

PROFESSIONAL EDITION RELEASE NOTES

NI Circuit Design Suite

Version 12.0

These release notes contain system requirements for NI Circuit Design Suite 12.0, and information about product tiers, new features, documentation resources, and other changes since NI Multisim 11.0.2 and NI Ultiboard 11.0.2.

NI Circuit Design Suite includes the NI Multisim and NI Ultiboard software products from the National Instruments Electronics Workbench Group.

Contents

Installing NI Circuit Design Suite 12.0	2
Minimum System Requirements	2
Installation Instructions	2
Product Activation	3
What's New in NI Circuit Design Suite 12.0	3
LabVIEW-Multisim Co-simulation	4
Multisim Snippets	4
Upgrades to Microchip Software	4
Database Improvements	4
Obsolescence Information	5
Component Filter	5
Improvements to Interface and Property Editing for Generic Diodes, Transistors and FETs	5
Digital Signal Analysis	6
Improved Text Handling in Multisim	6
Ability to Load Last File on Multisim Startup	6
Buses and Connectors Added to Sheet Properties Color Scheme	6
Ability to Configure Pin and Gate Swapping Added to Multisim	6
Display of Net/Component Attributes in Status Bar	6
Thumbnails	6
Improvements to DXF Import	7
Improvements to Parts Position Tab	7
Improved Shortcut for Toggling Layers in Ultiboard	7
Layers Tab is Default on Ultiboard Startup	7
Force Vectors Off on Ultiboard Startup	8
Improvements to Polygon Functionality in Ultiboard	8
Connectivity Check and Warn on Single Pins	8

Amalgamation of Component Help into Multisim Help	9
Bug Fixes	9
Product Tier Details	9
Localization	15
Documentation.....	15

Installing NI Circuit Design Suite 12.0

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

Minimum System Requirements

To run NI Circuit Design Suite 12.0, your OS must be one of:

- Windows XP 32-bit edition.
- Windows Vista 32-bit or 64-bit edition.
- Windows 7 32-bit or 64-bit edition.
- Windows Server 2003 R2 (32-bit) or 2008 R2 (64-bit) edition.



Note Circuit Design Suite 12.0 does not support Windows NT/Me/98/95/2000, Windows XP x64, or the Windows Server non-R2 editions.

Your system must also meet or exceed the following:

- 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 2010 or 2011 is required.

Installation Instructions

The NI Circuit Design Suite 12.0 installer installs both products in the suite: NI Multisim and NI Ultiboard.

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

By default, the NI Circuit Design Suite installation program copies files to <Program Files>\National Instruments\Circuit Design Suite 12.0 after you complete the following steps:

1. Insert the NI Circuit Design Suite CD into the CD-ROM drive. If the CD startup screen does not appear, select **Run** from the Windows **Start** menu and run `setup.exe` from your CD drive.

2. Follow the instructions in the dialog boxes.



Note To install the NI Multisim LabVIEW Co-Simulation Plug-in, you must already have NI LabVIEW 2011 (32-bit) and NI LabVIEW Control Design and Simulation Module 2011 installed on the machine. You can install the Co-Simulation Plug-in at a later time by first installing LabVIEW 2011 and the Control Design and Simulation Module 2011 and then re-running the Circuit Design Suite 12.0 installer.

Product Activation

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

If you do not have a valid license, the product will run in Evaluation Mode and continue to prompt you to activate 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 Software Note to Users* included with your NI Circuit Design Suite package.

What's New in NI Circuit Design Suite 12.0

This document describes the following feature additions or improvements to NI Circuit Design Suite 12.0:

- LabVIEW-Multisim co-simulation.
- Multisim snippets.
- Upgrades to Microchip software.
- Database improvements.
- Improvements to interface and property editing for generic diodes, transistors and FETs.
- Digital signal analysis.
- Improved text handling in Multisim.
- Ability to load last file on Multisim startup.
- Buses and connectors added to sheet properties color scheme.
- Ability to configure pin and gate swapping added to Multisim.
- Display of net/component attributes in status bar.
- Thumbnails.
- Improvements to DXF import.
- Improvements to Parts Position tab.
- Improved shortcut for toggling layers.
- Layers tab is default on Ultiboard startup.
- Force vectors off on Ultiboard startup.
- Improvements to polygon functionality in Ultiboard.
- Connectivity check and warn on single pins.

