Cadence OrCAD Capture

Production-proven PCB schematic entry and powerful Component Information System

Cadence® OrCAD® Capture includes numerous features that facilitate PCB schematic entry. An industry standard in PCB schematic entry, it is one of the most popular systems because of its intuitive use model and out-of-the-box capabilities. With OrCAD Capture CIS, component information can be accessed from both online and centralized part databases—expediting circuit creation.

Whether used for designing a new analog circuit, revising digital schematic diagrams for an existing PCB, or implementing hierarchical block design, OrCAD Capture delivers everything needed to take circuit design from concept to production. Seamless interfaces establish robust data paths plus integration with OrCAD PCB Editor for physical PCB design and with Cadence PSpice® A/D for analog/digital circuit simulation. OrCAD Capture CIS integrates the OrCAD Capture schematic design application with the added capabilities of a Component Information System (CIS), allowing easy access to component databases and part information.

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 through integration with a robust CIS that promotes preferred current parts
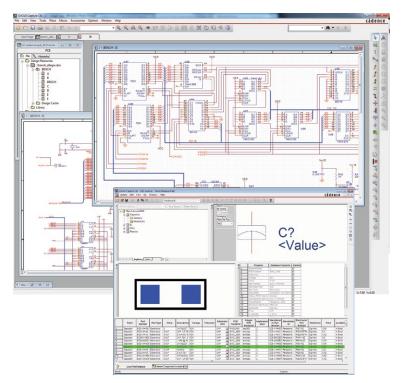

Figure 1: OrCAD Capture and OrCAD Capture CIS provide powerful capabilities to enter, modify, and verify schematic circuits including part management, flat and hierarchical design, and circuit reuse

- Reduces time spent researching parts and enables intelligent component selection with access to MRP, ERP, and PLM data

- Provides access to more than two million parts with Cadence ActiveParts, for greater flexibility when choosing design components

Features

Schematic Editor

The flat and hierarchical schematic page editor of OrCAD Capture builds on the OrCAD legacy of fast and easy schematic editing. It 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 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.

Ease of Use

The schematic page editor combines an intuitive user interface with functionality and features that enhance usability and speed for accomplishing design tasks and publishing design data. The autowire capability, for example, automates the often tedious and time-consuming task of wiring signal pins. Wiring between component pins is as simple as selecting a starting pin and a destination pin and letting the software automatically and quickly add the connection. Context-sensitive menus, OLE support, custom colorization of wires/nets/parts, and a tabbed/dockable interface all provide a better user experience.

Design Reuse

The re-use of existing logical circuits that have already been tested and proven is one of the best ways to reduce cycle time and maximize quality. Having already been placed, routed, and validated on a previous design, the effort that went into the original design can be preserved. Typical examples include power supply modules, RF circuit designs, multichannel circuits (I/O, drivers, etc.), and memory.

Design Rule Check

The configurable design rule check (DRC) feature in OrCAD Capture allows comprehensive verification of the design before committing to downstream design processes saving the time and cost of ECOs later in the design cycle. Design rules checks include reports of duplicate parts, invalid design packaging, and electrical violations.
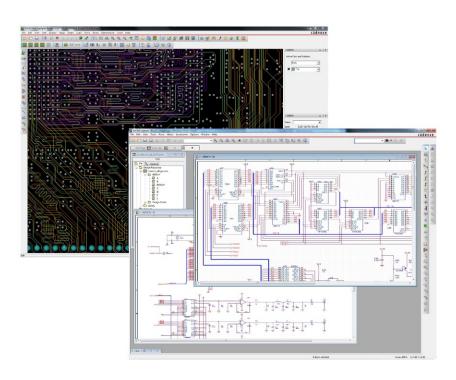

Figure 2: Tight product integration provides cross-probing/cross-placing and accurate data passing between OrCAD Capture and OrCAD PCB Editor

Component Information System

The Component Information System (CIS) is a central part of the OrCAD Capture design 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.

Relational Data Support

CIS allows for the creation and use of relational tables in the component parts database. These relational tables have a one-to-many relationship with the part information (primary) tables. The relational database may contain a vendor table with multiple vendor/manufacture part numbers for a single company part number in the electrical (e.g. resistor) table. With this structure, search and query for data across the primary and relational tables is possible.

Part Selection

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, 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.

Internet Component Assistant

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 manufacturing characteristics, and be automatically retrieved for the schematic. The free Cadence ActiveParts online electronic database—accessible through ICA—contains more than two million parts. With ActiveParts, users can search for and select parts based on specific criteria, and preview parts before placing them in a schematic.

