The Leading, Global Supplier of Circuit Simulation Software and Related Tools for Professional Design

Electronics Workbench is an international market leader, delivering the world’s most widely used circuit design software. The Company has over 15 years of EDA experience and was one of the pioneers of PC-based electronics design. Today, Electronics Workbench boasts an installed base of more than 180,000 seats.

The Electronics Workbench product portfolio includes software for schematic capture, circuit simulation (SPICE, VHDL and patented co-simulation), PCB layout and autorouting. SUU, our integrated Support and Upgrade Utility, automatically checks for and installs software updates over the web, ensuring that your software is always at its most capable level.

Our users benefit from truly unique product features, industry-leading ease-of-use and complete integration between the tools that comprise our product suites. In addition, our software works seamlessly with National Instruments LabVIEW and SignalExpress, bridging the gap between electronics design and test.

Electronics Workbench is located in Toronto, Canada with a European headquarters in Newbury, UK. The Company also has sales and support operations in Brazil, China, Germany, Japan and the Netherlands. A network of highly competent technical distributors offers sales and support services in 30 additional countries throughout the world.

Unique Functionality, Comprehensive Portfolio

MULTICAP SCHEMATIC CAPTURE
Multicap is an exceptional capture program, suitable for pure schematic entry, driving simulation or feeding PCB layout.

A well-designed user interface reduces the time and steps needed for repetitive schematic entry tasks. Powerful wiring algorithms enable automatic wiring as well as part dragging without losing connections. Parts are easy to find and quick to place, making you more efficient. All devices are stored in an integrated database along with their simulation models and PCB footprints so that your schematics are automatically ready to simulate at the click of a single button, and are easily transferable to PCB layout. Page 6.

MULTISIM SIMULATION & CAPTURE
Multisim is a powerful simulation tool that includes a complete, fully integrated version of Multicap for design entry leading to simulation. Verify circuits and locate errors before they propagate further down the design flow and become costly mistakes. Investigate circuits using the industry’s only virtual instruments and “change-on-the-fly” interactive simulation. Gain true understanding of your circuit’s performance with a comprehensive suite of analyses. Multisim helps optimize designs and minimize time-to-market.

Every PCB design should now include simulation as a key step. Page 9.

ULTIBOARD PCB LAYOUT
Ultiboard combines power with ease of use to quickly get error-free boards to production.

Ultiboard’s user interface is carefully designed so that the most critical repetitive tasks such as trace editing and component placement can be performed quickly, with minimum steps and fewer mouse clicks. In addition, Ultiboard boasts one of the industry’s only real-time systems so that all DRC, ratsnest, polygon and force vector information is updated in real-time as you work, for instant feedback and greater efficiency. Page 14.

ULTIROUTE AUTOROUTING
Ultiroute is the only autorouting program to employ “Progressive Routing” – breakthrough technology that combines grid and gridless (shape-based) routing techniques into a single engine. Get better board layouts with higher completion, shorter trace lengths and fewer vias through gridless routing, while most traces are still placed on the grid for easy editing.

With Ultiroute, you have complete control over all configuration factors affecting routing strategy, and can stop, adjust and restart routing at any time. Page 18.

TIMESAVING FUNCTIONALITY FROM ELECTRONICS WORKBENCH:

Capture
• Modeless placement & wiring
• Fast retrieval Parts Bins on the desktop
• Auto and manual wiring
• Rubberbanding on part moves/rotates
• Industry-leading bus support and the world’s only “Bus Vectors”
• Auto-connect passives
• Full support for design variants and hierarchical designs
• Comprehensive circuit-annotation capabilities

Simulation
• Instant simulation of any schematic
• Interactive “change-on-the-fly” simulation
• Intuitive virtual instruments, including NI VIs and "real" Agilent® and Tektronix® instruments
• Circuit Wizards self-generate circuitry to match user specifications
• Patented co-simulation of SPICE and VHDL
• Integrated with National Instruments LabVIEW and SignalExpress
• Automatic SPICE model makers
• Measurement Probes annotate circuit with dynamic values
• Simulation Advisor and Simulation Profiles
• Interactive, virtual and animated components
• Comprehensive suite of analyses with optional user-defined expressions

Layout and Autorouting
• “Push and shove” parts and traces
• Component nudging with trace rubberbanding
• Real-time Ratsnest and DRC
• “Follow-me” trace placement
• Automatic Connection Machine
• Interactive autorouting – stop, adjust and restart as required
• Progressive routing – industry’s only combined grid and gridless routing engine
The popularity of circuit simulation continues to grow, and at Electronics Workbench, simulation is the core of what we do.

In a recent survey, 80% of designers said that they would like to simulate before proceeding to PCB layout. In today’s demanding market, it’s easy to understand why.

Our truly unique approach to circuit simulation – featuring full interactivity, intuitive virtual instruments, patented co-simulation and a comprehensive suite of analyses – have made us the undisputed market leaders in this increasingly important field.

