

Welcome to our Demo CD



Electronics WORKBENCH

Design Solutions for Every Desktop



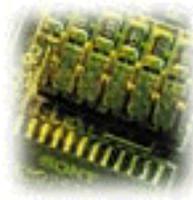
About Us



Multisim
Schematic Capture,
Simulation &
Programmable Logic



Ultiboard
PCB Layout



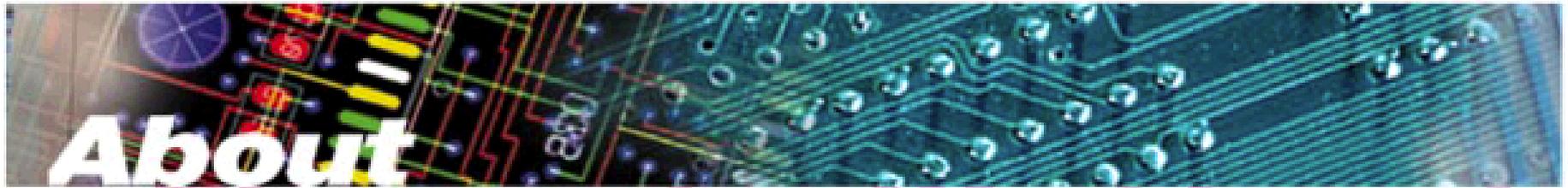
Ultriroute
Autorouting &
Autoplacement



Install
Product Demo



Contact
Electronics Workbench



Electronics Workbench

- **Electronics Workbench develops and markets the world's most popular circuit design tools with over 130,000 users.**
- **The company (previously Interactive Image Technologies) was incorporated in 1982.**
- **Corporate Headquarters are in Toronto, Canada with our US office in NY and our European office located in the Netherlands.**
- **Products are sold through a direct sales force in North America and through regional distributors internationally (for a listing see the “Contacting Electronics Workbench” section of the demo).**
- **Electronics Workbench is a pioneer in the EDA industry, with over 12 years of market leadership.**



PREVIOUS

BACK



NEXT

Exit

Main



Electronics Workbench

The Products

- **Electronics Workbench products are the most widely used circuit design tools in the global electronics industry.**
- **A broad portfolio of products cover all major steps in circuit design flow - from start to finish (schematic capture, SPICE simulation, VHDL/Verilog design entry and simulation, FPGA/CPLD synthesis, PCB layout, autorouting).**
- **Products are available in 8 languages (more than any other EDA product).**
- **Pioneering the future of mixed signal design with leading-edge co-simulation technology that combines SPICE, VHDL, and Verilog (Patent Pending).**





Electronics Workbench

The Mission

- **Design Solutions for Every Desktop**
 - We deliver design tools with broad functionality and scalable, easy-to-understand pricing options to serve all needs and budgets.
- **Value Leader**
 - We have always been dedicated to making leading edge technology both affordable and easy to use.
- **Tight Integration between tools**
 - We understand the importance of a smooth exchange of data between various tools in your design flow. Electronics Workbench products provide tight integration for all stages in your design.



SCHEMATIC CAPTURE, SPICE SIMULATION, AND PROGRAMMABLE LOGIC

- As the most recent version of EWB V5 schematic capture and simulation, the world's most popular circuit design tool, Multisim V6 is used by more designers than any other software of its kind. 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 less time than it takes to install and configure most other programs.

[Multisim overview with video clip examples](#)

[Product brochure in PDF format](#)

[Product documentation in PDF format](#)



THE INDUSTRY LEADER IN DESIGN ENTRY AND ANALOG/DIGITAL SIMULATION

- **Intuitive design entry lets you spend less time entering schematics and HDL code, and more time on real design.**
- **Fast, accurate simulation gives you immediate insight into circuit behavior.**
- **Multisim combines powerful functionality with industry-leading ease-of-use: a key reason that its the world's most popular circuit design tool.**
- **Co-simulation of SPICE, VHDL, and Verilog (Patent Pending) means that your boards can contain any combination of digital and analog parts from simple passives to complex ICs, FPGA's and CPLD's.**

