NI Ultiboard Basics: PCB Layout Course

Overview
The NI Ultiboard Basics: PCB Layout course introduces you to the Ultiboard environment. This course prepares you to transfer Multisim schematic netlists to Ultiboard and design a printed circuit board for export to production. Topics include design setup, precise part and trace placement, trace routing and the optimization and use of autoplacement and autorouting. Students will learn how to prepare final designs for manufacturing and export to industry-standard file formats. The hands-on approach of the Ultiboard Basics course steps you through PCB creation, from Multisim netlists to Gerber files, teaching the application of necessary skills to quickly become productive with Ultiboard.

Duration
Instructor-led classroom: One (1) day
Instructor-led online: Two (2) half days

Audience
• New users and users preparing to layout, route and export PCB designs using Ultiboard or Circuit Design Suite
• Users and technical managers evaluating Ultiboard or Circuit Design Suite

Prerequisites
• Experience with Microsoft Windows®
• Basic knowledge of Electronics theory
• Basic knowledge of PCB Layout theory
• Basic knowledge of Multisim

NI Products Used During the Course
• Ultiboard Power Professional
• Multisim Power Professional

After attending this course, you will be able to:
• Understand the features of the Ultiboard user interface
• Transfer designs from Multisim to Ultiboard
• Apply efficient part placement procedures
• Apply efficient trace placement procedures
• Create custom footprints (landpatterns)
• Create copper areas and power planes
• Work with design constraints
• Forward and Back Annotate changes to and from Multisim
• Prepare your design for manufacturing

Registration
Register online at ni.com/training or call (800)433-3488 Fax: (512)683-9300 info@ni.com

Outside North America, contact your local NI Office. Worldwide Contact Info: ni.com/global

Part Number
910758-xx
-01 NI Corporate or Branch
-11Regional
-21 Onsite (at your facility)
-69 Online

Suggested Next Courses
Multisim Basics: Schematic Capture and Simulation
NI Ultiboard Basics: PCB Layout Course Outline

Introduction
This lesson introduces the Ultiboard graphical user interface (GUI) and configuration options. Topics include:
- What is Ultiboard?
- The Design Process
- The Ultiboard GUI
- Setting Environment Preferences
- Spreadsheet View
- Selection Filter
- Workspace Area

Transfer and Board Design Setup
This lesson explains how to transfer designs from Multisim to Ultiboard and ways of creating or placing a board outline for your PCB. Topics include:
- Transfer from Multisim
- Check for Virtual Components
- Board Layer Technology
- Via Support
- Creating a Board Outline
- Import a Board Outline from DXF
- Board Wizard

Parts and Placement
This lesson explains how to place parts inside the board outline and how to create, edit and manage footprints using the database. Topics include:
- Manual Part Placement
- Using the Part Sequencer
- Using the Autoplacer
- Placement Tools
- Keep-in / Keep-out Areas
- Footprints
- Footprint Properties
- In-Place Part Edit
- Database Manager
- Footprint Creation
- Component Wizard

Design Setup before Routing
This lesson covers working with and changing the netlist; you also learn how to propagate changes to Multisim and set up design constraints. Topics include:
- Netlist Editor
- Forward and Back Annotation
- Pin and Gate Swap
- Trace Settings and Clearance Setup
- Renumber RefDes

Traces and Copper Areas
This lesson introduces all the trace-routing methods and guidelines for working with copper. Topics include:
- The Connection Machine
- Follow-me Router
- Manual Routing
- Autorouter
- Placing Vias
- Working with Traces
- Powerplanes and Copper Areas
- Thermal Relief
- Polygon Splitter
- Net Bridges

Preparing for Manufacture
This lesson explains final steps you can take to better prepare your design for manufacturing and how to export your design to Gerber format. Topics include:
- Corner Mitering and Tear Dropping
- Connectivity and Design Rules Check (DRC)
- Text and DXF Import
- Mounting Holes and Dimension Lines
- Gerber and NC Drill
- Parts Centroid and Bill of Materials
- Gerber Viewer
- 3D View

ni.com/training