ADD MULTISIM 9 TO YOUR DESKTOP NOW AND ENJOY THESE BENEFITS:
- **Save time and money.** With Multisim, every circuit you draw is automatically ready for simulation, allowing you to begin testing at the earliest possible stage of the design flow – during schematic capture. Catch and correct errors almost before they happen.
- **Become a simulation expert – fast.** With virtual instruments that look and operate exactly like real-world devices, Multisim is completely intuitive. Just connect to your schematic and start making measurements. It’s that simple!
- **Make design changes during simulation.** Circuit changes in Multisim require only a few mouse-clicks, encouraging experimentation and improvement. Variable devices can even be changed while simulating to instantly see the impact on signals.
- **Draw from a comprehensive component library, or easily create new models.** If you can’t find the device you need in Multisim’s 16,000 part library, you can easily import external models written in SPICE or VHDL. Alternatively, the built-in Model Makers quickly create SPICE models from databook values.

MULTISIM EASILY OVERCOMES THE MOST COMMON SIMULATION OBJECTIONS
Don’t let these outdated opinions about circuit simulation stand in the way of substantial increases in your productivity:

“**Simulation software is too hard to use.**”
Multisim is incredibly easy to use. Every schematic you draw is automatically a single mouse-click away from simulating.

“**Simulation software is too expensive.**”
Multisim is available from under $2,300, providing powerful and complete simulation capabilities at a cost-effective price.

“**Simulation software is not capable of simulating my entire design.**”
Multisim easily simulates digital, analog and mixed-mode circuits and, using patented technology, also co-simulates VHDL devices.

“**Simulation software doesn’t have models for all of my components.**”
All 16,000 devices in Multisim’s library come with matching simulation models. Additional models are easily imported or created from databook values with Multisim’s built-in Model Makers.
The Electronics Workbench product family was designed from the start to leverage the benefits of tight integration. While each of our products also operates as a stand-alone application, it is in combination that they truly shine.

We have embedded simulation within capture and autorouting within layout, to provide you with instant, on-demand access to these important tools. Enjoy the unparalleled power and flexibility of a truly interactive design environment. Every schematic you draw is instantly ready for simulation. Sophisticated Design Rules can be easily created during capture or layout, and will be recognized and respected by all downstream steps. Advanced layout features, such as component nudging with trace rubberbanding, are made possible through automatic, real-time calls to the autorouting engine. And changes made at any stage in the design process can be annotated – either forwards or backwards.

The Component Database System stores all component data in a single repository accessible from all Electronics Workbench products. User, corporate and master library levels operate transparently across products. Any database search can access all possible component parameters, including user fields.

MULTISIM AND NI LABVIEW / SIGNALEXPRESS DELIVER INTEGRATED DESIGN AND TEST

Multisim seamlessly integrates with LabVIEW 8 and the SignalExpress interactive measurement software, increasing your productivity and reducing time to market by bridging the traditional disconnect between popular design and test tools.

- Enhance Multisim’s simulation capabilities by using real-world data, acquired by LabVIEW, as the source that drives your circuit during testing.
- Use LabVIEW to graphically create fully customized virtual instruments for use in the Multisim environment, as a supplement to the software’s 20 supplied instruments.
- Seamlessly import Multisim simulation results into LabVIEW or SignalExpress to easily compare real-world and simulated data. Use this data as a benchmark during the project’s testing and debug phases to improve efficiency and enhance interdepartmental communications.

SELECT A RECOMMENDED SUITE

We have created a number of recommended Suites and offer them at best-in-class pricing. If you don’t need the power of simulation and autorouting, then choose the Electronics Workbench PCB Suite. Combine Multicap, the market’s most efficient capture program, and Ultiboard, our easy-to-use layout software, into a powerful, integrated suite.

If you create more complex designs, then you are ready for our complete Design Suite. Integrating Multicap, Multisim, Ultiboard and Ultiroute, our Design Suite delivers a complete, powerful and tightly integrated end-to-end solution.

OR BUILD YOUR OWN!

If our recommended Suites don’t match your exact requirements, then build your own! With Electronics Workbench, your options are always open.
MultiCAP 9 – Schematic Capture

MultiCap is the industry’s most intuitive, powerful schematic capture program. MultiCap’s innovative and timesaving features, including modeless operation, powerful autowiring and a comprehensive database organized into logical parts bins on your desktop, allow designs to be captured almost as fast as they can be conceptualized.

Rrepetitive tasks are optimized, leaving you freer to create, test and ultimately perfect your designs, resulting in superior products and minimizing time-to-market.

PROJECT MANAGER
Using a familiar tree control structure, MultiCap’s Project Manager provides an easy and intuitive way to navigate and manage files. The Project Manager collates schematics, simulation data, PCB files and documentation together into a single project.

HIERARCHY
MultiCap has full hierarchical and flat multi-sheet design, with an intuitive hierarchy viewer that allows you to efficiently navigate all instances of nested circuits.

Changes to a hierarchical block can be applied to just that instance, or to all instances of that block throughout the schematic, allowing refinements made to a circuit propagate to all other circuits making reference to it. Portions of a circuit can be easily saved as a subcircuit for future reuse, facilitating reusability of common circuits and ensuring consistency among groups of designers.

