Orcad Layout

Printed circuit board design

With the Orcad Layout family, your team can create and electronically share PCB data across the entire design process. Start with complete component information automatically retrieved via Capture CIS, add design constraints and other properties and electronically exchange the information with products in the Layout family. You can also exchange floorplanning information including board outlines, mounting holes, height keepins and keepouts, and routing keepins and keepouts. After component placement, send designs to 3D modeling and thermal modeling systems to ensure that they will fit and perform as expected.

The Orcad Layout family includes three products. You can mix and match different versions of Layout within your company, purchasing only the level of functionality each team member requires.

Orcad Layout Plus includes all the features of Orcad Layout and includes a gridless autorouter. It is an excellent choice for: designers of complex, high-density, mixed-pitch, multi-layer boards; designers of boards with high analog content; engineering departments with a high throughput requirement; and service bureaus.

Orcad Layout includes all the features of Orcad Layout Engineer's Edition and an embedded grid-based autorouter. It is a good choice for designers of less-complex boards, or companies with lower volumes of new board starts. In addition, the interactive routing features make Orcad Layout an ideal design tool in cases where most of the routing is performed manually.

Orcad Layout Engineer's Edition is intended for engineers or PCB designers who do not require an autorouter, or who use an external autorouter. It is a great choice for engineers performing critical placement and routing; engineers reviewing designs received from others; designers of RF/microwave designs; engineering teams that use an outside service bureau to create boards; or for EDA librarians who need to create and maintain corporate libraries.

Inter-Tool Communication Reduces Rework and Streamlines the Entire Board Design Process

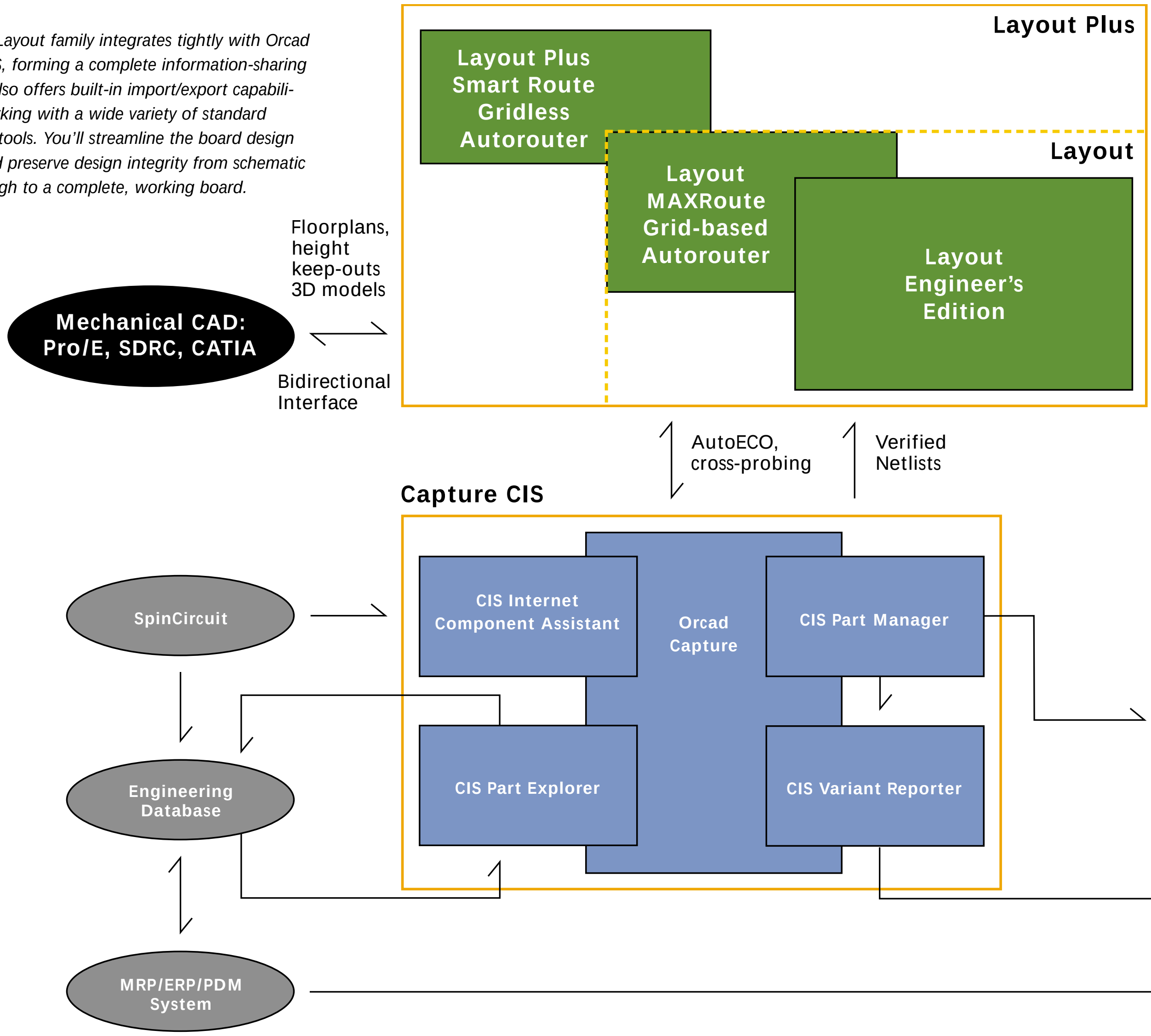
Most problems in PCB designs can be traced to incorrect component information, or poor communication between engineers and board designers.