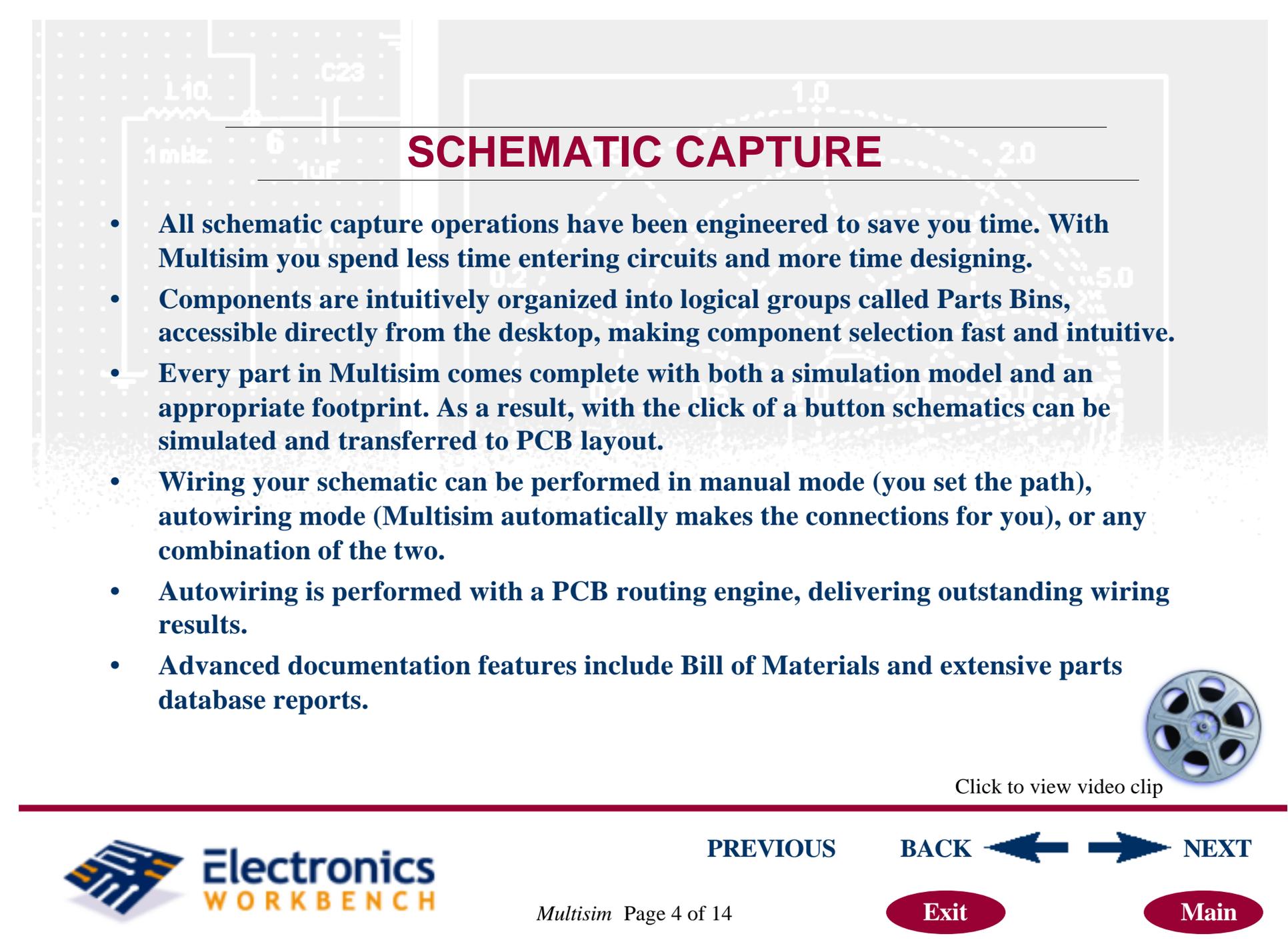


THE MOST COMPREHENSIVE CIRCUIT DESIGN TOOL AVAILABLE:



- Schematic Capture
- Extensive Component Database
- SPICE Simulation
- VHDL Design Entry, Simulation and Debug
- Verilog Design Entry, Simulation and Debug
- RF Design Kit
- Virtual Instruments, Analyses and Post-Processor
- Project/Team Design Kit
- FPGA/CPLD Synthesis





SCHEMATIC CAPTURE

- All schematic capture operations have been engineered to save you time. With Multisim you spend less time entering circuits and more time designing.
- Components are intuitively organized into logical groups called Parts Bins, accessible directly from the desktop, making component selection fast and intuitive.
- Every part in Multisim comes complete with both a simulation model and an appropriate footprint. As a result, with the click of a button schematics can be simulated and transferred to PCB layout.
- Wiring your schematic can be performed in manual mode (you set the path), autowiring mode (Multisim automatically makes the connections for you), or any combination of the two.
- Autowiring is performed with a PCB routing engine, delivering outstanding wiring results.
- Advanced documentation features include Bill of Materials and extensive parts database reports.



