INTEGRATED TECHNOLOGIES

Cadence addresses every aspect of your electronic design with the most comprehensive set of integrated design technologies in the industry (see Figure 1). Each Cadence Design Technology is made up of a flexible set of tools that form end-to-end solutions throughout the following areas:

- System-level design
- Functional verification
- Emulation and acceleration
- Synthesis/place-and-route
- Analog, RF, and mixed-signal design
- Custom IC layout
- Physical verification and analysis
- IC packaging
- PCB design

Cadence® PCB Design offers a complete range of integrated PCB design solutions for both individual and team-based environments. Scalability is also a key offering. At any point in the design flow, you can scale to the next level of performance preserving your investment in libraries, design data, and expertise.

DESIGN TECHNOLOGY CONVERGENCE

Converging design technologies such as digital logic, RF, and analog circuitry are becoming an integral part of PCB design. As a result, PCB engineers must now consider a host of new challenges throughout all stages of PCB development. System-on-a-chip (SoC), system-in-package (SiP), and other complex silicon technologies are also requiring advanced solutions. With high-speed (sub nanosecond rise times, high percentage of nets requiring constraints) designs now accounting for over half of all PCB systems developed, critical issues such as constraint development and management must be considered in ways never before imagined.

Today, design teams need to develop PCB systems faster, more economically, more reliably, and with greater collaboration from front to back — all the way to manufacturing. Using non-integrated “point tools” will only lead to increased design cycles, altered product specifications, and unacceptable delays. To stay competitive, you need an integrated set of solutions that provide critical functions across the entire design flow.

Figure 1: The integrated Cadence solution
INCREASED DEMAND FOR HIGH-SPEED PCB

Cadence PCB Design includes Constraint Manager, the industry’s first truly integrated and unified constraint management system. Managing complex constraints is more critical than ever. It’s not uncommon to see a 5,000-net PCB with over 50% of the nets constrained today — compared to only 5% of the nets constrained a few years ago.

Complex high-speed PCB designs also require advancements in power delivery. In fact, current demand has been growing faster than the reduction in supply voltage. Noise that was tolerated on a 3.3 V system cannot be tolerated in a 2.5 V or a 1.8 V system. To create a stable and reliable design, engineers need to explore system power delivery requirements during the conceptual design phase, define powerplane decoupling requirements based on system specification requirements, and drive the PCB layout process. SPECCTRAQuest™ Power Integrity provides the design team with an industry-proven solution.

TEAM-BASED PCB DESIGN SOLUTIONS

Cadence PCB Design offers a full range of integrated team-based solutions. Entry-level PCB solutions are available to teams of all sizes who want to design complex, high-volume PCB systems. When teams want to scale to a more powerful high-speed solution, they can leverage their existing tools preserving their original investment. Predominantly complex high-speed, high-volume solutions are also available for medium to large sized corporations looking for immediate, leading-edge, high-speed PCB solutions (see Figure 2).

- Industry’s only unified constraint management system improves time-to-market by limiting the number of iterations between engineering, layout, and prototype
- Scalable, front-to-back PCB design solutions
- Advanced technology in signal integrity design and analysis solves complex design issues
- Team-based design reuse modules reduce design cycles by weeks or months depending on the complexity of the module

Figure 2: High-speed PCB design flow
“Cadence proved that it can address our requirements of high-density and high-speed interconnection design solution space, with a scalable solution that can address current and future technology demands imposed by system and chip designers.”

S.L.N. Murthy
CEO, ECAD Technologies Ltd.

---

**CONCEPT HDL SCHEMATIC ENTRY**

Concept® HDL is a mixed-level design capture system that fuses robust HDL design facilities with a wealth of schematic capture and part selection capabilities — supporting both behavioral and structural descriptions. It includes an integrated graphical editor for structural capture and serves as an interface to any text editing system for HDL-based design. Seamlessly integrated with all the downstream tools required in the development cycle, Concept HDL provides much-needed flexibility in the design capture process.

- Provides an integrated FPGA design flow with Synplicity’s Synplify and other major FPGA vendors
- Tight integration with Cadence NC-Sim simulators and PSpice® allows concurrent analog simulation and debugging — speeding the design verification process up to 25%
- Cross-highlighting function with Allegro® PCB layout makes it easy to verify logical and physical designs
- Allows import of RF design from Agilent’s ADS product solutions
- Integrates with Constraint Manager to manage high-speed constraints across the design flow

---

**PCB LIBRARIAN EXPERT**

PCB Librarian Expert is central to Concept HDL and Allegro component information and library management solutions. Using the Part Browser utility, PCB Librarian Expert assembles all the tools necessary in all areas of part management — creation, editing, validation, and source control management. It also publishes data to the engineering desktop.

At the heart of PCB Librarian Expert is Part Developer, which swiftly walks the user through the part creation and validation process.

