



CADENCE ORCAD CAPTURE CIS PRODUCTION-PROVEN PCB SCHEMATIC DESIGN ENTRY AND POWERFUL COMPONENT INFORMATION SYSTEM

Cadence® OrCAD® Capture CIS integrates robust schematic design capabilities with the features of a component information system (CIS) for fast, universal PCB design entry. The added capabilities of a CIS provide easy access to component databases and part information. This boosts productivity by reducing the amount of time spent researching for or reusing existing parts, manually entering part information, and maintaining component data. Whether used for designing a new analog, digital, or mixed-signal circuit, revising schematic diagrams for an existing PCB, or implementing hierarchical block design, OrCAD Capture CIS delivers everything needed to take circuit design from concept to production.

SCHEMATIC EDITOR

The flat and hierarchical Schematic Page Editor of OrCAD Capture CIS combines an intuitive interface with the features and functionality needed to speed design tasks and facilitate circuit creation. For larger, more complex designs, OrCAD Capture CIS supports multi-sheet and hierarchical designs. It also makes hierarchical designs easy to traverse and ensures that all connections are maintained accurately throughout the design. Seamless interfaces establish robust data paths and integration with OrCAD PCB Editor for physical PCB design and with Cadence PSpice® A/D for analog/digital circuit simulation. Seamless bi-directional integration with OrCAD PCB Editor enables synchronization and cross-probing/placing between the schematic and the board, and automated engineering change orders (ECOs) backannotate layout changes, gate/pin swaps, and changes to component names or values.

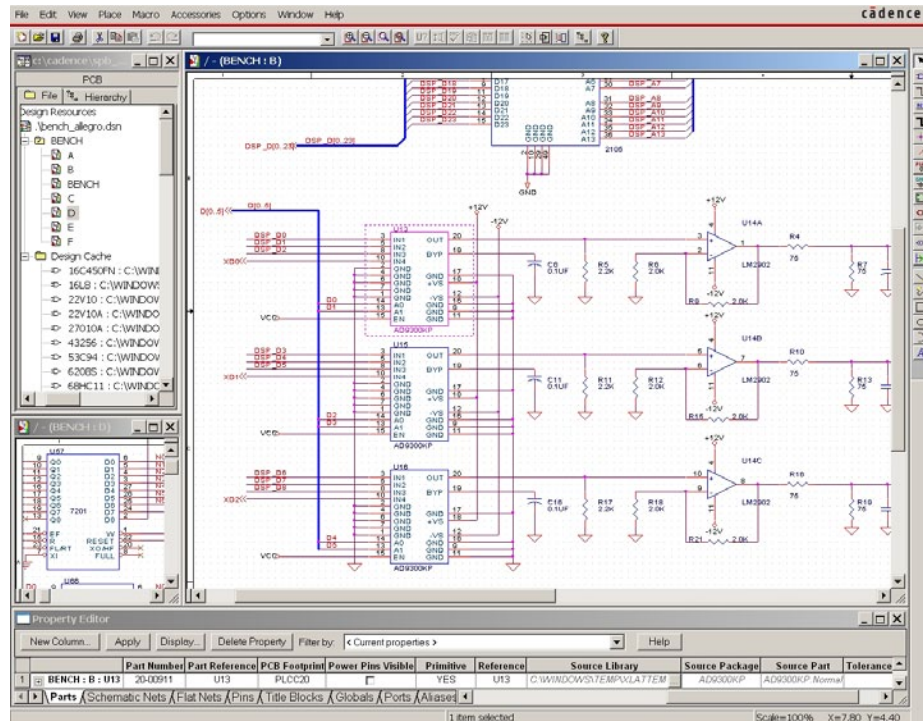


Figure 1: OrCAD Capture CIS provides powerful capabilities to enter, modify, and verify schematic circuits including part management, flat and hierarchal design, and circuit reuse

COMPONENT INFORMATION SYSTEM

The component information system (CIS) is a central part of the OrCAD design entry solution. It automatically synchronizes and validates the externally sourced data with the schematic design database. CIS works with any database that complies with Microsoft's ODBC standard to directly access data in an MRP, ERP, or PLM system, or in an intermediate database dedicated to engineering component data.