Click to view video clip



COMPREHENSIVE COMPONENT DATABASE

- Multisim includes an industry leading component library with over 16,000 parts.
- All parts come complete with schematic symbols, part numbers, simulation models, footprints for layout, and electrical parameters that completely describe their behavior.
- An advanced database structure delivers simplified part selection while providing complete flexibility to organize parts and edit their properties and values.
- A sophisticated SQL search capability lets you locate parts based on any parameter in the database: price, electrical properties, availability, manufacturer, packaging, or any user-defined parameter.
- Multisim has more ways to add models than any other program on the market (SPICE/VHDL/Verilog import, C-code modeling, Model Makers, Sub-circuits).
- State-of-the-art Model Makers automatically generate SPICE models from data book values.



Click to view video clip



CO-SIMULATION

- **Multisim delivers a breakthrough in simulation technology by simulating circuits containing a mix of SPICE, VHDL, and Verilog modeled components.**
- **This revolutionary technology gives you:**
 - The world's leading mixed analog/digital simulator that combines the best of multiple simulation technologies.
 - Once you have a completed FPGA/CPLD design (for which Multisim is ideally suited), Multisim also provides an ideal testbench for your FPGAs and CPLDs by simulating these chips together with the actual PCB components to which they connect.
 - The ability to use HDL models of complex digital ICs which cannot be modeled in SPICE (e.g. microcontrollers, memory, etc.) along with standard SPICE modeled components as part of a complete PCB design.



Click to view video clip



SPICE

- **Multisim SPICE is your best choice for analog and digital design, whether used as a stand-alone simulator, or co-simulating with Multisim VHDL and Multisim Verilog.**
- **Based on industry standard and widely proven Berkeley SPICE with XSPICE enhancements.**
- **Only program that includes virtual instruments that operate just like their real-world counterparts.**
- **When you adjust the settings on virtual instruments, Multisim automatically issues appropriate SPICE commands.**
- **The world's only interactive SPICE simulator lets you change circuit values while simulation is running and instantly see the reflected change.**



Click to view video clip



VHDL

- **Multisim VHDL is your best choice for FPGA/CPLD design using VHDL.**
- **Can be used as a stand-alone simulator, or co-simulating with Multisim SPICE and Multisim Verilog.**
- **Ideal for modeling complex digital ICs (e.g. microcontroller, memory, etc.) that cannot be built using SPICE.**
- **Intelligent source code editor automatically color-codes and indents your code for easy reading and writing.**
- **Project manager neatly organizes and automatically handles file ordering and dependencies.**
- **Waveform Viewer makes it easy to observe, isolate, and investigate signals.**
- **Fast, accurate IEEE 1076-93 simulation with powerful debugging to easily identify and correct errors.**



Click to view video clip



VERILOG

- **Multisim Verilog is your best choice for FPGA/CPLD design using Verilog.**
- **Can be used as a stand-alone simulator, or co-simulating with Multisim SPICE and Multisim VHDL.**
- **Ideal for modeling complex digital ICs (e.g. microcontroller, memory, etc.) that cannot be built using SPICE.**
- **Project manager provides easy navigation and coordination of source files and their dependencies.**
- **Fast, accurate IEEE 1364 simulation at the behavioral, gate, and switch level.**
- **Flexible, easy-to-use waveform viewer to easily probe critical regions in your code.**
- **Advanced debugging features make it easy to locate errors and track variables and expressions.**



Click to view video clip



RF DESIGN KIT

- **Multisim is the only all-purpose design tool with enhancements and features for you to design at higher frequencies.**
- **Spectrum Analyzer lets you examine your circuit's frequency domain characteristics.**
- **Network Analyzer automatically determines a network's S, Y, H, and Z parameters.**
- **Impedance matching can be performed automatically or interactively through Smith Charts.**
- **RF model library is included with parts you need for higher frequency designs.**
- **RF Model Maker automatically generates new RF models from databook values.**
- **Automatic calculation of noise figures, power gains, and voltage gains.**