- Create, edit, and validate logical and physical component data
- Error-checking routines ensure parts are error-free and guaranteed to pass the packaging process.
- Accelerates productivity by leveraging millions of internet-hosted, pre-created library symbols
- Enforced library specifications ensure the consistency of published library data
- Links enterprise information directly to the Concept HDL user so component selection decisions are made using correct and current information
**SPECCTRAQUEST SI EXPERT**

SPECCTRAQuest™ Signal Integrity (SI) Expert is an integrated high-speed design and analysis environment for electrical engineers creating digital PCB systems and IC Package designs. SPECCTRAQuest SI Expert allows users to explore and resolve electrical performance-related issues in all stages of the design cycle. By exploring and making trade-offs between timing, signal integrity, crosstalk, power delivery, and EMI, designers can optimize electrical performance and reliability before committing the final design for manufacture.

- Consists of three integrated pieces — signal and topology exploration, timing driven floorplanning, and transmission line and crosstalk simulation
- Full pre- and post-route signal integrity analysis can be performed on partially and fully completed designs
- Tight integration with Constraint Manager

**HIGH-SPEED PCB ANALYSIS**

High-speed PCB analysis consists of a variety of SPECCTRAQuest technologies aimed specifically at distinct high-speed design and analysis environments (see Figure 3). SPECCTRAQuest for PCB design is comprised primarily of SPECCTRAQuest SI Expert, which enables the digital design engineer to explore, simulate, predict, and characterize high-speed signal integrity requirements in all stages of the design cycle. SPECCTRAQuest SI Expert also includes a number of optional signal integrity libraries.

SPECCTRAQuest Power Integrity is an add-on option to SPECCTRAQuest SI Expert. It provides an integrated design and analysis environment for users who want to develop clean, noise-free power delivery systems.

SPECCTRAQuest for IC packaging places total integrated analysis capabilities into the hands of the package engineer early in the design cycle. It merges electrical and physical design into one easy-to-use environment, along with capabilities designed specifically to support IC package exploration, definition, and implementation.
“With intense time-to-market pressures faced by the industry, designers need flows that work seamlessly, and the Cadence PCB design environment fills this need.”

Harry Bartley
Applications Engineer, Tektronix

ALLEGRO PCB LAYOUT

The Allegro series of solutions enhance designer productivity during the placement and routing stages of PCB design. The more advanced Allegro solution offers support for electrical and high-speed design rules and methodologies managed through a revolutionary, spreadsheet-based Constraint Manager. Built-ins support high-speed design rules, real-time DRC set, shape-based interactive interconnect editing, and manufacturing output support for fab and assembly drawings. The Allegro series provides designers a production-proven environment for complex PCB design.

- High-speed constraint driven PCB layout and routing enables clear communication of electrical and timing design intent between engineer and designer
- IntelliUSE interactive routing supports push, shove, hug, and heads-up DRC for reduced design time
- Quickplace component and reuse module floorplanning reduces component placement cycles by as much as 50%
- Integrates with Constraint Manager to manage high-speed constraints across entire design flow
- Highly customizable to meet specific design needs

SPECCTRA AUTOROUTER

SPECCTRA® placement and routing is designed to handle high-speed, high-density PCB systems that require complex design rules. SPECCTRA employs powerful, shape-based algorithms to make the most efficient use of the routing area delivering increased productivity and shortened design cycles. The Smart Route feature looks at a design and draws from a library of proven strategies based on the design’s context. It then configures and executes the router as required, leaving the designer with more time to focus on more complex design issues.

- Designed to handle high-density PCB designs and complex IC Packaging
- SPECCTRA autorouter option is the standard for high-speed PCB and MCM autorouting — a proven solution for large, dense, multilayer, high-speed boards
- Design rule set and functionality depth is the best in the industry
- According to independent benchmarks, SPECCTRA is the fastest autorouter with the highest completion rates for complex, constrained boards
THE NEED FOR INTUITIVE TOOLS

In today’s competitive market, small teams and individual board designers have little room for error. The challenge is to take a project to manufacturing as efficiently and as skillfully as possible. To be successful, you need a set of intuitive tools that allow you to move from schematic capture to simulation to layout and back again without any miscues along the way.

Orcad PCB Design has a long history of addressing the needs of individuals and small teams. Orcad PCB solutions deliver easy-to-use product capabilities at an unprecedented value. Today, over 160,000 PCB professionals enjoy Orcad PCB solutions worldwide.

INDIVIDUAL PCB DESIGN SOLUTIONS

Orcad PCB Design offers a complete, highly integrated package of design solutions (see Figure 4). Ideal for price conscious individuals or small organizations, Orcad PCB Design affords designers the ability to collaborate freely on any number of projects from prototyping to PCB design production. For increased functionality, users can scale to a more powerful high-speed, team-based PCB solution mixing and matching solutions as needs dictate — keeping the original investment intact.

- Includes the de-facto standards — Orcad Capture® design entry for schematics and PSpice® simulator for analog and mixed-signal simulation
- Upgrade/scalability possibilities to high-speed PCB design allows improved productivity and IP protection
- Built-in component information system in Capture CIS saves up to 70% of time searching for correct information
“PSPice Advanced Analysis gives me the capability to quickly optimize my amplifier or filter designs in either the time or frequency domain. As a result, the time expended to improve a design has been decreased by up to 50 percent.”