With easy access to component databases and part information, designers can reduce the amount of time spent researching parts. CIS allows users to identify, utilize, and design with preferred components. Parts can be queried based on their electrical, physical, or corporate characteristics, and then be automatically retrieved for use in the schematic. Adding components directly from company databases minimizes errors in bills of materials (BOMs) and parts lists, and allows control of part usage for industry directives such as RoHS and WEEE. Electrical part tables can be related to vendor part tables, allowing relational database support and relational queries.

CIS also features the Internet Component Assistant (ICA) for accessing component information in online databases. As with corporate database access, online parts can be queried based on electrical, physical, or manufacture characteristics, and automatically retrieved for the schematic. The free Cadence ActiveParts online electronic database, accessible through ICA, contains more than 2 million parts. With ActiveParts, users can search for and select parts based on specific criteria, and preview parts before placing them in a schematic. Users have direct access to additional part supplier databases through ICA.

BENEFITS

- Offers full-featured schematic editing for fast, intuitive design capture
- Boosts efficiency in schematic editing of complex designs through hierarchical and variant design capabilities
- Accelerates the design process and lowers project costs by integrating with a robust CIS that promotes preferred, current parts
- Increases design productivity through auto-zoom/search/navigation features, cross-probing and cross-placement with PCB Editor, and integration with PSpice A/D

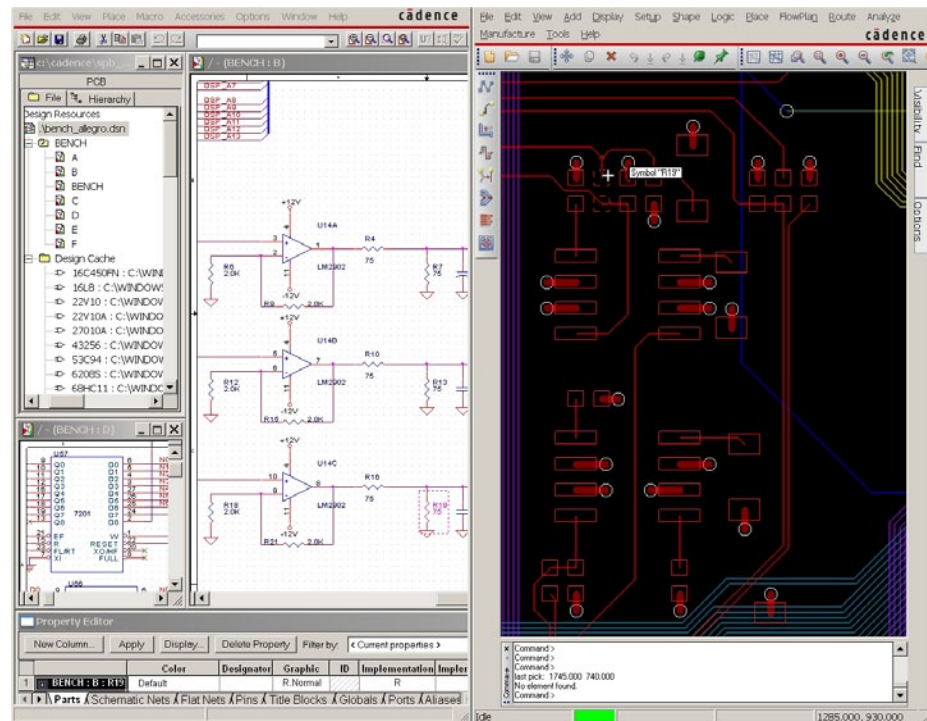


Figure 2: Tight product integration enables cross-probing and accurate data passing between OrCAD Capture and OrCAD PCB Editor

- Reduces time spent researching parts and enables intelligent component selection with access to MRP, ERP, and PLM data and support for relational databases
- Offers greater flexibility when choosing design components and easier access to direct vendor part data with ActiveParts and the ActiveParts Portal, which provides access to more than two million parts
- Automates the integration of field programmable gate arrays (FPGAs) and programmable logic devices (PLDs) with a bi-directional data flow through FPGA export/import, thereby reducing design time

FEATURES

FULL-FEATURED SCHEMATIC EDITOR

OrCAD Capture CIS, with its flat and hierarchical Schematic Page Editor, is based on the OrCAD legacy of fast, intuitive schematic editing. The Schematic Page Editor combines a standard Windows user interface with functionality and features specific to the engineer for accomplishing design tasks and publishing design data.

