

OrCAD Layout

ADVANCED, RULES-DRIVEN PCB DESIGN FOR SIMPLE TO COMPLEX PRINTED CIRCUIT BOARDS

KEY FEATURES FOR ORCAD LAYOUT:

- Grid-based 16-layer autorouter
- Grid-based auto-interactive routing
- Grid-based push-and-shove
- Grid-based single layer autorouting
- Automatic test point generation
- Automatic ECO
- Intelligent copper pour
- Design reuse
- Routing density display
- Pro/ENGINEER, SDRC, Solid Edge (IDF 2.0) interface
- Gerber RS-274D, extended Gerber (274X) output
- GenCAD, GenCAM interface
- IPC-D-356 interface
- ODB++ interface
- 68 x 68 inch maximum board size
- 30 total layers
- 16 simultaneous routing layers
- 8,000 components per board
- 10,000 nets per board
- 32,000 connections per board
- 16,000 connections per net
- 8,000 different component symbols per board
- 3,200 pins per component
- 1,000 different padstacks
- 250 different via types
- 250 characters per reference designation
- 250 characters per net name
- 1/60 mil or 1 micrometer base resolution
- 1/60 (1 minute) degree component rotation

OrCAD® Layout® offers PCB designers and design teams the power and flexibility to create and share data and constraints across the design flow. OrCAD Layout delivers all the capabilities designers need from netlist to place-and-route to final output. The ease-of-use and intuitive capabilities of OrCAD Layout provide for quick startup and rapid learning right out of the box.

OrCAD Layout incorporates a spreadsheet-based constraint management system that gives designers easy access to board rules. Powerful floorplanning and advanced placement functionality provides greater control over component placement. Auto-interactive routing is ideal for hand-routed designs. The design rule checking (DRC) route engine easily pushes and shoves vias and traces, avoiding routing obstacles. Intelligent copper pour supports mixed-voltage power planes. Both batch DRC and online DRC are available. Online DRC checking can be toggled on or off and works in conjunction with the auto-interactive router. Additionally, you can choose from standard reports or create customized reports to suite your individual needs. Intuitive post-processing tools with an output preview provide quick and error-free output. OrCAD Layout is ideal for individuals and small to medium design teams creating printed circuit boards from prototype to production.

INTEGRATION WITH ORCAD CAPTURE AND CAPTURE CIS SCHEMATIC TOOLS

Component properties, pin properties, and net properties can be assigned in OrCAD Capture® or OrCAD Capture CIS and then passed to OrCAD Layout. Inter-tool communication supports cross-probing of components and nets between OrCAD Capture and OrCAD Layout. You can specify spacing rules, flag critical nets, and specify other key design constraints on the schematic and pass that information to OrCAD Layout. AutoECO forward annotates all changes made in the schematic to the board while back annotation performs a full database comparison and creates a back annotation file containing changes made in the board database to the schematic. Bi-directional property passing allows board-level design rules to be passed back and forth.

INTER-TOOL COMMUNICATION STREAMLINES THE ENTIRE BOARD DESIGN PROCESS

Many problems in PCB design can be traced to incorrect component information or poor communication between engineers and board designers. Specifying an obsolete component or one that has an unexpectedly long lead time can result in expensive rework. Even a simple communication error can trigger an extra board turn and delay a project significantly. This can happen, for example, when a designer changes an IC package or the wattage of a resistor without changing the corresponding PCB footprint and company part number. Such errors are easy to miss, especially when design teams communicate via email messages and handwritten notes. If discovered early in the design process, these errors can be easily avoided. If discovered after the board has been fabricated, assembled, or shipped in a product these problems are far more expensive to correct. Through the CIS Option for OrCAD Capture and its tight integration with OrCAD Layout, you can eliminate these types of errors. You start the board design process with up-to-date and reliable component information, enabling you more correct first-pass designs that will streamline your process to meet critical product release windows.

AUTOMATIC ECO AND BACK ANNOTATION

When you implement changes to connectivity, component information, or design constraints in the schematic, the AutoECO feature detects the change and processes it as an ECO. OrCAD Layout also passes changes in reference designators, pin, and gate swaps, along with physical component placement information, back to the schematic.