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, problems may be tens or hundreds of times more expensive to correct.

Through Capture CIS and its tight integration with Orcad Layout, you can eliminate these sources of errors. You'll start the board design process with more complete information, produce more correct first-pass designs and streamline your entire process.

The Orcad Layout family integrates tightly with Orcad Capture CIS, forming a complete information-sharing system. It also offers built-in import/export capabilities for working with a wide variety of standard third-party tools. You'll streamline the board design process and preserve design integrity from schematic entry through to a complete, working board.

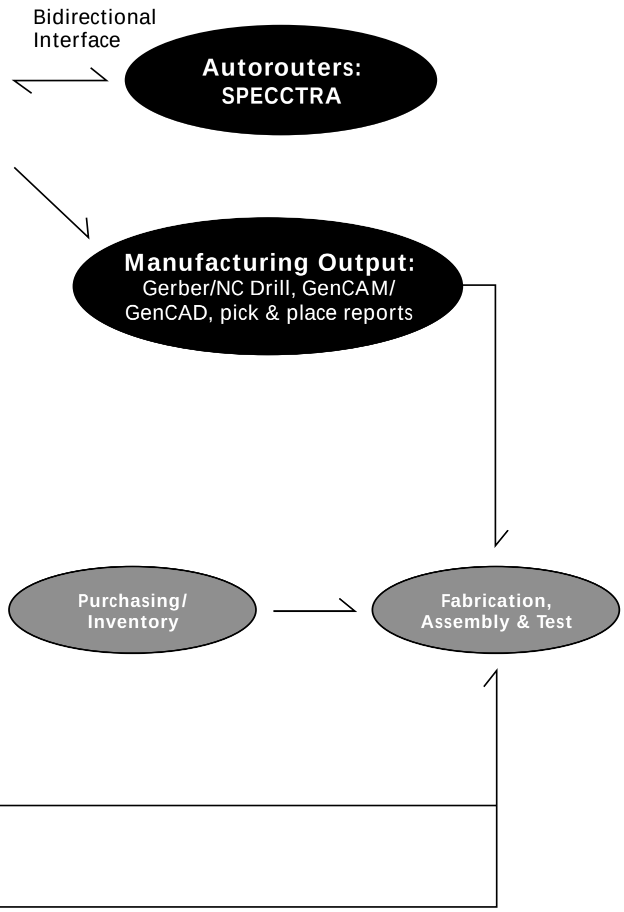


Automatic Retrieval of Correct Component Data

Capture CIS gives your design engineers online access to up-to-date, approved component information from a central database such as a corporate MRP system. As you place parts on the schematic, the system automatically retrieves part numbers, footprints and all the other part data necessary to complete the board. The CIS system automatically flags unapproved parts, allowing you to correct problems before they can cause design errors downstream.

Passing Design Rules, Constraints and Key Physical Data

You can specify spacing rules, flag critical nets and specify other key design constraints on the schematic, then pass them to Orcad Layout. Or, you can use Layout Engineer's Edition to perform critical placement and routing, then pass the design to a PCB design specialist for completion.



Interface with Mechanical CAD Systems

Import data from mechanical CAD systems such as Pro/ENGINEER®, SDRC, CATIA and Solid Edge into 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 3D modeling. Layout then processes subsequent changes as mechanical ECOs, automatically importing the changes as needed.

Auto-ECO and Back-Annotation

When you change connectivity, component information or design constraints in the schematic, Orcad Layout's Auto-ECO feature detects the change and processes it as an ECO. Layout also passes changes in reference designators, pin and gate swaps, physical component placement information and design constraints back to the schematic.

