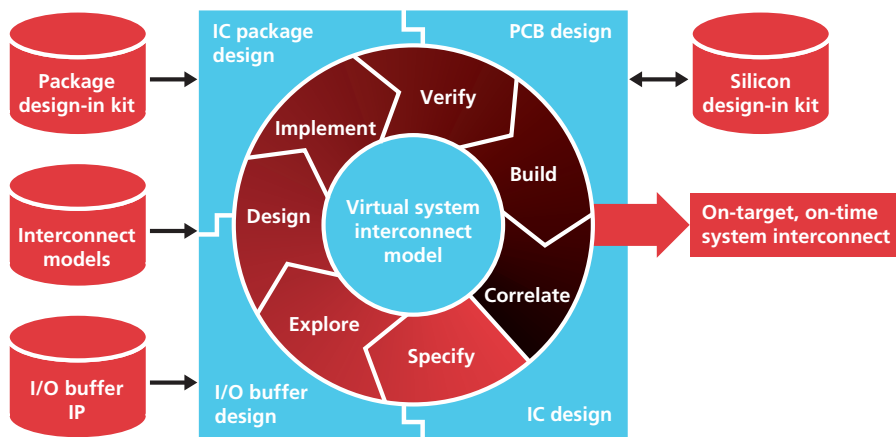


# ALLEGRO PCB LIBRARIAN 610

**AUTOMATED LIBRARY PART CREATION, VALIDATION, AND MANAGEMENT**

Cadence® Allegro® PCB Librarian 610, a 600 series product within the Allegro system interconnect design platform, is an advanced library development environment. It provides an all-encompassing library toolbox that speeds up the creation, validation, and management of large pin-count EDA library parts—from days to minutes. Allegro PCB Librarian automates part creation tasks and eliminates manual validation.



*The Allegro system interconnect design platform*

## 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.

## ALLEGRO PCB LIBRARIAN 610

With large pin-count devices becoming the norm for off-the-shelf components, the creation of new parts by librarians can take hours when you add up the time needed to manually create each graphical pin, associate each logical pin to a physical number, and then assign each pin to a slot within the packaged device. The resulting bottleneck could easily be reduced to minutes per part, or even seconds per pin, by automating part creation tasks and eliminating manual validation.

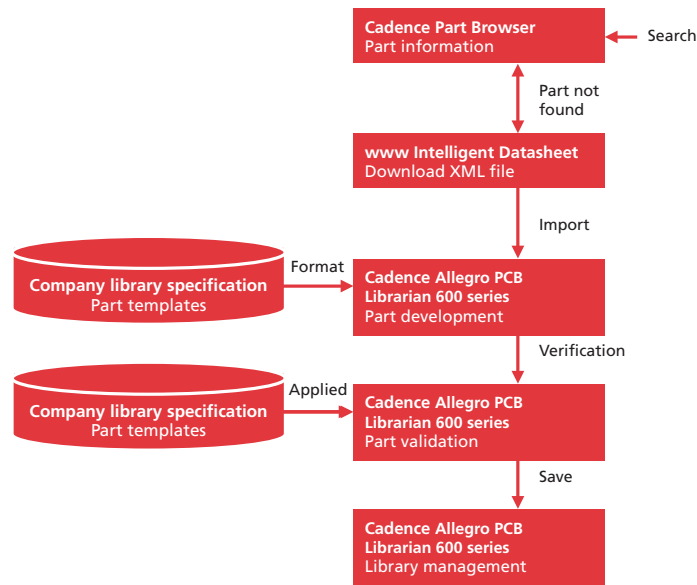
Allegro PCB Librarian 610 streamlines library part creation and validation to speed up the design process. Librarian 610 reduces the pressures normally associated with the library creation process—regardless of the demands created by the engineering team.

## BENEFITS

- Enables librarians to efficiently develop and maintain consistent libraries using XML data-driven symbol-generation technology
- Reduces symbol creation time by automatically interpreting Internet-derived pin and package data, eliminating time-consuming re-entry
- Ensures current and quality reference data from part manufacturers by leveraging the timeliness of the Internet
- Enhances the ability to copy various data formats from PDF datasheets using Paste Special to insert pin information directly into Allegro PCB Librarian 610, while enforcing error checking routines during importing

## KNOWLEDGE-DRIVEN PART DEVELOPMENT

The ability of Allegro PCB Librarian 610 to automatically leverage component information published on the Internet drastically reduces the amount of time it takes to develop a part. Adherence



*The Allegro PCB Librarian 600 series is the complete solution for the development, validation, and management of CAD data*

to your library standards during part development ensures correct, consistent, and informative CAD libraries.

