

## Information about the workshop

This workshop goes through the basics of Capture CIS(Schematics), PCB design and PSpice(Simulation on schematic).

### Workshop content

This workshop contains everything needed for at beginner to start with OrCAD. The workshop goes through the basics of OrCAD and contains different exercises for both schematics, PCB Design and simulation with PSpice.

The workshop also goes through a bit more advanced stuff such as the highspeed design functionalities.

What you will learn:

- How to create a circuit in Capture
- How to simulate on your circuit with PSpice
- How to manage your components with CIS(Component Information Search) and CIP(Component Information Portal)
- How to do a smoke analysis on your schematics.
- How to create a PCB design from A-Z.
- How to see your board in a 3D view
- How to use the 3D canvas to optimize your design
- How to design a highspeed PCB
- The highspeed PCB design functionalities

### Why this workshop?

This workshop will help you to start a schematic, simulation and PCB design the correct way. This is created for all our customers and free users of OrCAD and Allegro.

### Who is this workshop created for?

This workshop is usually meant for the beginner level customer/user. The highspeed exercise can be used by advanced users.