

ALLEGRO DESIGN WORKBENCH

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

ALLEGRO PCB DESIGN WORKBENCH

Allegro PCB Design Workbench is a collaborative design and data management solution for PCB design. Tightly integrated with Allegro Design Entry HDL, Allegro PCB SI, and Allegro PCB Editor, Design Workbench provides a configurable work environment that lets designers create standard design methodologies across different design disciplines. This promotes the use of common design tools and best practices across design teams.

Parametric component search ties into your company's preferred components database, promoting the use of approved and preferred parts and reducing component research time by as much as 75 percent. The part request system allows designers to submit, manage, track, and query status of new component requests and engineering change orders (ECOs) for existing parts.

Work-in-progress (WIP) data management supports collaborative team design that lets users control changes and maintain history of design revisions, and it provides access to the most current design data to all design team members. "Where-used" functionality lets designers see where components are used in production designs and it manages schematic and board files separately to facilitate concurrent design. Collaboration technology includes secure shared workspaces for both local and globally dispersed design teams.

BENEFITS

- Improves quality and reduces board spins by providing common access to "known-good" library data
- Eliminates design errors due to out-of-date or defective libraries by interactively synchronizing logical and physical reference libraries with logical and physical design projects
- Enables concurrent design and reduces development time by managing and vaulting schematic and layout data separately during the design process
- 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

ALLEGRO PCB COLLABORATION WORKBENCH

Allegro PCB Collaboration Workbench is a secure design collaboration solution that enables design teams to share data and ideas with internal and external partners, suppliers, and customers—regardless of geographic location. By enabling teams to work online in shared workspaces, Collaboration Workbench reduces the time and cost spent searching for information that normally resides on an individual's desktop. It also permits team members to view and mark up designs and related documents without the need for expensive CAD tool licenses and, when done, to return those comments and revisions to the design owner. All exchanges between engineers during design reviews are captured and stored as part of the design history.

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

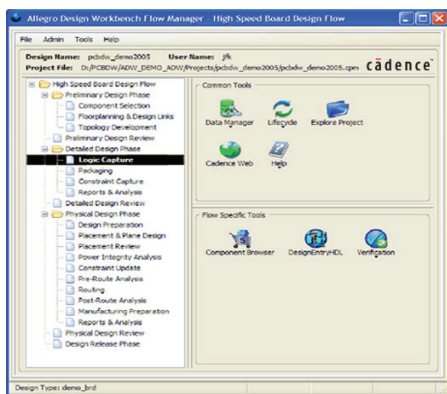


Figure 1: Configurable design flow guides users through the design process and brings appropriate design tools to the user's desktop in each step of the process

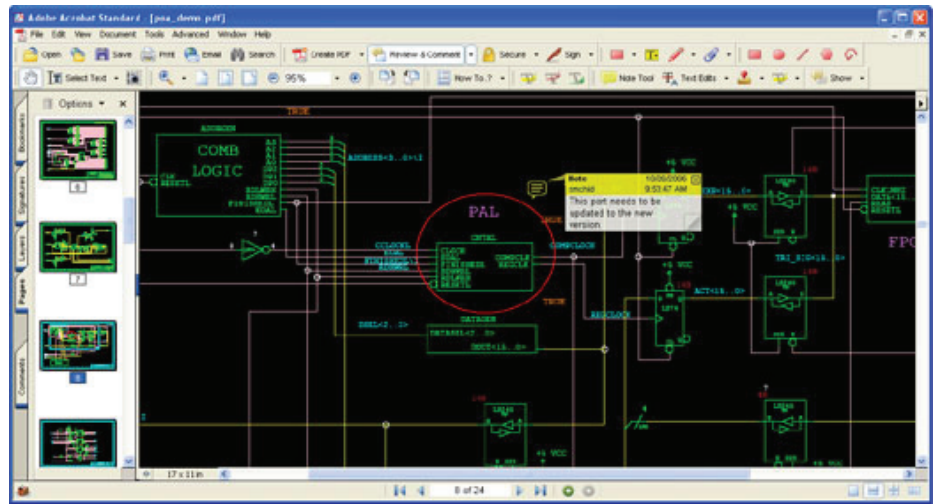


Figure 2: View and mark up schematic and PCB files

ALLEGRO PCB LIBRARY WORKBENCH

Allegro PCB Library Workbench is a library development and management environment that enables PCB librarians to create, validate, manage, and distribute 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, Library Workbench automatically creates revisions and distributes the updated design libraries to company or specified design sites. This keeps all design centers up to date with the latest component and library information. Library Workbench works in conjunction with Allegro PCB Librarian and incorporates all of the capabilities of 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
- Decreases design ECOs by verifying the accuracy of logical symbols and physical footprints using automatic library part validation

- Eliminates design errors due to out-of-date or defective libraries by automatically synchronizing logical and physical reference libraries across the enterprise
- Increases librarian productivity and company purchasing power by eliminating redundant components and suppliers

FEATURES

Configurable work environment

Set up a standard design methodology across the company by defining standard design flows across a variety of design types, such as standard, 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 helps shorten learning curves and makes casual users more productive, ensuring that important steps and checkpoints are not missed.

