PROFESSIONAL EDITION RELEASE NOTES

NI Circuit Design Suite

Version 10.1.1

These release notes contain system requirements for NI Circuit Design Suite 10.1.1, and information about product tiers, new features, documentation resources, and other changes since NI Multisim 10.1 and NI Ultiboard 10.1.

NI Circuit Design Suite includes the familiar NI Multisim and NI Ultiboard software products from the National Instruments Electronics Workbench Group. The NI Multisim MCU Module functionality is now included with NI Multisim.

Contents

Installing NI Circuit Design Suite 10.1.1

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

Minimum System Requirements

To run NI Circuit Design Suite 10.1.1, National Instruments recommends that your system meet or exceed the following requirements:

- Windows 2000 Service Pack 3 or later, Windows XP, Vista, or 64-bit Vista.
- 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 8.2.*x* or 8.5.*x* is required.

Installation Instructions

The NI Circuit Design Suite 10.1.1 installer installs both products in the suite: NI Multisim and NI Ultiboard.

If you have not installed version 10.1.0, the installer installs that version first, followed by version 10.1.1. If you have already installed version 10.1.0, the installer installs version 10.1.1, which automatically updates version 10.1.0 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.

Unless you specify another location during installation, the NI Circuit Design Suite installation program copies files to `<Program Files>\National Instruments\Circuit Design Suite 10.1` 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 drive.
2. Follow the instructions in the dialog boxes.

Product Activation

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

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 Software Note to Users* included with your NI Circuit Design Suite package.

What's New in NI Circuit Design Suite 10.1.1

This document describes the following feature additions or improvements to NI Circuit Design Suite 10.1.1:

- Toolbars in Multisim and Ultiboard can be locked in place.
- Multisim's component search is always advanced.
- Ability to choose units for RLC components.
- Multisim instruments and grapher allow default black or white background.

- Multisim does not automatically rewire large pin-count components.
- Improved parameter suppport for semiconductor devices.
- Added support for Cadence® PSpice® temperature parameters.
- Improvements to SPICE DC convergence algorithms.
- New components from leading manufacturers.
- Add All and Remove All functions added to Ultiboard's Group Editor.
- Custom land patterns in Ultiboard automatically synchronize to Multisim.
- Renumbering footprints in Ultiboard includes top-to-bottom setting.
- Snap-to-pin in Ultiboard is only active during trace placement.
- Improvements to Ultiboard's Gerber viewer.
- Bug fixes.

Locking Toolbars

In previous versions of Multisim and Ultiboard, the location of docked toolbars would sometimes change if the application was resized or closed while minimized.

This has been fixed for this distribution. By default, docked toolbars are now locked in place. You can unlock the toolbars from the **Options** menu.

Advanced Multisim Component Search

The functionality of the **Search Component** dialog box, which is accessed by clicking **Search** in the **Select a Component** dialog box, has changed.

The **Advanced** button has been removed from the dialog box—the **Function**, **Model ID**, **Model Manufacturer**, and **Footprint Type** fields are now always visible.

Units for RLC Components

When placing resistors, inductors or capacitors (R, L, or C components) in Multisim with the **Select a Component** dialog box, you can set the value of these components to a numeric string with an optional metric suffix, for example, "k", "n".

In versions 10.1.0 and prior, values entered in this way were reformatted without warning—for example, if you entered "2000M", the value would be reformatted to "2G". This behavior was not desirable, as manufacturers do not always use the most obvious metric unit to describe their components in price lists and specifications. For example, components with values of "1k" and "1000" are often from different manufacturers.

Units are no longer converted—the value and unit displayed on the schematic and in reports now display as entered during component placement.

Note You can also edit the value of an R, L, or C component from the **Value** tab of the component's properties dialog box. The value remains as entered.

Refer to the *Multisim Help File* for a list of the supported suffixes.

Default Background Color for Instruments and Grapher

You can now select either black or white default background color for the Multisim Grapher and instruments.

No Automatic Rewire of Large Pin-Count Components

As with previous versions of Multisim, components with a large pin count can be set to autowire on move from the **General** tab of the **Preferences** dialog box. This is done using the **Autowire on move, if number of connections is fewer than** checkbox, and its associated field.

In past versions of Multisim, if this checkbox was disabled, or if a component with more than the set number of connections was moved, the moved component would keep the connections, but the wires would be drawn in straight, usually diagonal lines that were very confusing to view.

Now, when the checkbox is disabled, or a component with more than the set number of connections is moved, the wires are disconnected from the component and remain in their original location.

Improved Parameter Support for Semiconductor Devices

The JFET, MESFET, MOSFET (level 1, 2, and 3) and BJT semiconductor models have been updated to support additional model parameters.