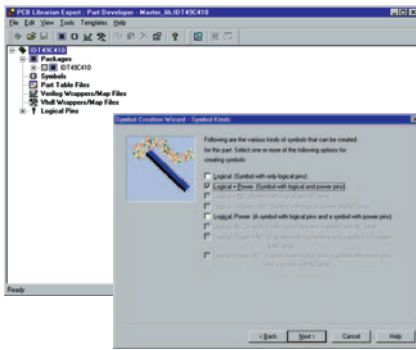
- Accelerates the development of CAD libraries by leveraging original Internet-driven source datasheets
  - Increases overall library accuracy by reducing the amount of manual data re-entry, which is susceptible to human error
  - Consolidates all the information, best practices, and tools needed to generate CAD libraries into one complete process
- By harnessing the power of Librarian 610, librarians around the world can generate accurate library parts quickly for use with Allegro Design Entry HDL and Allegro PCB Editor.

## CORRECT BY SPECIFICATION

All companies strive to promote consistent and correct library content.

However, most library specifications are manually applied to the library creation process, making it difficult to realize these goals. With Librarian 610, you can change your library specification easily from a manual post-process to an automated and consistently applied pre-process.

- Promotes standardized, reusable library content by automatically applying corporate or industry specifications during the creation of the library elements
- Imposes standards reflecting your company's library creation specifications
- Ensures integrity across all library elements by enforcing consistent formatting
- Embeds specifications, reducing the learning curve and increasing the productivity of new employees

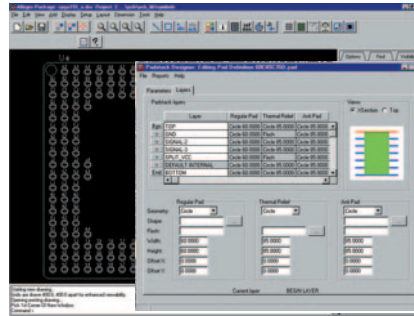


Symbol Wizard quickly guides you through the creation process

## ALLEGRO DESIGN ENTRY HDL SYMBOL CREATION

Allegro Design Entry HDL parts are automatically generated from the Internet-derived source datasheet information. Schematic symbols, physical part tables, bill-of-material parameters, and logical pin-to-physical-pin associations are easily automated, freeing librarians to put the finishing touches on each view to ensure a quality result.

- Accelerates the time required to derive schematic symbol representations by using XML data-driven symbol creation
- Allows automatic formatting or conversion of logical pins—from scalar to vector pins or vector to scalar pins—and package information to meet library specifications
- Minimizes project delays by rapidly creating schematic symbols, physical pin data, and part table data
- Ensures success during logical design when using a Cadence Verilog® or VHDL-based simulator by auto-creating the simulation wrappers and pin mappings
- Speeds symbol creation with a user-definable template and a form-driven interface
- Defines pins quickly by using pin form definitions that handle pin ranges
- Provides expanded formats for the creation of Allegro Design Entry HDL parts, making it easier to import power pins, assertion characters, and other library data formatted within a



Padstack Editor provides visual confirmation and efficient form editing

comma-separated value file, and also allows mapping of data columns

- Leverages HDL models to drive part creation—creates parts utilizing port information within Verilog, VHDL, and Synopsys PTM models
- Structures information for creating library parts within a spreadsheet-like UI—users have configurable formats for importing data, and it simplifies the development, modification, and organization of associated symbol data (including the partitioning of large pin-count packages into multiple heterogeneous or asymmetrical symbols)
- Manages subsequent data import and ongoing ECOs of parts to minimize rework and track modifications made after original part creation
- Streamlines property editing through the use of predefined property locations, the ability to generate templates, and the addition of a wizard-like command structure that guides the user through the required tasks

## ALLEGRO PCB EDITOR PADSTACK AND FOOTPRINT CREATION

The physical symbol part creation process is focused on increasing productivity during the generation of your Allegro PCB Editor footprints and padstacks in accordance with your manufacturing technology requirements. It is specifically designed to simplify the ever-increasing pin-counts and package complexity in today's designs.

