The World's Most Widely Used Circuit Design Solution

multiSIM

Schematic Capture, Simulation & Programmable Logic

Electronics
## WORKBENCH

DESIGN SOLUTIONS FOR EVERY DESKTOP

multi**SIM**

The Worldwide Standard For Excellence In Electronics Design

As the world's most popular circuit design tool, Multisim is used by more designers than any other software of its kind. It will reduce development times and help you produce higher quality circuits—we guarantee it! This powerful EDA tool provides all the advanced functionality you need to take designs from specification to production. And because the program is so easy to use, you will be producing designs with Multisim in the time that it takes to install and configure most other programs.

Design Solutions for Every Desktop

Multisim is a complete system design tool which offers schematic entry, a comprehensive component database, SPICE simulation, VHDL/Verilog entry and simulation, RF capabilities, post-processing features and seamless transfer to PCB layout. An easy-to-use design tool that delivers the advanced functionality needed for high-quality designs, Multisim is available in configurations to serve all levels of designers. Its scalable functionality and pricing combined with a variety of team design features make Multisim attractive to both individuals as well as teams of engineers who can now collaborate by designing with a common EDA tool.

```
                          ┌──────────────┐  ┌──────────────┐
                          │  BILL OF     │  │    PCB       │
                          │  MATERIALS   │  │   LAYOUT     │
                          └──────────────┘  └──────────────┘

                            ADVANCED      ┌──────────────┐            ┌──────────────────┐
┌──────────────┐            SEARCH        │  SCHEMATIC   │            │      CO-         │
│  COMPONENT   │ ◄────────────────────►   │   CAPTURE    │ ◄──────►   │   SIMULATION     │
│  DATABASE    │                          │              │            │ SPICE VHDL VERILOG│
└──────────────┘                          └──────────────┘            └──────────────────┘
                                                                                  ┌──────────────────┐
                                                                                  │    VIRTUAL       │
┌──────────────┐            ┌──────────────┐  ┌──────────────┐                    │  INSTRUMENTS     │
│  COMPONENT   │            │    VHDL      │  │   VERILOG    │                     └──────────────────┘
│   EDITOR     │            │ DESIGN ENTRY │  │ DESIGN ENTRY │                    ┌──────────────────┐
└──────────────┘            └──────────────┘  └──────────────┘                    │    ANALYSES      │
                                                                                  └──────────────────┘
┌──────────────┐                                                                  ┌──────────────────┐
│   SYMBOL     │                                                                  │      POST        │
│   EDITOR     │                                                                  │   PROCESSOR      │
└──────────────┘                                                                  └──────────────────┘
                            ┌──────────────┐
┌──────────────┐            │  FPGA/CPLD   │                                       ┌──────────────────┐
│    MODEL     │            │  SYNTHESIS   │                                       │    GRAPHER       │
│    MAKER     │            └──────────────┘                                       └──────────────────┘
└──────────────┘
```
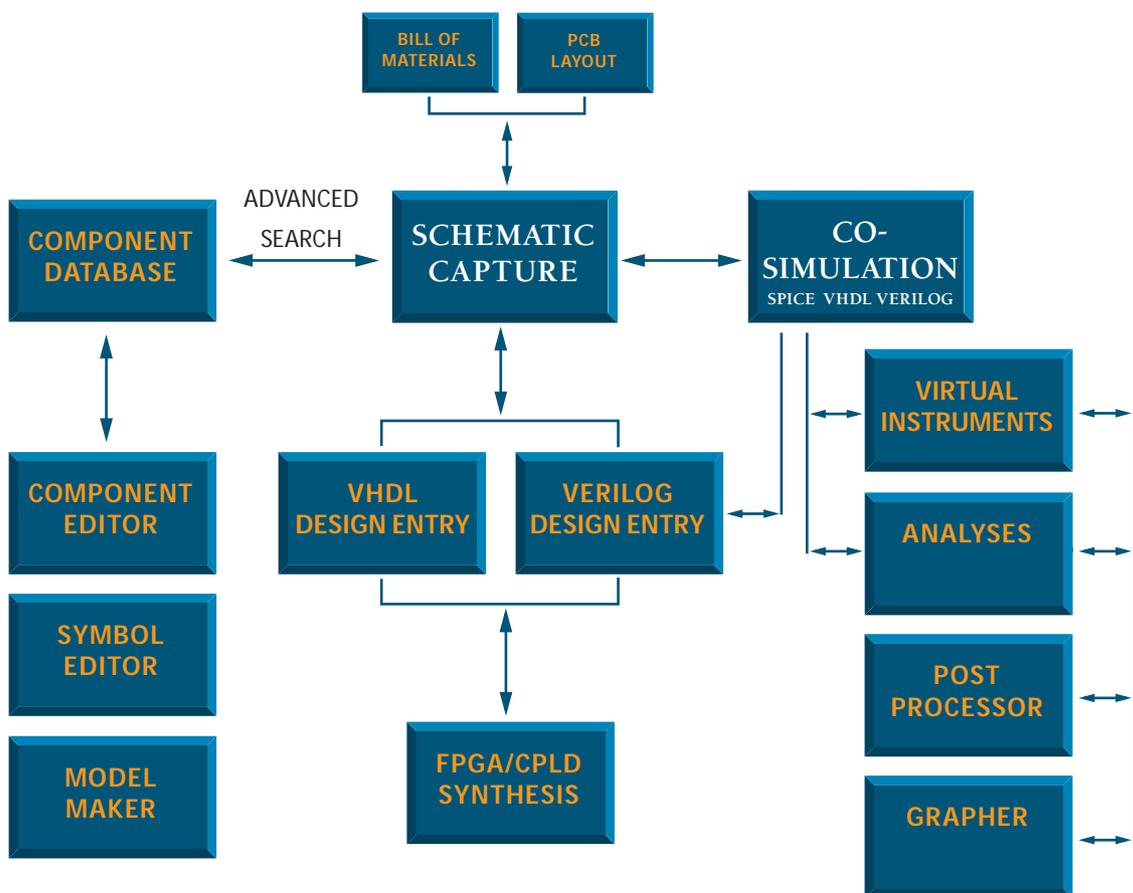
Co-Simulation Combines SPICE, VHDL & Verilog

Multisim offers the unique ability to design circuits containing any combination of SPICE, VHDL and Verilog modeled elements. A co-simulation core coordinates the communication between SPICE, VHDL, and Verilog simulation engines, so that Multisim's co-simulation is automatic and completely transparent to you.

