

RELEASE NOTES

NI Circuit Design Suite

Version 10.0

These release notes contain system requirements for NI Circuit Design Suite 10.0, and information about product tiers, new features, documentation resources, and other changes since Multicap 9.0, Multisim 9.0, Ultiboard 9.0, and Ultiroute 9.0.

NI Circuit Design Suite includes the following familiar Electronics Workbench software products: NI Multisim, NI Ultiboard, and the NI Multisim MCU Module (formerly MultiMCU).

Contents

Installing NI Circuit Design Suite 10.0	2
Minimum System Requirements	2
Installation Instructions	2
Product Activation	3
What's New in NI Circuit Design Suite 10.0	3
Mouse-Click Support for Interactive Components	4
Convergence Assistant	4
Increased Quality and Breadth of the Component Database	4
New Components from Leading Manufacturers	4
Generic Power Simulation Parts	4
Bipolar Sources	5
Graphical LCD	5
Single Symbol Representations of Standard Logic Components	5
Enhancements to Passive Components	5
Extended SPICE Modeling Capabilities	5
Parameterized SPICE models	5
Improved Support of Behavioral Sources	5
Support for BSIM 4 Parameters	6
Enhanced Data Visualization	6
Advanced Functionality of Static Probes	6
Add Traces to Grapher after Running Analyses	6
Display Initial Conditions on the Schematic	6

Current Probe Instrument	6
Enhanced Analysis Capabilities.....	6
Extended Language Support and File Management in the MCU Module.....	7
Improved Export to Mentor Graphics PADS®	7
Improvements to Speed and Quality of NI Ultiboard	7
Advanced Options for Exported Data Interpolation	7
Miscellaneous Features	8
Unicode Characters.....	8
NI Installation and License Management.....	8
Product Tier Details.....	8
Documentation	14

Installing NI Circuit Design Suite 10.0

This section describes the system requirements and installation procedures for NI Circuit Design Suite.

Minimum System Requirements

To run NI Circuit Design Suite 10.0, National Instruments recommends that your system meet the following requirements:

- Windows 2000 Service Pack 3 or later, or Windows XP
- Pentium 4 class microprocessor or equivalent (Pentium III class minimum)
- 512 MB of memory (256 MB minimum)
- 1.5 GB of free hard disk space (1 GB minimum)
- Open GL® capable 3D graphics card recommended (SVGA resolution video adapter with 800×600 video resolution minimum, 1024×768 or higher preferred)
- To develop custom LabVIEW based instruments for use in Multisim, LabVIEW 8.0.x or higher is required

Installation Instructions

The NI Circuit Design Suite 10.0 installer installs all products in the suite: Multisim, Ultiboard, and the Multisim MCU Module.

National Instruments recommends that you close all open applications before you install NI Circuit Design Suite.

Unless you specify another location during installation, the NI Circuit Design Suite installation program copies files to <Program Files>\National Instruments\Circuit Design Suite 10.0 after you complete the following steps:

1. Insert the NI Circuit Design Suite CD into the CD-ROM drive. If the CD startup screen is not visible, select **Run** from the Windows **Start** menu and run `setup.exe` from your CD.
2. Follow the instructions in the dialog boxes.

Product Activation

When you run a product in the NI Circuit Design Suite for the first time, it will prompt you to activate a license for that product.

Note: To run the Multisim MCU Module, place a component from the MCU Module group on a Multisim circuit or open a Multisim file that contains a component from the MCU Module group.

If you do not activate a valid license, the product will run in Evaluation Mode and continue to prompt you to activate a license on each subsequent run. Evaluation Mode is valid for 30 days following the first run of the product.

For information about how to activate your software product, please refer to the *Activation Instructions for National Instruments Products Note to Users* included with your NI Circuit Design Suite 10.0 package.

What's New in NI Circuit Design Suite 10.0

This document describes the following new features of NI Circuit Design Suite 10.0:

- Mouse-Click Support for Interactive Parts
- Convergence Assistant
- Increased Quality and Breadth of the Component Database
- Extended SPICE Modeling Capabilities
- Enhanced Data Visualization
- Extended Analysis Capabilities
- Extended Programming and File Management in the MCU Module
- Improved Export to Mentor Graphics PADS®
- Improvements to Speed and Quality of NI Ultiboard
- Advanced Options for Exported Data Interpolation
- Miscellaneous Features

Mouse-Click Support for Interactive Components

NI Multisim 10.0 lets you use your mouse to control interactive components during simulation. You can click on switches to toggle them, push keypad buttons with the mouse, and adjust the value of the variable components, such as potentiometers, with a slider bar. You may also continue to use keyboard controls for these devices.

