

NI Multisim Basics: Schematic Capture and Simulation

Overview

The National Instruments Circuit Design Suite (Multisim and Ultiboard) equips the professional PCB designer with world-class tools for schematic capture, interactive simulation, board layout, and integrated test. This course teaches the fundamentals of the Multisim integrated capture and simulation design environment. Students learn how to build a schematic and evaluate circuit performance through interactive simulation and advanced analyses while creating custom capture and simulation parts. Educators also benefit from additional customizable content specifically for electronics education. At the end of the NI Multisim Basics course, students can design and simulate a circuit that is ready for board layout and routing. The hands-on format of this course is the quickest way to become productive with Multisim.

Duration

Two (2) Days

Audience

- New users and users preparing to capture and simulate circuits using Multisim or Circuit Design Suite
- Users and technical managers evaluating Multisim or Circuit Design Suite

Prerequisites

- Experience with Microsoft Windows®
- Basic knowledge of Electronics theory

NI Products Used During the Course

- Multisim Power Professional 11.0
- Ultiboard Power Professional 11.0

After attending this course, you will be able to:

- Understand the Multisim user interface
- Use Multisim to capture circuit schematics
- Use interactive simulation to check your design
- Use virtual instruments and analyses
- Apply modular design with subcircuits, hierarchical blocks or multi-page designs
- Create custom title blocks
- Properly document your circuit designs
- Work with design variants
- Create custom components
- Co-simulate MCU projects along with SPICE
- Transfer your design to PCB Layout software

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

910756-xx
-01 NI Corporate or Branch
-11 Regional
-21 Onsite (at your facility)
-69 Online

Suggested Next Courses

- Ultiboard Basics: PCB Layout

NI Multisim Basics: Schematic Capture and Simulation

Schematic Capture

This lesson introduces the Multisim graphical user interface (GUI) and the Schematic Capture process to start your circuit design. Topics include:

- What is Multisim?
- The Design Process
- Setting environment preferences
- The Multisim GUI
- Components
- Placing components
- Wiring components

Simulation and Virtual Instruments

This lesson explains the simulation capabilities of Multisim as well as the concept-check features. Topics include:

- Types of simulation
- What are simulation models?
- Virtual Instruments
- Measurement Probe
- Circuit Wizards

Analyses

This lesson teaches how you can perform advanced analyses to verify your circuit design. Topics include:

- Analyses in Multisim
- Configuring analyses
- Using custom expressions
- The Grapher

Results and Post-processing

This lesson introduces the Post-processor and ways to use results from analyses to further calculate and examine data. Topics include:

- The Post-processor
- Configuring the Post-processor
- Viewing results

Advanced Schematic Capture

This lesson introduces the features that help you create professional-looking schematics and correctly-documented circuit design. Topics include:

- Graphic Annotations
- Placing Comments
- Using the Description Box
- Title Blocks
- Title Block Editor

Communication and Transfer

This lesson teaches best practices for creating meaningful reports and methods for transferring your design to PCB Layout software. Topics include:

- Reports in Multisim
- Bill of Materials
- Transfer to PCB Layout
- Netlist Export

Projects and Design Sharing

This lesson teaches you to professionally architect your design; you learn to use design blocks and buses, check for electrical failures and use the Project View.

Topics include:

- Design Blocks
- Hierarchical View
- Subcircuits and Hierarchical Blocks
- Multi-page Design
- Electrical Rules Check (ERC)
- Buses
- Team Design with Multisim
- Project Management with Multisim

Multisim Basics: Schematic Capture and Simulation

Design Variants

This lesson introduces Design Variants, teaching you how to properly configure and use variants, as well as use cases. Topics include:

- What are Variants?
- Variant Manager
- Using the Design Toolbox
- Variant Mapping

Custom Components

This lesson teaches the process for customizing or editing components and creating new components.

Topics include:

- Tools for editing components
- Component Properties
- The Component Wizard
- The Symbol Editor
- Model Makers
- Database Manager
- Using existing databases
- Component import and export

MCU Co-simulation

This lesson introduces the Multisim MCU Module.

Topics include:

- The MCU Module
- MCU Workspace
- Code Manager
- Source Code Editor
- Memory View
- Debugging Features

Educational Features*

This optional lesson introduces the Educational features of Multisim, mostly suitable for an academic classroom setup. Topics include:

- Using Component Faults
- Rated Virtual Components
- Ladder Diagrams
- Forms
- Circuit Description Box
- Circuit Restrictions
- 3D Breadboard
- 3D ELVIS Breadboard

** Typically covered when the audience is Academically-oriented, optional when the audience is Professional. Topics discussed are available in the Educational edition of Multisim, however not all of them are available in the Base, Full or Power Pro edition.*