COMBINING THE VERY BEST OF SPICE, VHDL & VERILOG

SPICE, VHDL, and Verilog each have their unique strengths such that various portions of your designs may be best modeled using different simulation languages. You can use VHDL, for example, to design and simulate your FPGAs or CPLDs. Then, Multisim allows you to combine SPICE, VHDL, and Verilog models into a single simulation of your entire circuit. As a result, Multisim delivers the world's best simulation engine, capable of accurately handling everything from basic analog components to the most advanced ICs.

A SINGLE TOOL HANDLES YOUR COMPLETE DESIGN NEEDS

Multisim eliminates the need to purchase multiple simulation tools from separate vendors. By using a single tool for your complete design, you won't have to worry about integration between programs. Best of all, Multisim's combined SPICE, VHDL, and Verilog simulation is less expensive than even one of these simulation products from other EDA vendors.

STAY COMPETITIVE AS ANALOG AND DIGITAL CONTINUE TO MERGE

Technology is changing such that the traditionally distinct worlds of digital and analog design continue to merge. Digital designers encounter analog effects from high-speed circuits which must also interface through analog components to the real world. At the same time analog engineers are performing more digital design than ever before because of the size and speed advantages that digital can offer over analog. Now, more than ever, there is a need for a single design tool equally well adapted to digital and analog design. Electronics Workbench meets this need with the world's most complete analog/digital design tool—Multisim.

CO-SIMULATION FEATURES

♦ Simulates SPICE, VHDL and Verilog separately and together

♦ Simulates circuits/systems containing both programmable logic and off-the-shelf devices

♦ Combines individual strengths of SPICE, VHDL and Verilog

♦ Eliminates the need to use separate tools for SPICE, VHDL and Verilog simulation

♦ Design, simulate and program your CPLDs/FPGAs, then simulate them as part of a complete PCB

"Board-level simulation has never really been commonplace among PCB designers because it couldn't handle all the components on the board. Multisim could very well herald the next step forward in PCB design methodology, because it can co-simulate both HDL and SPICE components, and it's available at a very affordable price."

Rita Glover,
EDA Industry Analyst and
President of EDA Today

With Multisim you can simulate designs containing a mix of SPICE, VHDL and Verilog.

VERILOG
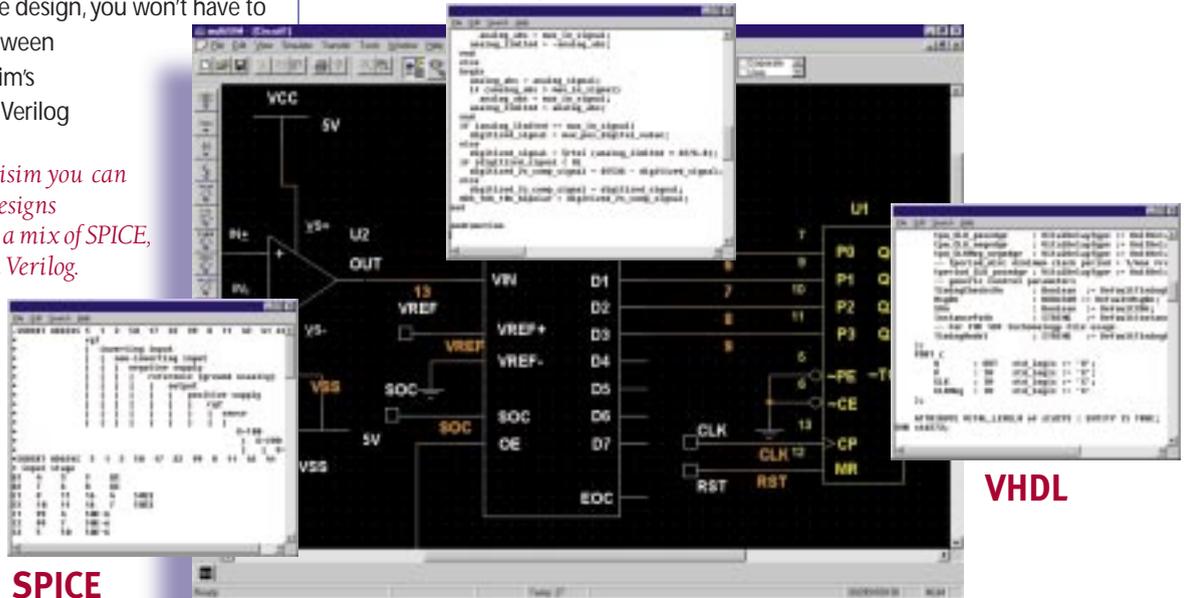
SPICE

VHDL

multiSIM

Full-Featured Schematic Capture

WORLD'S EASIEST-TO-USE SCHEMATIC CAPTURE SAVES YOU TIME

Multisim offers the world's most intuitive schematic capture for fast, flexible design entry. With Multisim you'll spend less time on schematic capture and more time on real circuit design. Wiring circuits in Multisim is simple compared with wiring in other tools. Just click on a source and destination pin—Multisim will automatically make the wiring connection for you.

MAKE YOUR DESIGNS STAND OUT WITH HIGH QUALITY SCHEMATICS

Multisim includes all the features you need to create professional schematics for documentation and reports. A built-in Symbol Editor allows you to edit the appearance of existing symbols or to create your own. You can also record all your schematic details with title blocks and fully customizable text and labels.
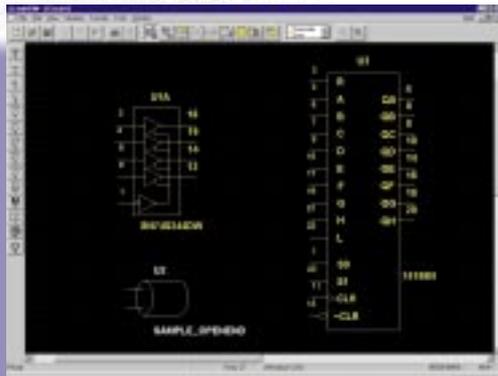
Multisim is the fastest and easiest way to produce high quality schematics.