Interfaces to Other Cadence Tools

For high-speed digital designs, the optional Cadence SPECCTRA Autorouter can be used through the included bidirectional interface. Orcad Layout designs can also be passed to Cadence ALLEGRO through the included Layout/ALLEGRO interface. (Note: At the time of this printing, the Layout/ ALLEGRO interface is available only on the Orcad update webpage.)

Output of Complete Data for Driving Automated Assembly and Test Systems

Drive downstream CAM/CIM tools from Mitron (GenRad), Fabmaster and others through Layout's GenCAD/GenCAM outputs. 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.

Features

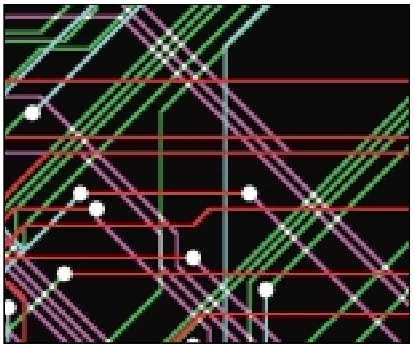
Component Placement*

Orcad's automatic and interactive component placement tools are found in Layout Plus only; the constraint management capabilities are present in all versions of Layout.

- Manually or automatically group components into clusters based on connectivity; these "super-components" allow you or the autoplacer to work with fewer objects, providing a high-level view of the overall placement strategy
- Drop a new component into a heavily populated area of the board; the "Auto-shove" capability automatically shoves other components out of the way to make room under full DRC control
- Place components in an arc or circle using polar coordinate entry; especially useful for creating IC test boards
- Matrix placement instantly places regular arrays of similar components, such as memory arrays or discrete components
- Component grouping makes it easy to move components as a group to simplify the creation of identical circuits; group assignment can be done at the schematic or board level
- Component locking/fixing temporarily or permanently prohibits components from being moved
- Import component locations specified from within Orcad Capture® or Capture CIS
- Define keep-in and keep-out areas by component group or by height, for floor planning
- Import complete mechanical information including all necessary floor planning constraints from systems such as Pro/ENGINEER and SDRC's I-DEAS™

SmartRoute Shape-Based, Gridless Autorouter*

Many of today's PC boards are mixed-pitch, with surface-mount components on both sides. These require gridless autorouters to assure full routability. The SmartRoute autorouter is designed to make short work of these more complex designs.



The SmartRoute autorouter performs diagonal routing which minimizes track length, via count and crosstalk.

SmartRoute automatically sets up the correct routing strategy for maximum completion rate. This technology produces finished boards that rival hand-routed boards created by expert PCB designers, both in performance and in appearance.

Unlike orthogonal routers which follow Manhattan rules and produce boards with longer tracks, higher via counts and higher levels of crosstalk, SmartRoute supports true diagonal routing which minimizes vias, track lengths and crosstalk.

SmartRoute is complete, with no add-ons or options to purchase. With it, you can produce high-quality, high-density boards of up to 16 routing layers.

- Fast, easy use with no complicated setup files
- Reads and writes Layout's native files, and recognizes rules passed from Orcad Capture or Capture CIS
- Routes complete boards, or individual components or nets under interactive control
- Automatically determines the proper bias (direction) that each layer should use based on connectivity
- Allows multiple designs to be batch-routed with a single command for overnight processing of multiple boards, or different versions of the same board

- Produces highly manufacturable boards through user selectable design-for-manufacture functions:
  - Automatically miters tracks during autorouting
  - Eliminates unnecessary vias
  - Removes jaggies on tracks to avoid unnecessarily long tracks
  - Centers tracks between pads to improve manufacturability

Choosing an Autorouter: Layout Plus or Layout?

The gridless, shape-based SmartRoute autorouter in Orcad Layout Plus is much faster than the gridded push-and-shove autorouter in Orcad Layout. The gridded Layout router can often complete the same complex boards as the SmartRoute router, but routing takes longer to complete and often requires more layers to find enough routing channels to complete a design.

However, Layout's autorouter is perfectly adequate for some kinds of design. For instance, analog designers who want to interactively control part or most of their routing can complete much of their design with the point-n-shoot routing feature.

