



SCHEMATIC + PCB
FRONT-TO-BACK FLOW

TRAINING SYLLABUS

SYLLABUS DETAILS (3-DAY):

Day One: Capture Essentials

Lesson 1: Getting Started with OrCAD Capture

In this lesson design flow will be discussed along with how to launch Capture; open an existing project; explore the User Interface; understand Project Manager structure; and select and edit objects.

Lesson 2: Setting Up Your Environment

In this lesson you will work with setting User Preferences and developing a Design Template.

Lesson 3: Creating Part Libraries

This covers the ability to re-characterize a circuit for a different set of performance goals. The Optimizer is given a list of components to vary in a circuit to achieve a different set of measured goal responses, continually altering these component values until the new goal sets are achieved (if that is possible with the given circuit arrangement). It can also operate on a given set of curves to essentially manipulate device values/parameters for a curve-fit situation.

Lesson 4: Creating Homogeneous Parts

This covers the about 98% of the same capability as the PSpice A/D tool, with the exception of the ability to split the positive versus negative tolerance ranges independently, and the ability to plot a cumulative histogram type of performance.

Lesson 5: Creating a Heterogeneous Part

This covers the ability to manipulate any number of device parameters of a circuit. With PSpice A/D, you only have the ability to manipulate (i.e. sweep) two parameters in a given simulation. With the Parametric Plotter, that limitation is lifted and allows any number of parameter manipulations in a simulation.

Day Two: PCB Editor Essentials

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.

Day Three: PCB Editor Essentials

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.

Day Three: PCB Editor Essentials (Con't)

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.