SEAMLESS ACCESS TO SIMULATION & PCB LAYOUT

In less time than it takes to enter a circuit in other schematic capture programs, with Multisim you get a schematic that's ready to simulate and export to PCB layout. That's because every part in the Multisim

database contains both a simulation model and a footprint value from that part's databook. With a single click of the mouse, you can verify your circuit's performance or transfer your design to layout—without any errors!

SCHEMATIC CAPTURE FEATURES

♦ Industry's most intuitive user interface
♦ Automatic and manual wire routing
♦ Wire drag support
♦ Advanced symbol editor
♦ Bus support
♦ Sub-circuit support
♦ Print preview
♦ Right mouse clicks access context sensitive commands
♦ Back-annotation from PCB layout
♦ Copy and paste to other Windows applications
♦ Multiple active circuit support
♦ Dockable toolbars
♦ Hierarchical schematic support
♦ Zoom to different views
♦ Off-page connectors and modules
♦ Multiple ground support
♦ Export to all major PCB formats
♦ SPICE import/export
♦ PSPICE import to read old PSPICE files
♦ Advanced BOM
♦ Virtual wiring
♦ Undo
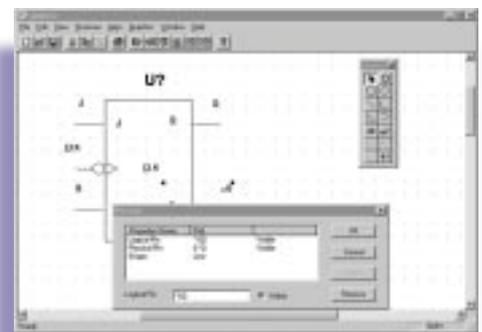
A built-in symbol editor lets you edit or create any component symbol.

Components & Models

ONE OF THE INDUSTRY'S LARGEST PARTS LIBRARIES WITH OVER 16,000 COMPONENTS

Multisim includes one of the industry's largest component libraries with over 16,000 parts. In addition to containing simulation models, these parts come complete with part numbers, symbols for schematic capture, footprints for layout, and all electrical parameters. Any of these part characteristics can be easily created or modified through Multisim's Component Editor.

ADVANCED & INTUITIVELY ORGANIZED PARTS DATABASE

Multisim stores its parts in a sophisticated, well-organized component database, providing you easy access to all your required devices through intuitive parts bins and component families—no more scrolling through long alphabetic lists. All parts placed in your circuit are stored in an "In-Use" list for instant, repeat access. In addition, Multisim provides you with the unique ability to perform advanced SQL searches for specific parts. For example you can search for a "BJT with a TO-220 package, a Vceo of 0.53-0.55 volts and a cost of less than 15 cents".

NO OTHER PROGRAM HAS AS MANY WAYS TO ADD NEW PARTS & MODELS

While Multisim offers one of the industry's most comprehensive and diverse component libraries, we recognize that it will never be possible to include every part you might wish to use. For this reason, we have equipped Multisim with more flexibility for adding parts than any other EDA product:

♦ BSPICE/ XSPICE/PSPICE import

♦ VHDL/Verilog import

♦ Generate models in C-code

♦ Use the built-in analog/digital Model Makers

♦ Model a part using a sub-circuit representation

♦ Copy and modify an existing model

CUTTING-EDGE MODEL MAKERS

Multisim's Model Makers are extremely sophisticated yet easy-to-use applications that automatically generate SPICE models from databook values. Multisim employs advanced algorithms that automatically calculate SPICE values, with optimal performance matching. Supported analog parts include Diodes, BJTs, MOSFETs, Zener Diodes, SCRs, and Opamps. Digital parts can be made with a simple-to-learn and easy-to-use descriptive language.

ANALOG/DIGITAL/HYBRID PARTS

♦ ADC	♦ GaAsFET	♦ Switch
♦ BJT	♦ IGBT	♦ TIL
♦ Capacitor	♦ Inductor	♦ Tiny logic
♦ Clock	♦ JFET	♦ Transfer function
♦ CMOS	♦ LED	
♦ Comparator	♦ MOSFET	♦ Transformer
♦ Connector	♦ Multiplier	♦ Triac
♦ Control functions	♦ Opamp	♦ TTL
♦ Crystal	♦ Optocoupler	♦ Vacuum tube
♦ Current Limiter	♦ Norton Amp	♦ Varactor
♦ DAC	♦ Pin diode	♦ Voltage reference
♦ Darlington	♦ Power MOS	♦ Voltage regulator
♦ Diac	♦ Potentiometer	♦ Voltage suppressor
♦ Diode	♦ Relay	
♦ Display	♦ Resistor	♦ Wideband Amp
♦ Divider	♦ Schottky diode	
♦ DMOS		♦ Zener diode
♦ Electromechanical	♦ Sources	

Multisim's Model Makers allow you to easily generate SPICE models from databook parameters.

multi**SIM**

Multisim SPICE

FAST, ACCURATE SPICE
SIMULATION

Multisim SPICE is the best way for you to simulate board-level analog and digital devices. The Multisim simulation engine is based on the industry standard and widely proven Berkeley SPICE, with enhancements for greater speed, accuracy and convergence. Multisim SPICE also includes XSPICE features to extend the capabilities of Berkeley SPICE for improved digital capabilities and behavioral modeling.
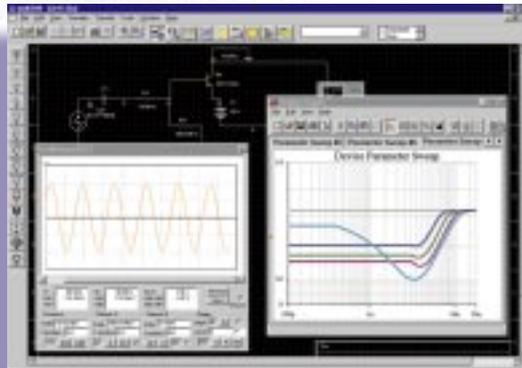
Multisim SPICE performs fast, accurate simulations of mixed analog and digital circuits.

WORLD'S EASIEST-TO-USE
SPICE TOOL

Multisim contains the world's easiest-to-use SPICE simulator. You will be simulating circuits with Multisim in the time it takes to install and configure most other applications.