Convergence Assistant

The Convergence Assistant adjusts simulation settings when a "Time Step Too Small" error occurs during interactive simulation. The assistant adjusts the minimum number of parameters required in order to allow convergence of the simulation. The assistant adjusts the following parameters:

1. Initial Condition
2. TMAX
3. RELTOL
4. RSHUNT
5. ITL1
6. Integration method
7. GMIN

Increased Quality and Breadth of the Component Database

NI Multisim 10.0 has a number of new additions and improvements to the component database. These include: around 1,000 new components from leading manufacturers, generic power simulation parts, new bipolar sources, a Graphical LCD, single symbol representations of standard logic components, and improvements to passive components.

New Components from Leading Manufacturers

NI Multisim 10.0 has approximately 1,000 new components with models from Analog Devices, Texas Instruments, and Linear Technologies. These additions include symbols, models, and IPC-standard landpatterns. Components include operational-amplifier, comparator, and voltage reference models.

Generic Power Simulation Parts

NI Multisim 10.0 includes models for all power simulation parts found in the "Switch-Mode Power Supply SPICE Cookbook" by Christophe Basso. These components include Buck, Boost, Buck-Boost, and PWM controllers. Their models include voltage and current mode controlled devices, and models for average and detailed transient operation.

Bipolar Sources

New bipolar pulse sources include both current and voltage sources.

Graphical LCD

A Graphical LCD is available for users who purchase the MCU Module in conjunction with NI Multisim. The command system for the Graphical LCD follows the Toshiba T6963C. The graphical LCD is a two-color device with 256 x 256 pixel display resolution. This device supports three modes of operation: text-only, graphics-only, and mixed text and graphics.

Single Symbol Representations of Standard Logic Components

In addition to the multi-section component representation of standard logic components such as logic gates and flip-flops, the component database now includes single symbol representations of common components. These single-symbol representations show the power and ground pins of these devices.

Enhancements to Passive Components

You can now change the value of any resistor, capacitor, or inductor placed on the schematic without replacing it. You can also assign a landpattern to any passive component. You can assign information about the type of component, for instance metal-oxide, and this information propagates to the Bill of Materials. The tolerance of the components is automatically available for Monte-Carlo and Worst Case analyses, and you can edit the tolerances in the spreadsheet.

An advanced non-linear inductor model lets you define the inductor characteristics based on datasheet values.

Extended SPICE Modeling Capabilities

NI Multisim 10.0 introduces enhancements to its SPICE modeling capabilities, including parameters in SPICE subcircuit models, improved support of behavioral sources, and support for BSIM 4 parameters.

Parameterized SPICE models

You may now define parameters in the .subcircuit line of SPICE macro-models in NI Multisim. The definition of parameters is as follows.

```
.subckt <subckt_name> <node_list> PARAMS: param_name = value, ...
```

You may then use the parameter name in place of a value in the macro-model. The value of the parameter is editable in the component dialog on the schematic.

Improved Support of Behavioral Sources

Behavioral sources now support nested instances of IF statements.

Support for BSIM 4 Parameters

NI Multisim 10.0 supports the standard BSIM 4 parameters for MOSFET models. BSIM 4 supports up to 400 parameters. More information about BSIM 4 is available at <http://www-device.eecs.berkeley.edu/~bsim3/bsim4.html>.

Enhanced Data Visualization

NI Multisim 10.0 includes a number of improvements to the way you configure and view results. These include: advanced functionality of the static probes, the ability to add traces to the Grapher after running a simulation, the ability to display the initial conditions of components on the schematic, a current probe instrument, and improvements to the memory and register displays of MCUs.

Advanced Functionality of Static Probes

Placed (static) probes now include a reference designator, which allows you to select another probe as a reference net. In previous versions of NI Multisim, all probes referenced ground. You can also use probe reference designators to select which traces to view in analyses.

Add Traces to Grapher after Running Analyses

You can add traces to the Grapher view after running an analysis and select what type of data you want NI Multisim to store.

Display Initial Conditions on the Schematic

You can choose to display the initial conditions of capacitors and inductors on the schematic.

Current Probe Instrument

The current probe instrument is a virtual representation of a real current probe that connects to an oscilloscope. You connect one end of the probe to a net on the schematic and the other to the input to an oscilloscope. You can set the ratio of amps to volts displayed on the instrument. Note that the units remain in volts on the oscilloscope.

Enhanced Analysis Capabilities

NI Multisim 10.0 now allows you to evaluate more expressions before and after running analyses. The definitions of the expressions are:

1. $\text{avg}(\mathbf{X})$ — Running average of the vector \mathbf{X}
2. $\text{avg}(\mathbf{X}, d)$ — Running average of the vector \mathbf{X} over d
3. $\text{envmax}(\mathbf{X}, n)$ — Upper envelope of the vector \mathbf{X} where n is the number of points on either side of a peak that must be less than the value for a peak to be identified

4. `envmin(X, n)` — Lower envelope of the vector **X** where *n* is the number of points on either side of a peak that must be less than the value for a peak to be identified
5. `grpdelay(X)` — Group delay of **X** with results in seconds
6. `rms(X)` — Running RMS average of vector **X**
7. `integral(X)` — Running integral of vector **X**
8. `sgn(X)` — The sign or signum of a real number. It is -1 for a negative number, 0 for the number zero, and 1 for a positive number.

