Electronics Workbench education

The World’s Most Popular Software for Electronics Education

multiSIM
Intuitive Schematics, Accurate Simulation and Programmable Logic

ultiBOARD
Powerful Printed Circuit Board Layout
Electronics Workbench

The World’s Most Popular Circuit Design And Simulation Tool

More electronics educators and electronics professionals rely on Electronics Workbench for circuit design, simulation and layout tools than any other vendor. We’ve become the design tool of choice for over 130,000 satisfied customers by offering a broad range of high-quality products at effective prices. Around the globe, our software has become the defacto standard in education (at all levels, from university and college to high school) and has been translated into 8 different languages, far more than any other electronics software program!

Electronics Software Solutions

The suite of Electronics Workbench software solutions includes: Multisim Schematic Capture and Simulation; and Ultiboard Printed Circuit Board Layout. Multisim, an industry leader in board level design, delivers the unique capability to co-simulate SPICE and VHDL together as part of one complete board level circuit.

At Electronics Workbench we’re committed to the education market and have developed products with specific features to meet the unique requirements of this demanding and growing sector. In addition to the Education versions of Multisim and Ultiboard, we have developed a wide range of complementary education resources including: circuit files that accompany leading electronics textbooks; computer-based training solutions; and books to enhance the use of our software. Of course, you can always feel confident utilizing Electronics Workbench products in your education environment, because our portfolio is one of the most widely used in the professional engineering community.

MULTISIM

ULTIBOARD

ADDITIONAL RESOURCES
Teaching using Electronics Workbench

Electronics Workbench software products have become the standard in supporting electronics teaching across a broad range of curriculum including: basic electricity, introductory and advanced electronics, circuit analysis, analog/digital circuit design, RF communications, and digital electronics. Electronics Workbench solutions enable educators to take students through the entire process of circuit analysis design, simulation, troubleshooting and PCB layout. Whether you teach college, university, high school, in a classroom or via remote distance learning, Electronics Workbench products can help transform your classes into an exciting environment where true learning occurs.

Learning using Electronics Workbench

Students learn faster and retain course content longer when they learn with Electronics Workbench. Multisim and Ultiboard are advanced and powerful software solutions that are both easy-to-learn, easy-to-use, and provide students a powerful computer-based learning environment that fits many diverse learning styles. Students can begin by using Multisim to work through virtual laboratory assignments, create and design circuits, simulate circuit behavior and troubleshoot for faults. The learning continues with Ultiboard which is used to complete the design cycle with printed circuit board layout and output to manufacturing.

Electronics Workbench Labs

In addition to the Education Edition, intended for use by educators in the development of curriculum, circuits, demonstrations, the Lab Edition is also available and is our most popular product. Deploying an Electronics Workbench Software Lab into your institution will add flexibility and new options in teaching electronics. As an example, Multisim’s complete virtual lab of instruments, analyses and components can be used in place of costly traditional hardware labs or as a complement to reduce the need for large quantities of hardware stations. Available in both networked and standalone configurations, Multisim Labs provide an economical alternative to complement hardware laboratory work including pre and post lab alternatives. When integrated with Ultiboard PCB Layout, students can design and test their circuit, transfer a circuit to Ultiboard, layout and route their board and then export files for actual board production using any of the most common techniques.

Affordable Student Edition

Affordable and with the features to complement the full package, the Student versions of Multisim and Ultiboard allow students to get the most out of their courses via at-home study and educators to get the most out of students.

“Electronics Workbench is a terrific aid to help students learn circuit design. It provides a learning experience that is very similar to the actual lab experience, allows a far greater variety of circuits to be studied and, of course, is far more convenient.”

Dr. Adel Sedra, University of Toronto
Multisim is the world’s leading educational schematic capture and simulation tool because it is the most powerful and easy to use product available on the market today. Students can typically place components, connect their first circuits, and run simulation within 20 minutes of being introduced to the product. It is completely intuitive—students spend time learning electronics, not a confusing user-interface.

In less time than it takes to enter a schematic using most other products, with Multisim you will have created a high-quality circuit that is ready to simulate. For example, only with Multisim can you drop parts onto wires and have the wire automatically open to make room for the part (and make the connection). As well, Multisim offers you the benefit of autowiring—just click on the pins you want to connect and Multisim automatically finds the best path through the schematic, to make the connection. In addition, you as the educator can set Multisim program options, such as circuit restrictions and passwords to appropriately challenge and test your students.

Five thousand industry-standard components are only a mouse click away from your virtual benchtop. Multisim includes one of the industries largest component libraries complete with part numbers, symbols for schematic capture, models for simulation and footprints for layout, as well as full electrical information. The new, well-structured component database offers a wide variety of part types including a diverse selection of analog, digital, RF, electromechanical and hybrid parts. You can easily locate any of these parts using the powerful, built-in search engine.