COMPONENT PLACEMENT

Components can be grouped into clusters based on connectivity. These “super components” allow you to work with fewer objects, providing a high level view of the overall placement strategy. Component grouping makes it easy to move components as a single object, simplifying the creation of identical circuits. Group assignment can be done at the schematic or board level. Components can be dropped into a heavily populated area of the board and the “auto-shove” functionality automatically shoves other components out of the way to make room. This is done with full DRC control. Component locking/fixing temporarily or permanently prohibits components from being moved. Component locations specified from within OrCAD Capture or OrCAD Capture CIS can be imported. Keep-in and keep-out areas can be defined by component group or by height for floorplanning. Complete mechanical information, including all necessary floorplanning constraints from systems such as Pro/ENGINEER and SDRC I-DEAS, can also be imported.

MANUAL ROUTING/EDITING

T-routing allows you start or stop routing on a pad, via, or track segment. Track width can be quickly changed on-the-fly during the routing process. Segment editing allows you to slide segments, under DRC control, quickly pushing segments together to maximize available board real estate. Sliding segments will jump over obstacles to valid locations. Segment splitting divides an existing segment into two segments or easier manipulation. Complete tracks can be moved to another layer under DRC control.

AUTO-INTERACTIVE ROUTING

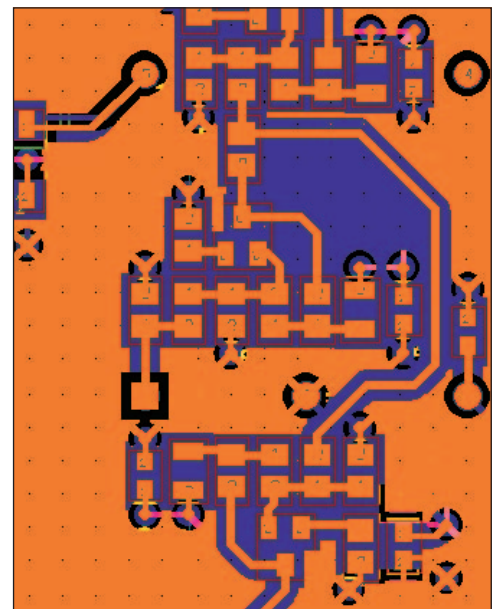
Interactive routing in OrCAD Layout offers powerful functionality and user interactive features. Use auto-path mode to select individual connections and steer a route interactively, with automatic shoving and clearing of surrounding tracks. Use auto-completion mode to steer a partial route, then let the autorouter complete it for you. Activate push/shove routing to steer routes interactively through congested areas as the router clears a routing path, while observing DRC requirements.

GRID-BASED AUTOROUTER

The autorouter in OrCAD Layout is especially useful when boards consist of similar-pitch components. The built-in, 16-layer, grid-based autorouter with push/shove capability, routes simple designs quickly with high-speed and high completion rates. Push/shove technology minimizes vias and allows higher routing density. It finds the optimal route for a track and then modifies existing routes and vias to clear space while adhering to all design rules.

AUTOMATIC FANOUT

An increasing number of designs today call for BGA and micro-BGA packages. OrCAD Layout automatically creates the complex interconnect these devices require. Fanouts can also be created for individual components or for an entire board. Users have control over the fanout distances, the ability to share/not share fanout vias, and control of fanout direction, either inside or outside of a component.



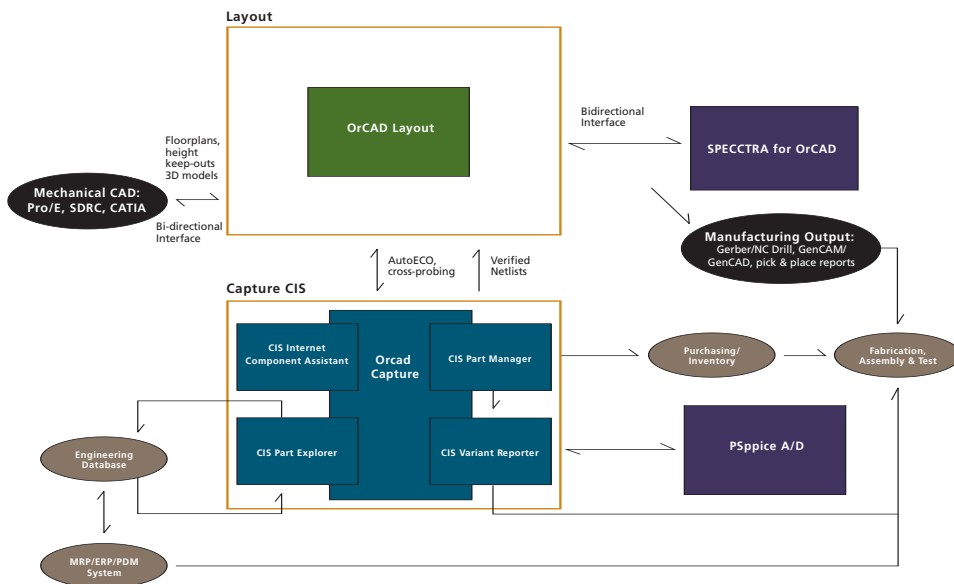
The intelligent copper pour is not segment-based so it won't create unconnected islands or other undesirable artifacts