WORKSPACE AND SPREADSHEET
Every aspect of MultiCap’s user interface can be customized for your own way of working. Create and position your own menus, toolbars and shortcuts, or edit the defaults that already exist.

MultiCap provides a spreadsheet view of all components and nets in your schematics, offering advanced viewing and editing of all schematic parameters and PCB layout design constraints. This highly powerful tool enables you easily sort components and rapidly change parameters for multiple objects in a single step, and also provides access to advanced functionality such as pin or gate swapping.

LIBRARIES
MultiCap’s large, integrated library includes schematic symbols, simulation models and footprints for over 16,000 devices. As a result, every schematic created in MultiCap is instantly ready for simulation and layout, literally with the click of a single button.

In addition to the master library that ships with the product, users can set up corporate or user libraries. In this way it is possible, for example, to identify a set of approved components that are available through the purchasing department, and that meet technical and cost requirements.

A Database Manager aids in managing and updating your libraries and parts, while a Component Wizard guides you through the creation of new devices. Any new components included in a schematic can be easily saved to the library for future use. Components can also be exported/imported from a spreadsheet for rapid editing.

You can add your own properties to parts by employing User Fields, either across the entire component database, or on a part-by-part basis.
MODELESS PLACEMENT AND WIRING
Modeless operation, a capability unique to Multicap, conserves significant time by eliminating the need to repeatedly toggle back and forth between placement and wiring modes – a redundant requirement of other schematic products.

WIRING
Wiring is the most common step in the capture process; it is often repeated thousands of times per design. Multicap’s modeless operation and other truly unique wiring functionality maximizes your efficiency during this time-intensive step.

Multicap’s powerful autowiring feature lets you simply click on a source and destination pin, instructing the program to make a connection for you. The autowiring algorithm strives to make wire paths intuitive and logical, placed as you would expect if you had drawn them yourself. You can even combine manual and automatic wiring to force the placement of some key wire segments, letting the program automatically connect the rest of the wire.

If you prefer, you can simply touch a component’s pin(s) to an existing wire, bus or other pin to automatically create a new connection, and then move the component into place – Multicap maintains the connection(s) and draws the wire(s) for you. You can even drop 2-terminal passives directly onto a wire and both connections will be made automatically, saving a huge amount of time as there are frequently hundreds of such devices added once the basic circuit is created.

Wire “rubber-banding” enables you to move or rotate connected parts without ever losing connections. Similarly, you can drag and reshape wires as you please. Or use virtual wiring to completely eliminate the need to physically draw the wire at all. Just label two points the same and Multicap will automatically connect them “virtually”.

MULTISECTION PARTS
Multicap automatically optimizes the use of any multisection components included in your design, efficiently packing sections into parts for export to PCB layout, even across hierarchies. Of course, full manual control is also available.

SYMBOL EDITOR
A sophisticated Symbol Editor provides the flexibility to build even complex parts quickly and easily. Customizable pins can be placed individually, or by using pin array placement to automatically add large numbers of pins with a few simple steps.

The Symbol Editor offers a spreadsheet view for advanced and fast editing of pins and Multicap’s Symbol Integrity Check quickly verifies new or edited symbols.
MultiCAP 9 – Schematic Capture

BUSES
Multicap 9 includes powerful bus support that will significantly reduce the time you creating, wiring and editing buses.

Configure any size bus (up to 128 bits) easily with auto-incrementing functionality that eliminates the need to define every bit. Assign bus bits to the appropriate range of pins on a symbol with a few easy steps using Multicap's Bus Wizard. Finally, wire the entire bus in one easy step with the industry's only bus "vector-connect" function.

VARIANTS
With Multicap, you can draw a single schematic for multiple product variants. Designs that include variants are compatible with Multisim, Ultiboard and Multicap's spreadsheet view and ERC functionality, allowing these circuits to be readily managed, completely tested and easily manufactured. Hierarchical variants are fully supported and all reports are variant selectable.

DESIGN CONSTRAINTS
Using Multicap, you can set PCB layout and routing constraints directly from your schematic, an especially useful option for higher speed designs. All such constraints specified during capture will be recognized and utilized by Ultiboard and Ultiroute.

REPORTS AND DOCUMENTATION
Multicap generates a wide variety of professional reports, all of which fully support design variants, hierarchical designs and other advanced options.

Annotate your designs by attaching tool-tip style notes to any point on your schematic. User-defined, switchable annotation layers facilitate organization and navigation between notes.

VARIANTS
With Multicap, you can draw a single schematic for multiple product variants. Designs that include variants are compatible with Multisim, Ultiboard and Multicap's spreadsheet view and ERC functionality, allowing these circuits to be readily managed, completely tested and easily manufactured. Hierarchical variants are fully supported and all reports are variant selectable.

ELECTRICAL RULES CHECK
A sophisticated Electrical Rules Check allows you to define what Multicap sees as a schematic error, automatically flagging incorrect connections as you work.

Multicap annotates your schematics with visible ERC error markers and facilitates rapid corrections with its zoom-to-error feature.

TRANSFER TO PCB LAYOUT
Multicap schematics are transferred and loaded into Ultiboard with a single button.