Click to view video clip



ANALYZING AND DISPLAYING RESULTS

- **Multisim offers the most comprehensive set of analysis and display capabilities in the EDA industry, including virtual instruments, analyses, and post-processor**
- **Virtual instruments operate exactly like their real world equivalents, and include:**
 - Oscilloscope, Function Generator, Multimeter, Bode Plotter, Network analyzer, Word Generator, Spectrum Analyzer, Logic Converter, Wattmeter, Distortion Analyzer, Logic Analyzer.
- **Advanced analyses allow you to investigate circuits in ways sometimes just not possible in the real world:**
 - DC Operating Point, Transient, AC Frequency Sweep, Fourier, Pole-Zero, Transfer Function, Worst Case, Monte Carlo, Trace Width, Noise, Distortion, Temperature Sweep, Model Parameter Sweep, Nested Sweep, AC Sensitivity, DC Sensitivity, Batched Analysis, User Defined.
- **A fully-customizable grapher is used to display, analyze, and output results.**
- **Using Multisim's OLE integration with Excel® and MathCAD®, you can export simulation data for further analysis.**



Click to view video clip



PROJECT AND TEAM DESIGN

- **Multisim's remote engineering support lets you share circuits with colleagues in real-time or design together (on the same circuit) live over the Internet.**
- **Multiple database levels enable you to configure custom parts libraries for an enterprise, project, or team.**
- **Customized user fields can hold information such as cost, lead-time, or preferred supplier for each part.**
- **Hierarchical support breaks complex projects into manageable units and provides automatic updating of higher/lower levels of your designs.**
- **Project Management is provided through easy-to-use, easy-to-navigate interface.**
- **Version Control feature protects against unauthorized changes.**



Click to view video clip



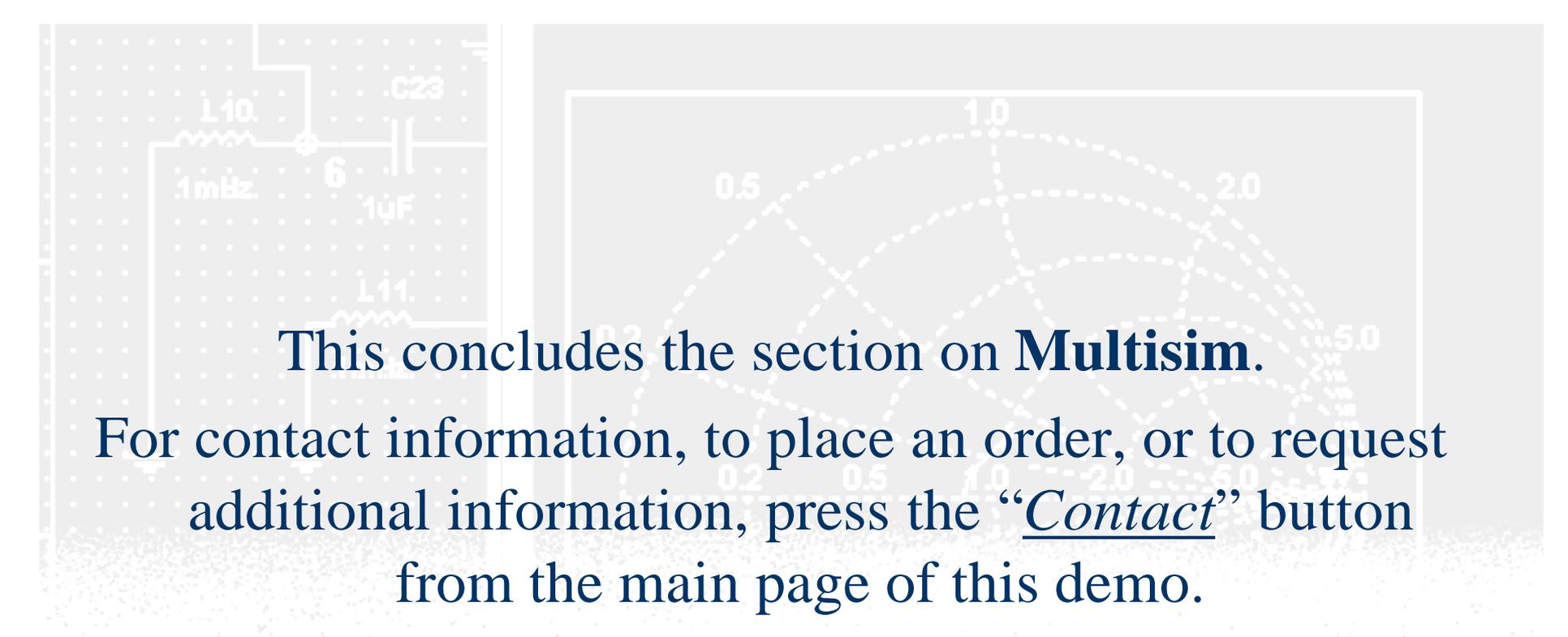
FPGA/CPLD SYNTHESIS

- **Multisim Synthesis supports all major vendors so that you can choose the best technology for each design.**
- **Tight integration with Multisim HDL ensures automatic handling of dependencies and permits merging of separate design units onto a single device.**
- **Extremely fast synthesis speeds and advanced optimization algorithms fit more code only less silicon.**
- **Supported devices include: Actel, Altera, AMD, Lattice, Lucent, Quicklogic, Xilinx.**