- View and edit multiple projects in a single session
- Access the latest parts via the Internet
- Use Label State for “what if” scenarios
- Launch the Property Spreadsheet Editor at the design or schematic level to edit or print your design properties
- Reuse design data by copying and pasting within or between schematics
- Select from a comprehensive set of functional part libraries
- Move pin name and number with in-line editing of parts
- Lock files in case the design is being opened by another user
- Place, move, drag, rotate, or mirror individual parts or grouped selections while preserving both visual and electrical connectivity
- Ensure design integrity through design and electrical rule checkers
- Create custom title blocks and drawing borders to meet your company specifications
- Insert drawing objects, bookmarks, logos, and bitmapped pictures
- Choose from metric or imperial unit grid spacing to meet all drawing standards
- Design digital circuits with the VHDL or Verilog® Text Editor
- Use non-linear auto-zoom to pan the canvas and leverage extensive find/search capabilities
- Enable dockable windows and non-model dialogues

INTERCHANGE ARCHITECTURE

Through its interchange architecture, Capture CIS accepts OrCAD plug-ins for programmable logic design and analog simulation. Users can create and simulate the entire design within a single environment. The centralized Project Manager organizes the entire design and process flow.

- Plug-in PSpice A/D simulators for analog simulation
- Interface with OrCAD PCB Editor place-and-route with full forward and backward connectivity
- Interface with Cadence Allegro® high-speed PCB layout solutions with full forward and backward connectivity
- Cross-probe nets and signals between schematic and simulation windows
- Cross-probe nets and parts between schematic and PCB layout
- Cross-place components in PCB layout from OrCAD Capture

COMPONENT INFORMATION SYSTEM (CIS)

OrCAD Capture CIS solves many of the problems associated with incorrect or incomplete component data. The key to the system is an embedded part selector that accesses information stored within corporate MRP or ERP systems and engineering databases. CIS synchronizes the externally sourced data with the schematic design database, automatically generating accurate BOMs. Through efficient management of a CIS within a PCB design process, users can eliminate production delays and lower costs. OrCAD Capture CIS can also reduce or eliminate a number of potential problems:

- **Board rework due to incorrect component data** - A simple task such as changing the wattage on a resistor without changing the corresponding footprint and part number can cause a mismatch between the schematic, PCB layout, and BOM. Such an out-of-sync design can easily trigger an expensive, unnecessary board turn.
- **Production delays caused by out-of-stock parts** - Inadvertently identifying and utilizing an obsolete component or an item with a long lead-time or incorrect supplier can cause an unplanned, costly delay. All too often, these problems aren't discovered until the product is on the verge of being manufactured.
- **Higher-than-anticipated component costs** - Without access to cost information during the design stage, it's easy to introduce high-priced parts and miss the opportunity to use equivalent lower-cost substitutes from the same or alternate supplier.
- **High inventory costs due to duplicate parts** - Redundant parts are often introduced into the inventory system when functionally equivalent parts are already available and in stock.

OrCAD Capture CIS is developed within the industry-standard schematic package of OrCAD Capture as a simple extension of the part selection process. It solves a critical problem for engineering organizations: selecting the best component available, based on both technical and business information, is accessible to engineers on a daily basis.

- Promote the reuse of company preferred components
- Reuse known good part data
- Quickly introduce new parts into the approval process
- Ensure design data integrity by leveraging part property validation
- Have confidence in the accuracy and completeness of BOMs and netlists
- Enable electronic sharing of information with purchasing, manufacturing, and other departments to eliminate delays and surprises and reduce product costs

PROJECT MANAGER

OrCAD Capture CIS allows you to manage the entire design using a sophisticated project management system. An expanding-tree diagram allows for efficient navigation and organization of design files, including those files generated by plug-in modules available from OrCAD.