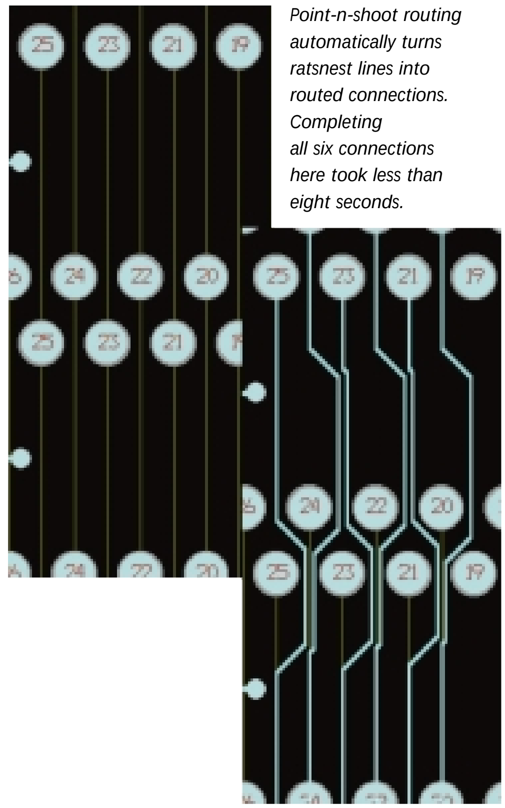
Grid-Based Autorouter*

Orcad's grid-based autorouter is especially useful when boards consist of similar-pitch components.

- Built-in 16-layer grid-based autorouter with push-and-shove capability routes typical designs quickly, with high speed and high completion rates
- Push-and-shove technology minimizes vias and allows higher routing density; it finds the optimal route for a track, then modifies existing routes and vias to clear space while adhering to all design rules

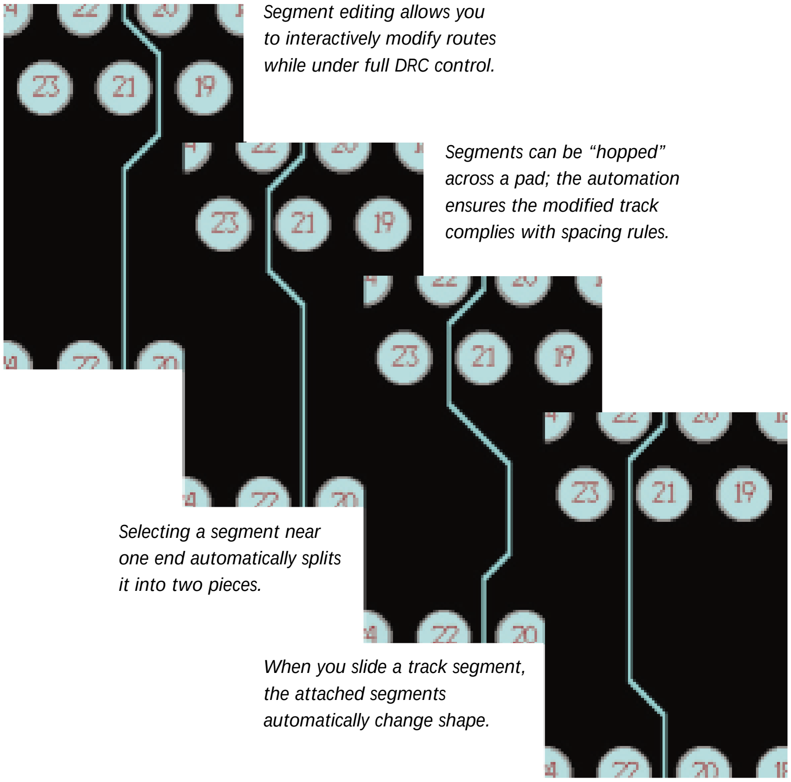
Interactive Autorouting*

Orcad's interactive routing offers powerful features not found in other PCB design tools. For example, with point-n-shoot routing you can simply double-click on a ratsnest line to automatically route the connection. This is especially useful on designs that have numerous short, simple connections. With point-n-shoot routing these types of connections can be routed very quickly.



Point-n-shoot routing automatically turns ratsnest lines into routed connections. Completing all six connections here took less than eight seconds.

- 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 interactive push-and-shove routing to steer routes manually through congested areas while the router clears a routing path, while observing DRC requirements



Segment editing allows you to interactively modify routes while under full DRC control.

Segments can be "hopped" across a pad; the automation ensures the modified track complies with spacing rules.

Selecting a segment near one end automatically splits it into two pieces.

When you slide a track segment, the attached segments automatically change shape.

