PCB EDITOR ESSENTIALS
TRAINING SYLLABUS

SYLLABUS DETAILS (2-DAY):

Day One

Lesson 1: Cadence® OrCAD PCB Editor User Interface
In this lesson, the user interface components of PCB Editor will be identified, users will learn how to navigate within the PCB Editor window and access UI features to tailor the tool to their individual needs

Lesson 2: Managing the OrCAD PCB Editor Work Environment
In this lesson, users will learn how to control the color and visibility of objects, use the Find Filter to locate objects and report information about them.

Lesson 3: Padstacks
In this lesson, users will learn how to use Pad Editor to create padstacks for typical pins, such as through-hole and surface-mount pads.

Lesson 4: Component Symbols
In this lesson, users will learn how to use the Package Symbol Wizard and the Package Symbol Editor to create package symbols (footprints).

Lesson 5: Board Design
In this lesson, users will learn how to create a board outline in the Mechanical Symbol Editor, how to import a DXF file of the board outline and how to create a Template board file using PCB Editor.

Lesson 6: Importing Logic Information into OrCAD PCB Editor
In this lesson, users will learn how to import logic information from an OrCAD Capture Schematic. They will learn the key setup choices for importing logic information into the PCB Editor layout environment.

Lesson 7: Setting Design Constraints
In this lesson, users will explore the design rules system and learn to apply physical and spacing design rules. They will also learn to add, change and delete properties of components and nets.

EMA | Design Automation®

Day Two

LESSON 8: Component Placement
In this lesson, users will do floorplanning of components using the ROOM property. They will also learn to use various commands to place components.

LESSON 9: Additional Placement Features
In this lesson, users will learn how to selectively turn on and off ratsnest lines, modify padstacks within a design, create and copy fanouts, create a library from a board file and perform cross placement between an OrCAD Capture schematic and the PCB Editor board file.

LESSON 10: Interactive Routing
In this lesson, users will learn how to define and display routing grids, add and delete traces and vias, how to use the slide command and how to use Glossing to clean up routed traces.

LESSON 11: Copper Areas and Positive Planes
In this lesson, users will learn to generate planes, split planes and copper pours. They will also learn to work with the Shape Editor application mode.

LESSON 12: Preparing for Post Processing
In this lesson, users will check their design for DRCs and unconnected pins, ready shapes for final artwork, rename reference designators and back-annotate changes made in PCB Editor to the OrCAD Capture schematic.

LESSON 13: Preparing the board Design for Manufacturing
In this lesson, users will learn generate and edit silkscreen layers, create reports, setup, preview and generate artwork files, generate NC Drill data and generate an IPC-2581 database for manufacturing.

EMA | Design Automation®