- Amalgamation of component help into *Multisim Help*.
- Bug Fixes.

LabVIEW-Multisim Co-simulation

Multisim includes detailed simulation models which can now be used to simulate electrical components directly in LabVIEW using the new Multisim Design VI.

In LabVIEW 2011, the Multisim Design VI represents a Multisim design file that you want to simulate. When you place the Multisim Design VI in a Control & Simulation Loop, Multisim and LabVIEW co-simulate in a highly integrated manner.

The NI Multisim LabVIEW Co-Simulation Plug-in includes the Multisim Design VI. It installs automatically when you run the NI Circuit Design Suite installer.



Note For the NI Multisim LabVIEW Co-Simulation Plug-in to successfully install, you must already have NI LabVIEW 2011 (32-bit) and the NI LabVIEW Control Design and Simulation Module 2011 installed on the machine.

Multisim Snippets

You can now save snippets of a Multisim design to reuse later or to share with other Multisim users in a .png image file. The image file shows a picture of your design and contains design information for all components in the snippet, including symbol, model, footprint and connected nets.

Upgrades to Microchip Software

The HI-TECH PICC compiler that is included with Multisim has been upgraded to version 9.82.

The Microchip MPASMWIN assembler is upgraded to version 5.42.

Database Improvements

Refer to <http://zone.ni.com/devzone/cda/tut/p/id/5607> for more detailed information about the components and models included in your edition of Multisim.

Manufacturers' component updates:

- Over 225 new and updated Analog Devices and National Semiconductor components.
- Over 1,800 new and updated NXP and Philips components.
- Over 90 new National Instruments connectors, including NI sBRIO connectors, NI DAQ connectors for E Series, S Series and X Series devices.
- Over 75 new industry standard connectors, including USB connectors, DSUB connectors and banana jacks.

As well as the above manufacturers' components, the database includes the following new devices:

- 25 parameterizable transformer components.
- 13 parameterizable machine components, including DC machines, induction machines, brushless DC machines and stepper motors, as well as a number of supporting components such as stepper motor drives, mechanical loads and speed sensors.
- Several simplified power semiconductor switches including transistors and thyristors for high-level simulation.
- Several new voltage-controlled generic switches including single pole double throw and single pole single throw.
- A single pole single throw switch with a customizable bounce pattern.
- Control signal to PWM generators.
- Phase angle controllers.
- Step and chirp signal voltage and current sources.
- Interactive DC voltage and current sources.

Obsolescence Information

To help guide designs, components in the Multisim database now contain information about component availability from some manufacturers.

The **Select a Component** browser now displays a link when a component has been marked obsolete by a manufacturer. Click **Obsolescence information** to display the **Component Obsolescence** dialog box.

While one manufacturer may have discontinued a component, a replacement may exist, or the component may still be available from another manufacturer. Some obsolete components will have manufacturer suggested replacements listed.



Note Consult the manufacturer's website for the most accurate and up-to-date information about component availability and replacements.

Component Filter

A filter has been added to the **Select a Component** dialog box that allows you to select whether or not to list components without models in the dialog box.

Improvements to Interface and Property Editing for Generic Diodes, Transistors and FETs

You can more easily access and modify the parameters of the discrete components so that the generic device can better match the operating conditions required in the design. All instance and model parameters are now easy to access.

You can also select the device model type for a given component.

Digital Signal Analysis

There is a new digital graph type in Multisim's grapher.

It can be used from various analyses and is able to display both native digital signals and analog signals that have been digitized using configurable digitization thresholds.

Improved Text Handling in Multisim

Now, when you place text in Multisim, a **Text** toolbar appears where you can select the font, text size, stroke weight and style, and adjust the text color and alignment.

You can also rotate and flip any text on the workspace, including a component's RefDes and description.

Alignment of objects, including text is available from the **Edit»Align** menu.

Ability to Load Last File on Multisim Startup

You can set Multisim to load the last file on startup from the **General** tab of the **Global Preferences** dialog box.

Buses and Connectors Added to Sheet Properties Color Scheme

You can adjust the color of buses and connectors from the **Colors** tab of the **Sheet Properties** dialog box.

Note the default color of buses and connectors has been changed from black to red for Multisim 12.