In addition to the comprehensive library included with Multisim, users can edit or create their own parts using the highly flexible Component Editor to modify, assign or create new footprints, symbols or models. Multisim offers more methods for obtaining simulation models than any other product of its kind—import SPICE models, create a subcircuit, utilize advanced C-code modeling, or take advantage of Multisim’s state-of-the-art Model Makers that automatically generate SPICE models from databook values. You can even use VHDL models for complex LSI/VLSI parts with the purchase of an optional VHDL Add-on module.
Multisim SPICE is the best way to simulate circuits—it’s 100% compliant with industry-standard SPICE, unlike some other proprietary technologies, such as PSpice. The Multisim simulation engine is based on the industry standard and widely proven Berkeley SPICE, with customization for greater speed, accuracy and convergence. Multisim SPICE also adds XSPICE features to extend the capabilities of Berkeley SPICE for improved digital capabilities and behavioral modeling.

Easy-to-use SPICE Simulation

Multisim's SPICE simulator is easy-to-use—simply build the circuit and with the click of the mouse you can begin simulating in less time than it takes to install and configure most other similar 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 software user guides.

Interactive Simulation—make circuit changes while you simulate

Multisim is the only program of its kind that allows you make changes to a circuit while a simulation is running, providing you with instant feedback. The importance of this capability to a students’ learning experience cannot be overstated. By modifying component parameters, during simulation, students can instantly see changes in the oscilloscopes display. For example, in classroom demonstrations or in the lab, the behavior of a circuit can be brought to life via the changing waveforms as a potentiometer is varied.

CO-SIMULATION COMBINES THE BEST OF SPICE AND VHDL

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

SPICE & VHDL each have their unique strengths such that various portions of your designs may be best modeled using different simulation languages. You can use VHDL to design and simulate your FPGAs/CPLDs or for modeling LSI/VLSI chips which cannot practically be modeled in SPICE. Then, Multisim allows you to combine SPICE & VHDL 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.
Bring in-class instruction to life

Multisim is the only software of its kind to provide a suite of virtual instruments, 9 in total. Introduce students to the world of electronic instrumentation with unbreakable virtual instruments that look and operate just like their real-world counterparts. Best of all, the need for expensive hardware is reduced, students learn without risk to their safety, and access to virtual instruments is extended to all students in the computer lab.

Begin by creating a circuit. Next, connect an instrument to the circuit in exactly the same way you would with real test equipment, make adjustments to the controls (SPICE commands are sent transparently to the simulation engine) and the appropriate signals are displayed on the face of the instrument.

With Multisim’s virtual instruments, SPICE quirks and peculiarities are hidden from view. Students can, therefore, tap into the full power of SPICE without having to focus on messy code and syntax.

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 S-parameters of networks, with Smith Charts and impedance matching.

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 preset 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.


Richard McCammack,
Teacher, O’Fallon Twp. High School
**POWERFUL ANALYSES**

**Powerful Analysis**

Multisim provides the most complete and flexible set of analysis and display capabilities of any design and simulation tool available.

A suite of 16 powerful analysis functions lets you investigate circuits in ways sometimes not possible in the real world. Students make use of Analysis Wizards to take full advantage of these sophisticated abilities. The analyses not only help students understand the behavior of circuits, but also allow them to gauge its stability and sensitivity.

**Post Processor**

Simulation results can be plotted to a fully customizable grapher, then mathematically manipulated using a wide range of functions (arithmetic, trigonometric, calculus, algebraic, boolean, logic). Graphs can be easily copied to student assignments or placed on a schematic to illustrate a signal.

Alternatively, use the powerful ability of OLE (included in Multisim) to export your data to math applications (e.g., Mathcad®) or spreadsheets applications (e.g., Microsoft® Excel) for further investigation and analysis.

**DESIGN AT HIGHER FREQUENCIES WITH MULTISIM RF**

With recent technological advances, RF (Radio Frequency) electronics has become very important within college and university electronics programs. To meet this requirement Multisim has been enhanced with capabilities to assist you in teaching the requirements of higher frequency circuits. The RF Design Module includes: RF component library (microstrips, waveguides, RF transistors, tunnel diodes, spiral inductors, inter-digital capacitors), RF Model Maker, Spectrum Analyzer, Network Analyzer, Network Matching Analysis, Noise Figure Analysis, tutorial and examples. If you teach any RF curriculum, you can’t do without Multisim.
MULTISIM VHDL: FOR PROGRAMMABLE LOGIC AND ADVANCED DIGITAL MODELING

Multisim VHDL

Teaching and learning about the latest in VHDL technology is now cost effective and easily accessible with Multisim VHDL. It is ideal for teaching the design and simulation of programmable logic (FPGAs/CPLDs) and other complex digital devices (processors, memory etc.). It is included in the Student version and is available as a modestly priced Add-on to the Education and Lab versions of Multisim.