Ruben Aguila
Engineer and Analog Designer, Litton Advanced Systems, Inc.

ORCAD CAPTURE DESIGN ENTRY

Because of its fast, universal design entry capabilities, Orcad Capture® schematic design is the world’s favorite design entry tool. Whether you’re designing a new analog circuit, revising the schematic diagrams for an existing PCB, or drafting a block diagram of HDL modules, Orcad Capture provides everything you need to complete and verify your design quickly.

- Intuitive interface and rich feature set makes Orcad Capture the de-facto industry standard for schematic entry
- Draft, view, and edit multiple schematics in a single session for true Windows multiprocessing capability
- Drag and drop schematic page between schematics and sessions to move and copy designs on the fly
- Includes a set of libraries with over 40,000 electronic parts and electro-mechanical symbols

CAPTURE CIS SCHEMATIC CAPTURE

Capture CIS is a comprehensive capture and management environment that offers powerful schematic capture capabilities. It features a built-in component information system (CIS) that reduces the amount of time spent researching and manually entering part data.

- Eliminates miscues by instantly classifying a selected part as obsolete or disapproved
- Easily access component information from a centralized part database and incorporate data into a company MRP/ERP system
- Validates parts in the design automatically with visual indication and notations of changes
**PSPICE ANALOG AND MIXED-SIGNAL SIMULATOR**

PSpice simulator is a full-featured analog and mixed-signal simulator supporting everything from high-frequency systems to low-power IC designs (see Figure 5). Advanced features enable “what if” analysis, allowing designers to explore various design configurations before committing to a specific implementation.

- The industry’s favorite analog and mixed-signal simulator
- Create designs, control simulations, and interpret the results within a single environment
- Choose from PSpice, PSpice A/D, and PSpice A/D Basics
- Select from a library containing over 14,000 analog and 1,600 digital devices
- PSpice Advanced Analysis option delivers higher yields, better reliability, lower component costs, and quicker time to market

**ORCAD LAYOUT PCB DESIGN**

Orcad Layout® solutions allow your team to create and electronically share PCB data across the design process. Start with complete component information from your preferred part database retrieved via Capture CIS, add design rules and constraints along with other essential properties — then exchange the information with your choice of solutions offered in the Orcad Layout series.

- Transmit design footprints, package types, and design rules
- Derive a quick and easy component placement adhering to the design constraints from Capture CIS
- Drive engineering changes rapidly from the schematic with the automatic ECO process and automatically back-annotate PCB changes to the schematic
- Route designs quickly with an automated router or use the more sophisticated SPECCTRA autorouter for more dense designs

Figure 5: PSpice plays a critical role in the OrCAD PCB design flow.
As silicon speeds increase, margins become tighter which means PCB professionals must now consider the interconnect from silicon to package to the printed circuit board. If you are a semiconductor company, you need to enable your customers to successfully design your chips into their PCB-based products. If you’re a systems company, you have to implement these chips on some very complex PCB systems. In some situations, you may be working with electronic manufacturing services (EMS) so you’re probably feeling pressure at both ends of the communication spectrum.

Because of the breadth and scalability of PCB Design solutions and tight integration with other Cadence technologies, PCB Design plays a key role in empowering the design chain whether you’re an IC, package, or board systems company.

Designing IC, package, and PCB concurrently can save significant time and money when compared to a traditional isolationist design flow. Product reliability can also be improved when all members of the design team reach across the design flow to consider IC, package, and board effects. Supporting this concept, Cadence PCB Design Solutions interface tightly with our IC Packaging technologies to allow for the design and analysis of the interconnect across all levels of the design, empowering a true silicon-package-PCB design chain.

For more information about Cadence PCB Design technology, log on to www.pcb.cadence.com. Or email us at pcbsalesinfo@cadence.com.
<table>
<thead>
<tr>
<th>CADENCE PCB DESIGN PRODUCTS</th>
<th>PRODUCT #</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Team-based PCB:</strong></td>
<td></td>
</tr>
<tr>
<td>Concept HDL</td>
<td>PX3000</td>
</tr>
<tr>
<td>SPECCTRAQuest</td>
<td>PX3100</td>
</tr>
<tr>
<td>PCB Librarian Expert</td>
<td>PX3500</td>
</tr>
<tr>
<td>Allegro</td>
<td>PX3700</td>
</tr>
<tr>
<td>SPECCTRA</td>
<td>PX3800</td>
</tr>
<tr>
<td><strong>Orcad PCB:</strong></td>
<td></td>
</tr>
<tr>
<td>Orcad Capture</td>
<td>PO1100</td>
</tr>
<tr>
<td>Capture CIS</td>
<td>PO1110</td>
</tr>
<tr>
<td>PSpice</td>
<td>PO1310</td>
</tr>
<tr>
<td>Orcad Layout</td>
<td>PO1410</td>
</tr>
</tbody>
</table>