Component browsing

Parametric component search ties into your company's preferred components database, providing access to approved and preferred parts. This helps lower costs and reduce inventory. Users can search and select parts based on parametric and business data and view schematic symbols, PCB footprints, and component data sheets during the selection process. The selected parts

are used to build a preliminary bill of materials (BOM) from which they can be added directly into the schematic.

Library version control

Allegro Design Workbench notifies users when changes to the design's library parts occur and allows users to decide when to accept a new part. The rollback capability allows users to revert to a previous part version if necessary. This feature works with both schematic and layout designs.

WIP data management

Allegro Design Workbench lets users manage the design while in development—known as work-in-progress (WIP) data management—as well as manage the completed design when released to production. WIP data management gives all team members access to the most current design data through a secure data vault. It further enables designers to control changes, keep a history of design revisions, explore what-if changes, and, if necessary, roll back 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 without fear of using out-of-date or mismatched file versions.

Check in and check out a design

All members of the design team may securely check out, review, and mark up designs and then check in their mark-ups for review by the team. All exchanges between engineers during design reviews are captured and stored as part of the design history. Users control access to this data, so only those authorized to see the data can access it.

Where-used functionality

As part status changes, or for order planning and inventory control, where used functionality allows users to see where components are used in production designs as well as WIP designs. It includes information about which designs use the specified component and the reference

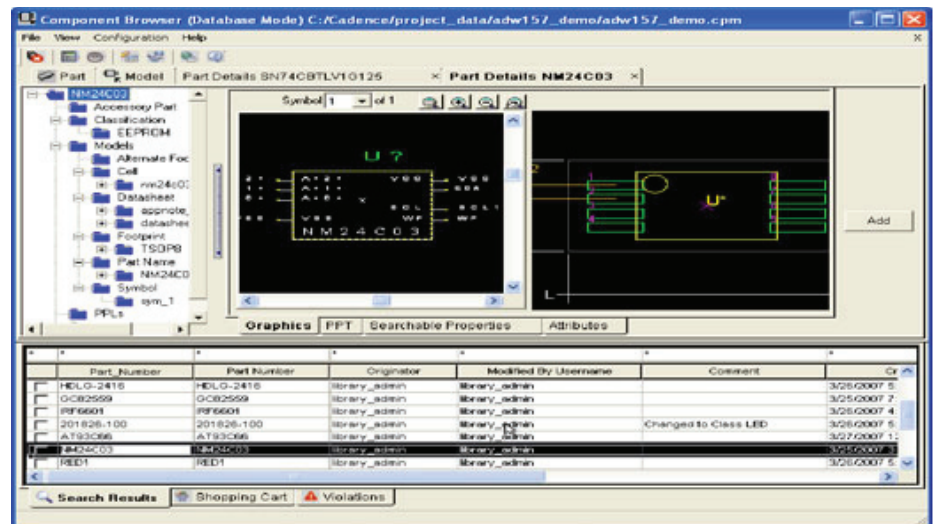


Figure 3: Parametric search brings all component data to the desktop

designators for the component in each design. This allows users to quickly assess the impact of component changes as well as plan for production needs.

BOM management

As parts are selected, they are automatically added to an initial 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. Users 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.

Browse, review, and annotate design

Team members and partners do not need to have Allegro Design Entry HDL, Allegro PCB SI, or Allegro PCB Editor tools to review and annotate their designs. Schematic PDFs and native board files

(.brd) can be viewed and annotated, capturing comments and mark-ups and returning them to the design owner. The design developer is notified when mark-ups are ready for review and can access them through the project workspace in the design vault.

Library development flow

The ability to set up standard part creation methodologies through a graphical user interface streamlines the library development process. Users 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 displays the tools appropriate for that step. This acts as a checklist, creating a shorter learning curve, improving productivity, and ensuring consistency in part creation. Library verification steps, with their appropriate tools, are built in to the flow to facilitate rapid verification of components.