Only Multisim contains a unique suite of virtual instruments that operate just like their real-world equivalents. When you adjust the controls on these instruments, SPICE commands are automatically issued to the simulation engine. As a result, you don't have to be a SPICE expert to take full advantage of this powerful simulation program. With Multisim SPICE, you'll spend more time designing and less time reading training manuals.
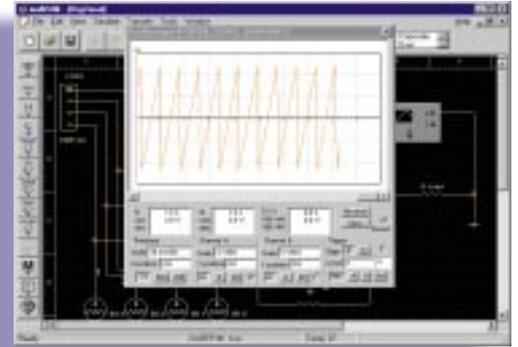
Use Multisim's Virtual Instruments to investigate circuits the same way you would in the real world.

MAKE CIRCUIT CHANGES
WHILE YOU SIMULATE

The world's only interactive simulator, Multisim SPICE enables you to make changes to your circuit while a simulation is running. For example, you can adjust a potentiometer and instantly observe changing waveforms on a virtual oscilloscope. This unique ability to make changes on the fly allows you to vary circuit properties to achieve optimal performance results.

SIMULATION FEATURES

♦ Industry-standard Berkeley SPICE simulation
♦ XSPICE enhancements to expand Berkeley SPICE3 capabilities
♦ Co-simulation with VHDL and Verilog
♦ Interactive simulation
♦ Extensive sources including DC, sine, pulse, piecewise linear, arbitrary, AM, FM
♦ Code modeling
♦ True mixed analog/digital simulation
♦ Advanced algorithms to resolve circuits with convergence problems
♦ Extensive simulation options for making speed/accuracy trade-offs

Electronics
WORKBENCH

Advanced RF Design Capabilities

BENEFITS OF MULTISIM RF

Multisim is the only general purpose simulator that has been enhanced to perform accurately at frequencies well above 100 MHz, where SPICE normally becomes unreliable. Multisim's RF Design Kit is comprised of a suite of tools critical to the success of designers working at higher frequencies, and includes a special parts library, RF Model Makers, RF Virtual Instruments and RF Analyses.

SPECTRUM ANALYZER

This powerful instrument allows you to view your circuit's characteristics in the frequency domain. Use the Spectrum Analyzer to investigate important details about signal harmonics, carrier levels and side-band frequencies.
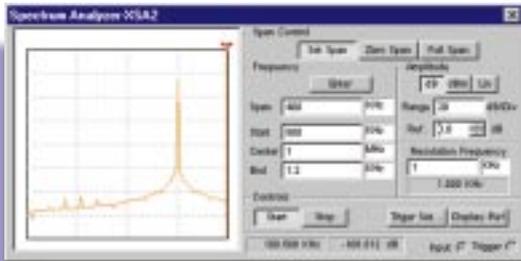
Multisim's Spectrum Analyzer displays the spectral composition of signals in your circuit.

NETWORK ANALYZER

The Network Analyzer displays your circuit's output versus input amplitude over a range of frequencies. From this data, Multisim automatically calculates S, Y, Z and H parameters.

CIRCUIT CHARACTERIZATION ANALYSIS

This analysis derives and identifies the characteristic properties of your RF circuits by automatically calculating the circuit's voltage and power gain. The characterization analysis also determines a noise figure for your circuit.

NETWORK MATCHING WITH SMITH PLOTS

Multisim RF plots network properties onto a Smith Chart and also generates stability and gain circles. It replaces the cumbersome task of performing impedance matching by hand and automates the process by offering interactive impedance matching through the Smith Chart.

Multisim RF performs automatic network matching using Smith Charts.

RF MODELS

Multisim contains an RF library with components unique to high frequency design. Models include:

- RF BJTs
- RF MOSFETs
- Schottky diodes
- Pin diodes
- Tunnel diodes
- Varactor diodes
- RF capacitor

- RF inductors
- Noiseless load resistor
- Noise source generator
- Transmission lines
- Striplines
- Microstrips Waveguides

RF MODEL MAKER

Multisim RF comes complete with a model-making tool to generate your own models for components not included in the RF library. This highly sophisticated tool automatically generates SPICE models from databook values for:

- RF Capacitors
- RF Inductors
- Transmission Lines

- Striplines
- Microstrips
- Waveguides

multi**SIM**

Analyzing & Displaying Results

MORE ANALYSIS & DISPLAY CAPABILITIES THAN ANY OTHER EDA TOOL

Multisim provides the most complete and flexible set of analysis and display capabilities of any EDA tool available. Choose from virtual instruments, advanced analyses or post processing. Alternatively, use the powerful ability of OLE (built into Multisim) to export your data to math applications or spreadsheets programs for further investigation and analysis.

VIRTUAL INSTRUMENTS OPERATE JUST LIKE THEIR REAL WORLD COUNTERPARTS

Multisim is the only software of its kind that includes a suite of virtual instruments that look and operate just like their real world counterparts. When you press a button on an instrument, an appropriate simulation command is issued and results are immediately displayed on the instrument's face. Since there's absolutely no learning curve to using these instruments, you'll be working productively the first day you start using Multisim. Virtual instruments let you take advantage of the full power of SPICE simulation without having to be a SPICE expert.

VIRTUAL INSTRUMENTS

Oscilloscope
Time base adjustable from nanoseconds to seconds with internal or external triggering.

Function Generator
Produces square, triangular, or sinusoidal waves. Adjustable frequency, duty cycle, amplitude, and DC offset.

Multimeter
Measures AC and DC current, voltage, resistance and decibel loss.

Bode Plotter
Measures frequency response of a circuit. Plots both gain or phase shift against frequency.

Network Analyzer
Measures the transfer function of networks over a range of frequencies.

Word Generator
Drives digital logic one word at a time, bursting through user-defined sets of data, or cycling continuously.

Logic Analyzer
Triggered internally or externally by recognition of a pre-set defined bit pattern.

Spectrum Analyzer
Measures amplitude versus frequency with adjustable span and amplitude range.

