

# EDUCATION EDITION RELEASE NOTES

# NI Circuit Design Suite

## Version 11.0.1

These release notes contain system requirements for NI Circuit Design Suite 11.0.1, and information about product tiers, new features, documentation resources, and other changes since NI Multisim 11.0 and NI Ultiboard 11.0.

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

## Contents

---

Installing NI Circuit Design Suite 11.0.1.....	2
Minimum System Requirements .....	2
Installation Instructions.....	3
Product Activation .....	3
What's New in NI Circuit Design Suite 11.0.1.....	4
Programming Xilinx PLD from Multisim .....	4
Connector Improvements.....	4
Continued Improvements to Forward and Backward Annotation ...	5
Simulating Indicator Added to Design Toolbox.....	5
Ability to Place a Component Directly from the Component Wizard .....	5
Improved Multisim File Load/Save Performance .....	5
Updated Component Database.....	5
Model Encryption .....	6
PCB Matrix Nominal Land Patterns Added to Ultiboard.....	6
All Layers Show in General Layers Tab in Ultiboard .....	6
Reorganization of the Ultiboard Help.....	6
Bug Fixes .....	6
What's New in NI Circuit Design Suite 11.0.....	6
Programmable Logic Design from Schematic.....	7
Better Ways of Working with Analog Simulation.....	7
SPICE Modeling Improvements.....	7
Direct Viewing of the SPICE Netlist.....	8
Additions and Improvements to Analysis .....	9

Improvements to Digital Simulation .....	9
Improvements to the Grapher .....	9
Updated Component Database .....	10
New Components from Leading Manufacturers .....	10
New Components to Integrate with NI Hardware and RIO Design .....	10
New Virtual Components.....	10
New Schematic Net System .....	10
Enhanced Projects and New Packing Function .....	11
All-New Forward and Backward Annotation.....	11
Additional Functionality of the LabVIEW-Multisim Instrument Interface.....	11
Integration with NI Example Finder.....	12
General Improvements to NI Multisim Usability.....	12
Improved Multisim Simulation Description in Help File.....	12
General Improvements to NI Ultiboard.....	12
File Compatibility with NI Circuit Design Suite 10.x .....	12
Bug Fixes.....	13
Product Tier Details .....	13
Localization .....	19
Documentation.....	20

## Installing NI Circuit Design Suite 11.0.1

---

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

### Minimum System Requirements

To run NI Circuit Design Suite 11.0.1, National Instruments recommends that your system meet or exceed the following requirements:

- Windows Vista/XP 32-bit editions.
- Windows Vista 64-bit edition.
- Windows 7 32-bit and 64-bit editions.



**Note** Circuit Design Suite 11.0 does not support Windows NT/Me/98/95/2000, or Windows XP x64.

- 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<sup>®</sup> 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 8.6.x or 2009 is required.

## Installation Instructions

The NI Circuit Design Suite 11.0.1 installer installs both products in the suite: NI Multisim and NI Ultiboard.

If you have not installed version 11.0, the installer installs that version first, followed by version 11.0.1. If you have already installed version 11.0, the installer installs version 11.0.1, which automatically updates version 11.0 in the same directory in which it was originally installed.

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 11.0 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** On completion of the NI Circuit Design Suite (NI CDS) 11.0.1 installation, the installer prompts you for the NI ELVISmx 4.x installation CD, which is included in your NI CDS package. The NI ELVISmx 4.x enables the NI ELVIS II functionality in Multisim. If you wish to install the NI ELVISmx 4.x software, follow the instructions in the prompt. Otherwise, you can install the NI ELVISmx 4.x software at a later time.