- Simplifies the creation of complex, high pin-count parts—such as BGAs and PGAs—by providing a wizard utility that efficiently guides the user through the step by step footprint creation process
- Eliminates the need to generate test designs and decipher textual reports during footprint creation by allowing users to interactively define, check, review, and edit multilayer padstacks within a single interface
- Provides consistent and comprehensive padstack definitions using centralized parameters that enable a single visual feedback of cross-section and top-view data
- Optimizes footprint creation to suit the soldering type (wave, reflow) and assembly considerations, such as through-hole or surface mounted parts on one or both sides of the board
- Fully integrates 2D drafting and dimensioning
- Quickly defines design constraints such as drawing details, cross sections, and other manufacturing output parameters

## PART VALIDATION

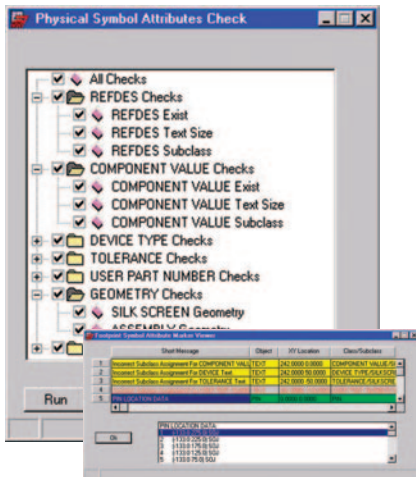
Allegro PCB Librarian 610 allows you to write and compile custom program and systems rules, and includes a user interface for creating rule files and executing rules.

Using the part validation system in Librarian 610, you can flush parts through an advanced rule checking and advisement system prior to release. This reduces costly design iterations by detecting a broad range of errors before they ever reach the design-in stage.

- Ensures parts are consistent and error-free using built-in error checking—quickly catches errors and prevents costly rework
- Guarantees that librarians always validate using configurable guidelines and limits the amount of manual intervention to reduce setup time
- Checks for incorrect constraints by discovering errors up front, before

library elements are released for general distribution

- Highlights errors with markers—user controllable parameters
- Verifies data flow through tools, assuring information is passed correctly from schematic to layout, schematic to simulation, part package, and backannotation information
- Ensures availability and validity of all required model views using process testing
- Delivers an extensive set of rules out of the box
- Compares interfaces to help control debugging differences
- Reduces dependence on script-based verification with online verification process, including final save



Process testing ensures the availability and validity of all required library parts

## LIBRARY MANAGEMENT

Librarian 610 features a built-in control for library data, preventing unmanaged or unauthorized changes, which allows for revisions to be managed effectively. A central master library with consistent and comprehensive library data frees your engineers to focus on what they do best—creating next generation products in a fast, error-free flow.

- Delivers controlled work-in-progress access to library data for creation and modification of individual library elements

- Eliminates library errors caused by colliding efforts
- Provides security of release libraries and access for all users
- Brings schematic logical representations, physical footprints, mechanical board outlines, simulation, and modeling information under centralized control
- Delivers the right library information to the fingertips of every engineer throughout your organization
- Ensures data integrity and reuse of library elements
- Automatically tracks changes made between part versions
- Intelligently identifies, tracks, and shows the details of changes made between different revisions of the same part

## PROJECT AND FLOW MANAGEMENT

Librarians face ever-increasing pressure to find new ways to maximize their productivity. Many companies attempt to incorporate productivity gains in flows and methodologies that, in reality, rarely get opened, updated, or used. New employees learn the existing procedures from their peers and teach this same information to their replacements. These are ineffective and costly training methods completely counter to best practices or maximizing productivity from librarians.

By using Allegro Project Manager as part of Librarian 610, you can better manage your library process creation. In addition, you can customize your flows and tailor them to unique aspects of your business model.

- Streamlines the library development process by utilizing procedural steps captured in a graphical user interface
- Decreases the cost of your library creation through the automated deployment of your best practices
- Speeds up your library process to deliver faster service to your engineers

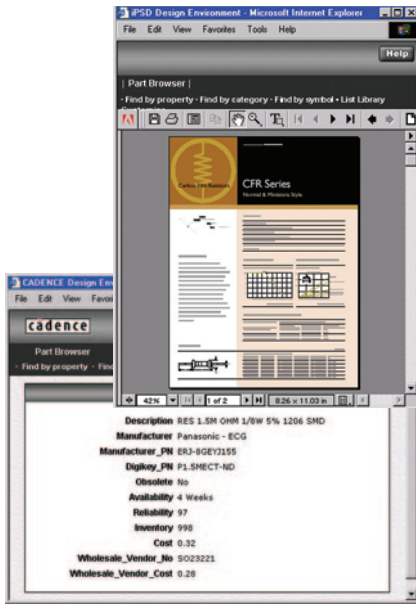


Project Manager provides easy access to the library development and validation process

## VENDOR INDEPENDENCE

Due to acquisitions and industry consolidations, the proliferation of vastly different vendor CAD tools throughout an organization is a reality for most corporate librarians. To support these third-party tools, Allegro PCB Librarian 610 has the ability to export to an XML-based Universal Data Format. Using this open format, any library creation process can be jump-started using pre-defined pin and package data. In addition, Librarian 610 is capable of generating symbols from .csv files with a flexible format, making it easy to support multiple CAD systems. Librarian 610 also exports symbols and validation of Allegro PCB Editor footprints. It can also import and export library part data from OrCAD Capture® Design Entry, effectively translating and validating OrCAD to Allegro Design Entry HDL symbol and chips data.

