

## ALLEGRO DESIGN WORKBENCH

The Cadence® Allegro® Design Workbench, an integral part of the Cadence Allegro system interconnect design platform, is a suite of products that provides a collaborative design environment to enhance the productivity of your entire design team. This integrated solution is specifically designed for board design groups, enabling cross-team collaboration, design and library management, and has been proven to increase engineering productivity up to 50 percent.

### THE ALLEGRO SYSTEM INTERCONNECT DESIGN PLATFORM

The Cadence Allegro system interconnect design platform enables collaborative design of high-performance interconnect across IC, package, and PCB domains. The platform's unique co-design methodology optimizes system interconnect—between I/O buffers and across ICs, packages, and PCBs—to eliminate hardware re-spins, decrease costs, and reduce design cycles. The Allegro constraint-driven flow offers advanced capabilities for design capture, signal integrity, and physical implementation. With silicon design-in kits, IC companies shorten new device adoption time and systems companies accelerate PCB design cycles for rapid time to profit. Supported by the Cadence Encounter™ and Virtuoso® platforms, the Allegro co-design methodology ensures effective design chain collaboration.

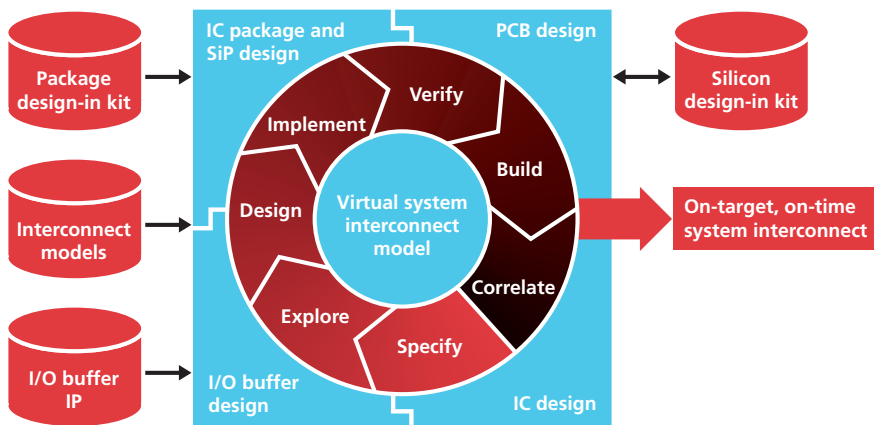


Figure 1: The Allegro system interconnect design platform

**Allegro PCB Design Workbench** — is a collaborative design and data management solution for board-level electronics design. Tightly integrated with Allegro Design Entry HDL, Allegro PCB SI, and Allegro PCB Editor, Allegro PCB Design Workbench provides a customizable work environment that lets you create standard design methodologies across different design disciplines. Parametric component search ties into your company's preferred components database, promoting the use of approved and preferred parts in your designs. Work-in-progress data management supports collaborative team design that lets you control changes, maintain history of design revisions, and provide access to the most current design data to all team members. To enable concurrent design, it also manages schematic and board files separately. Collaboration technology includes secure shared workspaces for local as well as globally dispersed design teams.

### BENEFITS

- Cuts training and support costs by providing a common user interface and design methodology across the enterprise
- Improves productivity of engineers, designers, component engineers, procurement, and others by expanding access to component information and design data
- Improves quality and reduces board spins by providing common access to 'known good' library data
- Enables concurrent design and reduces development time by managing and vaulting schematic and layout data separately during the design process

**Allegro PCB Collaboration Workbench** — is a secure, scalable workspace management solution that enables companies to use the Web to collaborate with internal and external partners, suppliers, and customers—regardless of geographic location. By enabling teams to work online in shared workspaces, the workbench reduces the time and cost spent searching for information that normally resides on a user's desktop. It also permits team members to view and mark-up designs and related documents without the need to tie up expensive CAD tool licenses and to return those comments and revisions to the design owner.

### BENEFITS

- Streamlines the design cycle by allowing non-CAD users to check-out, review and mark-up data, then check-in their mark-ups for review, by the design owner
- Speeds ECO resolution by saving all design revisions separately to create a log of mark-ups for future troubleshooting

**Allegro PCB Library Workbench** — is a library development and management environment that allows librarians to create, validate, manage, and disperse library parts and their associated data for use with Allegro Design Entry HDL, Allegro PCB SI, and Allegro PCB Editor. As parts are created or modified, the workbench automatically distributes the updated design libraries to company-specified design sites. This keeps all design centers up to date with the latest component and library information. The Allegro PCB Library Workbench incorporates all of the capabilities of the Allegro PCB Librarian and the Allegro PCB Design Workbench, allowing the librarian to act as a super-user. This permits the librarian to test the library elements in the same environment that is used in production and perform all of the tasks that a designer will perform when using the libraries.

