OrCAD PCB DESIGNER
OrCAD PCB DESIGNER with PSpice

FULL-FEATURED SUITES INCLUDE PRODUCTION-PROVEN AND NEW ORCAD TECHNOLOGY

OrCAD® PCB Designer suites are complete, scalable, cost-effective solutions for the design of complex PCBs. They offer increased productivity and design performance along with improved overall design quality. The powerful, tightly integrated design solutions feature both new technologies and best-in-class capabilities from OrCAD. Included are OrCAD Capture®, OrCAD PCB Editor, and SPECCTRA® for OrCAD.

A COMPLETE PCB DESIGN SOLUTION
OrCAD PCB Designer contains everything you need to take a PCB design from concept to production. It includes design capture, library tools, a PCB editor, an auto/interactive router, as well as interfaces for manufacturing, mechanical CAD, and translators for other PCB systems.

FUTURE-PROOF SCALABILITY
Unlike other PCB solutions, OrCAD PCB Designer has the ability to grow with your needs and technology challenges. It provides a feature-rich, fully scaleable solution. As your level of design challenges and sophistication grows, OrCAD PCB Designer does also, providing an easy upgrade path from the OrCAD product line to the Cadence® Allegro® 200 Series and, then, the Allegro 600 Series, without the need to translate design databases or change use models.

SUITE OVERVIEW
OrCAD Capture
OrCAD Capture is the world’s leading schematic entry solution. Intuitive, easy to use, with sophisticated part search mechanisms, this is the tool of choice for rapid design capture. Whether you’re designing a block diagram or a complex PCB, FPGA, or CPLD, OrCAD Capture gives you everything you need to get your design capture work done quickly and accurately. If parts get modified in either the schematic or the database, you can update them with a touch of a button and generate a complete, accurate parts list or netlist at any time. Tight bidirectional integration with OrCAD PCB Editor provides cross-highlighting and cross-probing as well as forward and backward annotation that ensures agreement between the schematic and the board layout in the event of any gate-swaps, pin-swaps, or changes to component names or values.

PSpice A/D (included in OrCAD PCB Designer with PSpice)
PSpice® A/D simulator has the world’s largest installed base of any PCB simulation product and is the most widely used tool for simulating analog and mixed-signal designs of any size. This sophisticated, native mixed-signal simulator performs functional simulations of digital parts and allows engineers to perform a broad range of PSpice analyses. It contains analog and digital parts ranging from IGBTs and pulse width modulators to DACs and ADCs. Tight integration with OrCAD Capture facilitates rapid design-and-simulate repetitive cycles allowing engineers to explore various design configurations before committing to specific implementation.
The designer can simply align any/all of the defined/designed re-use module or subcircuit with graphical component preview or from a pre-allocation, schematic sheet/block, from a browser Quickplace allows the designer to place by room layer counts are to be minimized and optimized.

The contraint-driven methodology of OrCAD PCB Editor drives a powerful and flexible set of placement capabilities, including interactive and automatic component placement. With floorplanning, you can divide boards into separate sections called “rooms.” Components or subcircuits can be assigned to a specific room by the designer during design entry or layout. Floorplanning is a critical task if routing, signal integrity, timing, and layer counts are to be minimized and optimized. Quickplace allows the designer to place by room allocation, schematic sheet/block, from a browser with graphical component preview or from a predefined/design re-use module or subcircuit. The designer can simply align any/all of the selected items along any or multiple edge(s) of the design as a palette for final individual, group, subcircuit, or room placement.

SPECTRA for OrCAD complements the layout facilities of OrCAD PCB Editor. It provides two powerful tools for interconnect routing: a route editor and an autorouter. Constraints and design rules from OrCAD PCB Editor — even those defined during design entry — are passed through to SPECTRA for OrCAD. The route editor and autorouter can route up to six signal layers concurrently, with no restriction on the number of components, component pins, or nets.

The route editor provides wire and via editing with its trace plowing, shoving, and ghosting features. As you route new traces, the plowing feature automatically pushes existing traces aside and routes around pins. Using the shoving feature, you can move trace segments or vias against existing traces and continue routing — over and through other pins and vias. To optimize manufacturability, the critic feature removes extra bends created during routing, either in a specific area or across the entire board, at your discretion.

The autorouter handles today’s complex routes and staggered-pin components with ease. Its diagonal routing algorithms, operating in either gridded or gridless mode, handle components of nonstandard dimensions that previously required manual routing, reducing productivity.

With direct embedded access through OrCAD PCB Editor or with designer customized autorouter commands, SPECTRA for OrCAD runs and executes the routing tasks while adhering to the design constraints. Routing results will be automatically read back into OrCAD PCB Editor. SPECTRA for OrCAD speeds routing task to completion, increasing productivity and shortening overall design cycle times.