The new parameters descibe temperature, DC, small signal, and noise characteristics of the semiconductor devices. They are:

- JFET:
  - ALPHA, BETATCE, ISR, M, N, NR, PB, VK, VTOTC, XTI.
- BJT:
  - CN, D, GAMMA, ISS, NS, QCO, QUASIMOD, RCO, TRB1, TRB2, TRC1, TRC2, TRE1, TRE2, TRM1, TRM2, VG, VO, XCJC2, XCJS.
- MOSFET:
  - GDSNOI, JSSW, N, NLEV, PBSW, TT, WB, XQC.

- MESFET:
  - BETATCE, CDS, EG, N, RG, TRD1, TRG1, TRS1, VTOTC, XTI, M, TAU, VDELTA, VMAX.

The BJT devices now also support lateral PNP (LPNP) models.

Refer to the *Multisim SPICE Reference* section of the *Multisim Help File* for complete information.

Added Support for Cadence® PSpice® Temperature Parameters

Support for T_MEASURED, T_ABS, T_REL_GLOBAL and T_REL_LOCAL Cadence® PSpice® temperature parameters has been added to: Resistor, Capacitor, Inductor, Diode, JFET, MESFET, MOSFET (level 1, 2, and 3) and BJT devices.

Refer to the *Multisim SPICE Reference* section of the *Multisim Help File* for complete information.

Improvements to SPICE DC Convergence Algorithms

Dynamic source-stepping and GMIN-stepping algorithms have been added to DC Operating Point calculations. These are used when ordinary algorithms fail to converge.

Refer to the *DC Convergence Assistance Algorithms* section of the *Multisim Help File* for more information.

New Components from Leading Manufacturers

There are 316 new components from Analog Devices and National Semiconductor in Multisim. The additions include symbols, models, and land patterns. The components are comprised of operational amplifiers, comparators, and RMS-to-DC Converters.

There are also 232 updated components from Analog Devices and National Semiconductor. These components have been updated to contain the latest models and package information.

In Ultiboard, there are 35 new land patterns for National Semiconductor packages.

Add All and Remove All Added to Ultiboard's Group Editor

The ability to add all and remove all parts or nets from a group has been added to Ultiboard's **Group Editor**.

Note Whether parts or nets are being added to, or removed from, a group depends on the tab selected in the **Edit Groups** dialog box. Refer to the *Ultiboard Help File* for complete information.

Custom Land Patterns in Ultiboard Automatically Synchronize to Multisim

When you create or edit a part with a custom land pattern (footprint) in Ultiboard, the change is now automatically synchronized in Multisim. This appears in the preview field of the **Select a Footprint** dialog box in Multisim.

Note If the database containing the part is already open in the **Select a Footprint** dialog box in Multisim when you create or edit the part in Ultiboard, you must select a different database and then reselect the original database to view the change.

Renumbering Parts in Ultiboard

The renumbering parts functionality has changed to allow footprints on PCBs to be renumbered by:

- Board side order.
- Renumbering direction and renumbering start location by board side.

Snap-to-Pin is Only Active During Trace Placement

Ultiboard has been modified so that "snap-to-pin" behavior is now only active during trace placement. This allows easier selection of non-pin elements during any other operation.

Improvements to Ultiboard's Gerber Viewer

Gerber files generated by other applications can now be viewed in Ultiboard. Refer to the *Ultiboard Help File* for more information.

Note Mathematical expressions are not supported.

Bug Fixes

Refer to the Readme file at `<Program Files>\National Instruments\Circuit Design Suite 10.1\documentation` for a list of bugs in version 10.1.1:

- `Readme_eng.html`—English Readme file
- `Readme_deu.html`—German Readme file
- `Readme_jpn.html`—Japanese Readme file

What's New in NI Circuit Design Suite 10.1

This document describes the following feature additions or improvements to NI Circuit Design Suite 10.1:

- The Multisim Automation API
- Enhancements to NI LabVIEW instruments in Multisim
- Additions to the component database
- Extended SPICE modeling capabilities
- NI Update Service
- Vista compliance
- Support for TDMS data files
- File compatibility with NI Circuit Design Suite 10.0.*x*
- New font rendering in NI Ultiboard

The NI Multisim Automation API

The NI Multisim Automation API allows the automation of simulation and analyses through a COM interface. This is useful when a number of simulation runs are required with different circuit settings. Examples include such tasks as component analysis, stress analysis, and fault analysis.

The Multisim Automation API lets you programmatically control a Multisim simulation without needing to view Multisim. Clients written in any COM-aware programming language, such as NI LabVIEW, Visual Basic, or C++ can access Multisim through this interface.