- Project Creation Wizard guides the process through all the resources available for a specific design flow
- Centralized management of all design data permits a seamless interchange of schematic data for OrCAD plug-ins and downstream flow
- Browser enables hierarchical navigation of the entire design structure and "visiting" specific selections instantly
- Archive capability ensures the portability of the entire design project

HIERARCHICAL DESIGN

OrCAD Capture CIS can easily organize and reuse duplicate circuitry through the use of hierarchical blocks.

- Update ports and pins dynamically for hierarchical blocks and underlying schematics
- Reuse OrCAD/Allegro PCB modules within or between schematics
- Enables a single instance of the circuitry for you to create, duplicate, and maintain
- Reference and reuse circuitry throughout the entire design on an unlimited basis
- Eliminate potential design connection errors through automatic creation of hierarchical ports
- Manage/edit all property occurrences from one location using a spreadsheet view of the entire reuse hierarchy tree

DATABASE INTEGRATION

OrCAD Capture CIS works with any database that complies with Microsoft's ODBC standard. Data can be directly accessed from an MRP, ERP, or PDM system and/or an intermediate database dedicated to engineering component data.

- Integrates easily with your existing MRP, ERP, or PDM systems
- Supports relational databases and relation queries
- Supports download of parts directly from vendor's site and stores them as parts in the local library
- Supplies standard drivers for ODBC-compliant databases including Microsoft Access, Visual FoxPro, SQL Server, and Excel
- Supports Japanese characters
- Honors regional numeric format setting from the Control Panel
- Database Creation Wizard automatically creates new preferred parts database populated with information imported from existing OrCAD schematics

PART SELECTION

Part Placement

Parts can be browsed from local or network-installed libraries. Users can review the part, modify its properties as necessary, and then dynamically place it within a design—all in the same sequence.

- Identify similar library schematic parts with the use of wildcards
- Store commonly reused parts by placing them directly from the project design cache
- Apply auto-numbered reference designators, according to preference settings

Placement from Preferred Part Databases

With the OrCAD Capture CIS interface to centralized part data, users can avoid tedious part searches and manual data entry.

- Access centralized and comprehensive part information without leaving the Capture CIS design environment
- Explore, query, and filter according to part-specification criteria, and identify a specific physical part
- Give preference to pre-approved, "qualified" parts that are already in the company's MRP or ERP system

- Graphically confirm the selection so that the part appears on the cursor for placement
- Eliminate hand-typing of part properties; any required information housed in the database is automatically annotated with the selected part

New Part Introduction and Tracking

OrCAD Capture CIS assists with the introduction of new parts into a part approval system.

- Easily create and place a new part by duplicating and editing an existing record in the component database
- Receive an automatic alert when a temporary part is assigned a company part number and promoted to preferred status