All 16,000 parts in the library have matching footprints so that you will never experience transfer errors. There is full forward and backward annotation between Multicap and Ultiboard so that any design changes are correctly propagated.

CIRCUIT ANNOTATIONS
Annotate your designs by attaching tool-tip style notes to any point on your schematic. User-defined, switchable annotation layers facilitate organization and navigation between notes.
MultiSIM 9 – Simulation & Capture

Multisim, the world's only interactive circuit simulator, allows you to design better products in less time. Multisim includes a completely integrated version of Multicap, making it the ideal tool for creating and then instantly simulating circuits.

Featuring unparalleled ease-of-use and packed with unique and powerful functions, Multisim boasts an impressive history that has made it the most successful simulator in the industry with over 180,000 users worldwide.

With Multisim, there is no reason not to simulate every design.

INTERACTIVE SPICE
You don’t have to be a SPICE expert to use Multisim. All SPICE netlists and commands operate behind the scenes, earning Multisim its well-deserved reputation as the most intuitive, user-friendly simulator available.

Simulation profiles allow you to quickly and easily configure, save and reuse complete SPICE parameter setups. Should the engine ever encounter convergence problems, Multisim’s Simulation Advisor provides you with detailed messaging and diagnostic suggestions.

And Multisim is the only interactive simulator in the world, allowing you to modify your schematic while simulation is running to explore “what if” scenarios in real-time.

MODEL LIBRARY
Multisim includes one of the industry's largest libraries with over 16,000 simulatable components. Even with such a large database, it’s fast and easy to find the part you want. And, unlike other tools, Multisim is fully mixed-mode, so that every part can be used in every circuit.

Multisim is the only product with interactive, animated and virtual parts – specialized devices that let you easily explore design alternatives during simulation:

- **Interactive components** allow you to adjust their values and instantly see the resulting changes while a simulation is running.
- **Animated parts**, such as LEDs and 7-segment displays, change their appearance in response to simulation results.
- **Virtual components** allow you to set their parameters to any value. Use virtual parts early in the design cycle, regularly changing and experimenting with their values, and replace them with real devices once you know what values are required to meet your design specifications.
**MultiSIM 9 – Simulation & Capture**

**ADDING MODELS**
Multisim provides more options for adding new simulation models to its already extensive database than any other product.

Model Makers deliver an easy way to automatically generate SPICE models from databook values and employing highly advanced algorithms for optimal performance matching. Manufacturer-supplied SPICE models (often available for download from their websites) are easily imported into Multisim’s database.

Digital parts can be created with an easy to learn descriptive language. Multisim can even simulate compiled C-code so that you can model devices using C-code scripts. And finally, using patented co-simulation, Multisim fully supports IEEE compliant VHDL models.

**INTERNET DESIGN SHARING**
Multisim’s Internet Design Sharing allows colleagues in different locations to work on the same circuit together in real-time. Users can be around the corner, across town or on opposite sides of the globe.

**SOURCES AND DATA ACQUISITION**
Multisim’s virtual function generator operates just like its real world equivalent and is an easy way to generate source signals. You can also change this instrument’s waveform shape, frequency, and amplitude while a simulation is running and instantly see the resulting changes.

Multisim’s library includes an extensive collection of sources. In addition, the piecewise linear source lets you define the shape of a signal, or use real data (acquired with NI LabVIEW or SignalExpress) as the source for your simulations, and allows you to ‘loop’ your data.

**CIRCUIT WIZARDS**
Multisim contains unique, advanced wizards for automating the creation of certain specialized circuits. Simply specify the desired characteristics and Multisim will generate the recommended circuitry to deliver the required functionality.

- 555 Timer Wizard
- Filter Wizard
- Common-Emitter Amplifier Wizard
- Op Amp Wizard
- MOSFET Amplifier Wizard
VIRTUAL INSTRUMENTS

Multisim has established a reputation for powerful yet easy-to-use simulation. The use of virtual instruments, with faceplates that look and operate just like their real-world counterparts, is one of the reasons why Multisim has risen to this leadership position.

Virtual instruments let you take advantage of the full power of simulation without having to be an expert in SPICE syntax. These instruments are wired into the schematic just like you would connect a real instrument on the bench. Like their real world counterparts, they are fully interactive so that you can change their settings while a simulation is running and instantly see new results. It’s completely intuitive.

Virtual Instruments created in NI LabVIEW can now be used within the Multisim environment as an alternative to the pre-built instruments provided with the software. Electronics Workbench has also partnered with leading instrument suppliers such as Agilent® and Tektronix® to provide simulated “real” instruments that look and operate exactly like these vendors’ real-world models.

- 16-Channel Logic Analyzer
- Agilent 34401A DMM
- Agilent 54622D Scope
- Agilent 33120A Waveform Generator
- Ammeter
- Bode Plotter
- Distortion Analyzer
- Dynamic Measurement Probes
- Frequency Counter
- Function Generator
- Multimeter
- Network Analyzer
- Oscilloscope (2 and 4-channel)
- Spectrum Analyzer
- Tektronix TDS 2024 4-channel Oscilloscope
- Voltmeter
- Wattmeter
- Word Generator
MultiSIM 9 – Simulation & Capture

ANALYSES
While virtual instruments give you the flexibility of a complete lab, Multisim’s comprehensive suite of 24 analyses (the most extensive selection available anywhere) lets you investigate circuits in ways just not possible in the real world.