## 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 11.0.1

---

This document describes the following feature additions or improvements to NI Circuit Design Suite 11.0.1:

- Programming Xilinx PLD from Multisim.
- Connector improvements.
- Continued improvements to Forward and Backward Annotation.
- A **simulating** icon has been added to the Design Toolbox.
- Ability to place a component directly from the Component Wizard.
- Improved Multisim file load/save performance.
- Updated component database.
- Model encryption.
- PCB Matrix nominal land patterns added to Ultiboard.
- All layers show in the General layers tab in Ultiboard.
- Reorganization of the *Ultiboard Help*.
- Bug fixes.

## Programming Xilinx PLD from Multisim

With version 11.0.1 of Multisim, you can program a Xilinx logic device from within the PLD Schematic interface. You must still download and install Xilinx ISE® Webpack™ 10.1 (with service pack 3). Multisim runs the Xilinx tools in the background and downloads the programming file to the device.

## Connector Improvements

The following improvements have been made to connectors in Multisim:

- The properties dialog boxes for all connectors now have a Display tab, where you can set whether or not to display the connector's name, or use the schematic global setting. The global settings for all connector types are found in the Circuit tab of Sheet Properties dialog box, in the Connectors box. The connector viewing options have also been added to the Visibility tab of the Design Toolbox.
- The On-page Connection dialog box has been renamed to On-Page Connector.
- The Global Connection dialog box has been renamed to Global Connector.
- When adding a global or on-page connector to a design, you must now enter a new name, or choose an existing name for a list of available target connectors.

For complete information on properties dialogs for the various connector types, refer to the *Multisim Help*.

## Continued Improvements to Forward and Backward Annotation

This release continues on the improvements made to forward and backward annotation in Circuit Design Suite 11.0.

Improvements include:

- An optional time stamp added to annotation files (@YYYY-mm-dd-time appended to file name before extension, as in Getting Started Final@2010-05-19-0928.ewnet.)
- The Forward Annotation and Backward Annotation dialogs have two new buttons that become active if one or more conflicts are present in the dialog. These are **Go to next conflict** and **Go to previous conflict**.
- An **Additional Information** box has been added to the Forward Annotation and Backward Annotation dialogs. Select a difference to see a more detailed description in this box. This box also provides information on how to resolve a selected conflict.

## Simulating Indicator Added to Design Toolbox

Previously, when multiple designs were open in Multisim, it was sometimes difficult to identify which design was simulating. This was inconvenient, as simulation of one design must be stopped before starting simulation of another.

To overcome this, a **simulating** indicator now appears in the Hierarchy tab of the Design Toolbox, beside the design that is being simulated.

## Ability to Place a Component Directly from the Component Wizard

Previously, when you created a component with Multisim's component wizard, it was necessary to navigate to the new component using the Select a Component dialog box to place the new component on the workspace.

The **Place this component** checkbox has been added to the final window of the component wizard so that a newly created component can be added directly to the workspace.

## Improved Multisim File Load/Save Performance

The speed at which Multisim loads and saves files has been improved for this release.

## Updated Component Database

NI Circuit Design Suite 11.0.1 includes new additions and improvements to the database. These include 33 new models and 38 new components from Analog Devices.

## Model Encryption

This edition of Multisim may include models that have been encrypted at the request of their manufacturers. When seen in the netlist viewer, **Encrypted Model** will show where the model syntax would normally appear.

## PCB Matrix Nominal Land Patterns Added to Ultiboard

Approximately 1,500 land patterns from the PCB Matrix SMN7351B library have been added to the Ultiboard database.

## All Layers Show in General Layers Tab in Ultiboard

The General layers tab in the PCB Properties dialog box has been modified to display all layers available in Ultiboard.

Also, a new Information layer called the Courtyard has been added. The Courtyard is the boundary, or space, around an IPC-compliant part. The Courtyard includes the minimum electrical and mechanical clearance of the component and land pattern boundaries.

## Reorganization of the Ultiboard Help

The *Ultiboard Help* has been reorganized to be more efficient and user-friendly.

The *NI Ultiboard User Manual*, which had essentially the same content as the *Ultiboard Help*, has been replaced by *NI Ultiboard Fundamentals*.