### BENEFITS

- Reduces the time to create, validate, and manage large pin count devices—from days to minutes—by employing an all-encompassing librarian toolbox
- Eliminates design ECOs by verifying the accuracy of logical symbols and footprints using automatic library part validation
- Eliminates errors due to out-of-date or defective libraries by automatically synchronizing logical and physical reference libraries across the enterprise
- Increases volume purchasing power by eliminating redundant components and suppliers

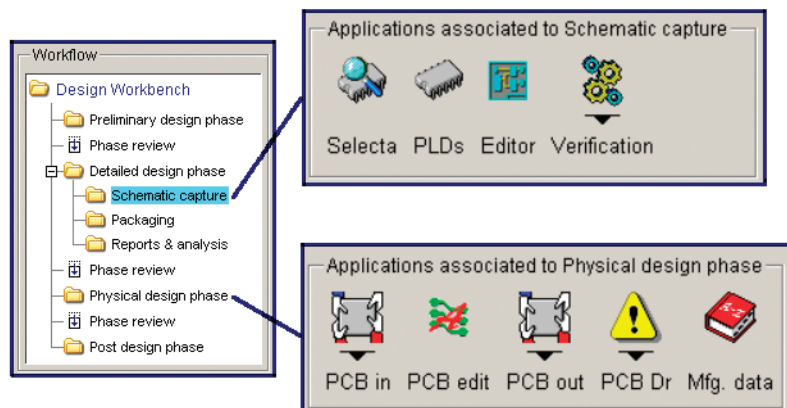


Figure 2: Customizable design flow brings design tools to your desktop at each stage of the design process

## FEATURES

### Customizable Work Environment:

Now you can set up a standard design methodology across your company by defining standard design flows across multiple types of designs, such as high-speed, analog, and prototype. Each flow is defined with access to appropriate design tools and aids for each step in the flow. These flows can act as a checklist that help shorten learning curves and make casual users more productive.

**Part Browsing:** The parametric component search ties into your preferred components database, providing access to approved and preferred parts. This helps lower costs and reduce inventory. Users can search and select parts from the database and view component data such as schematic symbols, PCB footprints, and component data sheets. The selected parts are used to build a preliminary BOM, from which they can be directly added into the schematic.

**Library Version Control:** The workbench notifies you when changes occur to libraries that are used in your design and allows you to decide when to accept a new part. The rollback capability keeps a copy of “old” parts and allows you to revert back to an old part if necessary. This feature works with both schematic and layout designs.

**Work-in-progress Data Management:** The Allegro PCB Design Workbench lets you manage the completed design as a whole as well as design data while in development, known as work-in-progress (WIP) data management. WIP data management gives all team members access to the most current design data. It further enables designers to control changes, keep history of design revisions, allow for what-if changes and, if necessary, roll-backward to previous versions. A key aspect of effective board-level design data management is the ability to manage the individual design files (BOM, schematic, and layout) separately and concurrently. This enables concurrent design and design reviews to proceed uninhibited by out-of-date or mismatched file versions.

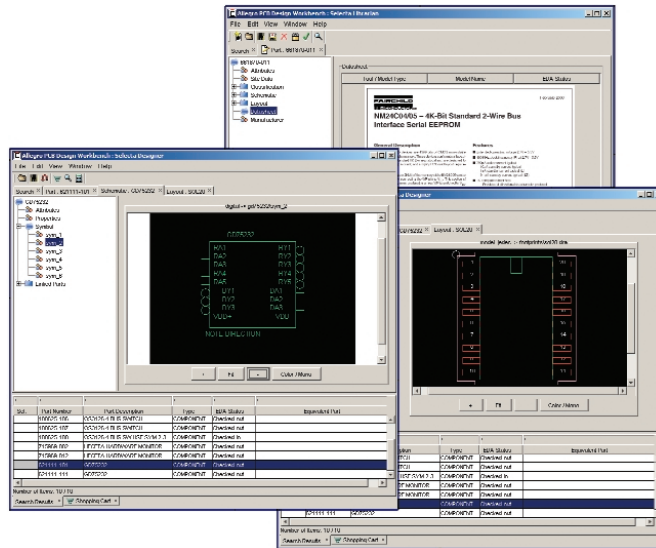


Figure 3: Parametric search brings all component data to your desktop

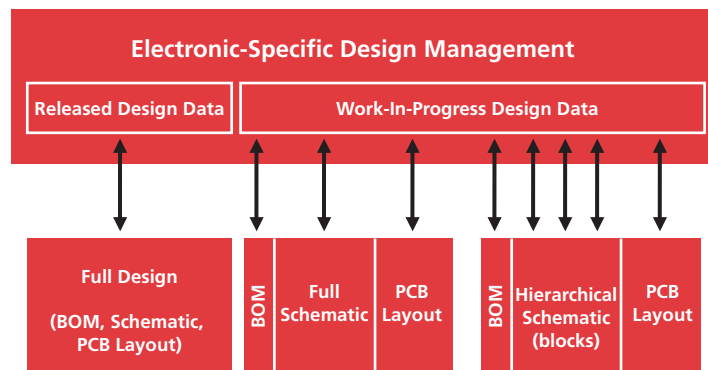


Figure 4: Work-in-progress data management permits concurrent engineering on the BOM, schematic, and layout

**Check-in and Check-out a Design:** All members of the design chain may securely check-out, review, and mark-up data and then check-in their mark-ups for review by the team. You control access to this data, so only those authorized to see the data can access it.