- Improves library consistency and reusability for all vendor tool processes by using a common data source for library creation
- Leverages the open standard, which reduces duplications and improves productivity
- Maximizes the reuse of existing CAD library IP while still ensuring optimization and validation to a third-party vendor design flow
- XML supports platform-independent protocols for the exchange of data
- XML is license-free and well supported—making it easier to develop software to handle specialized information



*Part Browser automatically retrieves part data for you, quickly and easily*

## COMPONENT INFORMATION WITH PART BROWSER

The web-enabled Part Browser publishes key component information to the desktop, allowing the engineer to make knowledgeable decisions regarding part selection. Electrical characteristics, CAD library data, datasheets, application notes, life-cycle status, and the quality, reliability, cost, and component availability are all readily accessible. Part Browser serves as the front end for the entire design process—streamlining part selection activities and saving you time.

Part Browser works with any JDBC-compliant MRP, ERP, or PDM system or database. The flexibility and scalability of Part Browser supports small design groups as well as geographically dispersed teams.

- Links CAD library and business data directly to the Allegro Design Entry HDL user, so component selection decisions are made using correct and current information
- Makes schematic creation more efficient and design data generation easier by automating part selection and part data entry
- Easily finds and reuses your company's pre-approved parts—those in stock and readily accessible—eliminating unnecessary new part introduction
- Identifies the part that best meets your design requirements and places the graphical symbol directly onto your schematic page—complete with all the appropriate property data with no further searching, selecting, or manual typing
- Previews the footprint, reviews the manufacturers' datasheets, double checks current stock levels and availability, and views many other purchasing and manufacturing considerations, reducing the risk of last minute surprises
- Integrates your choice of database for optimal flexibility

## OPERATING SYSTEM SUPPORT

- Red Hat Linux 7.3, 8.0
- Windows 2000 with Service Pack 2, XP Professional
- Sun Solaris 7, 8, 9
- HP-UX 11.0, 11.11i
- IBM AIX 5.1

## CADENCE SERVICES AND SUPPORT

- Cadence application engineers can answer your technical questions by telephone, email, or Internet—they can also provide technical assistance and custom training
- Cadence certified instructors teach over 70 courses and bring their real-world experience into the classroom
- Over 25 Internet Learning Series (ILS) online courses allow you the flexibility of training at your own computer via the Internet
- 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 change release information, technical documentation, solutions, software updates, and more

## 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).

## Allegro PCB Librarian 610 features

Primary feature	Secondary feature	Allegro PCB Librarian
Allegro Project Manager	Library project creation	•
	Library development workflow	•
Library management	Work-in-progress management	•
	Centralized visibility in library data	•
	Source control check-in/out	•
Vendor independence	Universal data format	•
	Import and export edaXML	•
	Import Si2 PinPak Std	•
Web based	Part Browser	•
	Internet-derived part data	•
Orcad Capture Library	Import "OLB" symbol format	•
Allegro Design Entry HDL part development	Manual	•
	Automated	•
Enforced library specifications	Validation rules part templates	•
Wrappers and pin map creation	Verilog	•
	VHDL	•
Allegro PCB Editor part development	Padstack creation and editing	•
	Footprint creation and editing	•
	Footprint Creation Wizard	•
Validation	Manual	•
	Automatic	•
	Custom rules	•
	Single part or entire library	•
	Conformity to part templates	•
	Verification of packaging flow from Allegro	•
	Design Entry HDL to Allegro PCB Editor	•
Component information management	Browsing of parametric data	•
	Graphical views of Allegro Design Entry	•
	HDL and Allegro PCB Editor parts	•
	Access to datasheet	•
	JDBC support	•
	RDBMS support	•
	Access to MRP/ERP business data	•
Additional licenses	Part Browser	(web download only)
	Allegro Design Entry CIS	• (NT only)
	Allegro PCB Editor	•
	Allegro Design Entry HDL	•
	CheckPlus	•
	Concept libraries	•
	Verilog simulation models	•
	PCB interfaces	•
	Mechanical interfaces	•