Click to view video clip



The background of the slide features a circuit diagram on the left and a graph on the right. The circuit diagram shows an AC voltage source labeled '1mHz' connected in series with an inductor 'L10', a capacitor 'C23' with a value of '10F', and another inductor 'L11'. A node between the capacitor and the second inductor is labeled '6'. The graph on the right shows a sinusoidal wave with a peak value of 1.0 and a period of 2.0. The x-axis is labeled with values 0.2, 0.5, 1.0, 2.0, and 5.0. The y-axis is labeled with values 0.2, 0.5, 1.0, 2.0, and 5.0.

This concludes the section on **Multisim**.

For contact information, to place an order, or to request additional information, press the “Contact” button from the main page of this demo.



POWERFUL PCB LAYOUT

Draw angle

- 90
- 45 + 90
- all angles

Method

- trace

- **Ultiboard offers all the advanced features you need to quickly produce reliable boards, but without the steep learning curve and high price tags normally associated with other high-end design tools.**

[Ultiboard overview with video clip examples](#)

[Product brochure in PDF format](#)

[Product documentation in PDF format](#)



UNIQUE COMBINATION OF POWER AND EASE-OF-USE

- **Ultiboard is a powerful, yet easy-to-use PCB layout tool that delivers the features and functionality you need to effectively build reliable boards.**
 - Real-time design rule check
 - 32 layer support
 - 1 nanometer resolution
 - Blind & buried vias
 - Component and trace dragging
 - Output to the formats you need



TIGHTLY INTEGRATED WITH MULTISIM

- Transferring your schematics from Multisim to Ultiboard is done with a simple click of a button.
- Design with confidence knowing that all important design details are transferred reliably.
- All of Multisim's 16,000 parts have correct footprint values so that you'll never encounter "footprint not recognized" errors.
- Schematic and PCB layouts are always synchronized with complete forward and back-annotation.



Click to view video clip



BOARD SETUP

- **Ultiboard supports almost any board shape.**
- **Board shapes can be generated using a built-in, easy-to-use editor, or imported from a file.**
- **Handles all board sizes from very small up to 50" x 50" in size.**
- **32-layer support, with or without power planes, handles even the most advanced technology boards.**
- **1nm internal resolution is one of the finest in the industry, so that you can design with as much precision as you need.**



Click to view video clip



COMPONENT PLACEMENT

- **Ultiboard's Force Vectors feature makes recommendations on where to place each component to achieve optimal part placement.**
- **Histograms show routing channel densities at cross-sections of your boards.**
- **Exact placement location and rotation angles can be issued for parts requiring precise placement.**
- **Ultroute, available as an option, provides complete autoplacement functionality (See "Ultroute").**



Click to view video clip



TRACE PLACEMENT

- **Ultiboard supports 90 degree, 45 degree or any angle trace routing.**
- **Trace widths and layers can be changed as you route.**
- **When routing from a pad, all other pins belonging to the same net are highlighted for easy identification.**
- **Making changes to your layout is easy with Ultiboard's powerful component and trace drag feature.**
- **Copper fill support with automatic void creation should you need to place traces or parts in copper regions.**



Click to view video clip



REAL-TIME DESIGN RULE CHECK

- **Ultiboard's real-time design rule check helps you produce error-free boards that always work on the first run.**
- **Discovers errors and violations as they occur, not when it's too late.**
- **Rules can be specified for trace, pad, component, and board outline clearances.**
- **Design Rule Check can be run in Full Mode (never allowing violations), Override Mode (alerts you to, but allows you to perform violations), or disabled altogether.**



Click to view video clip



MANUFACTURING SUPPORT

- **Ultiboard can improve manufacturing yields by automatically replacing 90 degree corners with 45 degree angles.**
- **A Delete Open Trace Ends and Delete Unused Vias feature cleans up boards before going to production.**
- **Output to the formats you need:**
 - Gerber (274X and 274D)
 - Pen plotters
 - Photo plotters
 - .DXF files
- **A renumber feature allows you to perform customized annotations of your boards.**