Through the API, you can:

- Open and close an existing circuit.
- Optionally inject a signal in place of an existing voltage or current source.
- Start, stop, and pause simulations.

- Read out simulation results from an existing static probe.
- Enumerate the components in the schematic.
- Replace components with components from the database.
- Get and set the values of resistor, capacitor, and inductor components.
- Enumerate variants.
- Get and set the active variant for simulation.
- Generate reports on the schematic, including a bill of materials and a netlist report.
- Produce an image of the circuit.

Notes Simulation settings used during API control simulations are taken from the Transient Analysis settings of the chosen circuit. Any changes to the Interactive Simulation settings are ignored.

The API can only change and replace components on the top level of a design; components in sub-circuits and hierarchical blocks cannot be altered.

Multisim component replacements may not always have the expected result—they should always be tested in an interactive session of Multisim to make sure that the behavior is what is required.

Multisim Fault settings cannot be changed through the API. Fault analysis requires the building of circuits with additional resistive elements and setting the elements to zero or large values.

Enhancements to NI LabVIEW Instruments in NI Multisim

LabVIEW instruments in Multisim let you create custom input, output, and input/output instruments for use in Multisim. Improvements in Multisim 10.1 include support for both input and output pins on the same instrument, continuous data out to Multisim from output instruments, and the ability for input instruments to affect the time step and interpolation method used in simulation.

Input and Output Instruments

Previous releases allowed input-only and output-only instruments. Input/output instruments have all of the capabilities of input-only and output-only instruments. Note, however, that LabVIEW instrument input data is lagging (meaning that it arrives at the instrument in discrete chunks after the simulator has produced the data), and streaming-out data is leading (meaning that the simulator requests the data in discrete chunks before it needs it). This means that tight feedback loops where a LabVIEW instrument's output depends instantaneously on its intput are not possible.

In general, it will take at least one extra set of continuous outputs through the "Update Data Get Output Values" event of the instrument before you can be sure that the input received by that instrument reflects an adjusted output signal.

Continuous Data from Output Instruments

Previously, data output instruments could generate a finite set of possibly repeating data. When creating an output instrument that generates data continously, remember that, in general, Multisim simulates circuits at a rate slower than the real-time behavior of an equivalent physical circuit. Therefore, if you create an instrument that continuously acquires real-world data using a data acquisition device, modular instrument, or other similar hardware, and use that data as a simulation source in Multisim, simulation will not be able to keep up with the acquired data because it will be running at a rate slower than the data acquisition.

It is the responsibility of the output instrument creator to generate sufficiently small chunks of data at a rate that allows Multisim simulation to keep up with the data acquisition.

Affecting the Time Step and Interpolation Method

LabVIEW samples and processes data at even time steps. SPICE generally operates at uneven time steps best suited to convergance and speed. When creating an input instrument, it is neccesary to resample the data to evenly spaced data for use in LabVIEW. You can now create an instrument that uses Force Step interpolation. (Force Step is not strictly an interpolation method. It forces the Multisim SPICE simulator to take additional timesteps that line up with the requested sampling rate.)

If a particular LabVIEW instrument allows you to set the sampling rate and/or interpolation method, Multisim 10.1 recognizes changes to these values during simulation.

Increased Quality and Breadth of the Component Database

Multisim 10.1 includes new additions and improvements to the database. These include around 300 new components from leading manufacturers, more than 500 updated components, and the latest generic power simulation parts.

New Components from Leading Manufacturers

There are around 300 new components with models from Analog Devices and Texas Instruments. The additions include symbols, models, and IPC compliant land patterns. These components include operational-amplifier, comparator, analog switch, and voltage reference models.

Generic Power Simulation Parts

Multisim 10.1 includes models for all freely available power simulation parts found in the latest release of "Switch-Mode Power Supplies Spice Simulations and Practical Designs" by Christophe Basso. These components include Buck, Boost, Buck-Boost, and PWM controllers.

Extended SPICE Modeling Capabilities

Multisim 10.1 introduces enhancements to its SPICE modeling capabilities. The SPICE parser has been redesigned and improved to better support multiple SPICE languages and advanced modeling. Multisim also supports more Cadence® PSpice® parameters. There is also improved XSPICE model error reporting.

Redesign and Improvement of the NI Multisim SPICE Parser

The SPICE parser in Multisim interprets the native SPICE net lists, reports syntax errors, and evaluates mathematical expressions. Enhancements to the parser include:

- Improved syntax and simulation compatibility for controlled sources.
- Table sources.
- Numerous compatibility improvements that enhance the ability to support Cadence® PSpice® and Intusoft IsSpice4™ syntax.
- Flexible component, net, terminal, and model names.
  - Names may contain any characters with the exception of white space and the following characters: `" ' ( ) { } [ ] , # $: ;`
- The ability to use expressions in place of any value in the netlist.
  - The two special exceptions to this are the BSIM `level` param and the `POLY` number in a controlled source.
- Improved support for expressions, including:
  - Support of inline `if` statements.
  - Improved support for `.param` statements.
  - Full support of single and nested `.func` statements.
  - Current through diodes and relevant controlled sources can now be used in expressions (for arbitrary sources).
  - Improved convergence for complex behavioral-modeling style circuits.