Expand the power of the pre-built analyses by entering your own expressions to manipulate the collected data and customize the results.

Multisim’s analyses provide you with an unrivaled ability to fully understand your circuit’s behavior and performance.

- 3dB Point
- AC and DC Sensitivity
- AC Frequency
- Batched Analyses
- DC Operating Point
- DC Sweep
- Distortion
- Fourier
- I-V Analyzer
- Model Parameter Sweep
- Monte Carlo
- Nested Sweep
- Noise
- Pole-Zero
- Temperature Sweep
- Trace Width
- Transfer Function
- Transient
- User-Defined Analysis
- Worst Case

GRAPHER
The grapher serves as a multi-purpose tool to view, adjust, save, print and export the results of a simulation analysis. You can even overlay traces from different simulation runs to compare results. All grapher results can be loaded into Microsoft Excel® or Mathsoft Mathcad® with a single button.

POST PROCESSOR
The Post Processor allows you to perform almost any mathematical or engineering calculation on simulation results and waveforms, and includes a comprehensive selection of:

- Arithmetic operators
- Trigonometric operators
- Calculus & Algebraic operators
- Boolean logic operators
- Vector operators
- Complex math operators

The possible uses for this tool are endless.

RF DESIGN
Multisim’s RF Design Module provides simulator enhancements and special features for high frequency design beyond 100MHz.

Most SPICE models are unreliable for RF designs because at higher frequencies, simulation behavior departs from databook values. Multisim RF contains a library of SPICE models designed to be accurate at high frequencies, devices unique to higher frequency circuits and a collection of timesaving RF Model Makers.

Multisim RF includes new virtual instruments including a Spectrum Analyzer and Network Analyzer.
MultiVHDL – VHDL Co-simulation

Multisim’s VHDL add-on has the power and features demanded by experienced users, but with exceptional ease-of-use.

MultiVHDL comes complete with flexible design entry (including a graphical state machine diagram editor), highly automated project management, powerful simulation, advanced waveform viewing, a Graphical Testbench Designer and comprehensive debug features. MultiVHDL can be used as a stand-alone application, or together with Multisim’s SPICE simulator using patented co-simulation.

Multisim with MultiVHDL

PATENTED CO-SIMULATION OF SPICE AND VHDL
Because different components are sometimes best modeled in different simulation languages, Multisim has developed a co-simulation engine that can analyze designs combining a mix of SPICE and VHDL modeled components. This technological breakthrough finally enables designers to simulate complete boards containing discrete, analog, glue logic, programmable logic and complex digital devices for the first time.

Multisim’s patented co-simulation technology seamlessly integrates these distinct simulation engines that interchange data in real time, displaying one consolidated result.

Co-simulation is an ideal testbench when writing VHDL code since you can test devices along with the very circuits in which they are designed to operate.
Ultiboard has been carefully designed to maximize your productivity. By optimizing the most common, repetitive tasks such as part and trace placement, we have dramatically reduced the number of keystrokes and mouse movements required to layout your design.

Ultiboard handles today’s higher speed designs with ease using constraint driven layout. And innovative features such as Real-Time design rule checking, “Push & Shove” components and traces, component nudging with trace rubberbanding, “Follow-me” trace editing and an Automatic Connection machine ensure that you rapidly complete an error-free board.

SEAMLESS SCHEMATIC INTEGRATION
Importing a schematic design into Ultiboard is fast and easy: a single click of a button in Multicap transfers the netlist into Ultiboard, populates the parts bin and adds the components to the layout workspace.

Ultiboard supports forward and backward annotation with Multicap schematics allowing you to make changes to all critical schematic information during layout. Use cross probing to automatically select a footprint in Ultiboard by highlighting the corresponding symbol in Multicap. And of course, Ultiboard is fully compatible with Multicap’s design variant functionality.

USER-FRIENDLY INTERFACE
Ultiboard’s user-friendly interface simplifies viewing and navigation. You can customize all toolbars, menus and shortcut keys, resulting in a comfortable and productive workspace.

Selection filters let you isolate components, traces, copper areas or any other object type, making it easier to select the element you want. Quickly annotate your design with “post-it style” notes and easily locate traces using the powerful Highlight Selected Net function.

The spreadsheet view lets you easily see and work with any type of objects – use this powerful feature to quickly edit information common to multiple elements simultaneously.

FAST BOARD OUTLINE CREATION
Ultiboard lets you create any shape board, including cutouts, up to 2 x 2 meters in size.

Import board outlines as DXF files from mechanical CAD programs, define your outline using the Advanced Board Outline Wizard, or make use of Ultiboard’s library of pre-defined, common board shapes.

DESIGN SET-UP
Ultiboard’s one-nanometer architecture, the industry’s leading resolution, ensures support for truly gridless placement of even the smallest objects.

