

ORCAD CAPTURE ORCAD CAPTURE CIS

The capability to provide fast and universal design entry makes OrCAD® Capture design entry the most widely used schematic entry system in electronic design today. Whether used to design a new analog circuit, revise a schematic diagram for an existing printed circuit board (PCB), or design a digital block diagram with an HDL module, OrCAD Capture provides the tools needed to enter, modify, and verify the PCB design. OrCAD Capture CIS integrates the OrCAD Capture schematic design application with the features of a component information system (CIS).

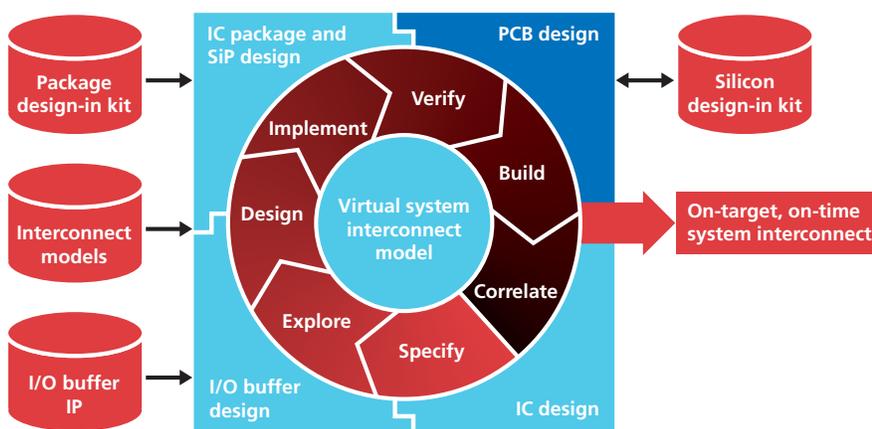


Figure 1: The affordable, high-performance OrCAD product line is easily scalable with the full complement of Cadence Allegro PCB design solutions

ORCAD PCB DESIGN TECHNOLOGIES

OrCAD products have a proven track record of innovation in the PCB personal productivity market. Available as stand-alone tools or in comprehensive suites, they allow designers to realize products from conception to manufacturing output. Easy to use and intuitive, they offer exceptional value, and OrCAD technology provides easy migration to the Cadence® Allegro® platform (see Figure 1).

ORCAD CAPTURE

OrCAD Capture is a complete solution for design creation, management, and reuse. Its ease-of-use allows designers to focus their creativity on design development rather than tool operation. The hierarchical Schematic Page Editor combines a Windows user interface with functionality and features specifically for design entry tasks and for publishing design data. Centralized project management provides seamless interchange of