Distortion Analyzer
Provides distortion measurements for signals. Measures both intermodulation distortion and total harmonic distortion.

Wattmeter
Performs power readings at any point in the circuit.

*Multisim's virtual oscilloscope works just like a real oscilloscope on your bench.*

POST PROCESSOR PERFORMS MANIPULATIONS ON RESULTS

Multisim's Post Processor allows you to manipulate simulation results using a full range of math functions. Supported mathematical functions include:

• Addition	• Interpolate	• Or
• Absolute Value	• Length	• Phase
• And	• Log	• Power
• Atan	• Logical Equivalent	• Real
• Boltz	• Magnitude	• Random
• Cos	• Max	• Root
• dB	• Mean	• Sin
• Derivative	• Min	• Subtraction
• Division	• Multiplication	• Tan
• Exponent	• Norm	• True
• Imaginary	• Not	

Electronics
WORKBENCH

FULLY CUSTOMIZABLE GRAPHER FOR PLOTTING RESULTS

The grapher is used to both analyze signals and produce high-quality outputs of simulation results. The grapher's features include: *customizable grids, crosshair cursors for precise measurements, customization of legends, titles, fonts, and colors, copy and paste to other Windows applications, easy zoom in and out.*
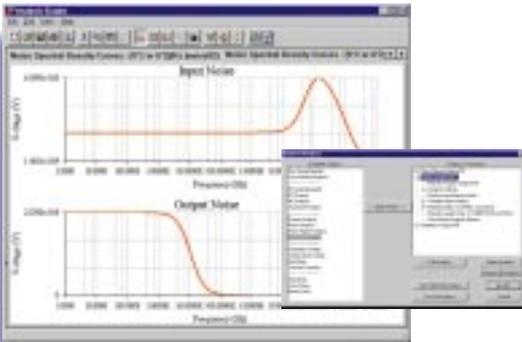
All simulation results are plotted to Multisim's fully customizable grapher.

MAKE MEASUREMENTS THAT ARE DIFFICULT OR IMPOSSIBLE TO PERFORM IN THE REAL WORLD

Whereas Multisim's virtual instruments give you all the flexibility of real lab work, a suite of powerful analysis functions lets you investigate your circuit in ways that are just not possible in the real lab.

ANALYSIS TOOLS

DC Operating Point
Determines the DC operating point of a circuit and produces a detailed report of voltages and currents at each circuit node.
Transient
Lets you understand the behavior of your circuit over a user-specified time range.
AC Frequency Sweep
Plots gain and phase responses over a specified range of frequencies.

User Defined Analysis
Enables you to create your own analyses to perform on circuits.
Fourier
Allows you to observe the spectral components signals in your circuit.

Noise
Determines the noise susceptibility of your circuits by calculating the RMS sum of noise contributions.

Distortion
Determines total harmonic distortion and reveals the spectral density of this distortion.

Temperature Sweep
Sweeps transient, DC, and AC response of any circuit over a range of operating temperatures.

Model Parameter Sweep
Performs transient, DC, and AC analysis while sweeping a component model parameter.
Nested Sweep
Runs an analysis within another analysis. For example, you can perform a temperature sweep on the results of a parameter sweep.

AC Sensitivity, DC Sensitivity
Displays sensitivity to a particular parameter and helps predict how variances in manufacturing can affect performance.
Pole-Zero
Determines the AC small-signal transfer function of a circuit and calculates the poles and zeros (points of instability).
Transfer Function
Derives the DC small-signal transfer function of a circuit and calculates the input resistance, output resistance and DC gain.
Worst Case
Determines the most extreme values to be expected in your circuit given the tolerances specified for each component.
Monte Carlo
Provides statistical analysis of the transient, AC and DC circuit response to random variations of device parameters.
Trace Width
Simulates current through all circuit traces and automatically assigns appropriate trace widths for PCB layout.
Batched Analysis
Allows you to schedule a sequence of analyses to be run.

EASILY TRANSFER DATA TO EXCEL & MATHCAD

Multisim contains OLE integration to automatically transfer your simulation results to Microsoft ® Excel and Mathsoft® Mathcad. Use this link to perform further analysis or investigation of your simulation data in these popular spreadsheet and math applications.

multi**SIM**

Project & Team Design

REMOTE ENGINEERING LETS YOU WORK WITH COLLEAGUES ACROSS DISTANT LOCATIONS

Multisim contains a unique Remote Control feature that allows you and your colleagues to work on identical circuits, live and in real time over the Internet or via an intranet. All participants see exactly the same screen and changes made by any participant are instantaneously seen by all others. This gives you the ability to work jointly with, or present designs to, colleagues or audiences anywhere in the world—as if you were all in the same room.

CREATE CUSTOMIZED CORPORATE, PROJECT OR INDIVIDUAL DATABASES OF PARTS

Multisim includes three levels of component databases. In addition to the Master Level database (the set of parts shipped with your product), you can configure two additional levels of component databases. The Project/Corporate Level database can be set up to include pre-approved parts allowing designers to easily identify preferred parts or alternatively to include the parts you need for a particular project. A User Level database allows you to configure your own custom libraries to include your favorite, specialty or custom parts. And with Multisim you can add customized fields to parts in the database for information such as: cost, lead-time, or preferred supplier.

Multisim lets you maintain multiple levels of parts databases to suit different user or project needs.

HIERARCHICAL DESIGN PARTITIONS AND MANAGES COMPLEX PROJECTS

Multisim's Hierarchical Design helps you break down complex projects into more manageable units—that way you can divide responsibility for different portions of a design so that they can be worked on at different times or by different members of an engineering team. Multisim provides a flexible and easy way to build and navigate through such complex hierarchical designs.

VERSION CONTROL PROTECTS AGAINST UNAUTHORIZED CHANGES

Multisim also includes a Version Control feature that automatically handles the coordination of critical files between members of a team. You can force files to be checked in and out to ensure that only one person has access to a particular portion of the design at any time. Multisim Version Control adds security to your designs and averts possible problems caused by multiple people working on a design in parallel.