Ability to Configure Pin and Gate Swapping Added to Multisim

Pin and gate swapping configuration has been added to the **PCB** tab of Multisim's **Sheet Properties** dialog box.

Display of Net/Component Attributes in Status Bar

When you select an element in Multisim 12, such as a component, net, bus or placed text, its attributes display in the status bar at the bottom of the screen.

Thumbnails

Multisim and Ultiboard now provide a number of ways to see thumbnail-sized previews of files, designs and windows.

A button in the **Design Toolbox** displays thumbnails of recently opened files that you can click to quickly open these files.

Another button, on the bottom-right of the workspace, displays thumbnails of all currently open files.

Thumbnails also appear when you hover the cursor over open designs in the **Hierarchy** tab of the **Design Toolbox**.

Multisim enhances this feature further to show the previews of the hierarchical blocks, subcircuits and parent designs.

You can also display thumbnails from off-page connectors, bus off-page connectors, HB/SC connectors, and bus HB/SC connectors. Hover the cursor over the desired connector and click on the magnifying glass icon that displays. A thumbnail of the connected design appears. Click the thumbnail to go to the complementary connector on the connected page.

Improvements to DXF Import

Ultiboard's **DXF Import Settings** dialog box has had a number of improvements made to it, and has been renamed to **DXF Import**.

Use this dialog box to import any of the following into Ultiboard:

- Board outlines which originate from a mechanical design.
- Other mechanical information.
- Graphics such as a company logo.
- Trace and polygon information.

The **DXF Import** dialog box contains the following sections:

- **Layer mapping**—Use to assign a layer from the DXF file to a specific layer in Ultiboard.
- **Scaling**—The controls in this section determine how physical data in the DXF file is interpreted on import.
- **Other options**—Contains infrequently used import options.

Improvements to Parts Position Tab

The read-only columns in the **Parts position** tab in the Ultiboard **Spreadsheet View** are now editable.

Improved Shortcut for Toggling Layers in Ultiboard

Press <F2> to toggle between mirror layers. For example, if the active layer is **Copper Top** and you press <F2>, the active layer becomes **Copper Bottom**. Press <F2> again to make **Copper Top** the active layer.

Press <Q> to return to the previous active layer. For example, if the active layer is **Silkscreen Top** and you switch to **Board Outline**, you can press <Q> to return to **Silkscreen Top**.

Layers Tab is Default on Ultiboard Startup

The **Layers** tab of the **Design Toolbox** now displays on Ultiboard startup.

Force Vectors Off on Ultiboard Startup

Force vectors are now switched off by default on Ultiboard startup.

Improvements to Polygon Functionality in Ultiboard

All filled copper shapes in Ultiboard are now copper areas, and include a **Copper areas** tab in their properties dialog box, where you can switch voiding off/on.



Note A void is an empty area around traces, copper areas, pins and other objects that prevents unintentional connection to copper areas.

The following commands have been removed:

- **Place»Copper area**—There is no longer any need to place unique copper areas, as all filled copper shapes are now treated as areas.
- **Design»Shape to area**—There is no longer any need to convert a shape to an area, as all filled copper shapes are now treated as areas.
- **Place»Shape**—All selections from this have been incorporated into **Place»Graphics** to be consistent with Multisim.

Previous versions could only draw voids in 45° increments. Ultiboard 12 uses smooth voiding without any loss of performance.

If you open an Ultiboard file that was created in an earlier version that includes voiding, a message appears in the **Results** tab of the **Spreadsheet View** advising you that the design uses 45° voids. The message includes a link to the new Upgrade tab of the PCB Properties dialog box, where you can convert the design to use smooth voids.



Note If you choose to update the design, the **Upgrade** tab will no longer appear for this design.

Connectivity Check and Warn on Single Pins

The ability to check for single-pin nets, that is, a net with only one pin attached to it, has been added to Ultiboard 12.

A single-pin error can occur:

- **In error**—for example, you create a net and do not name it to match an existing net. In this case, a connection error for the single-pin net is desired.
- **Intentionally**—for example, you want to create a testpoint without a footprint, and instead modify the silkscreen on the top layer to expose the net. Or, if you design an on-the-board antenna, which would also be a single-pin net. In these cases, you would not want connection errors to display, so you would set the nets to be excluded from single-pin checking.