Intelligent Source Code Editor

Features such as those below are especially important when teaching VHDL, because unlike SPICE, learning the correct syntax is essential to developing proficient VHDL skills. Reading and writing code with Multisim is easy with the help of a built-in text editor that color-codes VHDL syntax and intelligently indents your code. A built-in Wizard simplifies and accelerates the writing of source code by assisting in the generation of shell code.

Project Manager Organizes Your Designs

All the pieces of a design, 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.

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 standard. 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.

features

- The best way to learn VHDL
- Built-in source file editor with color-coded keywords
- Entity/architecture templates
- Design Wizard to automatically generate shell code
- Powerful source level debugging
- Graphical waveform viewer to display results
- Detailed explanations of warnings and errors
- Complete VHDL IEEE 1076-93 support
- Highly automated file compilation and ordering
- Testbench Wizard to create templates for test code
FEATURES DESIGNED SPECIFICALLY FOR EDUCATION

Whether you teach university, college or high school electronics, in the classroom or delivering training via distance education, Multisim can help to transform your program into an exciting multimedia learning environment. The following education features of Multisim allow teachers to personally customize Multisim for use in the classroom by students, in the laboratory, or at home.

Program Options
Customizing Multisim to deliver your curriculum is critical to its overall success in your program. Powerful program options allow you to customize what a student has access to or sees within the program. Program options such as enable/disable printing, and hide instruments and component properties all enable you to control the students interaction with Multisim.

Circuit Restrictions
Circuit Restrictions permit you to further customize a specific circuit to control what the student sees with that specific circuit. For example, Multisim allows you to hide faults within a circuit, lock up a subcircuit, hide component values, hide parts bins or lock out specific analyses.

Passwords
A password can be assigned to a circuit to restrict access to its components, values, program options, and circuit restrictions that have been set during the original design of the circuit.

Sub-circuits
Sub-circuits or “black boxes” can be created to simplify a circuit or to hide a group of components from view. Sub-circuits can be used very effectively to develop student problem solving skills by assigning “solve the sub-circuit” problems.

Troubleshooting
Multisim is a powerful tool in the teaching of troubleshooting skills. Using Program Options and Restrictions, troubleshooting circuits can be created for a wide range of exercises. Create and then open a circuit, short a circuit or mislabel components. Hide component values, labels or even parts bins.

Remote Control
Multisim contains a unique Remote Control feature that allows a Multisim user to setup a bi-directional link with another user over which each person can control what the other user sees (eg. a student and a tutor). Remote control also permits a real-time, point-to-multipoint internet or intranet broadcast. Create a schematic, make changes to your design, or display simulation results and session members will see on their monitors exactly what you are doing on your PC. The new capability gives the opportunity to provide training, instruction or tutorial support at a distance, no matter whether your student needs help in his home, around the corner or around the world.

CIRCUIT SETS ENHANCE LEADING ELECTRONICS TEXTBOOKS

Many leading electronics textbooks now include CD-ROM’s containing circuits, pre-built in Multisim, to match the sample circuits found inside the book. In fact, many authors construct, design and test the circuits published in their textbooks using Electronics Workbench Multisim. Generally there are more than 100 Multisim-compatible circuits on each CD. This valuable enhancement enables students to immediately simulate circuits, transforming textbooks and print form schematics into live, interactive circuits.

“As a textbook author, I have found that Electronics Workbench is one of the most effective tools for bridging the gap between theory and the real world. I strongly recommend Electronics Workbench.”

Tom Floyd,
Author
Ultiboard is a powerful printed circuit board (PCB) layout and routing program that delivers the features and functionality needed to teach and support your lab and curriculum requirements. It is the perfect compliment to Multisim and is one of the most popular products of its kind in the professional electronics industry, just like Multisim. Ultiboard offers a unique combination of advanced functionality and exceptional ease of use in the tradition that has made Electronics Workbench the leading provider of electronics education design tools. It produces the top-quality board layouts that you would expect from a high-end design tool.

**Engineered for ease of use**

Ultiboard is both easy to learn and easy to use. You can double-click on most items to edit their properties without having to go through tedious menu commands. Making changes in Ultiboard is also simplified since components and traces can be dragged without breaking their copper connections. A unique “birds eye” window displays an aerial view of your board—highlight any area in the window and Ultiboard will zoom to the corresponding portion of the board.

Multisim Schematic Capture and Ultiboard printed circuit board layout are seamlessly integrated. This produces an extremely easy-to-use, integrated laboratory environment that reduces costs in both time and money. One click seamless transfer, as well as forward and back annotation between schematic and board layout (including a netlist compare command), result in shorter student learning curves and more productive learning time.