Click to view video clip



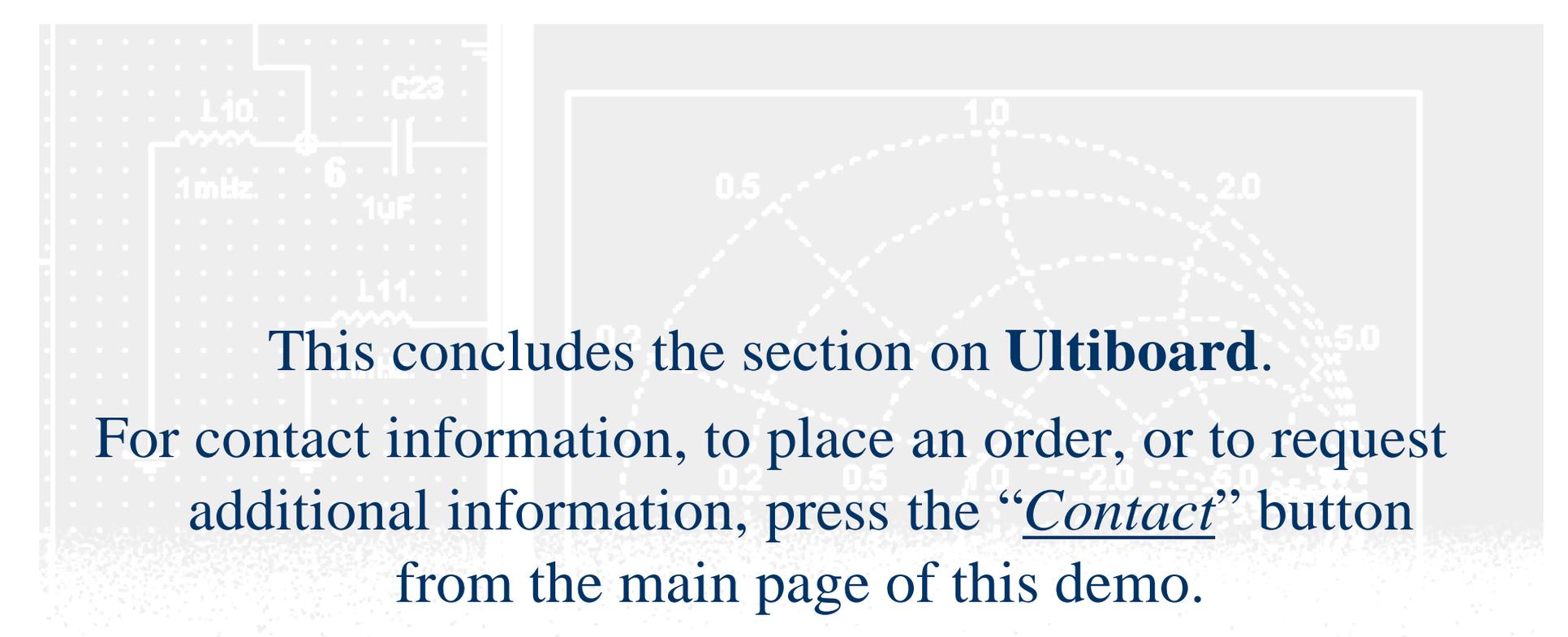
AUTOROUTING

- **Ultiboard includes a built-in standard autorouter.**
- **Routine performance and strategy can be customized with many control options: via reduction, pin/gate swap, layer directions, costing factors, trace hugging, 45 degree pad entry, via offset.**
- **Can be operated in single-pass mode, for simple boards, or rip-up-retry for more advanced applications.**
- **Ultroute is available as an option for high-end autorouting (See “Ultroute”).**



Click to view video clip





This concludes the section on **Ultiboard**.

For contact information, to place an order, or to request additional information, press the “Contact” button from the main page of this demo.