NOW ALL TEAM MEMBERS CAN USE A COMMON DESIGN PROGRAM

As the world's most comprehensive general purpose design tool, Multisim meets the needs of analog, digital, programmable logic and RF designers. Multisim creates an open framework where team members can all work from a common schematic and parts database. Team members can simulate their designs alongside modules produced by their colleagues. Working with a common EDA tool ensures tighter design integration and increased group performance by encouraging interaction, collaboration, and the sharing of information between team members.

VHDL & Verilog Design and Simulation

IDEAL FOR BOTH HDL EXPERTS AND NOVICES

Whether you are a novice to HDL design or an experienced user, you'll find Multisim to be the best program at any price. Multisim's HDL functionality comes complete with flexible design entry, highly automated project management, powerful simulation, advanced waveform viewing, and comprehensive debug features. Multisim's VHDL or Verilog simulation can be used separately or together with Multisim's SPICE simulator. And despite this advanced functionality, these applications are incredibly easy-to-use.

THE TOOL OF CHOICE FOR FPGA/CPLD DESIGN

If you've been designing FPGAs or CPLDs using either VHDL or Verilog, then Multisim is the program for you! Both Multisim VHDL and Multisim Verilog come with a rare combination of advanced functionality and ease-of-use that had made Electronics Workbench the EDA vendor of choice for over 130,000 designers. For FPGA/CPLD design, Multisim should be your primary tool of choice. And whether you use Multisim's HDL functionality alone, or together with Multisim SPICE and Multisim RF, our product will help you design better programmable logic devices in less time.

SIMULATE FPGAs/CPLDs AS PART OF YOUR PCB DESIGN— THE IDEAL TESTBENCH

Because of Multisim's ability to co-simulate SPICE, VHDL, and Verilog, you can now verify the behavior of FPGAs/CPLDs (usually HDL models) connected to board-level devices (usually SPICE models) all as parts of the same PCB. There's no better way to verify the performance of your FPGAs/CPLDs than to simulate their true performance in your circuit.

THE BEST WAY TO GET STARTED USING HDLs

If you've been thinking about using HDLs to improve your designs, but couldn't afford the time to learn a new language, then Multisim has a solution for you! By simulating HDL models along with SPICE models, Multisim lets you ease into HDL design. Get started by replacing simple gates with HDL models. In time, you'll be simulating complex digital chips using HDL models alongside the analog and glue logic portions of your boards.
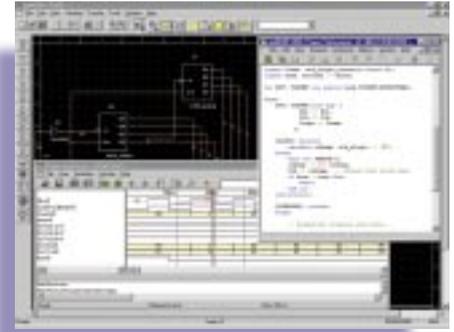
Both Multisim VHDL and Multisim Verilog offer high-end functionality while maintaining exceptional ease-of-use.

USE HDLs TO SIMULATE PARTS THAT CANNOT BE MODELED IN SPICE

VHDL and Verilog are not just for programmable logic! Most complex digital devices (eg. microprocessor memory) are described using HDLs, and cannot be practically modeled using SPICE. Multisim breaks through this SPICE limitation by enabling you to simulate SPICE and VHDL/Verilog models together. Add HDL models (from the Multisim library or an external source) to your circuits without requiring any HDL knowledge at all. For the very first time, you can now include parts in your PCB-level simulation that would be impossible to model in SPICE.

VHDL & VERILOG FEATURES

♦ The easiest way to get started using HDLs
♦ Simulate complex digital parts that can't be modeled in SPICE
♦ Simulate VHDL and Verilog parts without having to understand HDL syntax
♦ Complete design tools with code editors, project managers, simulation, waveform viewers and debugging
♦ Co-simulate with SPICE
♦ Full standards compliance

multiSIM

Multisim VHDL

INTELLIGENT SOURCE CODE EDITOR

Reading and writing code with Multisim VHDL is easy with the help of a built-in text editor that automatically color-codes VHDL syntax and intelligently indents your code. A built-in Design and Testbench Wizard simplifies and accelerates the generation of source code by automatically writing a large amount of custom shell code for you.

PROJECT MANAGER ORGANIZES YOUR DESIGNS

All the pieces of your designs, from source files through to model libraries, are neatly organized within Multisim's VHDL Project Manager. The Project Manager supports complex hierarchical file structures, and automatically handles file dependencies, ordering, and compilations for your files.
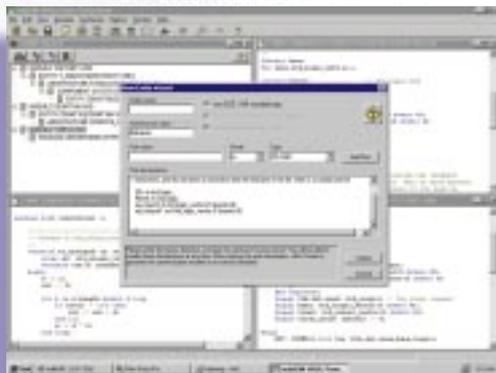
Multisim VHDL includes design wizards to reduce the amount of standard code you need to write.

WAVEFORM VIEWER FOR ANALYZING RESULTS

Multisim VHDL comes equipped with a powerful waveform viewer that allows you to observe signals and easily isolate and investigate state transitions or other areas of interest. The waveform viewer has cursors that can be used to measure the distance between signal events. Waveforms can be printed directly from the viewer to any Windows-compatible printer.

ACCURATE SIMULATION FOR RELIABLE RESULTS

Multisim VHDL delivers fast, high-performance simulation and fully supports the IEEE 1076-93 language. Multisim VHDL has been optimized for accelerated speed when using IEEE 1164 standard logic. Built-in VITAL support allows you to use SDF files for post-layout simulation and take into account real delay values. Multisim includes an automatic Dependency Make feature so there is no need to manually keep track of dependencies and ordering for correct compilation.

EASILY LOCATE & FIX ERRORS WITH ADVANCED DEBUGGING

Debugging your code with Multisim VHDL is made easy with the ability to set breakpoints, step the simulation one line at a time, and view the order in which VHDL statements are executed. The program also offers detailed and intelligent explanations of warnings and errors to help you quickly detect and correct problems.