Improved SPICE Model Error Reporting

More intelligent error messages are presented for errors relating to SPICE models.

Component terminals, parameters and expressions are now checked in depth before the simulation begins. Simulation errors display in the Simulation tab of the Spreadsheet View and identify the Multisim components that are causing the issue.

Note With the exception of the Simulation tab and the Results tab, the Spreadsheet View is not available in the Base or Full editions of Multisim. Refer to the *Multisim Help File* for information about these tabs.

Improvements to Cadence® PSpice® Compatibility

Multisim 10.1 now supports the following Cadence® PSpice® parameters for MOSFETS and BJTs:

Parameter	Description	Device
m	Device multiplicity parameter	MOSFET
Rg, Rb	Gate and Bulk Ohmic Resistance	MOSFET
Rds	Drain-Source Shunt Resistance	MOSFET
W, L	Width and Length support on .model line	MOSFET
Nk	Knee current roll-off coefficient	BJT

NI Update Service

The new NI Update Service helps keep your National Instruments software and drivers up-to-date. NI Update Service, which replaces the existing Support and Upgrade Utility (SUU), checks for and electronically delivers software updates for your NI software. You can launch NI Update Service at any time to check for updates or determine the most recent time updates were installed.

Windows Vista Compliant

NI Circuit Design 10.1 is Windows Vista Compliant. Version 10.0.1 was Vista compatible but continued to write to the Program Files directory during run-time. Multisim 10.1 follows Vista Compliance Guidelines.

Better Storage of User Fields from the Master Database

User fields associated with components stored in the Master database are now stored separately. You will see no visible impact when editing and working with user fields in Multisim. This will, however, help to preserve data as you migrate from version to version. This is also required for Vista compliance.

Support of TDMS Data Files

Multisim now supports TDMS files. TDM voltage and current sources can reference TDM streaming data files and the Multisim Grapher can export TDMS data.

Files are Compatible with NI Circuit Design Suite 10.0.*x*

Files from NI Circuit Design Suite 10.0.*x* and 10.1 are backward and forward compatible. File extensions remain the same; files saved with either version will open in both versions.

New Font Rendering in NI Ultiboard

Fonts placed on layers on the board in Ultiboard are now drawn using anti-grain geometry. This provides faster font rendering and better anti-aliasing.

Localization

NI Circuit Design Suite 10.1 is localized for English, German, and Japanese. The system locale setting determines the default language used by the software.

To change the default language the software uses, select **Options»Global Preferences**, click on the **General** tab, select the desired locale from the **Language** combination box, 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:

- Release Notes
- Getting Started with NI Circuit Design Suite

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

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
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

Functionality	Base	Full	Power Pro
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
Enhanced model support	X	X	X
Cadence® PSpice® model simulation*	X	X	X
Speed/Accuracy tradeoffs	X	X	X
Simulation Advisor	X	X	X

Functionality	Base	Full	Power Pro
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		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
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

Functionality	Full	Power Pro
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
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

Functionality	Full	Power Pro
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
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
Topology: shortest, daisy chain, star		X

Functionality	Full	Power Pro
Prioritize routing order		X
Route an individual net		X
Automatic bus routing		X
Differential Pair routing		X
Group autoplace		X
Group autoroute		X
Optimization		X
Pin number limit	1,400	Unlimited
Maximum number of layers	4	64

Documentation

NI Circuit Design Suite 10.1 includes a complete documentation set featuring printed and electronic resources for your reference.

The following printed and electronic resources are available:

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

The following electronic resources are available in PDF files:

- *Multisim User Manual*
- *Ultiboard User Manual*

Note Because the NI Multisim MCU Module functionality is now included with NI Multisim, the *Multisim MCU Module User Guide* has been discontinued. Refer to the *Multisim User Manual* for information about Multisim MCU. The *Multisim Component Reference Guide* has been discontinued. Refer to the *Component Reference Help File* for information about components.

To access the User Manuals, select **Start»All Programs»National Instruments»Circuit Design Suite 10.1»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, from the **Start** menu, select **Start»All Programs»National Instruments»Circuit Design Suite 10.1»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* (access from the Symbol Editor)
- *Multisim Title Block Editor Help File* (access from the Title Block Editor)

 374478D Jan09