Extended Language Support and File Management in the MCU Module

The MCU Module, formerly MultiMCU, is now available to the professional market. This module supports C-code in addition to Assembly language. It has a code manager that lets you use multiple files to define the operation of the microcontrollers in the design. You can have header files and use libraries. You can also load in externally assembled binary files and view them in disassembled format.

Improved Export to Mentor Graphics PADS®

You can export NI Multisim schematics to Mentor Graphics PADS®. The exported netlist file follows the Mentor Graphics specification. Every NI Multisim component includes generic landpatterns, which means you can transfer without dropping any nets and then select the appropriate landpattern once in PADS. You can also map your existing PADS landpatterns to the components in NI Multisim.

Improvements to Speed and Quality of NI Ultiboard

NI Ultiboard 10.0 contains enhancements to the quality of the product that include improvements to the speed of trace-placement and the ability to select whether or not to plate through-holes. Exported Gerber files do not contain mosaics in the polygons. Quality improvements in the landpatterns include: pin mappings from symbols to IC pin-outs and landpattern shapes and sizes in the database. All new landpatterns now follow IPC standards.

Advanced Options for Exported Data Interpolation

When exporting simulation data from NI Multisim to other NI data formats such as LVM or TDM files, you can choose the interpolation technique that best suits the signal. You can also control the interpolation method used when sending simulation data to NI LabVIEW based instruments running inside of NI Multisim. The interpolation methods include:

- Coerce

- Linear Interpolation
- Spline Interpolation

Miscellaneous Features

Some of the other features added to the new suite include Unicode character support and NI installation and license management.

Unicode Characters

All products in NI Circuit Design Suite 10.0 now support Unicode characters. This feature allows you to use Cyrillic and Asian fonts inside the products.

NI Installation and License Management

All products in NI Circuit Design Suite adhere to the standard method used to install and activate National Instruments software. You can activate the software automatically via the internet, or manually via a web browser, phone call, or email.

Product Tier Details

The following lists the schematic capture functionality available in Multisim Base, Full, and Power Pro editions:

Functionality	Base	Full	Power Pro
Customizable GUI	X	X	X
Modeless part placement and wiring	X	X	X
Fast retrieval parts bins	X	X	X
User defined fields	X	X	X
Advanced symbol editor	X	X	X
Auto and manual wiring	X	X	X
Virtual wiring by node name	X	X	X
Fast auto-connect passives	X	X	X
Rubber banding on part move	X	X	X
Replace multiple components at once	X	X	X
Bus-Vector connect	X	X	X

Functionality	Base	Full	Power Pro
Project manager	X	X	X
Hierarchical design	X	X	X
Multisheet design	X	X	X
Circuit annotations	X	X	X
Comments on schematic	X	X	X
Electrical rules check	X	X	X
Title block editor	X	X	X
Forward/Back annotation	X	X	X
Export to Mentor PADS layout	X	X	X
Advanced search	X	X	X
Variant support			X
Spreadsheet view			X
Design constraints			X
Zoom to selected part			X
Pin and gate swap			X
Customizable BOM			X
Advanced reports			X
Cross-probing with Ultiboard			X
ERC scope setting			X
Mark no-connect pins			X
Database import/export			X
Component database	Partial	Partial	Complete

The following lists the simulation functionality available in Multisim Base, Full, and Power Pro editions:

Functionality	Base	Full	Power Pro
Interactive simulator	X	X	X
Fully mixed-mode A/D simulation	X	X	X
Standard SPICE 3X5/XSPICE	X	X	X

Functionality	Base	Full	Power Pro
Enhanced model support	X	X	X
PSPICE model simulation*	X	X	X
Speed/Accuracy tradeoffs	X	X	X
Simulation Advisor	X	X	X
Convergence Assistant	X	X	X
Virtual, interactive, animated parts	X	X	X
Mouse click support for interactive parts	X	X	X
Measurement Probes	X	X	X
Component Wizard	X	X	X
NI measurement data file sources	X	X	X
NI measurement data file export	X	X	X
NI LabVIEW VIs as instruments and sources		X	X
Simulation Profiles		X	X
Postprocessor		X	X
Expressions in analyses		X	X
Add traces to Grapher post analyses		X	X
Rated components		X	X
Insert faults into components		X	X
Op-Amp Wizard			X
555 Timer Wizard			X
Filter Wizard			X
CE Amplifier Wizard			X
Model makers			X
Switch mode power supply generics			X
RF Design Module			X
Nested sweeps			X
C-Code modeling			X
Virtual Instruments	4	15	22