VHDL FEATURES

♦ Built-in source file editor with color-coded keywords
♦ Entity/architecture templates
♦ The best way to learn VHDL at your own pace
♦ Powerful source level debugging
♦ Graphical waveform viewer to display results
♦ Detailed explanations of warnings and errors
♦ Complete VHDL IEEE 1076-93 support
♦ Design Wizard to automatically generate shell code
♦ Highly automated file compilation and ordering
♦ Testbench Wizard to create templates for test code
♦ Optional Synthesis from Electronics Workbench

Electronics
WORKBENCH

Multisim Verilog

PROJECT MANAGER STREAMLINES DESIGNS

Multisim Verilog comes with a sophisticated Project Manager which automatically handles the coordination between dependent files, relieving you of this cumbersome task and allowing you to focus on more important design issues. The Project Manager includes a Hierarchy Explorer for easy navigation of projects containing multiple levels of hierarchy. Multisim Verilog also comes with a built-in code editor for writing your source files. The editor supports reading and writing of Verilog code by indicating line numbers and identifying stop, start and breakpoints.
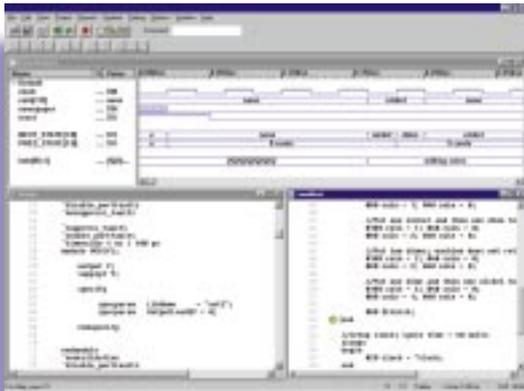
Multisim Verilog comes complete with everything you need to perform advanced Verilog design.

HIGH-PERFORMANCE SIMULATION

This fast, accurate simulator supports the entire IEEE 1364 standard language. The application delivers high-performance simulation of Verilog at the behavioral, gate, and switch level. All SDF capabilities are incorporated to allow back annotation of both functional and conditional (post layout) delay values for simulation. Multisim Verilog supports Analog Behavior Modeling (AHDL), an improved means of analog modeling which also accommodates integer and floating point I/O pins.

ANALYZE RESULTS WITH FLEXIBLE WAVEFORM VIEWER

For viewing simulation results, the graphical Waveform Viewer (or Data Analyzer) delivers both flexibility and ease of use. The Waveform Viewer includes controls for graphically navigating simulation results so that you can easily locate and focus on critical regions. Markers can be placed on the Waveform Viewer to take measurements and readings from the screen.

DEBUGGING MADE EASY

Debugging is made easy with the ability to set breakpoints or incrementally step through your code. Multisim Verilog facilitates monitoring state values with a unique 'watch window' which can be set to track any variable and expression as you single step through a simulation. Because the application incorporates a Save All feature which saves the netlist topology and all simulation results, you almost never need to re-simulate for each debugging task.

VERILOG FEATURES

- ♦ Standard IEEE 1364 support
- ♦ Analog Behavior Modeling (AHDL)
- ♦ Incorporates the full capabilities of SDF
- ♦ Fast simulation performance
- ♦ Built-in Project Manager
- ♦ Accurate simulation results
- ♦ Support of Verilog behavioral, gate and switch level
- ♦ Interactive, advanced debugging features
- ♦ Easy-to-learn and easy-to-use
- ♦ Graphical waveform viewer
- ♦ Advanced Debugging
- ♦ Hierarchy Explorer
- ♦ Built-in source code editor

FPGA/CPLD Synthesis

ADVANCED SYNTHESIS FOR ALL MAJOR FPGA & CPLD VENDORS

Multisim's sophisticated Synthesis Module will take your HDL code and produce an optimized netlist for FPGAs and CPLDs from most major vendors, including Actel, Altera, AMD, Lattice, Lucent, Quicklogic and Xilinx. Because the application uses advanced algorithms for size optimization, you will fit more code into less silicon. Multisim's Synthesis Module offers extremely fast synthesis speed, so you spend less time waiting and more time designing.

TIGHTLY INTEGRATED WITH MULTISIM VHDL

Multisim's Synthesis Module is the easiest way to transfer your HDL designs into actual hardware. The Synthesis Module is tightly integrated with Multisim VHDL so that exporting your HDL designs to Synthesis is a highly automated task. A File Manager lets you choose the designs you wish to synthesize and Multisim automatically handles any dependencies. Because Multisim Synthesis is so tightly linked with Multisim HDL, synthesis iterations are fast and error-free. Multisim Synthesis also synthesizes partial designs, allowing you to merge netlists during the place and route process. This valuable feature allows you to fit HDL blocks from different locations in your design into a single silicon device.

OUTPUT SUPPORT FOR ALL LEADING VENDORS

Multisim's unique support of all major FPGA and CPLD formats differentiates it from products which write only to their own proprietary netlist format. Because every design is unique, Multisim Synthesis gives you the freedom to choose the best silicon technology for each particular need.

KEY FEATURES

- Supports model generation
- Facilitates hierarchical synthesis
- Automatic I/O and macro-call interference
- Advanced size and speed optimization techniques
- Produces standard EDIF and XNF netlist formats
- Tightly integrated with Multisim HDL for fast, error-free synthesis
- Support for Actel, Altera, Lattice, Lucent and Xilinx
- Tight integration minimizes lengthy synthesis iterations

SUPPORTED DEVICES & VENDORS

- Actel - Series 1, 2, 3 and 32
- Altera - All devices
- AMD - All MACH devices
- Lattice - PLSI
- Lucent - Orca
- Quicklogic - All devices
- Xilinx - 3K, 4K, 4Ke, 7K and 9K
- Simple PLD devices (via a CUPL output interface)
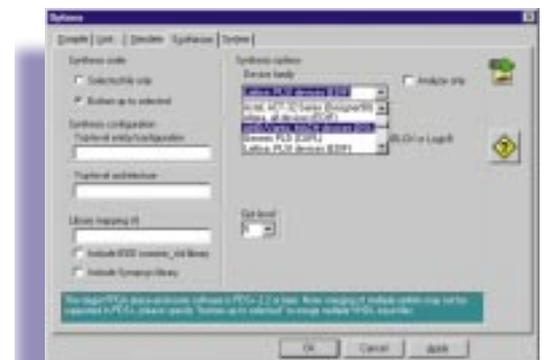
Multisim's Synthesis uses advanced algorithms for size optimizing, so you can fit more code on less silicon.