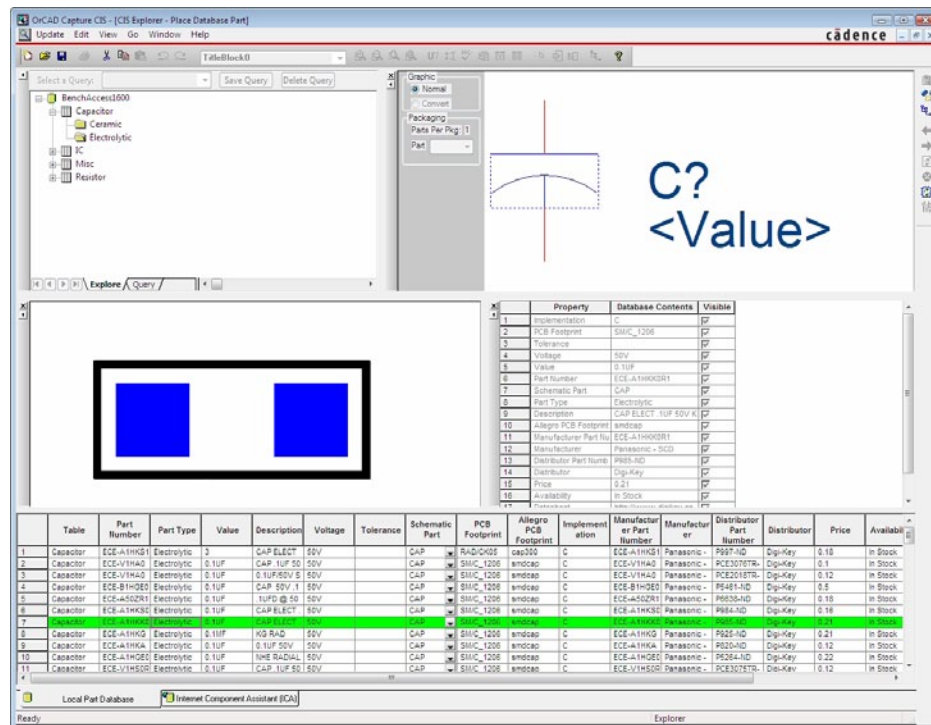


Figure 3: Visibility and access to complete part information ensures accurate and informed selections, reducing the risk of delays later in the design process

PROPERTY MANAGEMENT

Browsing and Property Editing

Users can access any or all part, net, pin, and title block properties—and make changes quickly—through a single spreadsheet viewer/editor.

- Select an individual element, grouped area, or entire page and then add/edit/delete part, net, or pin properties
- Define selections quickly through pick lists and true/false checkboxes
- Globally apply specific property names, as required by personal outputs or netlists, across all designs in order to maintain consistency, reduce manual errors, and eliminate multiple re-entry
- Browse and instantly visit any part, net, hierarchical port, off-page connector, bookmark, or design rule error marker from a single reference point

Property Verification

OrCAD Capture CIS can automatically verify the schematic part data against the corresponding records in a centralized part database.

- System automatically detects and prompts you to update properties containing incorrect data
- Colored indicators clearly flag any part with unresolved problems
- Tool Tips specify the exact nature of any problems
- Temporary parts are visually tracked until fully approved and part numbers have been assigned
- Direct links to the centralized database allow for resolving or retargeting problem parts
- CIS delivers clear, visual confirmation that the component data in the design is current and accurate

Part Re-linking and Updating

Users have the flexibility to modify, substitute, or update the initially chosen parts as the design progresses.

- Update part information by re-linking the part to the database and refreshing the property data
- Replace a part by simply re-directing your link to a different database entry
- Automatically substitute the part's graphical representation during the update process; OrCAD Capture CIS handles the part replacement and maintains reference designators and connectivity where applicable

RESEARCH AND DOWNLOAD NEW PART DATA FROM THE INTERNET

The Internet Component Assistant provides access to the ActiveParts Portal to download schematic symbols and parametric component data from the Internet without leaving the design application. The ActiveParts Portal provides free access to the Cadence ActiveParts database as well as the SupplyFrame online component database. The ActiveParts Portal allows users to search for and select parts based on specific criteria, and to preview parts before placing them in a schematic.

DESIGN VARIANTS

The Design Variants capability allows users to manage unlimited board assembly variations without having to maintain duplicate schematics or manually edit BOMs.

- Reduce the number of files to handle by maintaining all design assembly variations within a single file
- Modify the core schematic design and see the change automatically reflected throughout all assembly variations
- Ensure the integrity of component data in each assembly through the property verification utility
- Identify substituted and non-placed components within each assembly, assisted by graphical indicators and comparison reports
- Generate schematic printouts and BOMs for each assembly at any point in the design process