Assign keep-in/keep-out areas to components only, traces only or both, including height restrictions. Intuitive layer stackup lets you easily define boards up to 64 layers.

Ultiboard allows you to create rule sets, saved in Technology Files, which are completely reusable and are “attached” to specific projects. These Technology Files are used to specify parameters required by your design or by your PCB manufacturer.
FAST, INTUITIVE PART PLACEMENT

Ultiboard features two intuitive drag-and-drop placement techniques. You can drag parts from just outside the board outline (where they appear after schematic import) or place them from a parts bin that lists all components. Real-time “Force Vectors” guide placement for routability, analyzing the ratsnest and advising where to move the component.

Alignment and Equi-spacing functions aid in the placement of regular arrays of components and ruler guides assist in accurate and precise placement. Improve your layouts with flexible, automated pin and gate swapping. The “Replicate Place” function allows you to rapidly duplicate the placement of a group of components.

Save time using the “Component Sequencer” – the next component will automatically appear on your cursor as soon as the previous component is placed.

The “Push and Shove” feature automatically moves interfering parts aside to let you place components in congested areas – it even features “Springback” from the board outline.

FOOTPRINT LIBRARY & EDITING

Ultiboard contains one of the industry’s largest footprint libraries and shares the same Master, Corporate and User database structure as the rest of the Electronics Workbench Product Suite. Quickly customize libraries, or add parts and User Fields with Ultiboard’s sophisticated library manager.

Use the Footprint Wizard to quickly create footprints for even the most complicated components, or import shapes as DXF files. Ultiboard supports polygon pads, blind and buried vias, microvias, advanced padstacks and vias inside SMD pads.

The powerful Fan-out Wizard quickly generates fans-outs for BGAs and similar large components, reducing frustration.

CONSTRAINT DRIVEN HIGH-SPEED DESIGN

Ultiboard provides a number of essential functions necessary to deal with the challenges of today’s high-speed designs.

Ultiboard automatically respects any design constraints set in Multicap, including trace length, topology, separation and parallelism. You can also constrain trace placement for high-speed boards from inside Ultiboard. And by using Ultiboard in conjunction with Ultiroute for your higher speed designs and you’ll be able to automatically route differential pairs and add net shielding to specified traces.
POWERSFUL TRACE PLACEMENT
Ultiboard includes three powerful trace placement techniques, providing the flexibility to meet a range of design complexities and user preferences. All are fully gridless and interactive.

Ultiboard’s unique “follow-me” trace placement combines the power and automation of an autorouter with the flexibility and control of manual placement—it is completely unique in the industry. Simply guide the path of the trace with your cursor, and the “follow-me” feature automatically routes from the last placed segment to your mouse.

Ultiboard’s “Connection Machine” effectively autoroutes an entire trace with a single click on a ratsnest line. Fast, gridless and precise, it routes the trace between the two nearest pads on the net.

Ultiboard also supports traditional manual trace placement, where you change the path of the trace with each mouse click.

UNIQUE TRACE FUNCTIONALITY
Ultiboard offers truly unique and robust features for working with traces.

Real-time calls to the Ultiroute autorouting engine enable component nudging without losing connections—placed traces automatically “rubberband” as you move the part!

“Push and Shove” trace placement moves already placed traces aside when routing through congested areas. Of course, you can also lock a trace segment (or whole trace) in place, to avoid having it move.

Ultiboard’s real-time ratsnesting updates the ratsnest lines continually as you move parts and traces. “From copper ratsnesting” displays ratsnest lines from the nearest copper point, not just from pins.

All trace properties can be changed while editing. And when you get close to a destination pad, a “magnetic” attraction effect pulls the trace in to the center of the pad, ensuring a completed connection.

The “Copy Route” feature duplicates a previously placed set of traces to quickly create multiple channels or reuse other parts of your design.

The Bus Placement feature makes bus placement a breeze by simultaneously routing traces for all bits at once (in parallel) as you place the trace for a single bit.

COPPER AREAS & POWER PLANES
Ultiboard’s highly advanced copper pour lets you create areas with any combination of shapes including polygons, arcs and cutouts.

Real-time polygon updating lets you plow traces through a copper area, creating the necessary void automatically as you place the trace. Hatching and thermal relief are fully supported.
REAL-TIME DESIGN RULE CHECKING
Real-time design rule checking (unique to Ultiboard) shortens the design cycle by flagging rule violations as they occur, letting you prevent mistakes rather than finding and fixing them later.

With the violations automatically highlighted during part placement, layout and routing, you can quickly nudge a component or trace to meet design rules as you work. Or you can choose to ignore violations and analyze them at the end of the session using the error log. Clicking an entry in the error log “jumps” to the violation clearly marked on the board, where you can either correct it or override the rule.

BUILT-IN AUTOROUTING
Ultiboard includes a built-in standard autorouter that is sufficient for basic boards. For most of today’s PCBs, and to enable some of the more advanced, timesaving features of Ultiboard, you’ll want to add Ultiroute to your desktop. Page 18.

