



Pulsonix Training

Location:

Pulsonix training is provided either on-site or off-site depending on your requirements at the time. Our course tutors are knowledgeable and will offer practical experience. Where the course is made outside a general workshop environment, we encourage users to attend with their own designs so we can work through their particular problems and issues.

Course Materials:

Each delegate is provided with a professionally printed set of course notes which are spiral bound and cover the course presentation and any additional course notes. These are handed out at the start of the course and are for the trainee to use to make notes in and retain after the course.

Certificates:

Each course member will be sent a certificate following attendance of each course.

Duration:

This depends on trainees experience with Schematic or PCB design CAD tools (or AutoCAD) and general Office applications. Some users have minimal experience but may have used some CAD tools at University say but with no PCB design experience. The course will also include why items are done a particular way, such as Track spacings and clearances for example etc.

Two days is enough to get started and cover the basics of the product for someone who has seen or used a CAD product before. This will get a group of people up to speed with most of the generally used features within the product. If users have less experience, then an extra day should be included to ensure all the basics are absorbed.

We find that when courses have been run over one day, this isn't enough for the attendee to learn the features, and remember them the next day. There is a lot to cover and two days provides enough time for reflection of the course the previous day. This is particularly important for retention of the information provided.

Course Contents:

Regardless of the duration the contents will attempt to cover in some level of depth:

Introduction to the User Interface

- GUI - User interface, Docking windows, Design Area, Status bar, Browser windows, Workbook mode, Find, Toolbars and Properties
- Product fundamentals - Move/Edit/Rotate/Copy/Mirror/Groups/Delete and interaction
- Mouse usage - Selection, Framing, Pan and Zoom
- Units, Working area and Grids
- Technology and profile files

Designing with the Schematic editor

- Drawing sheets
- Adding components from the Library. Adding homogenous and heterogeneous Parts. Adding Parts from Add Component and the Parts Browser
- Using the Component bin
- Adding Connections - Nets, Net Names, Net Classes
- Adding Busses
- Adding Connectors
- Adding Documentation symbols for power and ground, sheet references, star points etc.
- Electrical rules checking
- Printing the Schematic to paper and PDF
- Transfer of the Schematic to the PCB design editor. Resolving errors and warnings.
- Hierarchy (optional if required)

Libraries – creation and management

- Library creation
- Creation of Symbols, PCB footprints and Parts
- Using the library creation wizards
- Using the Library Manager
- Styles and Technology files
- Multi-gated Parts, connectors, multi-pin mapping, wires and relays

Using the PCB design editor

- Adding board outlines, using the tools like Type Coordinate and Type Offset to do this
- Importing DXF files to create a board outline
- Adding components, translate from Schematic and interactively adding Parts
- Using the component bin
- Placing components
- Adding/using areas and pour templates
- Routing the design
- Copper pour
- Design Rules Checking DRC - Online and Batch checking
- Spacing rules
- Net Classes/Net Styles
- Technology files, styles and naming
- Slotted pads and user defined pad shapes
- Construction lines
- Replace Part and back annotation
- Forward annotation and Engineering Change Orders (ECOs)
- Manufacturing outputs from PCB using CAM Plot
 - CAM Plots for Gerber, Excellon, PDF and Windows
 - Combining layers, using the plot wizard, Step & repeat, Scaling plots and Plot Settings
 - OBD++ Output
- Reports and Report Maker
- 3D Viewer, setting up libraries and using
- STEP I/O, fundamentals of interfacing to MCAD systems