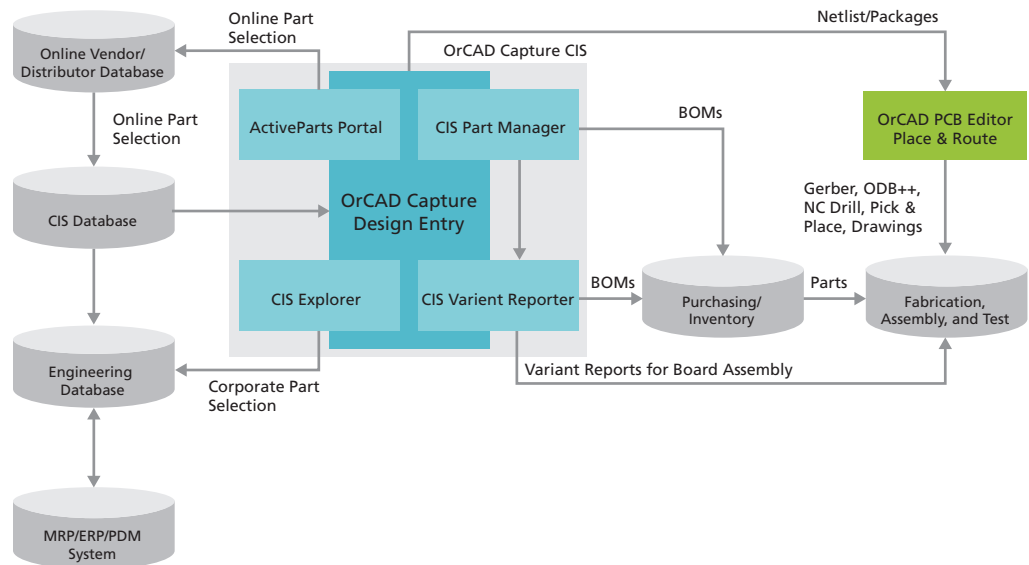


Figure 4: OrCAD Capture CIS provides a flexible and scalable solution throughout the design process

DOCUMENTATION

Standard Capture Documentation

OrCAD Capture CIS creates basic BOM outputs by extracting information contained in the schematic database.

- Automatically package parts with reference designators prior to report generation
- Preview, print, and save reports; save formats include tab delimited, comma delimited, and Microsoft Access
- Re-sort columnar data by simply clicking on the field header in the report preview window
- Directly export to Microsoft Excel; a single selection before preview will automatically open a local copy of Excel and import the report selections

Advanced CIS Documentation

Integrated within OrCAD Capture CIS, the Crystal Decisions reports print engine can analyze part data up front in the design process and prevent ECOs later in the design process.

- Apply pre-defined Crystal Decisions reports templates containing queries, formulas, and filters
- Output costed BOMs that include subtotaled sections
- Identify obsolete, low-inventory, or long lead-time parts with a preference status
- Produce quality presentations of reports that include logos, graphics, and automatically generated date-and-time stamps
- Communicate changes with notifications using integrated email
- Publish HTML documents to the Internet or an intranet

CONNECTIVITY RESOLUTION

As parts and wires are placed, the underlying connectivity is maintained at all times.

- Get visual confirmation when terminating nets on pins
- Automatically create or delete junction points using manual override

- Have complete flexibility when moving and editing segments, wires, or entire nets
- Generate connectivity utilizing net names to reduce schematic wiring
- Quickly create control buses using signal aliases
- Efficiently manage net properties, individually and globally, through spreadsheet editors
- Avoid potential connectivity issues using online design rule checking (DRC)

LIBRARY AND PART EDITOR

The OrCAD Library Editor is directly accessible from the OrCAD Capture CIS user interface. Users can create and edit parts in the library or directly from the schematic page without interrupting the workflow.

- Accelerate schematic part creation and editing using intuitive graphical controls
- Modify existing parts to quickly create new parts
- Speed the development of pin-intensive devices using spreadsheet and pin array utilities
- Reduce clutter on schematics using based "vector" pins
- Drag and drop between libraries and schematic design cache to speed creation and maintenance of master library sets
- Revise a single part on the schematic or update all occurrences
- Revise control tracks part edits at the schematic level
- Control visibility and connectivity of power and ground pins at the schematic level