Functionality	Base	Full	Power Pro
Analyses	0	15	19
Simulated Agilent instruments	0	1	3
Simulated Tektronix instrument	0	0	1
Multisim MCU module	Add On	Add On	Add On

* Does not support all PSpice syntax

The following lists the layout functionality available in Ultiboard Full and Power Pro editions:

Functionality	Full	Power Pro
Gridless Follow-me placement	X	X
Push and Shove part placement	X	X
Push and Shove trace placement	X	X
Real-time & from copper ratsnest	X	X
Auto-alignment	X	X
Real-time polygon update with voiding	X	X
Keep-in/Keep-out areas	X	X
Forward/Backward annotation	X	X
Real-time DRC	X	X
Jump to Error	X	X
64 layers and 1 nanometer resolution	X	X
Polar Grids	X	X
Customizable layer viewing	X	X
Split power-planes	X	X
Comprehensive Footprint Wizard	X	X
Enhanced 3D visualization with print	X	X
Full screen mode	X	X
Gerber, DXF, IPC-D-356A, SVG output	X	X
Dimensions on PCB and Landpatterns	X	X
Dimensions in Database Manager	X	X

Functionality	Full	Power Pro
User annotations	X	X
Net bridges	X	X
3D visualization inside circuit board		X
Turn off ratsnest for selected nets		X
Load and save technology files		X
Cross-probing with Multisim		X
Variant Support		X
Component Placement Sequencer		X
Place components in array		X
Unplace all components		X
Ruler bar alignments and measurements		X
Save PCB Design as a component		X
Permanent grouping		X
Pin & gate swapping		X
Gridless Connection Machine		X
High-speed constraint driven layout		X
Multiple clearances		X
Net topology choices		X
Equispace trace support		X
Differential Impedance Calculator		X
Transmission Line Calculator		X
Microvias		X
Test point insertion		X
Automatic tear-dropping		X
Pin necked trace support		X
Automatic jumper insertion		X
Copy Route & Replica Place functions		X
In-place footprint editor		X

Functionality	Full	Power Pro
Mechanical CAD		X
Export 3D info in 3D IGES, DXF formats		X
Copper amount report		X
Test point report		X
Number of pins supported	1,400	Unlimited
Spreadsheet view	Limited	Complete

The following lists the autorouting functionality available in Ultiboard Full and Power Pro editions:

Functionality	Full	Power Pro
Autoplacement	X	X
Pin & gate swapping	X	X
Fully customizable cost factors	X	X
Progressive Routing	X	X
Interactive autorouting	X	X
Constraint driven routing	X	X
Follows keep-in/keep-out criteria	X	X
Manual pre-placement: components, vias, traces	X	X
Auto Block Capacitor recognition	X	X
SMD mirroring	X	X
Net shielding	X	X
Automatic testpoint insertion	X	X
Trace rubberbanding	X	X
Advanced BGA fan-out		X
Topology: shortest, daisy chain, star		X
Prioritize routing order		X
Route an individual net		X
Automatic bus routing		X

Functionality	Full	Power Pro
Differential Pair routing		X
Group autoplacement		X
Group autorouting		X
Optimization		X
Pin number limit	1,400	Unlimited
Maximum number of layers	4	64

Documentation

NI Circuit Design Suite 10.0 includes a complete documentation set featuring printed and electronic resources for your reference.

The following printed and electronic resource is available:

- Getting Started with NI Circuit Design Suite Guide

The following electronic resources are available in PDF files:

- Multisim User Guide
- Multisim Component Reference Guide
- Multisim MCU Module User Guide
- Ultiboard User Guide

To access the User Guides, select Start > All Programs > National Instruments > Circuit Design Suite 10.0 > Documentation and then select the file of interest.

The following online help files are available from the installed software Help menu and from the Start Menu:

- Multisim Professional Edition Help File
- Ultiboard Help File

To access the Help Files, select Start > All Programs > National Instruments > Circuit Design Suite 10.0 > Documentation and then select the file of interest.

The following online help files are available from the installed software Help menu:

- Component Reference Professional Edition Help File
- Multisim Symbol Editor Help File
- Multisim Title Block Editor Help File

National Instruments, NI, ni.com, and LabVIEW are trademarks of National Instruments Corporation. Refer to the Terms of Use section on ni.com/legal for more information about National Instruments trademarks. Ultiboard is a registered trademark and Multisim and Electronics Workbench are trademarks of Electronics Workbench. Other product and company names mentioned herein are trademarks or trade names of their respective companies.

© 2007 National Instruments Corp. All rights reserved.

374478A-01

Jan07