

## 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**<sup>™</sup>  
CHANNEL PARTNER