Manual Routing

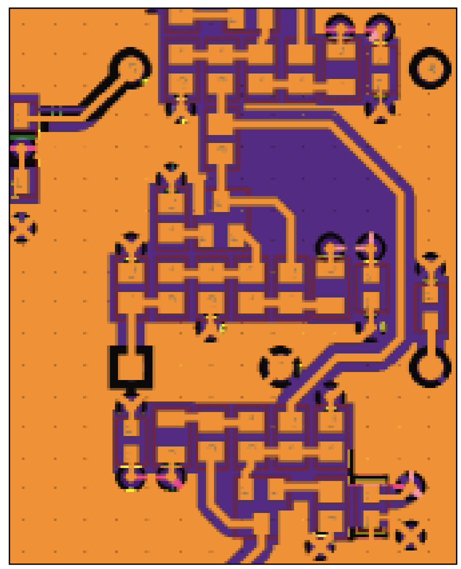
- "T-routing" lets you start or stop routing on a pad, via or track segment
- Track width can be quickly changed on-the-fly
- Segment editing allows you to slide segments under DRC control, quickly pushing segments together to maximize available board real estate
- Segments can jump over vias and other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

High-Speed Rules and Critical Route Support

- Specify pins as source, load or terminator for high-speed nets
- Designate nets as high speed to facilitate automatic daisy chaining, which keeps source and terminators in place while reconnecting the loads in the shortest fashion
- Use track spacing-per-layer and track width-per-layer settings to control impedance in microstrip or stripline designs

Copper Pour, Power Planes and Copper Areas

- Intelligent copper pour accurately places copper areas without creating copper islands
- Copper pour fully supports split-power and ground planes
- 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 high-current designs



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

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
- Clicking on error coordinates in the query window automatically zooms to that error

Other Automated Functions

Orcad Layout products have 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.

- Automatic test point generation lets you designate existing vias and through-hole pins as test points, and indicate whether to allow test points under components*
- User-selectable manufacturing optimization features enable you to: miter 90-degree corners, optimize vertices, optimize pad exits, eliminate acute angles, optimize shared tracks, and optimize shared vias.

Reports and Post-Processing

Layout has an extensive report list, and also 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.

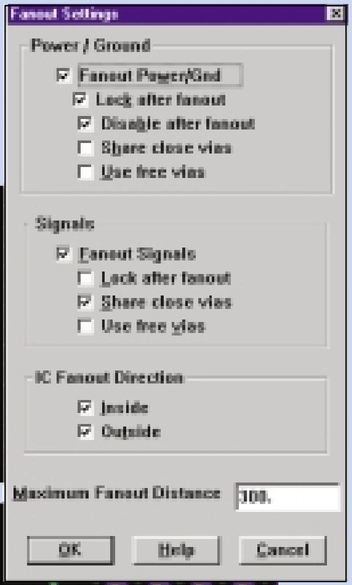
- Create a custom post-processing setup with a mix of outputs, by selecting outputs by individual layers in the post-processing spreadsheet
- Preview the post-processing setup in the built-in previewer, then save it in a file for fast recall and revision
- Generate pre-defined standard reports that include parts, component insertion, cross reference, net length, connection, unused pins, rename/swap, test points and pick-and-place reports
- Create an unlimited number of user-defined reports, drawing from any data in the spreadsheet-accessible board database

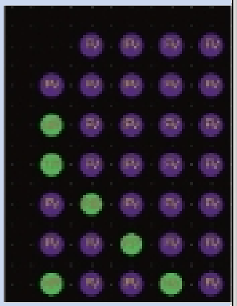
Auto-Assisted Fanout Supports High Pin-Density Devices

More and more designs call for BGA and microBGA packages. Layout automatically creates the complex interconnect that these devices require.

- Fanouts can be created for a whole board, or for individual components
- Complete control of fanout distances, ability to share/not share fanout vias, and fanout direction (in/out)
- Automatic full-board fanout promotes uniform density, making it easy to add additional tracks; this produces boards with up to 40% fewer vias than most other routing software
- Vias on close pins of the same signal can be optionally shared
- Vias on fanout stubs can be specified as standard (system) vias or free (user) vias
- Automatic fanout of BGAs and microBGAs supports fanout vias in the pad, or with short, 45-degree stubs (dogbone fanout)

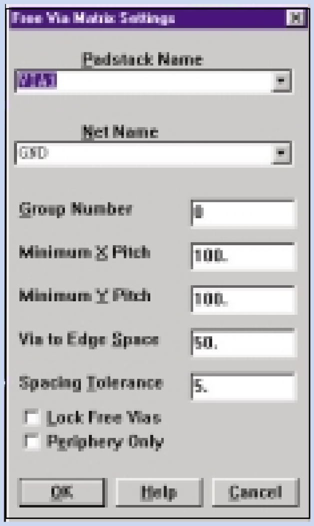
The fanout tool is highly configurable, accessed through a dialog box.