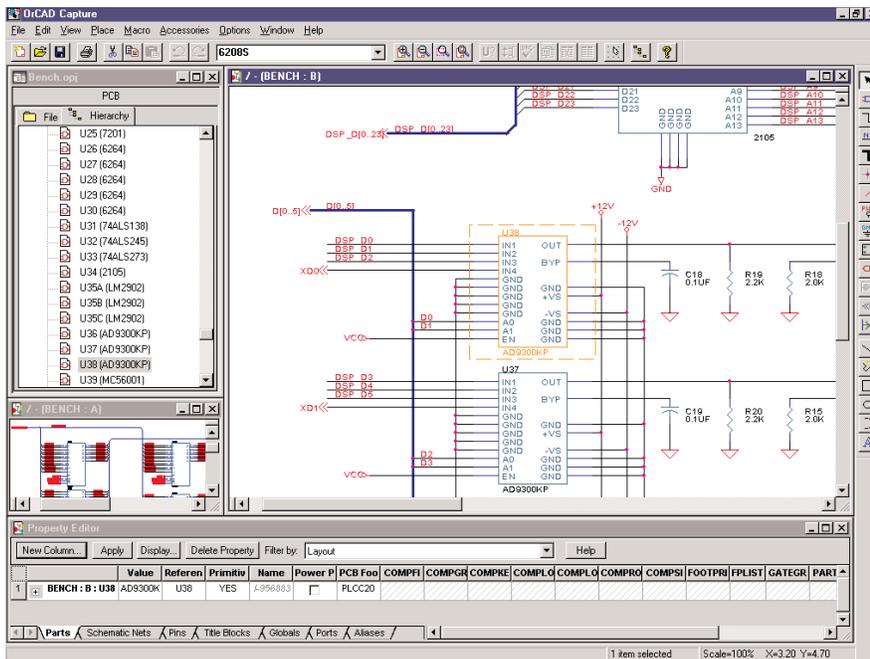


Figure 2: Find and select parts or nets quickly from the Project Manager. The multi-window interface makes navigation across hierarchy easy

schematic data for circuit simulation, board layout, and signal integrity analysis. A configurable design rule check (DRC) mechanism helps eliminate costly engineering change orders (ECOs). Basic bill of materials (BOMs) outputs are created from data contained in the schematic database.

BENEFITS

- Provides fast, intuitive schematic editing
- Boosts schematic editing efficiency by design reuse
- Automates the integration of FPGA and PLD devices
- Makes changes quickly through a single spreadsheet editor
- Imports and exports virtually every commonly used design file format
- Reduces delays caused by out-of-stock parts (CIS)
- Promotes reuse of preferred components (CIS)
- Encourages reuse of known good part data (CIS)
- Makes reuse of duplicate circuitry easy through hierarchical blocks (CIS)

FEATURES

SCHEMATIC EDITOR

The full-featured schematic editor allows users to view and edit multiple schematic designs in a single session. Design data is easily reused by copying and pasting within or between schematics. Parts are quickly selected from a comprehensive set of functional part libraries. Configurable design and electrical rule checkers ensure design integrity. In-line editing of parts allows pin name and number movement. A user interface has been provided to add critical constraints for users of the OrCAD Capture to OrCAD PCB Editor flow.

PROJECT MANAGER

The Project Manager simplifies organizing and tracking the various types of data generated in the design process. An expanding tree diagram makes it easy to structure and navigate design files, including those generated by PSpice® simulators, OrCAD Capture CIS, and other plug-ins. A Creation Wizard guides users through all the resources available for a specific design flow. Users can navigate the entire schematic structure

and instantly open specific elements—a schematic page, part, or net—with the hierarchy browser (see Figure 2).

HIERARCHICAL DESIGN AND REUSE

OrCAD Capture boosts schematic editing efficiency by enabling subcircuit reuse—without having to make multiple copies. Using hierarchical blocks, simply reference the same subcircuit multiple times. Automatic creation of hierarchical ports eliminates potential design connection errors. Ports and pins can be updated dynamically for hierarchical blocks and underlying schematics. Added navigation utilities recognize block boundaries and accessibility using keyboard shortcuts.

THE LIBRARIES AND PART EDITOR

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

Intuitive graphical controls speed schematic part creation and editing. New parts are created quickly by modifying existing ones. New parts can also be created from spreadsheets. A Library Part Generator automates the integration of field programmable gate arrays (FPGAs) and programmable logic devices (PLDs) into the system schematic. The Generate Part feature simplifies the creation of core FPGA library parts for high-pin-count devices. These parts can be split into multiple parts.

EASY DATA ENTRY

Designers can access all part, net, pin, and title block properties, or any subset, and make changes quickly through the OrCAD Capture spreadsheet Property Editor. It simply requires selecting a circuit element, grouped area, or entire page, and then selecting add/edit/delete part, net, or pin properties.

ORCAD CAPTURE CIS

OrCAD Capture CIS is designed to reduce production delays and cost overruns through efficient management of components. It reduces the time spent searching existing parts for reuse, manually entering part information content, and maintaining component data. Users search parts based on their electrical characteristics and OrCAD Capture CIS automatically retrieves the associated part. Flexible and scalable, the solution is quickly implemented.