**BOM Management:** As parts are selected, they are automatically added to a preliminary BOM. This communicates design intent early in the design process, allowing component engineers and procurement to estimate cost, status, availability, and other parts-related information. You can promote the BOM to a WIP data vault for review and automatically route it to reviewers. The reviewers can mark it up and add comments, which are then routed back to the

owner (and others if desired) for review and approval. All transactions between engineers during the review are captured as part of the design history. After the logical design is completed and the BOM has quantities, it can be promoted again for review and mark-up where cost estimates can now be applied. Finally, when the layout is complete, the final BOM is stored together with the design for use in procurement and manufacturing.

**Secure Collaboration Workspace:** Design teams, manufacturing, and other partners can share design data, perform what-if scenarios and work together through discussion threads, meeting management tools, and instant messaging.

## Browse, Review, and Annotate

**Design:** Team members and partners do not need to have Design Entry HDL, PCB SI, or PCB Editor tools to review and annotate designs. The system's view and mark-up technology displays the design and lets design team members annotate and capture comments and mark-ups, and return them to the design owner. The design owner is notified when mark-ups are ready for review.

**Library Development Flow:** The ability to set up standard part creation methodologies in a graphical user interface streamlines the library development process. You can define standard flows for multiple types of parts, each with a different flow and access to different tools (e.g., schematic symbols vs. layout footprints). Selecting a step in the flow prompts the interface to display the tools and aids appropriate to that step. This can act as a checklist, creating a shorter learning curve, making part-time librarians more productive, and insuring consistency in part creation.

**Multi-site Library Distribution:** The workbench maintains a central master library of preferred parts and known-good library data that is automatically dispersed to various design sites by updating them on a predefined schedule. This keeps all design sites up to date with the latest library additions and changes, ensuring designers have access to the most current library data.

## SPECIFICATIONS

### SYSTEM REQUIREMENTS

- Software requirements
  - PSD 15.1
  - Java 1.4.2
  - Oracle 9i
  - IE 6.X or Netscape 7.x
- Hardware requirements
  - PC Client
    - 2.0Ghz Pentium processor
    - 1GB RAM memory
    - 30GB hard disk
  - PC Server
    - 2.0Ghz Pentium processor
    - 2GB RAM memory
    - 40GB hard disk (IDE or SCSI)
  - Unix Client
    - Sufficient to run Cadence design tools
  - Unix Server
    - UltraSparc II
    - 2GB RAM memory
    - 40GB hard disk

### PLATFORM/OS

- Sun Solaris 2.8
- Windows 2000 with Service Pack 3

### CAPACITY

- Up to 150 concurrent users per server

## CADENCE SERVICES AND SUPPORT

- Customer-focused solutions that increase ROI, reduce risk, and achieve your design goals faster
  - Collaborative approach and design infrastructure—virtual teaming
  - Proven methodology and flow tuned to your design environment
  - Design and EDA implementation expertise
- Product and flow training to fit your needs and preferred learning style
  - Over 80 instructor-led courses—certified instructors, real world experience
  - More than 25 Internet Learning Series (iLS) online courses
- Cadence customer support that keeps your design team productive
  - Cadence applications engineers provide technical assistance
  - SourceLink® online support gives you access to software updates, technical documentation, and more—24 hours a day, seven days a week

## FOR MORE INFORMATION

Contact Cadence sales at 1.800.746.6223 or visit [www.cadence.com](http://www.cadence.com) for additional information. To locate a Cadence sales office or value-added reseller (VAR) in your area, visit [www.cadence.com/contact\\_us](http://www.cadence.com/contact_us).

CONFIGURATION MATRIX	PCB DW	PCB CW	PCB LW
Integrated with Design Entry HDL, PCB SI, and PCB Editor	X		X
Customizable work environment	X		X
Part browsing	X		X
Library version control	X		X
Work-in-progress data management	X		X
Check-in and check-out a design	X	X	X
BOM management	X	X	X
Secure collaboration workspace	X	X	X
Browse, review, and annotate design		X	X
Library development flow			X
Multi-site library distribution			X

© 2004 Cadence Design Systems, Inc. All rights reserved. Cadence, the Cadence logo, Allegro, SourceLink, and Virtuoso are registered trademarks, and Encounter is a trademark of Cadence Design Systems, Inc. All others are properties of their respective holders.

5528 05/04