MANUFACTURING & OUTPUT FILES
Ultiboard produces fully customizable output in all popular formats, including:

- Gerber 274-X and 274-D
- IPC-D-356A
- DXF and SVG
- BOM, Centroid and Drill files

Before going to production, clean up your board and reduce manufacturing costs using Ultiboard’s built-in optimization routines. You can also renumber components using your preferred strategy to easily locate parts on finished boards.

3D VISUALIZATION
Ultiboard’s 3D Viewer brings your design to life, providing a realistic 3D rendering of a complete, populated board, including components, traces, pads and silkscreen. You can even view the traces, pads and vias on the internal layers of your board!

Preview designs before production, and share them with clients or other design team members. Print the 3D rendering or export it in a variety of standard file formats.

MECHANICAL CAD
Ultiboard also allows you to export your complete PCB design, including 3D information, in DXF or IGES format for use in any popular mechanical CAD program.

For simpler projects, Ultiboard comes with a built-in Mechanical CAD module, allowing you to quickly draw accurate front panels, enclosures and other mechanical pieces.
UltiROUTE 9 — Autorouting

For all but the most basic PCBs, the density and complexity of today’s designs make manual component and trace placement techniques impractical. For these designs, Electronics Workbench offers Ultiroute — a state-of-the-art autorouting and autoplacement tool.

Ultiroute easily handles complex designs including large multi-layer boards and high-pin-count components, ensuring high circuit performance and lower production costs for these demanding projects by uniquely combining the best of gridless (shape based) and grid-based routing.

SEAMLESS INTEGRATION WITH ULTIBOARD

Placement and routing are no longer treated in a linear fashion during the design process. To maximize your productivity, you need instant, interactive access to autorouting at any time. That is precisely why we have embedded Ultiroute within Ultiboard, where it appears as a new menu item after installation.

This architecture provides you with ready access to all of Ultiroute’s powerful functionality at any time during the placement and layout process — much as simulation is instantly available from Multisim during schematic capture. And Ultiroute uses all of the design rules set for the project within Multicap and Ultiboard.

Ultiroute’s tight integration with Ultiboard also enables unique and powerful new functionality within Ultiboard. Exciting features such as part nudging with trace rubberbanding rely on automatic, real-time calls to autorouting engine — transparently enhancing your productivity.

ADVANCED AUTOPLACEMENT

Ultiroute’s autoplacement algorithms provide high-density component placement, fitting more parts onto limited board space, while optimizing for routability.

You have complete control over autoplacement with the ability to pre-place parts, constrain certain components to pre-defined areas, set separation values and enable surface-mount device mirroring and pin and gate swapping. Ultiroute also uses height restrictions to create intelligent keep-out areas based on component z-axis values.

DESIGN ANALYSIS OPTIMIZES ROUTING RESULTS

An autorouter’s settings will affect results differently for each unique design. Ultiroute uses Design Analysis to examine each design file before routing and adjusts the router settings based on the specific characteristics of the design, allowing it to easily surpass the performance of programs that use the same “fixed” default settings for every board.
INTERACTIVE ROUTING
Ultiroute provides you with the ability to interrupt the routing process at any time, allowing you to adjust the routing parameters or edit traces manually if necessary. Then have Ultiroute continue routing with the new conditions. This powerful interactivity allows for timely user intervention and results in improved control and better layouts.

BEST OF GRID-BASED & GRIDLESS AUTOROUTING
Ultiroute’s unique “Progressive Routing” algorithm routes on grid whenever possible, automatically switching to gridless routing as necessary to complete the trace. As a result, you get all the benefits of gridless autorouting for difficult to route traces, with the convenience of having most traces located on grid for easy and efficient manual editing when required.

Ultiroute’s capability to utilize both grid and gridless routing is truly unique and is one of the key reasons for its consistently impressive results.

FULLY CUSTOMIZABLE OPTIONS
Ultiroute’s flexibility gives you complete control over the autorouting process. Choose to route an entire board or just a selected net or subset of components.

You can define anything from grid size and maximum via count to preferred routing directions for each layer of the PCB. You also have complete control over the cost factors used by the routing algorithm.

ADVANCED FEATURES FOR IMPROVED RESULTS
Ultiroute boasts a broad range of powerful features that combine to deliver superior routing results.

Choose to route a net with a specified topology (normal, daisy-chain or star), and set limits for minimum and maximum connection lengths. Set routing layers for individual nets.

Ultiroute automatically routes differential pairs, creates fan-outs for complicated components and can insert testpoints automatically for you. Net shielding is automatically added to specified traces. And Ultiroute easily handles microvias, vias inside SMD pads, custom pad shapes and multiple powerplanes.

Only Ultiroute provides Automatic Bus Routing – specify the pin for each bit and Ultiroute will route the entire bus exactly as you would expect, with each bit’s trace parallel to the others.