“Electronics Workbench allows us to take our students through the entire process of design, simulation, troubleshooting and PCB layout. They’re ecstatic when they actually build a functioning board!”

Dr. Maciej Ciesielski,
University of Massachusetts
Support for any board size and shape

With Ultiboard you can define almost any shape board up to 50" x 50" in size and with multiple layers. To define a board shape you can utilize the built-in Board Editor or import outlines from a file.

Placing Components

Strategic component placement is a critical skill to be learned by students. Ultiboard helps you train students in efficient part placement in three ways: first, a unique Force Vector Option analyses ratsnest information and places arrows on each component to suggest recommended distance and direction to move each part; second, a Histogram feature indicates routing channel densities; and third, Ultiboard allows you to precisely place and lock parts to exact coordinate locations.

Editing

Ultiboard automatically selects the proper footprint for all parts from its large library of over 3000 shapes. A shapes editor allows for the editing of existing footprints and creation of new shapes.

Trace Placement and Design Rule Check

Highly flexible trace placement

Ultiboard also simplifies trace placement by automatically highlighting all the pins connected on the same net when you begin laying a trace from a pin. As well, Ultiboard suggests which pin should be connected next.

Design Rules

Ultiboard provides a rich set of design rule features, including a Real Time Check that will identify violations as they are made. Ultiboard also simplifies the selection of design rule properties by offering four of the most commonly used design settings by designers and production houses—easy, standard, dense, and very dense.

Autorouting

Ultiboard includes two autorouters, each offering options for optimizing and customizing performance, including a single pass autorouter for simple boards and a ripup-and-retry algorithm for more complex boards.

Manufacturing Support

Extensive Outputs Options

Outputs can be sent to pen plotters, photoplotters or to any Windows compatible printer. Design data can be output in all popular formats: Gerber, extended Gerber and .DXF.

Board Cleanup

Ultiboard includes several features to clean up and improve board layout before production, if you are using a manufacturing facility to build your PCB including: chamfer corners, delete unused vias and traces and component renumbering.
More educators rely on Electronics Workbench for circuit analysis, design, and simulation software than any other vendor. As the only company to design our products specifically for the education market, our software has become the design tool of choice for over 130,000 customers because it is highly effective in education environments, economical, easily integrated into a curriculum and most importantly, easy-to-use. Students spend time learning electronics, not a confusing user-interface. As well, students tend to learn faster and retain course content longer when using Electronics Workbench software.

Best of all, students will be learning on software that is used pervasively across all sectors of the professional engineering community. We offer two primary products, Multisim and Ultiboard, both available in three versions: Education, Lab and Student Editions. Call us today and join the growing majority of educators who choose Electronics Workbench for their electronics software solution.

**Multisim**

Multisim, the latest release of the Electronics Workbench flagship product, delivers intuitive schematic capture, accurate simulation and analysis, along with programmable logic. It provides industry-leading power with all the features and functionality you’ll need in an easy-to-learn and easy-to-use package.

Educators can prepare and present interactive circuits, changing values on-the-fly and demonstrating the changing behavior of a circuit with virtual instruments. The complex math and theory of electronics is simplified and brought to life through design, simulation and experimentation. Students can create circuits, analyze pre-built circuits, work through virtual laboratory assignments, simulate circuit behavior and troubleshoot for faults.

**features**
- Analog/digital SPICE simulation
- Powerful schematic editor
- 9 virtual instruments
- 5000 parts/model library
- Symbol editor
- 16 analyses
- Post processor
- RF design module
- Circuit restrictions
- Password protection
- Sub-circuits
- Troubleshooting
- Remote control
- VHDL design/debug (optional)

**Ultiboard**

Ultiboard offers all of the advanced features needed to teach students printed circuit board design, but without steep learning curve or high price tag often associated with other high-end design tools. Seamlessly integrated with Multisim, Ultiboard provides you with an across the curriculum design solution for your electronics courses.

Ultiboard allows you to teach students the practical skill of turning a theoretical circuit design into a real-life PCB product, by creating everything needed to physically construct your circuit board.

**features**
- Powerful interactive editing
- Real-time design rule check
- 3000+ library shapes
- Density histograms
- Shapes editor
- Board size (any shape)
- 32 layer support
- 1 nm internal resolution
- Built-in autorouting

---

**Print resources available from Electronics Workbench**

The following print resources are available from Electronics Workbench for educators to assist in the integration of Electronics Workbench products into your curriculum:

- Practical Teaching Ideas for Electronics Workbench V5
- 150 Circuits for use with Electronics Workbench V5
- Troubleshooting With Electronics Workbench
- Electronics For Physics Students
- Understanding Electricity
- The CEA Study Guide
- Electronic Circuit Design Using Electronics Workbench
- Troubleshooting Analog Circuit