AUTOROUTING AND AUTOPLACEMENT

Draw angle
 90
 45 + 90
 all angles
Method
 trace

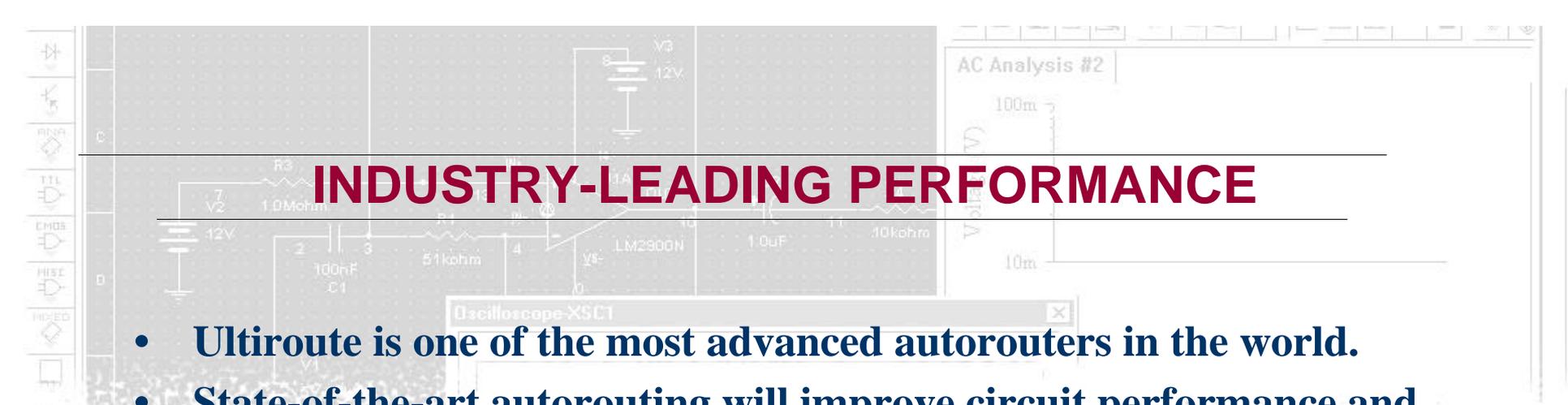
- **Ultroute combines advanced autoplacement with state-of-the-art autorouting for optimal layout of your printed circuit boards. No matter what the complexity of your designs, you'll be amazed at the way Ultroute will place and route your boards, consistently improving circuit performance and reducing production costs.**

[Ultroute overview with video clip examples](#)

[Product brochure in PDF format](#)

[Product documentation in PDF format](#)





INDUSTRY-LEADING PERFORMANCE

- **Ultiroute is one of the most advanced autorouters in the world.**
- **State-of-the-art autorouting will improve circuit performance and reduce manufacturing costs.**
- **The only autorouter that uses a combined grid and gridless routing strategy that gives you the best features of each approach.**
- **By improving your layouts and reducing production costs, Ultiroute pays for itself many times over.**
- **Eliminates the need to perform time-consuming and tedious manual wiring.**
- **Demonstrates superior routing to Spectra® V6 in benchmark tests (with fewer vias and shorter copper lengths)**



AUTOPLACEMENT

- **Autoplacement strategically places parts to achieve optimal routing, thereby considerably improving routing results.**
- **SMD parts can be placed on the underside of the board.**
- **Components requiring precise positioning can be pre-placed before autoplacement.**
- **Fully customizable control parameters include: part separation, pin/gate swap, SMD fanout, cluster placement and automatic block capacitor recognition.**



Click to view video clip



AUTOROUTING

Draw angle

- 90
- 45 + 90
- all angles

Method

- trace

- **Ultiroute is the only autorouter that always routes on-grid when possible, but will place traces off-grid where necessary.**
- **Delivers the efficiency and performance of a gridless autorouter.**
- **With most traces located on-grid, it is easy to edit and manipulate post-routing results.**
- **Supports manual wire pre-placement and power layer routing.**
- **Routing performance can be controlled with fully-customizable routing options.**
- **Default routing parameters are automatically set based on an analysis of your board's unique properties to always give near optimal routine performance.**
- **Customizable parameters include: via offset, pin/gate swap, SMD via pre-placement routing cost parameters, routing grid size and maximum via setting.**