MANUFACTURING OPTIMIZATION
After achieving 100% completion, Ultiroute’s fully customizable cleanup routine lets you refine your board to optimize the layout, reduce production costs and improve manufacturing yield.
A Complete Integrated Design Flow

**Capture**
- MULTICAP 9 FEATURES
  - Customizable GUI
  - Modeless part placement & wiring
  - Fast retrieval parts bins
  - Auto and manual wiring
  - Fast auto-connect passives
  - Rubber-banding on part moves
  - Bus vector connect
  - Project Manager
  - Hierarchical & Multi-sheet design
  - Circuit Annotations
  - Electrical rules check
  - Advanced Symbol Editor
  - Title Block Editor
  - Forward/Backward annotation Component Database (# of parts)
  - 11,000
  - 16,000
  - Spreadsheet view
  - n/a
  - Variant support
  - n/a
  - Constraint-driven layout from capture
  - n/a
  - Pin and gate swapping
  - n/a
  - Advanced parts search
  - n/a
  - User-defined fields
  - n/a
  - Customizable BOM
  - n/a
  - Schematic statistics report
  - n/a
  - Spare Gates report
  - n/a
  - Cross-probing with Uliboard
  - n/a
  - ERC scope setting
  - n/a
  - Import/export database
  - n/a

**Simulate**
- MULTISIM 9 FEATURES
  - World's only interactive simulator
  - Fully mixed-mode A/D simulation
  - Standard SPICE 3X5/XSPICE
  - Enhanced model support
  - Speed/Accuracy tradeoffs
  - Simulation Advisor
  - Simulation Profiles
  - Virtual, interactive, animated parts
  - PSpice import
  - Dynamic Measurement Probes
  - Component Editor Wizard
  - Internet Design Sharing
  - Co-simulation of SPICE/HDLs
  - Post Processor
  - NI LabVIEW & SignalExpress integration
  - NI LabVIEW VIs as Instruments
  - Virtual Instruments
  - 14
  - 20
  - Analyses
  - 17
  - 24
  - Simulated Agilent instrument
  - 1
  - 3
  - Simulated Tektronix instrument
  - 0
  - 1
  - Model Library
  - 11,000
  - 16,000
  - Circuit Wizards
  - n/a
  - Model Makers
  - n/a
  - Multisim RF module
  - n/a
  - Nested Sweeps
  - n/a
  - C-Code Modeling
  - n/a
  - MultivHDL
  - n/a

**Layout**
- ULTIBOARD 9 FEATURES
  - Gridless “Follow-me” placement
  - Push & Shove part & trace placement
  - Real-time & from copper ratsnest
  - Real-time polygon update with voiding
  - Keep-in/Keep-out areas
  - Forward/Backward annotation
  - Real-time ERC with “Jump-to”
  - Comprehensive Footprint Wizard
  - Enhanced 3D visualization with print
  - Gerber, DXF, IPC-356A output
  - Number of pins supported
  - 1,400
  - 1,400
  - Spreadsheet view
  - n/a
  - Cross-probing with Multicap/Multisim
  - n/a
  - Variant Support
  - n/a
  - Component Placement
  - n/a
  - Sequencer
  - n/a
  - Pin & gate swapping
  - n/a
  - Gridless Connection Machine
  - n/a
  - Bus routing
  - n/a
  - High-speed constraint driven layout
  - n/a
  - DfI Impedance/Trans Line Calculators
  - n/a
  - Net bridges
  - n/a
  - Microvias
  - n/a
  - Copy Route & Replica Place functions
  - n/a
  - In-place footprint editor
  - n/a
  - Mechanical CAD
  - n/a
  - Export 3D info in 3D IGES, DXF formats
  - n/a

**Autorouting**
- ULTIROUTE 9 FEATURES
  - Complete integration with Uliboard
  - Autoplacement
  - Pin & gate swapping
  - Topology: shortest, daisy chain, star
  - Fully customizable cost factors
  - Progressive Routing
  - Interactive autorouting
  - Constraint driven routing
  - Keep-in/Keep-out criteria: components, traces, groups, height
  - Manual pre-placement components, vias, traces
  - Cluster placement support
  - SMD mirroring
  - Net shielding
  - Automatic testpoint insertion
  - Part nudging with trace rubberbanding in Uliboard
  - 1,400
  - unlimited
  - Maximum number of layers
  - 4
  - 64
  - Advanced BGA fan-out
  - n/a
  - Prioritize routing order
  - n/a
  - Route an individual net
  - n/a
  - Automatic bus routing
  - n/a
  - Differential Pair routing
  - n/a
  - Route Group function
  - n/a
  - Optimization
  - n/a

---

For pricing information please contact Electronics Workbench or speak with your local distributor.

electronicsworkbench.com  •  800.263.5552  •  info@electronicsworkbench.com

**Worldwide Offices:**
- Austria: 49 89 741 31 38 31
- Belgium: 31 0 35 694 444
- Brazil: 11 3262 3599
- Canada: 416 977 5550
- China: 86 21 6555 7838
- France: 33 1 56 98 16 76
- Germany: 49 89 741 31 38 31
- Ireland: 44 1635 572 403
- Japan: 81 3 5472 2338
- Netherlands: 31 0 35 694 444
- Switzerland: 49 89 741 31 38 31
- U.K.: 44 1635 572 403

© 2005 Electronics Workbench Corporation. All rights reserved. Other product and company names listed are trademarks or trade names of their respective companies.

BRProV91105