Amalgamation of Component Help into Multisim Help

Component Reference Help has been discontinued and its contents have been incorporated into *Multisim Help*.

Bug Fixes

Refer to the Readme file for a list of issues fixed in version 12.0.

All readme files are located at <Program Files>\National Instruments\Circuit Design Suite 12.0\documentation.

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
Autowire when parts when brought together	X	X	X
Autowire when part pin placed on a wire	X	X	X
Fast retrieval parts bins	X	X	X
User defined fields	X	X	X
Component editing	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
Buses	X	X	X
Bus Vector connect	X	X	X
Circuit restrictions	X	X	X
Project manager	X	X	X
Project packing	X	X	X
Hierarchical design	X	X	X

Functionality	Base	Full	Power Pro
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 component search	X	X	X
Variant support			X
Snippets – creating			X
Snippets – opening	X	X	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 3F5/XSPICE	X	X	X
Enhanced model support	X	X	X

Functionality	Base	Full	Power Pro
Cadence® PSpice® model simulation*	X	X	X
Speed/Accuracy tradeoffs	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
LabVIEW-Multisim Co-simulation		X	X
Simulation Profiles		X	X
Grapher	X	X	X
Grapher – Digital Display		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	16	20
Simulated Agilent instruments	0	1	3
Simulated Tektronix instrument	0	0	1
Multisim MCU		X	X
Multisim Automation API			X
* Does not support all Cadence® 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
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
User annotations	X	X
Net bridges	X	X

Functionality	Full	Power Pro
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
Mechanical CAD		X
Export 3D info in 3D IGES, DXF formats		X
Copper amount report		X

Functionality	Full	Power Pro
Test point report		X
Number of pins supported	2,000	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
Topology: shortest, daisy chain, star		X
Prioritize routing order		X
Route an individual net		X
Automatic bus routing		X
Differential Pair routing		X
Group autoplacement		X
Group autoroute		X
Optimization		X
Pin number limit	2,000	Unlimited
Maximum number of layers	4	64

Localization

NI Circuit Design Suite 12.0 is localized for English, German, and Japanese. The system locale setting determines the default language used by the software.

To change the language the software uses, select **Options»Global Preferences**, click on the **General** tab, select the desired language from the **Language** drop-down list, and restart the application.

The following items are not localized, and remain in English:

- LabVIEW instruments.
- Layer names in both NI Ultiboard and the NI Multisim Spreadsheet View.
- Agilent and Tektronix simulated instruments.
- Sample files.
- MCU functionality: source file names, code/comments within source files, and compiler/linker messages.

The following documentation is available in English, German, and Japanese:

- *NI Circuit Design Suite Release Notes.*
- *Getting Started with NI Circuit Design Suite.*

User manuals and help files are not localized, and remain in English.

Documentation

NI Circuit Design Suite 12.0 includes a complete documentation set featuring printed and electronic resources for your reference.

The following printed and electronic resource is available:

- *NI Circuit Design Suite Release Notes.*

The following are available from the installed software Help menu and from the Start menu:

- *Getting Started with NI Circuit Design Suite.*
- *NI Multisim Fundamentals.*
- *NI Ultiboard Fundamentals.*
- *Multisim Help.*
- *Ultiboard Help.*

To access the above, choose the desired link from the Help menu, or select **Start»All Programs»National Instruments»Circuit Design Suite 12.0»Documentation** and then select the file of interest.

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

- *Multisim Symbol Editor Help* (access from the Symbol Editor).
- *Multisim Title Block Editor Help* (access from the Title Block Editor).

LabVIEW, National Instruments, NI, ni.com, the National Instruments corporate logo, and the Eagle logo are trademarks of National Instruments Corporation. Refer to the *Trademark Information* at ni.com/trademarks for other National Instruments trademarks. Other product and company names mentioned herein are trademarks or trade names of their respective companies. For patents covering National Instruments products/technology, refer to the appropriate location: **Help»Patents** in your software, the `patents.txt` file on your media, or the *National Instruments Patents Notice* at ni.com/patents. Refer to the *Export Compliance Information* at ni.com/legal/export-compliance for the National Instruments global trade compliance policy and how to obtain relevant HTS codes, ECCNs, and other import/export data.

© 2007–2012 National Instruments Corporation. All rights reserved.