SPECCTRA INTERFACE

The SPECCTRA® for OrCAD autorouter interface is embedded in the autorouting selections of OrCAD Layout, providing a seamless interface to the autorouter. The interface allows users to easily move PCB data and constraints to and from the SPECCTRA autorouter. SPECCTRA for OrCAD is tightly coupled with OrCAD PCB Editor, providing a rapid and seamless path for design autorouting. Routing strategy and setup is performed directly inside OrCAD PCB Editor using router setup dialogs. The interface then takes the defined routing setup, strategy, and all the design rules, and executes the router in a batch mode. The router's progress can be tracked interactively from within OrCAD PCB Editor. Once the routing strategy is completed, the routed interconnect is automatically imported back into OrCAD PCB Editor. Routing functions can be performed on a single net, a group of nets, or the whole design, allowing for the maximum in routing flexibility.

COPPER POUR, POWER PLANES, AND COPPER AREAS

Intelligent copper pour fully supports split-power and ground planes. Connections to the copper pour will only be created if the voltages of the pin and copper match. Copper pours can be overlaid and controlled with a Z-order rule which specifies the pouring order of the individual copper pours; the highest Z-order is poured last, leaving clearance between itself and the previous copper pour. Connectivity is automatically updated through copper pour, power planes, tracks, copper areas, and “free tracks.” Pads and vias can be designated as flood fill for direct hit on planes in high-current designs.



The OrCAD Layout family integrates tightly with Capture CIS and offers built-in import/export capabilities for working with a wide variety of standard third-party tools

ERROR CHECKING

Online DRC for routing and placement can be toggled on and off at any time. Double-clicking in the error-marker spreadsheet zooms to the error in the board window. Also, clicking on error coordinates in the query window automatically zooms to that error in the board window.

Online DRC works in conjunction with the interactive routing commands. Routing with 'Allow DRC Error' enabled can speedup manual routing throughput. If a DRC error is detected, you get real-time error notification.

REPORTS AND POST-PROCESSING

OrCAD Layout has an extensive report list and outputs customized reports. Reports can be saved as files and/or viewed directly on screen. Design post processing can be directed to a printer, or to Gerber or DXF files. All output settings are stored in a spreadsheet that can be saved and revised. This allows you to preview and select the proper output for each layer before creating the actual output files. OrCAD Layout also produces more than 20 standard reports including fabrication drawings, assembly drawings, and pick-and-place reports. Reports can also be created and customized for individual needs. A built-in output preview provides the user with the ability to examine the output before the actual files are created.

INDUSTRY INTERFACES AND OUTPUT

Interface with mechanical CAD systems

Import data from mechanical CAD systems such as Pro/ENGINEER, SDRG, CATIA, and Solid Edge into OrCAD Layout to place board outlines, height keep-outs, and other mechanical data in the PCB database. After placing the components, you can return the design to the mechanical system for 3-D modeling. OrCAD Layout then processes subsequent changes as mechanical ECOs, automatically importing the changes as needed.

Output of complete data for driving automated assembly and test systems

Drive downstream CAM/CIM tools from Mitron (GenRad), Fabmaster, and Valor (ODB++) through the GenCAD/GenCAM and ODB++ outputs available in OrCAD Layout. These include complete component information obtained via Capture CIS. Create all the output your design team needs with a comprehensive set of standard ASCII reports as well as user-customized reports.

Interfaces and translators

Interface to Quantic Omega software products facilitates simulation and analysis of high-speed designs, including pre-and post-route signal integrity simulation.

- Bi-directional IDF 2.0 interface supports 3-D modeling through Pro/ENGINEER and SDRG mechanical CAD systems

- IPC-D-356 netlist capability interfaces to downstream fabrication and test equipment
- GenCAD (bi-directional) and GenCAM along with Valor ODB++ format interface to Mitron's CIMBridge, and FABmaster, and other fabrication and test applications

OTHER AUTOMATED FUNCTIONS

OrCAD Layout has numerous automated functions which eliminate much of the drudgery of PCB design. For example, you can unroute or unplace a board with a single command. You can perform full or partial design rule checks, and the system will automatically remove design errors and error markers from the board upon demand. Other automated functions include test point generation, manufacturing optimization (gloss), component renaming, and back-annotation. Additionally, added database functions ensure integrity for database archival.