*NI Ultiboard Fundamentals* contains a high-level overview of the Ultiboard functionality.

## Bug Fixes

Refer to the Readme file for a list of issues fixed in version 11.0.1.

The Readme file is located at <Program Files>\National Instruments\Circuit Design Suite 11.0\documentation\Readme\_eng.html.

## What's New in NI Circuit Design Suite 11.0

---

This document describes the following feature additions or improvements to NI Circuit Design Suite 11.0:

- Programmable logic design from schematic.
- Better ways of working with analog simulation.
- Improvements to digital simulation.

- Additions and improvements to analysis.
- Improvements to data visualization and manipulation in the Grapher.
- Updated component database.
- New schematic net system.
- Enhanced Projects and New Packing Function.
- All new Forward and Backward Annotation.
- Additional functionality in the LabVIEW-Multisim Instrument Interface.
- Integration with NI Example Finder.
- General improvements to Multisim usability.
- Improved Multisim simulation description in help file.
- General improvements to Ultiboard.
- File compatibility with Circuit Design Suite 10.x.

## Programmable Logic Design from Schematic

You can create Programmable Logic Designs in NI Multisim 11.0. Over 100 basic components give you the opportunity to generate a schematic design that produces VHDL code. You can then use the code with manufacturer tools to target a programmable logic device.

The components include 44 logic gates and a collection of buffers, adders, comparators, decoders and encoders, multiplexers and demultiplexers, generators, latches, flip flops, counters, and shift registers.

The default setup allows you to target the NI Digital Electronics FPGA Board with an onboard XILINX Spartan 3E.

The designs are scalable and can include hierarchical blocks/subcircuits and user-created custom parts.

## Better Ways of Working with Analog Simulation

Analog Simulation improvements include extended SPICE modeling capabilities, direct viewing of the SPICE Netlist, and more analysis capabilities.

### SPICE Modeling Improvements

NI Multisim 11.0 includes the following improvements to SPICE simulation:

- Improvements to S-Domain modeling with new controlled sources: a Laplace Function Block and a Frequency Table Function Block.

- Updated and new core models including:
  - Updated to version 4.6.3 of BSIM.
  - Support for the VBIC BJT device model.
  - Support for the BSIMSOI v4.0 model.
  - Support for the EKV v2.6 MOSFET model.
- Support for integrals, SDT(), and derivatives, DDT(), inside Arbitrary Sources. These functions evaluate the time integral and time derivative of their arguments, respectively.
- The operators \*\* and ^ are evaluated with the absolute value of the base in real valued floating point expressions.
- An inductor can be coupled with an indefinite number of inductors. There is also a new virtual coupling inductor.
- An improved Current-Controlled Switch.
- Improved support for Diode parameters that model advanced effects including:

Parameter	Description
IKF	High-injection knee current.
ISR	Recombination current parameter.
IBVL	Low-level reverse breakdown knee current.
NBV	Reverse breakdown ideality factor.
NBVL	Low-level reverse breakdown ideality factor.
NR	Emission coefficient for ISR.
TBV1	Linear BV temperature coefficient.
TBV2	Quadratic BV temperature coefficient.
TIKF	Linear IKF temperature coefficient.
TRS1	Linear RS temperature coefficient.
TRS2	Quadratic RS temperature coefficient.

## Direct Viewing of the SPICE Netlist

NI Multisim is a graphical SPICE simulation environment, that allows you to simulate circuits without the need to look at the SPICE netlist. If you are an advanced user, you may find it is useful to view the SPICE netlist. Version 11.0 introduces a SPICE Netlist Viewer.



## Additions and Improvements to Analysis

NI Multisim 11.0 introduces a new analysis, easier access to device current while setting up analyses, and support for interactive components and the time delay switch in analyses.

### AC Single Frequency Analysis

AC Single Frequency Analysis creates a text output of the voltage, current, or power phasor at a given frequency. The phasor is displayed as a magnitude and phase or a real and imaginary pair.