Interactive etch editing
Provides real-time, shape-based, any angle, push/shove routing, which enables you to choose between “shove-preferred” and “hug-preferred” modes. The real-time, embedded, shape-based routing engine optimizes the route by either pushing obstacles or contour-following obstacles while dynamically jumping vias or component pins. Custom, controllable, on-the-fly smoothing automatically tunes the route for manufacturability during routing or route editing.

OrCAD PCB Editor makes it easy to generate any type of manufacturing output you need.

**Comprehensive powerplane creation and editing capabilities**

Today’s designs often have multiple power and ground sources as well as many areas of sensitive circuitry that necessitate localized maximum copper fill areas for shielding. Also, production designs demand complex powerplanes to keep layer counts to a minimum and reduce costs. The creation time and effort can be an intense, challenging and time consuming task. To meet this challenge, OrCAD PCB Editor provides the industry’s most powerful and comprehensive powerplane creation and editing capabilities. These include user-defined split planes, negative or positive views of internal layer planes, and extensive options for user-defined, copper pour partial-planes. Dynamic shapes offer real-time copper pour plowing/healing functionality. Shape parameters are hierarchical and can be applied at three different levels. Parameters are structured into global, shape instance, and object level hierarchy. Traces, vias, and components added to a dynamic shape will automatically plow and void through the shape. When items are removed, the shape will automatically fill back in. Dynamic shapes do not require batch autovoiding or other post-processing steps after edits are made.

**Manufacturing and fabrication outputs**

All this functionality and sophistication would be wasted if the path to PCB manufacturing and fabrication is too narrow. OrCAD PCB Editor tackles this side of the equation by giving you a wide range of manufacturing and fabrication outputs. It can generate a full suite of phototooling, bareboard fabrication, and test outputs including Gerber 274x, NC drill, and OrCAD in a variety of formats. OrCAD PCB Editor also addresses special requirements with a custom data extraction tool that supports total integration with in-house or vendor-specific manufacturing environments. Additionally, OrCAD PCB Editor supports embedded ODB++ output, a capability jointly developed with Valor Computerized Systems. The ODB++ data format enables you to create accurate and reliable manufacturing data for high-quality, Gerber-less manufacturing.
KEY FEATURES

OrCAD Capture unlimited undo/redo
- Schematic part and PCB footprint alternative picklists
- Management utility for schematic property validation
- Reports with sorting, subtotaling, and selection criteria formulas
- Graphical, flat, and hierarchical design schematic Page Editor
- Design archiving and project management process flow system
- Online design rule check
- Unlimited user-defined properties
- EDIF graphics and EDIF netlist interface
- Direct HDL output for Verilog® and VHDL simulation
- Direct integration with PSpice

PSpice A/D
- Circuit entry/integration with OrCAD Capture
- Large simulation model library
- Interactive, graphical PSpice Stimulus Editor
- Digital stimuli for signals, clocks, and buses
- View simulation bias results directly on the schematic
- Simulation view and setup control
- Analog analysis
- Mixed analog/digital simulation
- Generation of symbols from models

OrCAD PCB Editor
- Unlimited layer support
- QuickPlace for floorplanning
- Automatic and interactive package, gate, and pin swapping
- Automatic and interactive design rule checking
- Component rotations in 0.001 degree increments
- IntelliUSE shape-based, shove, or hug routing
- Dynamic shapes
- Variant design support
- Direct ODB++ output
- Gerber 274 & 274-X
- Automatic dimensioning
- Interfaces for DXF and IDF
- PADS, P-CAD® import
- Support for split planes
- Multiple undo/redo
- Slot support

SPECCTRA for OrCAD
- Shape-based or grid-based autorouting
- Manual or automatic attachment of nets to plane
- Graphical routability predictor
- 45-degree routing option
- Automatic legal pin-swap for improved routability
- “DO” file command/control language
- Interactive push/shove route editing
- Legal via hole visualization
- Interactive bus/bundle routing
- Follow-cursor routing
- Support for single-sided designs with jumper links
- Layer set routing

SYSTEM REQUIREMENTS
- Pentium 4 (32-bit) equivalent or faster
- Minimum 256MB RAM (512MB recommended)
- 300MB swap space (or more)
- CD-ROM drive
- 32,768 color Windows display with minimum 1024 x 768 (1280 x 1024 recommended)

SALES, TECHNICAL SUPPORT, AND TRAINING
The OrCAD product line is owned by Cadence Design Systems, Inc. and supported by a worldwide network of value-added resellers (VARs) or distributors. For sales, technical support, and training, contact your local VAR. For a complete list of authorized VARs, visit www.orcad.com.

PRICING INFORMATION
For product pricing and availability, contact the OrCAD VAR nearest you. To locate a VAR in your area, visit www.orcad.com.