USER INTERFACE

The OrCAD Layout user interface has highly intuitive menus that reflect the board design process, making it easy for occasional users to remain productive. Online help with extensive hypertext cross references makes it easy to find the answers you need. A spreadsheet-based interface provides access to all objects in the design. You can display and print components, drill chart information, error objects, footprints, nets, obstacles, text, padstacks, post-processing data, and full boards statistics. Queries with intelligence give you the capability to quickly retrieve information about objects in a design. Clickable "hotlinks" in the query instantly jump to objects in the design. Context sensitive pop-up menus give you easy access to commands.

INTELLICAD FROM CADOPIA

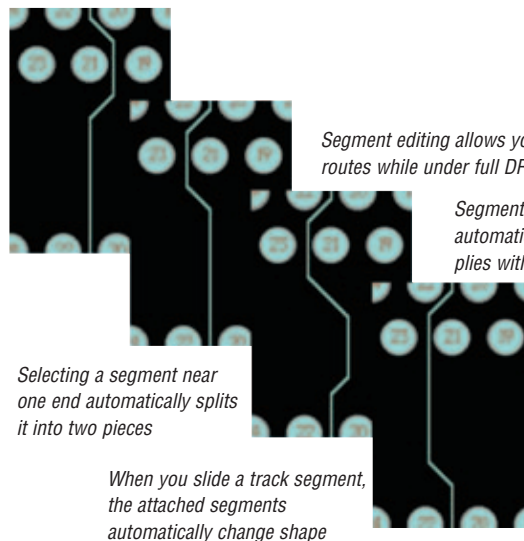
OrCAD Layout includes IntelliCAD which provides the ability to input, manipulate, and export mechanical CAD files. Open, create, modify, and review mechanical CAD files with IntelliCAD.

- Custom menus provide tools for creating board outlines; floorplanning objects; mounting holes; height keep-ins and keep-outs; autorouter keep-outs; and other similar PCB-oriented objects
- Associative dimensioning adjusts dimension lines automatically when a measured object changes
- Full read/write supports DWG, DXF, and GCD files — all AutoCAD format includes support for shape and font files

WISE SOFTWARE GERBTOOL

GerbTool is a complete CAM station that lets you view, edit, enhance, and verify Gerber files and is included with OrCAD Layout. This includes Gerber files created by other PCB layout packages.

- Automatic pad removal deletes unused pads from inner layers
- Automatic silkscreen removal removes silkscreen from pads and vias
- Draw/flash conversion changes "draws" to "flashes" for photoplotting
- Teardropping (filleting) reshapes pad exits to improve manufacturability
- Panelization, venting, and thieving save time and money by allowing you to do your own manufacturing prep work
- Gerber-to-DXF and DXF-to-Gerber translation supports reverse engineering
- Graphical netlist compare for the connectivity of your Gerber and drill files against an IPC-356 netlist to "close the loop" before committing to manufacturing



- Aperture list converter reads Gerber files from any CAD system
- Includes converters for many CAD packages
- Allows 4,000 different apertures per aperture list
- Supports user-definable aperture list converters
- Reads and writes Gerber 274D and Extended Gerber (274X), FIRE9XXX, EIE, Barco DPF, IPC356, and NC Drill files
- Supports HPGL, PostScript, LaserJet, Excellon, Sieb & Mayer, and most CAD aperture list formats

SYSTEM REQUIREMENTS

- Pentium 4 (32-bit) equivalent or faster
- Windows XP Professional, Windows XP Home Edition, or Windows 2000 (SP4)
- 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)

TECHNICAL SUPPORT

Contact your EMA Design Automation support engineer for questions on any of the OrCAD products.

support.ema-eda.com
techsupport@ema-eda.com
585-334-6001

SALES, PRICING INFORMATION, AND TRAINING

Contact your EMA sales representative for exact pricing information and details on internet training or classroom training in your area.

EMA Design Automation
225 Tech Park Drive
Rochester NY 14623
877-362-3321
www.ema-eda.com

OrCAD®

©2005 Cadence Design Systems, Inc. All rights reserved.
In the U.S. and numerous other countries, OrCAD, the OrCAD logo, OrCAD Capture, OrCAD Layout, and SPECCTRA, are registered trademarks of Cadence Design Systems, Inc. All others are properties of their respective holders.