### Device Current and Power in Analyses

In NI Multisim 11.0, you can choose the current or power of devices as analysis outputs from the main analysis output dialog. Previous versions of the software required advanced setup for these values.

### Configurable Devices in Analyses

Devices such as switches, potentiometers, variable capacitors, and inductors now send their set values to the analysis engine. Further, the configurations of the time delay switch now affect the analyses.

## Improvements to Digital Simulation

Digital simulations will run with increased speed, accuracy, and convergence due to improvements to the pin models. In real mode the power supplies will control the output voltage levels. There is also increased accuracy in mixed-mode simulations through increased time-step accuracy of the simulation engine and inertial delays.

## Improvements to the Grapher

A number of improvements were made to the Multisim Grapher including additional capabilities, a new option for displaying parameter sweep data, and new export file types.

New Grapher capabilities introduced in NI Multisim 11.0 include:

- **Annotations**—You can add annotations to the graph. Both data labels which display selected coordinates of a graph and general text labels are supported forms of annotation.
- **Simulation points**—You can optionally display the actual simulation points.
- **A Smart Legend**—This legend is dockable and allows you to toggle the display of traces.

NI Multisim 11.0 allows you to display parameter sweep results in a graph or table format.

You can now save Grapher data in two additional file formats: Comma Separated Variables, CSV, files and ASCII Delimited File, DLM files. CSV files are easy to open in spreadsheet applications and DLM files are easy to open with math scripting applications such as NI LabVIEW MathScript.

## **Updated Component Database**

NI Circuit Design Suite 11.0 includes new additions and improvements to the database. These include approximately 550 new components from leading manufacturers, new NI Hardware connectors, new virtual components, and more than 650 updated components.

### **New Components from Leading Manufacturers**

There are approximately 550 new components with models from Texas Instruments, Linear Technologies, and Microchip. The additions include symbols, models, and IPC standard land patterns.

### **New Components to Integrate with NI Hardware and RIO Design**

There are new connectors added to enable faster board design that will integrate with Single Board RIO as well as M-Series Devices. There are a total of 26 new NI connectors with both symbols and IPC standard land patterns.

### **New Virtual Components**

The addition of new virtual components makes simulations easier for both digital and analog applications. They include three digital sources, a transport delay component, a virtual coupling inductor, and two control function blocks for s-domain modeling. The additions include symbols and models.

## **New Schematic Net System**

NI Circuit Design Suite 11.0 introduces a new What You See Is What You Get (WYSIWYG) net system. The net names are consistent in capture, simulation, and layout, making schematics and artwork easier to read and identify. Net names pass through all levels of hierarchy and through multipage designs. You can define global nets by placing global connectors; this will allow the creation of arbitrary and unlimited power net names. On-page connectors replace the hidden virtual connections.

## Enhanced Projects and New Packing Function

NI Multisim 11.0 features the implementation of Project Packing. Project Packing allows users to easily share projects between computers and networks. The files linked to the project are packaged into a single file for distribution.

## All-New Forward and Backward Annotation

NI Multisim 11.0 introduces a fully interactive forward and backward annotation system. You now have visibility into and control over what changes take place between the schematics and the layout of a design. Both forward and backward annotation are invoked from within Multisim or Ultiboard and provide the same user experience. An annotation file is produced during annotation; you can save and share these files.

The following data are annotated from NI Multisim to NI Ultiboard:

- Multisim Components (ID, RefDes, connected net names, land patterns, IDs of multi-section peers, variant memberships, subcircuit/hierarchical block memberships, and part group memberships).
- Nets (ID, net name, components-pin pairs on the net, all net properties, net groups memberships).
- Variants (Name, enabled/disabled for export).
- Layers.