Complement Multisim

 Electronics WORKBENCH

ultiBOARD PCB Layout

POWERFUL PCB LAYOUT

Ultiboard offers all the features you need to quickly build reliable boards. Tightly integrated with Multisim, Ultiboard is available in a range of prices with some of the most advanced functionality on the market. And because Electronics Workbench offers a wide range of Ultiboard configurations, there's one that's right for you, no matter what your design or budget needs. All of this with the ease-of-use you have come to expect from Electronics Workbench.

TIGHT INTEGRATION WITH MULTISIM

Tight integration between Multisim and Ultiboard means that you can design with the confidence of knowing that all important and essential data will be effectively and automatically transferred from your schematics to Ultiboard. Forward and back annotation of design files ensures that circuit changes in Multisim will be reflected in Ultiboard, and vice versa. Multisim includes a unique Trace Width Analysis option which pre-calculates appropriate trace widths based on simulation results of currents in your circuits, and communicates this information to Ultiboard—a key advantage of selecting your PCB layout and simulation tools form Electronics Workbench.

FLEXIBLE COMPONENT PLACEMENT AND TRACE EDITING

Placing and editing your PCBs is fast and easy with Ultiboard. A Force Vectors feature indicates optimal locations for part placement. Once parts have been placed and routed, you can use the Reroute While Move function to reposition components without losing trace connections. Ultiboard comes with all the key features you'd expect from an advanced PCB tool: 32 layer support, power and ground plane support, built in editor for board outlines and component shapes, blind and buried via support, routing channel histograms, and graphical part searching capability.

POWERFUL DESIGN RULE CHECK

The real time Design Rule Check helps you prevent expensive errors by monitoring the size and clearance of pads, vias, and traces. Ultiboard notifies you of design violations as they are made, so there's no need to run batch check programs which only discover errors long after they have been made.

OUTPUT TO THE FORMATS YOU NEED

Ultiboard writes to all popular file formats to handle all your manufacturing and production needs. Creating output files is an easy and highly automated process. Output formats include: Gerber, .DXF, numerical control drill files, and plotter files.

FAST, HIGH-COMPLETION AUTOROUTING

Create smaller, more efficient boards with Ultiboard's powerful autorouter that includes advanced options so that you can customize performance, speed, and cost trade-offs to suit your needs. The autorouter is designed to handle high-density, multilayer boards, but can also be used for routing just a single component, a single trace, or any user-defined area.

ULTIBOARD FEATURES

♦ Powerful, interactive editing
♦ Real-time design rule check
♦ Over 3000 library shapes
♦ Density histograms to indicate routing densities
♦ Built-in shapes editor
♦ Chamfer corners option
♦ Any shape board up to 50"x50"
♦ 32 layer support
♦ 1 nanometer internal resolution
♦ Built-in autorouting
♦ Output for Gerber, .DXF and others
♦ Via reduction option
♦ Tight integration with Multisim

multiSIM Schematic Capture, Simulation & Programmable Logic

The World's Most Popular Circuit Design Tool

More designers rely on Electronics Workbench for circuit design tools than on any other EDA vendor. We've become the design tool of choice for over 130,000 satisfied customers by offering a broad range of powerful products at a variety of price points. With the release of Multisim, we are leading the way in board-level design with the unique ability to co-simulate SPICE, VHDL and Verilog. Co-simulation enables you to simulate all analog, digital (from simple logic to complex processors) and programmable logic (FPGAs/CPLDs) devices, not only separately, but also together as part of one complete board-level circuit.
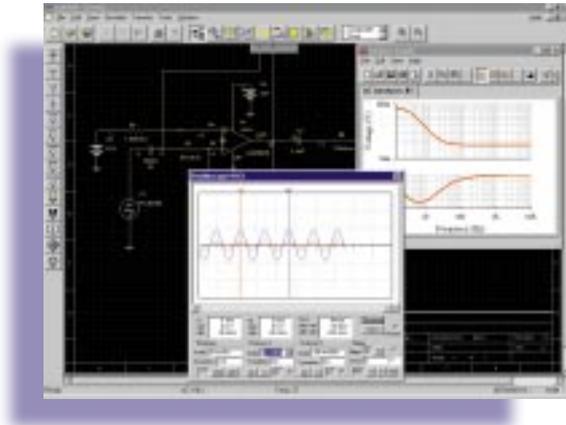
FEATURES

FEATURES	Professional	Power Professional
ADVANCED SCHEMATIC CAPTURE	✔	✔
SYMBOL EDITOR	✔	✔
PSPICE/XSPICE/BSPICE IMPORT	✔	✔
AUTO & MANUAL WIRING	✔	✔
SPICE ANALOG/DIGITAL SIMULATION	✔	✔
COMPONENT EDITOR	✔	✔
COMPLETE ONLINE DOCUMENTATION	✔	✔
INDUSTRY'S EASIEST USER INTERFACE	✔	✔
VIRTUAL INSTRUMENTS	9	11
MODEL LIBRARY	12,000	16,000
ANALYSES	14	19
COMPONENT DATABASE	BASIC	ADVANCED
PART SEARCH	BASIC	ADVANCED
BILL OF MATERIALS (BOM)	BASIC	ADVANCED
MODEL MAKER (ANALOG & DIGITAL)	OPTIONAL	✔
HDL DESIGN/ DEBUG AND SIMULATION	OPTIONAL	✔
RF DESIGN KIT	OPTIONAL	✔
TEAM DESIGN KIT	OPTIONAL	✔
MODEL EXPANSION PACKAGES	OPTIONAL	✔
POST PROCESSOR	N/A	✔
NESTED SWEEP ANALYSIS	N/A	✔
CODE MODELING	N/A	✔
BATCHED ANALYSIS	N/A	✔
USER DEFINED ANALYSIS	N/A	✔
PROGRAMMABLE LOGIC SYNTHESIS MODULE	OPTIONAL	OPTIONAL
ULTIBOARD PCB LAYOUT	OPTIONAL	OPTIONAL