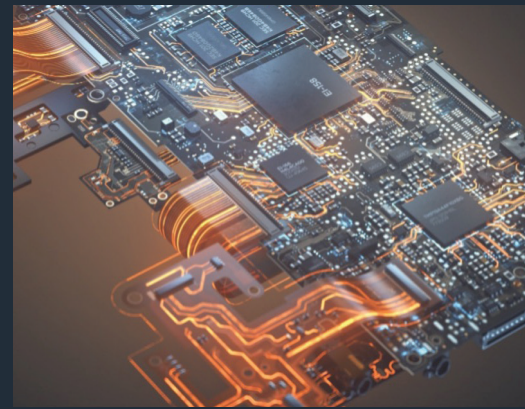
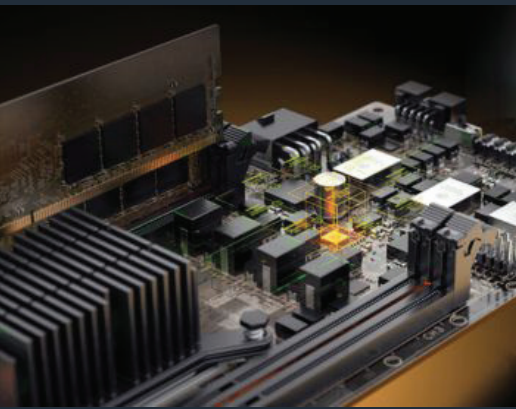


**100% UP-TO-DATE
WITH TRAININGS**



Training PCB Design Essentials in Altium Designer

Get all you need to know to successfully design PCBs. Reduce design time and costs through faster learning and increased productivity with Altium Designer.

TRAINING

Successful PCB Design in Altium Designer

Description

Increased productivity with Altium Designer starts with a sound knowledge of the basic tools and workflows in the design environment. This training course provides instructions on the entire design capture process, starting schematic capture, project, and library creation, moving to PCB layout, and finally generating production data. Whether you are starting as a designer or need a refresher to get up to speed, this training will make you a professional with Altium Designer helping you to efficiently use the tool by covering the following topics.

Content

- Altium Designer UI Overview
- Library creation
- Schematic capture and validation
- PCB setup
- Component placement
- Layout routing
- Design rule checks and PCB validation
- Outputs generation: Bill of Material, manufacturing data, assembly data, documentation

Aim

Ability to create a PCB from schematic to PCB layout and production data generation

For more information contact us directly:



Computer Controls AG
hello@ccontrols.ch
www.ccontrols.ch/ch_en/contact-us

COMPUTER
CONTROLS