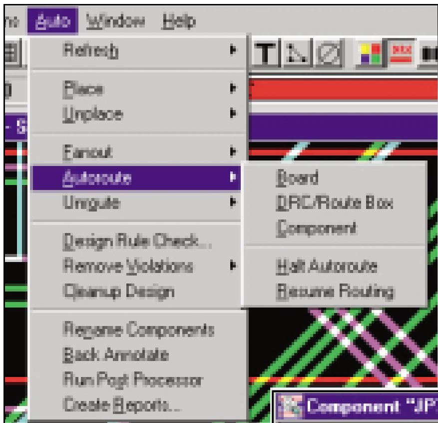


Fanout patterns can be embedded directly in a microBGA's pads, which simplifies routing.

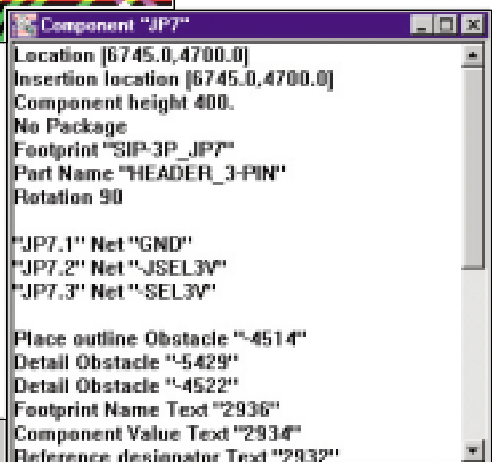
Layout automatically generates a 45-degree "dogbone" or "lollipop" fanout patterns.



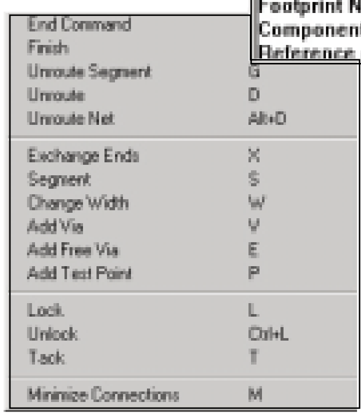
The Free Via Matrix Settings dialong lets you configure matrices, placing free vias evenly spaced in a copper area or only on the periphery.



Layout's menus closely follow the PCB design process, which makes it expecially intutive for electrical engineers and other team members who aren't full-time layout specialists.



The Query feature allows you to locate any object in the board database, view all entries related to that object, select an entry, then zoom to its location in the board layout.



Context-sensitive pop-up menus, accessed via the right mouse button, offer fast access to common procedures, such as routing options.

Library and Footprint Management

- Library manager streamlines library and footprint management:
  - Graphical library browser activates and deactivates libraries
  - Parts from all enabled libraries are sorted alphabetically for easy selection
  - Selected parts are displayed instantly, ready for editing
- Online footprint editor allows footprints to be directly edited on the board
- Extensive library includes more than 3,000 footprints and 60 board outlines

User Interface

- Highly intuitive menus 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
- Spreadsheet interface provides access to all objects in a design; you can display and print components, drill chart information, error objects, footprints, layers, nets, obstacles, text, padstacks, packages, post-processing data and full board statistics
- Query capability quickly retrieves information about objects in a design; clickable "hotlinks" instantly jump to objects in the board display
- Context-sensitive pop-up menus give instant access to commands
- Online interactive tutorial quickly brings you up to speed

Integration with Orcad Capture and Capture CIS

- Component, pin and net properties can be assigned in Orcad Capture/Capture CIS, then passed to Orcad Layout
- Intertool communication supports cross-probing of components and nets; select nets or components in Orcad Capture/Capture CIS or Orcad Layout and see them highlighted in the other program automatically
- Auto-ECO forward-annotates all changes made in the schematic to the board
- Back-annotation performs a full database comparison and creates a back-annotation file containing changes made to the board

- Bi-directional property passing allows board-level design rules to be passed back and forth

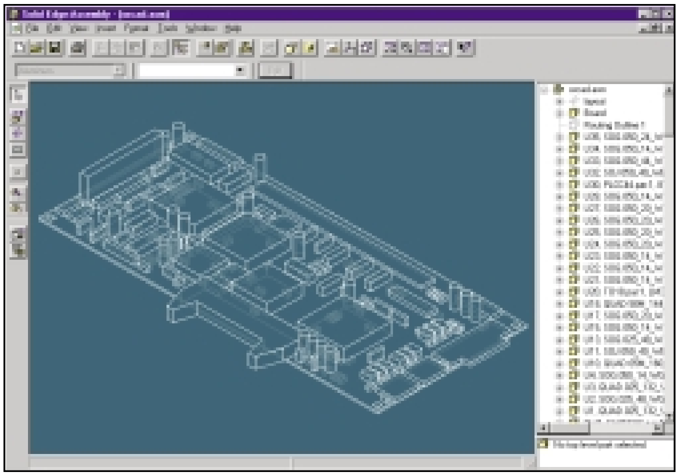
Mechanical Interfaces

The ability to input, manipulate and export mechanical CAD files is included with all Orcad Layout products. All versions ship with a complete version of Orcad Visual CADD, a 2D drafting tool customized for PCB work.