OrCAD Capture CIS is ideal for individual design teams or multi-site teams who need to collaborate across multiple locations, OrCAD Capture CIS gives designers access to correct part data early in the design process and enables complete component specifications to be passed to board designers and other members of the design team, reducing the potential for downstream errors. It provides access to cost information so designers can use preferred, lower cost, and in-stock parts. The embedded part selector accesses information stored in MRP/ERP systems and engineering databases and synchronizes externally sourced data with the schematic design database, so bills of materials can be automatically generated (see Figure 3).

ARCHITECTURE, DATABASE INTEGRATION

Through Interchange Architecture, OrCAD Capture CIS accepts OrCAD plug-ins for programmable logic design and analog simulation. Users create and simulate an entire design in a single environment. OrCAD Capture CIS works with any database that complies with Microsoft's ODBC standard. Users can directly access data in an MRP, ERP, or PDM system and/or an intermediate database dedicated to engineering component data.

ONLINE COMPONENT LIBRARY

ActiveParts™, a fully integrated online component library, allows designers to search and select components from more than two million parts. This

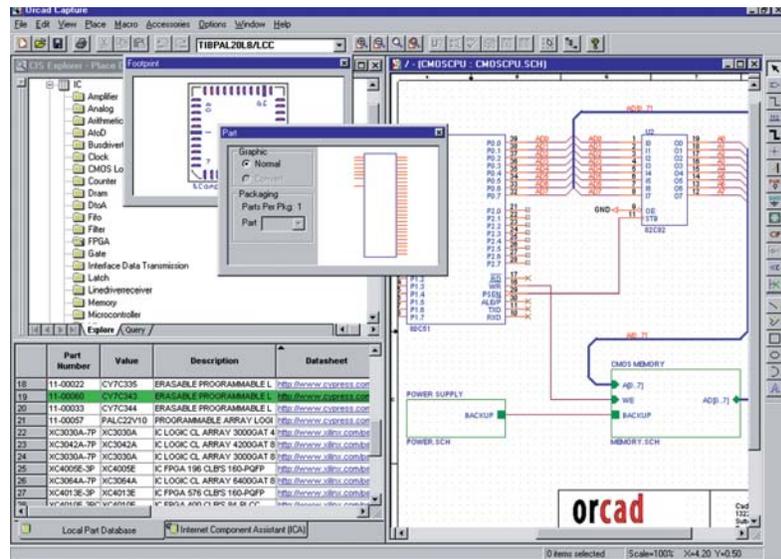


Figure 3: Visibility into complete part information ensures informed part selection, reducing the risk of delays later in the design process

significant improvement in access to parts provides users with a whole new level of design flexibility by allowing them to search online.

EXTENDED CIS DOCUMENTATION

OrCAD Capture CIS also allows a new level in report generation. Instead of limiting designers to just those properties that reside in the schematic, OrCAD Capture CIS draws from the extensive wealth of information that resides in the Preferred Part Database. Designers can generate BOMs using up-to-date, comprehensive, and complete information, and create reports through the Crystal report engine.

ONLINE HELP

- Find the answer you need by searching the online help system, and navigate quickly between related topics with extensive "hypertext" cross references
- Get up to speed quickly with the OrCAD Capture and OrCAD Capture CIS online interactive tutorial

SYSTEM REQUIREMENTS

- Pentium 4 (32-bit) equivalent or faster
- Windows XP Professional, Windows XP Home Edition, Windows 2000 (SP4), or Windows Server 2003

- 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 Cadence Channel Partners. For sales, technical support, or training, contact your local Cadence Channel Partner. For a complete list of authorized Cadence Channel Partners, visit www.cadence.com/partners/channel_partner/index.aspx.

PRICING INFORMATION

For product pricing and availability, contact the Cadence Channel Partner nearest you. For a complete list of authorized Cadence Channel Partners, visit www.cadence.com/partners/channel_partner/index.aspx.