ORCAD CAPTURE COMPARISON GRID

COMPONENT INFORMATION SYSTEM - FEATURE SUMMARY	OrCAD CAPTURE	OrCAD CAPTURE CIS
ODBC-compliant component database and MRP, ERP, and PDM integration		•
Interface to relational databases and database management systems		•
Browse manufacturers' specification datasheets (URL, UNC)		•
Centralized part number and part information management system		•
Japanese character set support in CIS Explorer		•
Graphical preview of schematic parts and OrCAD / Allegro PCB footprints		•
Database query for part selection and parametric property annotation		•
Schematic-to-database linking for property and schematic part re-placement		•
Schematic part and PCB footprint alternative pick lists		•
Management utility for schematic property validation		•
Temporary part tracking system		•
Automatic part number allocation for approved temporary parts		•
Report preview/print and save in Word, Excel, Access, or delimited formats		•

COMPONENT INFORMATION SYSTEM - FEATURE SUMMARY (continued)	OrCAD CAPTURE	OrCAD CAPTURE CIS
Report output containing both schematic and linked database properties		•
Reports with sorting, subtotalling, and selection criteria formulas		•
Centralized and sharable templates for report standardization		•
Crystal Decisions Reports print engine		•
Integration with company email and Lotus Notes		•
HTML interface for Internet and intranet report publishing		•
Download part information through the ActiveParts Portal		•
ASSEMBLY VARIANTS - FEATURE SUMMARY	OrCAD CAPTURE	OrCAD CAPTURE CIS
Assembly variations on a fabricated PCB		•
Unlimited board assembly specifications within a single schematic design		•
Part substitutions and part “not present” requirements definable per variation		•
Comparison reports for any combination of variants		•
Options for outputting/printing variant reports and schematics		•
SCHEMATIC ENTRY AND CIRCUIT DESIGN - FEATURE SUMMARY	OrCAD CAPTURE	OrCAD CAPTURE CIS
Unlimited undo/redo and Label State command	•	•
Dynamically update hierarchical blocks and underlying schematics	•	•
OrCAD / Allegro design reuse support	•	•
Graphical, flat, and hierarchical design Schematic Page Editor	•	•
Design archiving and project management process flow system	•	•
Standard Windows user interface and functions including cut/copy/paste	•	•
Floatable/dockable tool bars and palettes for common editor functions	•	•
Macro recorder for replay of complex schematic edits and customization	•	•
VHDL text editor	•	•
Verilog text editor	•	•
Online design rule check for data flow, packaging, and connectivity	•	•
Import/export for property data and reusable circuitry blocks	•	•

SCHEMATIC ENTRY AND CIRCUIT DESIGN - FEATURE SUMMARY (continued)	OrCAD CAPTURE	OrCAD CAPTURE CIS
Unlimited user-defined properties for parts, nets, pins, and title blocks	•	•
Support for multi-gate logic and distributed-pin and relay-type devices	•	•
TrueType fonts for display and printing	•	•
Adjustable color palette for display and plotting	•	•
Print offset and scaling controls	•	•
Forward- and back-annotation of PCB layout properties and pin-and-gate swaps	•	•
Graphical schematic part and library editor	•	•
Cross-probing and cross-placement with OrCAD / Allegro PCB Editor	•	•
More than 44,000 library parts including IEEE and IEC standard styles	•	•
EDIF graphic transfer and export of netlist format	•	•
MicroSim Schematics translation interface	•	•
30+ PCB layout netlist interfaces	•	•
Imperial or metric unit support	•	•
Automatic schematic backup and recovery	•	•
Interactive tutorial and online help system	•	•
Non-linear editor for auto-zoom as user pans the canvas	•	•
Dockable windows and multiple project navigation	•	•
FPGA bi-directional support	•	•

SALES, TECHNICAL SUPPORT, AND TRAINING

The OrCAD product line is owned by Cadence Design Systems, Inc. and supported by a worldwide network of Cadence Channel Partners (VARs). For sales, technical support, or training, contact your local Cadence Channel Partner (VAR). For a complete list of authorized Cadence Channel Partners (VARs), visit www.cadence.com/Alliances/channel_partner.

cadence[®]

Cadence Design Systems, Inc.

CORPORATE HEADQUARTERS

2655 Seely Avenue
 San Jose, CA 95134
 P: +1.800.746.6223 (*within US*)
 +1.408.943.1234 (*outside US*)
 F: +1.408.943.5001
www.cadence.com