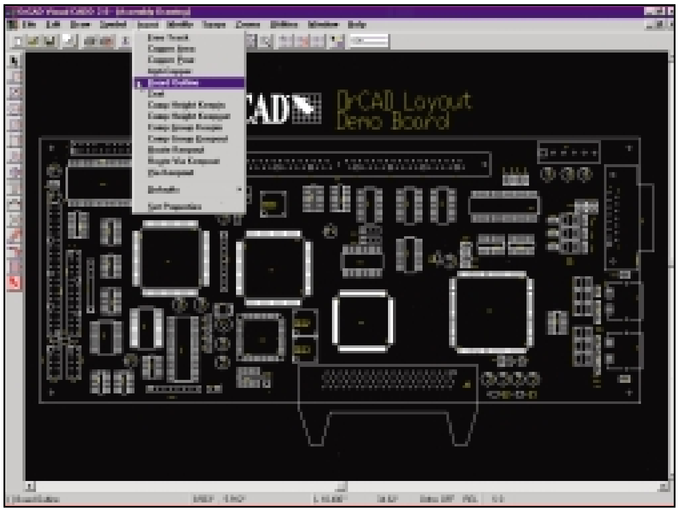
Layout also supports 3D modeling through the IDF interface. This interface allows bi-directional interchange of the mechanical data, component placement and height information. A board can be modeled in a 3D modeling system and a revised design sent back to Layout. Layout then automatically accepts the change and treats it as a mechanical ECO. This eliminates the need to hand-enter changes, and to subsequently remodel and review them.

- Create DXF files with complete floorplanning criteria
- Import complete PCBs, including connections, padstacks and components from DXF or Gerber
- Review the contents of complex DXF files, and selectively import only the information that is necessary to the board design
- Open, create, modify and review CAD files with the Orcad Visual CADD 2D design and drafting program:
  - 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
  - Unlimited undo/redo speeds editing
  - Associative dimensioning adjusts dimension lines automatically when the measured object changes
  - Full read/write supports DWG, DXF and GCD files; AutoCAD (Rel 12) format includes support for shape and font files

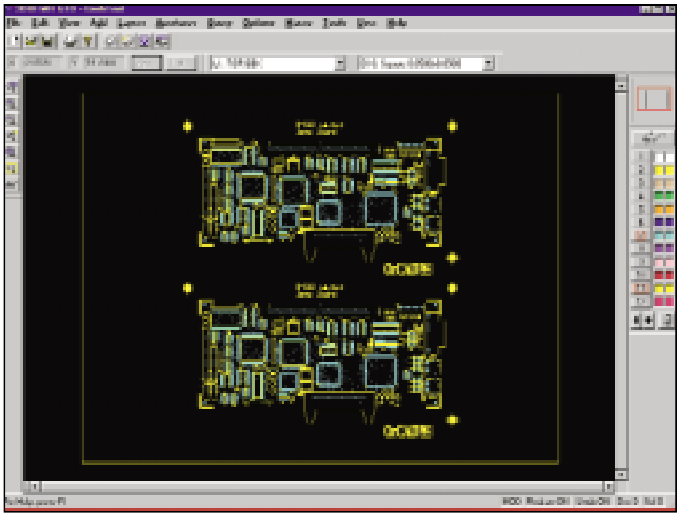
Special menu commands in Visual CADD automatically create complex board objects.



All Layout products import board outlines from mechanical CAD packages. After placing components, the design data can be exported for 3D modeling.



All Orcad Layout products include Orcad Visual CADD for importing, creating and documenting mechanical design data.



Orcad GerbTool panelizes boards, creates complex layers and automatically removes silkscreen from pads.

PC Boards from other systems are quickly imported into Layout, with the PCB Import Wizard.

Manufacturing Support

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

- Automatic pad removal reduces potential for clearance problems by deleting 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
- Aperture list converter reads Gerber files from any CAD system, for use with legacy Gerber files:
  - Includes converters for many CAD packages
  - Allows 4,000 different apertures per aperture list
  - Supports user-definable aperture list converters
- Support for multiple standards promotes compatibility with many design or manufacturing environments:
  - 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
- Can compare the connectivity of your Gerbers and drill files against an IPC-356 netlist to “close the loop” before committing to manufacturing.