Multi-site library distribution

Allegro Design Workbench maintains a central master library of preferred parts and associated known-good library data that is automatically distributed to various design sites as new parts and updates appear in the library. This keeps all design sites up to date with the latest library

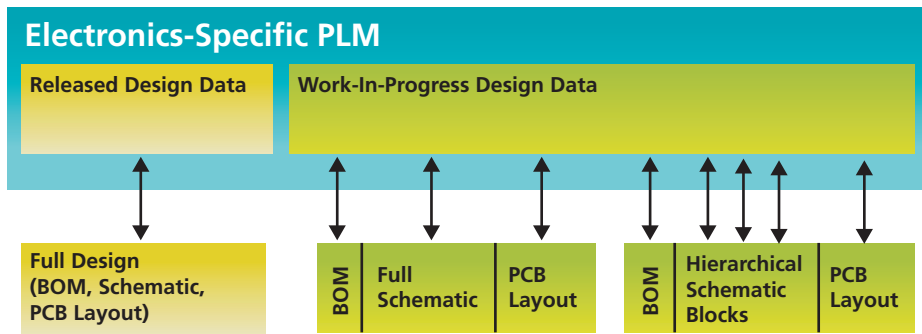


Figure 4: WIP data management permits concurrent engineering on the BOM, schematic, and layout

additions and changes, ensuring that all designers have access to the most current library and component information.

Regulatory compliance

Regulatory compliance directives (such as RoHS) are a top concern for electronics designers. Allegro Design Workbench captures RoHS and other regulatory compliance component information, making it searchable in the component browser. Designers can search for compliant parts in the library or specify a preferred parts list that contains only compliant parts. Where-used functionality can be employed to find non-compliant parts in previously completed designs.

Part request system

The part request system permits users to request new parts and ECOs to existing parts. It provides complete traceability of component history and permits designers to browse part requests, query status, and navigate between part requests and parts/models. It is configurable to

company-specific requirements and methods. The part request system can also be configured to trigger notifications to team members as part status changes. Additionally, part request supports electronic sign-offs.

Scalable solution

Allegro Design Workbench is available in both a library management configuration (Allegro Design Workbench XL) and a library and design data management configuration (Allegro Design Workbench GXL).

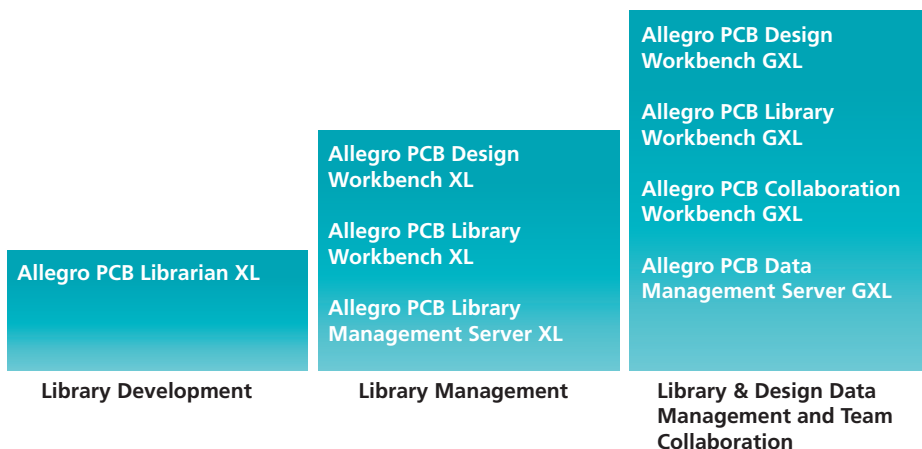


Figure 5: Allegro Design Workbench offers a scalable solution

SPECIFICATIONS

SYSTEM REQUIREMENTS

- Software requirements
 - SPB 15.7 or 16.01
 - Java 1.4.2
 - Oracle 9i/10g (XL or GXL) or MySQL 4.1.7 (included in XL)
 - Enovia MatrixOne 10.6.2 (GXL only)
 - IE 6.x or Firefox 1.5
- Hardware requirements
 - Client configurations
 - 2.0GHz Pentium processor
 - 1GB RAM memory
 - 30GB hard disk
 - Server configurations
 - Windows 2003
 - 3.0GHz Pentium processor
 - 4GB RAM memory
 - 100GB hard disk (IDE or SCSI)
 - Solaris 8, 9, 10
 - Sun Ultra 45 UltraSPARC IIIi CPU
 - 8GB RAM memory
 - 250GB SATA hard disk

PLATFORM/OS

- Windows XP Pro
- Windows 2003
- Solaris 8, 9, 10

CADENCE SERVICES AND SUPPORT

- Cadence applications engineers can answer your technical questions by telephone, email, or Internet—they can also provide technical assistance and custom training
- SourceLink® online customer support gives you answers to your technical questions—24 hours a day, 7 days a week—including the latest in quarterly software rollups, product release information, technical documentation, software updates, and more
- Cadence certified instructors teach more than 80 courses and bring their real-world experience into the classroom
- More than 25 Internet Learning Series (iLS) online courses allow you the flexibility of training at your own computer via the Internet

FOR MORE INFORMATION

Contact Cadence sales at 1.800.746.6223 or visit 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

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