Design Variants

With the design variants capability, designers can manage unlimited board assembly variations without having to maintain duplicate schematics or manually edit BOMs. This reduces the number of files by maintaining all design assembly variations within a single design. Substituted and/or unplaced components within each assembly are displayed through graphical indicators. Schematic variants with BOMs can be generated at any point in the design process.

TCL Programming Language

The Tcl programming language, embedded with OrCAD Capture, enables custom scripting support and the power to add features and to customize the user interface, command structure, and design database. Through scripting customization, automation can be applied to manual processes to complete tasks faster, and difficult operations can be streamlined. Custom features that do not exist natively can be created or downloaded as app from the OrCAD Capture Marketplace, further enhancing and extending the OrCAD environment.

OrCAD Capture Marketplace

OrCAD Capture Marketplace, embedded within OrCAD Capture, provides a web conduit that delivers OrCAD-specific information directly to engineer's fingertips. This OrCAD-centric product and design information includes component information, simulation models, and PCB services. It also includes OrCAD-centric multimedia (application notes, whitepapers, product tips, movies, etc.), which can reduce ramp-up time on new features and releases. Within OrCAD Capture Marketplace, Cadence provides an industry-first online store for apps, which enables powerful productivity and customized capabilities. With access to the online store, PCB designers can find and download apps that give greater control over the design processes and flows through new and enhanced features or capabilities.
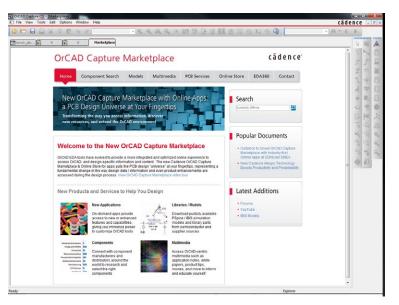

Figure 3: OrCAD Capture Marketplace provides a web conduit that delivers OrCAD-specific information directly to engineer's fingertips

FPGA Design

OrCAD Capture, together with OrCAD FPGA System Planner, addresses the challenges that engineers encounter when designing large pin-count FPGAs on the PCB board—which includes creating the initial pin assignment, integrating with the schematic, and ensuring that the device is routable on the board. They deliver a complete, scalable technology for FPGA-PCB design-in and co-design that automates creation of optimum "device-rules-accurate" pin assignment, symbol creation, and flow. By replacing manual, error-prone processes with automatic pin assignment synthesis, this unique placement-aware solution eliminates unnecessary physical design iterations while shortening the time required to create optimum pin assignment.

Design-In

OrCAD Capture supports the FPGA design flow with the ability to quickly import and/or create FPGA symbols and components. With an ever-increasing pin count and complexity for FPGA parts, the easy to use GUI-based options of OrCAD Capture can be used to create single and multi-section FPGA parts based on the device I/O pin files. Support for split parts, power pin visibility, pin shape, and pin group management provide flexibility to tailor symbol creation to the design

needs. FPGA components can also be exported using the export FPGA dialog box. The export FPGA completes the bi-directional link between FPGA designers and PCB designers.

Co-Design

OrCAD FPGA System Planner provides a complete, scalable solution for FPGA-PCB co-design that allows users to create an optimum correct-by-construction pin assignment. FPGA pin assignment is synthesized automatically based on user-specified, interface-based connectivity (design intent), as well as FPGA pin assignment rules (FPGA rules), and actual placement of FPGAs on PCB (relative placement). With automatic pin assignment synthesis, users avoid manual error-prone processes while shortening the time to create initial pin assignment that accounts for FPGA placement on the PCB (placement-aware pin assignment synthesis). This unique placement-aware pin assignment approach eliminates unnecessary physical design iterations that are inherent in manual approaches.

OrCAD FPGA System Planner is integrated with both OrCAD Capture and OrCAD PCB Editor. It reads and creates OrCAD Capture schematics and symbols. In addition, a floorplan view uses existing footprint libraries from OrCAD PCB Editor. Should placement change during layout,

pin optimization using FPGA System Planner can be accessed directly from OrCAD PCB Editor.

OrCAD PCB Flow Integration

Seamless bi-directional integration with OrCAD PCB Editor enables synchronization and cross-probing/placing between the schematic and the board. Automated engineering change orders (ECOs) backannotate layout changes, gate/pin swaps, and changes to component names or values. OrCAD Capture CIS comes with a library of schematic symbols and associated VHDL models, as well as the EDIF schematic and other CAD vendor netlist interfaces.

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 channel partner. For a complete list of authorized channel partners, visit www.cadence. com/Alliances/channel_partner.

cādence®

Cadence is transforming the global electronics industry through a vision called EDA360. With an application-driven approach to design, our software, hardware, IP, and services help customers realize silicon, SoCs, and complete systems efficiently and profitably. **www.cadence.com**