Interfaces and Translators

All of Orcad’s Layout products include all of the interfaces and translators you need to communicate and verify your design. New translators are added regularly, and updated versions can be found on the Orcad website (www.orcad.com).

- Interface to Cadence SPECCTRA is especially useful for designing high-speed boards
- Interface to HyperLynx products facilitates simulation and analysis of high-speed designs, including pre- and post-route signal-integrity simulation
- Bi-directional IDF 2.0 interface supports 3D modeling through Pro/ENGINEER and SDRC mechanical CAD systems
- IPC-D-356 netlist capability interfaces to downstream fabrication and test equipment
- GenCAD (bi-directional) and GenCAM formats interface to Mitron’s CIMBridge and FABmaster and other fabrication and test applications

Major Feature Summary for Layout

	Layout Plus	Layout	Layout Engineer's Edition
Component placement			
Cluster placement	■		
Automatic placement	■		
Auto-interactive placement	■		
Shape-based gridless autorouting			
16-layer autorouter	■		
Auto-interactive routing	■		
Diagonal routing	■		
Gridless push-n-shove capability	■		
Grid-based autorouting			
16-layer autorouter	■	■	
Auto-interactive routing	■	■	
Single-layer autorouting	■	■	
Grid-based push-n-shove	■	■	■
General features			
3,000+ footprint library	■	■	■
Auto test point generation	■	■	Manual
Auto ECO	■	■	■
Intelligent copper pour	■	■	■
Routing density display	■	■	■
PCB translators			
Bi-directional: Orcad PCB386+	■	■	■
Semi-Bidirectional: CadStar, PCAD, Tango, Protel, PADS, PCBoards	■	■	■
Export only: Cadence Allegro	■	■	■
Schematic netlist interfaces			
Orcad Capture, Capture CIS	■	■	■
Autorouter Interface			
Cadence SPECCTRA	■	■	■
Mechanical interfaces			
Orcad Visual CADD 2D Drafting (DXF/DWG, ACAD v12 Compatible)	■	■	■
Pro/ENGINEER, SDRC, Solid Edge, CATIA (IDF 2.0)	■	■	■
CAM interfaces			
Orcad GerbTool™ – full function Gerber Editor	■	■	■
Gerber RS-274D, extended gerber (274X)	■	■	■
GenCAD, GenCAM	■	■	■
IPC-D-356	■	■	■

Specifications for all Layout Products

- 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

System Requirements

- Intel Pentium™ or equivalent processor.
- Windows 2000®, Windows NT® 4.0 (with Service Pack 3 or later installed), Windows 98® or Windows 95®.
- 32 MB RAM minimum (additional memory improves performance); SmartRoute requires 64-128 MB RAM for large, complex designs
- 60-120 MB hard disk space recommended for installation and design files
- 800 X 600 minimum display resolution (1024 X 768 recommended)

Year 2000 Compliance

All Orcad family products for Microsoft Windows 2000, Microsoft Windows NT, Windows 98, and Windows 95 are Year 2000 Compliant.

Product Support

Every Orcad product comes with:

- One year of technical support via phone, email and fax
- Access to the Orcad-family Internet-based technical information and support connection at www.orcad.com
- A one year free subscription to product updates
- Training workshops and other services are also available. Contact Cadence PCB Systems Division at 1-800-671-9505 or a Cadence International Reseller for more information.

Upgrade Policy

Each Orcad Layout product contains the full functionality of the lower-priced versions. You can load existing designs into a higher-functionality product and use it without conversion or modifications.

When you purchase an Orcad Layout product, you will receive a generous trade-in allowance if you later decide to upgrade to a version with higher functionality.

Pricing and Ordering Information

In North America:

- Contact Cadence PCB Systems Division at 1-800-671-9505
- Email to pcbinfo@cadence.com
- Fax to (503) 671-9501

Outside of North America:

- Contact your Cadence International Reseller
- Email to intl@orcad.com

For a list of resellers visit our website at www.orcad.com/contact/contact.asp

cadence®

North America
PCB Systems Division
Portland, OR 97223 USA
phone: 503.671.9500
website: www.orcad.com

Europe
Cadence Design Systems, Ltd.
Bracknell, Berkshire
United Kingdom
phone: +44 1344 866 533
website: www.orcad.co.uk

Japan
Cadence Design Systems, Japan
Kohoku-ku, Yokohama
Japan
phone: +81 45 475-6330
website: www.orcad.co.jp

Asia
Cadence Design Systems Asia, Ltd.
Kowloon, Hong Kong
phone: 852 2377 7111
website: www.cadence-asia.com

60-31-821 PG 07/00 25K