulation errors
- Setting up and running parametric analysis
- Performing temperature analysis
- Configuring and running Monte Carlo analysis
- Simulating using Analog behavioral modeling
- Using digital components in a design
- Combining analog and digital components in designs
- Using performance analysis and creating goal functions
- Setting up and running worst-case analysis
- Setting up and running noise analysis

Pspice / AMS Simulator advanced analysis

Duration: 1 days
Time: 9:00 – 17:00

Description: The PSpice advanced analysis training is a two-day course extensive covering sensitivity analysis, circuit optimization, Monte Carlo analysis, and smoke / stress analysis

Audience: Engineers seeking maximum productivity in minimum time

Prerequisites: Know the basic analysis of Pspice AD. Proficiency with Windows and standard Windows applications.

Course Agenda:

- Introduction to PSpice AA
- Model Understanding in PSpice AA
- Measurements
- PSpice AA Sensitivity Analysis
- Circuit optimization and optimization algorithms
- AA PSpice Monte Carlo Analysis / tolerances
- Smoke / stress analysis