Click to view video clip



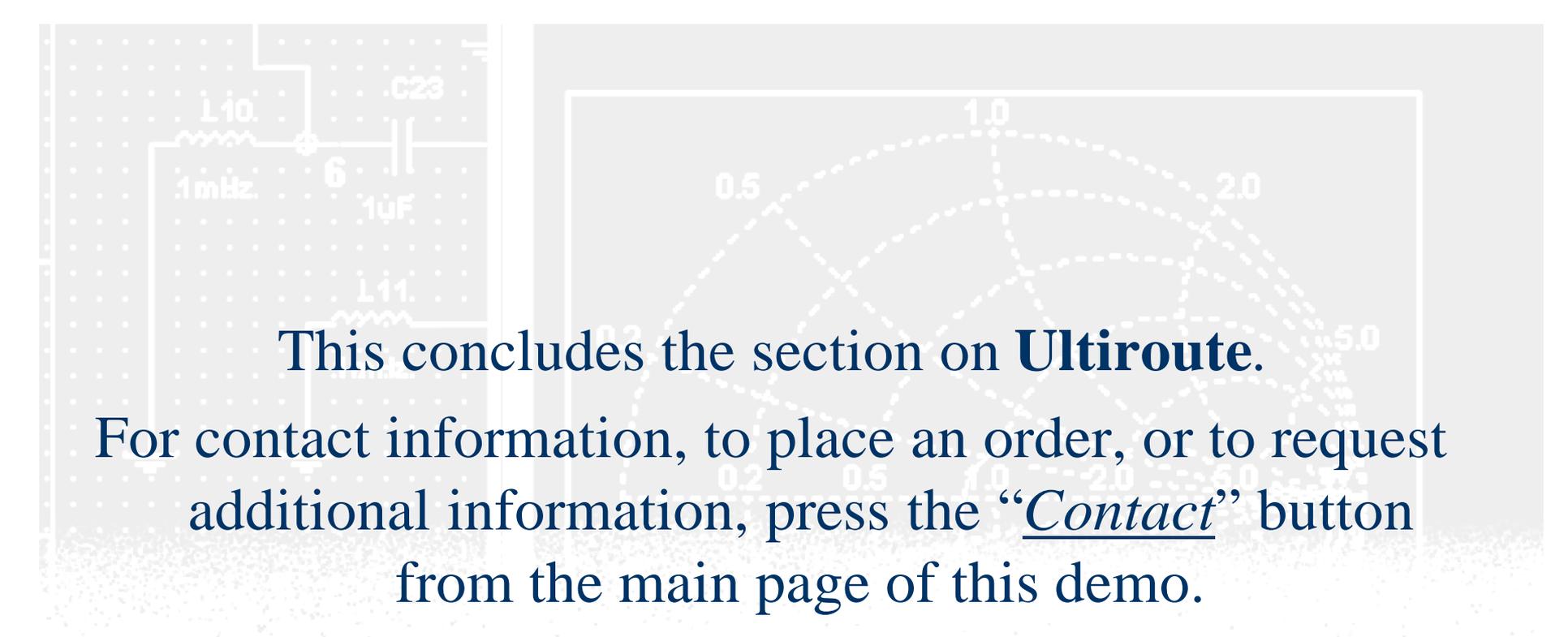
MANUFACTURING OPTIMIZATION

- **Ultiboard lets you optimize your layouts even after achieving 100% completion.**
- **Optimization will improve signal layouts, reduce production costs, and improve manufacturing yields.**
- **Optimization features include: Via reduction, Corner mitering, Trace smoothing, and Cross-net cleanup.**



Click to view video clip



The background of the slide features a circuit schematic on the left and a graph on the right. The schematic shows a series circuit with a 1mHz AC source, an inductor labeled L10, a capacitor labeled C23 with a value of 10uF, and another inductor labeled L11. The graph on the right shows a series of overlapping bell-shaped curves, likely representing resonance curves for different frequencies, with labels 0.5, 1.0, 2.0, and 5.0 indicating frequency values.

This concludes the section on **Ultiroute**.

For contact information, to place an order, or to request additional information, press the “Contact” button from the main page of this demo.





Demo *Download*

- From the list below, please choose the demonstration to be installed on your computer

Schematic Capture, Simulation & Programmable Logic

Printed Circuit Board Layout

Advanced Autorouting & Autoplacement





Contact Information

North America

Web: <http://www.electronicsworkbench.com>

Phone: 1-800-263-5552

416-977-5550

Fax: 416-977-1818

Email: info@electronicsworkbench.com

Electronics Workbench USA

908 Niagara Falls Boulevard, Suite #068

North Tonawanda, NY

14120-2060

Electronics Workbench Canada

111 Peter Street, Suite 801

Toronto, Ontario, Canada

M5V 2H1

International Distributors



PREVIOUS

BACK



NEXT

Exit

Main



For more information on our products visit the
Electronics Workbench home page at
<http://www.electronicsworkbench.com>

Exit demo?

1999 Interactive Image Technologies Limited, Electronics Workbench and the Electronics Workbench logo are trademarks or registered trademarks of Interactive Image Technologies Ltd. All other names are trademarks of their respective owners.