The following data are annotated from NI Ultiboard to NI Multisim:

- Ultiboard Parts (ID, RefDes, land-pattern, connected net names).
- Nets (ID, net name, land-pattern-pin pairs on the net, all net properties).
- Layers.

## Additional Functionality of the LabVIEW-Multisim Instrument Interface

LabVIEW instruments in Multisim allow you to create custom input, output, and input/output instruments for use in Multisim. Improvements in Multisim 11.0 include support for:

- AC Analysis.
- DC Sweep Analysis.
- Passing a number of new parameters:
  - Current.
  - Differential Voltage.
  - Inductance.

- Resistance.
- Capacitance.
- Diode voltage and voltage and current pairs.
- BJT voltage and current pairs.

## Integration with NI Example Finder

The NI Example Finder allows users to find examples by either browsing folders or searching with keywords. Descriptions of the files are presented prior to opening the design.

## General Improvements to NI Multisim Usability

Throughout NI Multisim a number of improvements were done to enhance the usability of the product. Some examples include:

- Ability to reorder Multipage Designs.
- Enabling the Title Block to return to the grid.

## Improved Multisim Simulation Description in Help File

The simulation section of the *Multisim Help* has been updated to include a detailed discussion of Multisim's simulation processes and methods.

## General Improvements to NI Ultiboard

Throughout NI Ultiboard a number of improvements were done to enhance the usability of the product. Examples include:

- Increasing the pin count limit of the versions as follows:
  - 1500 pin limit in NI Ultiboard Education.
  - 550 pin limit in NI Ultiboard Student.
- Streamlining the process of exporting Gerber files. There is no more need to define apertures for each exported layer.

## File Compatibility with NI Circuit Design Suite 10.x

Files from NI Circuit Design Suite 10.x, and earlier, are forward compatible with 11.0. New features of 11.0 prevent version 11.0 files from opening in previous releases.

## Bug Fixes

Refer to the Readme file at <Program Files>\National Instruments\Circuit Design Suite 11.0\documentation for a list of issues fixed in version 11.0:

- Readme\_eng.html—English Readme file.
- Readme\_deu.html—German Readme file.
- Readme\_jpn.html—Japanese Readme file.

## Product Tier Details

---

The following lists the schematic capture functionality available in Multisim Student and Education editions:

Functionality	Student	Education
Customizable GUI	X	X
Screen-capture utility	X	X
Comments on schematic	X	X
Circuit annotations	X	X
Modeless part placement and wiring	X	X
Fast retrieval parts bins	X	X
Auto and manual wiring	X	X
Virtual wiring by node name	X	X
Rubber banding on part move	X	X
Fast auto-connect passives	X	X
Subcircuits	X	X
3-dimensional breadboarding	X	X
Virtual NI ELVIS I and II	X	X
Virtual NI MyDAQ	X	X
Embedded questions—view and respond	X	X
Forward/Back annotation with Ultiboard	X	X
Cross-probing with Ultiboard	X	X

<b>Functionality</b>	<b>Student</b>	<b>Education</b>
Bus-vector connect		X
Spreadsheet view		X
Design constraints		X
Advanced search		X
Zoom to selected part		X
Corporate database		X
User defined fields		X
Save components to database from workspace		X
Multiple circuits open		X
Embedded questions—create and edit		X
Electrical rules check		X
Graphically mark no-connect pins		X
Hierarchical designs		X
Multisheet designs		X
Project manager		X
Project packing		X
Reports—including bill of materials		X
Pin and gate swap		X
PLD/VHDL output	X	X
PLD hierarchical blocks		X
Export to Mentor PADS layout		X
Device library	Partial	Complete
Maximum components in design	50	Unlimited

The following lists the simulation functionality available in Multisim Student and Education editions:

<b>Functionality</b>	<b>Student</b>	<b>Education</b>
Interactive simulation	X	X
Fully mixed-mode A/D simulation	X	X
Standard SPICE 3F5/XSPICE	X	X
Enhanced model support	X	X
Cadence® Pspice® model simulation*	X	X
Speed/Accuracy tradeoffs	X	X
Simulation advisor	X	X
Convergence assistant	X	X
Virtual, interactive, animated parts	X	X
Mouse click support for interactive parts	X	X
Rated components	X	X
Insert faults into components	X	X
Measurement Probes	X	X
Component Wizard	X	X
NI measurement data file sources	X	X
NI measurement data file export	X	X
NI LabVIEW VIs as instruments and sources	X	X
Microphone & speaker	X	X
Circuit restrictions	X	X
Grapher & Postprocessor	X	X
RF design kit	X	X
Circuit wizards		X

<b>Functionality</b>	<b>Student</b>	<b>Education</b>
C-Code modeling		X
Description box synced with simulation		X
Ladder diagrams/components		X
Model makers		X
Load and save simulation profiles		X
Virtual Instruments	22	22
Analyses	12	20
Multisim MCU	X	X
Multisim Automation API		X
* Does not support all Cadence® PSpice® syntax		

The following lists the layout functionality available in Ultiboard Student and Education editions:

<b>Functionality</b>	<b>Student</b>	<b>Education</b>
Push and Shove trace placement	X	X
Push and Shove part placement	X	X
Real-time & from copper ratsnest	X	X
Real-time polygon update with voiding	X	X
Forward/Backward annotation	X	X
Cross-probing with Multisim	X	X
Real-time DRC	X	X
64 layers and 1 nanometer resolution	X	X
Comprehensive Footprint Wizard	X	X



<b>Functionality</b>	<b>Student</b>	<b>Education</b>
Enhanced 3D visualization with print	X	X
User annotations	X	X
Full screen mode		X
Gerber, DXF, IPC-D-356A, SVG output		X
Dimensions on PCB and Landpatterns		X
Dimensions in Database Manager		X
Net bridges		X
3D visualization inside circuit board		X
Turn off ratsnest for selected nets		X
Gridless follow-me placement		X
Load and save technology files		X
Polar Grids		X
Customizable layer viewing		X
Split power-planes		X
Keep-in/Keep-out areas		X
Place components in array		X
Unplace all components		X
Ruler bar alignments and measurements		X
Auto-alignment		X
Save PCB Design as a component		X
Permanent grouping	X	X
Pin & gate swapping		X
Multiple clearances		X

<b>Functionality</b>	<b>Student</b>	<b>Education</b>
Jump to Error		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 DXF format		X
Copper amount report		X
Test point report		X
Customization of report generation		X
Multiple open documents		X
Number of pins supported	550	1,500
Spreadsheet view	Limited	Complete

The following lists the autorouting functionality available in Ultiboard Student and Education editions:

<b>Functionality</b>	<b>Student</b>	<b>Education</b>
Fully customizable cost factors	X	X
Progressive Routing	X	X
Interactive autorouting	X	X

Functionality	Student	Education
Constraint driven routing	X	X
Manual pre-placement: components, vias, traces	X	X
Auto Block Capacitor recognition	X	X
SMD mirroring	X	X
Trace rubberbanding	X	X
Follows keep-in/keep-out criteria	X	X
Pin number limit	550	1,500
Maximum number of layers	2	4

## Localization

---

NI Circuit Design Suite 11.0.1 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 locale from the **Language** drop-down list, and restart the application.

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

- LabVIEW instruments.
- NI ELVIS 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 11.0.1 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 electronic resources are available in PDF files:

- *Getting Started with NI Circuit Design Suite*
- *NI Multisim Fundamentals*
- *NI Multisim for Education*
- *NI Ultiboard Fundamentals*

To access the User Manuals, select **Start»All Programs»National Instruments»Circuit Design Suite 11.0»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 Help*
- *Ultiboard Help*

To access the help files from the **Start** menu, select **Start»All Programs»National Instruments»Circuit Design Suite 11.0»Documentation** and then select the file of interest.

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

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