

PCB Libraries for Cadence Allegro and OrCAD

With powerful IPC-7351A viewing and calculation tools

PCB Libraries for Cadence Allegro

- Over 14,000 common prebuilt surface mount parts
- Includes LP Wizard Suite

PCB Libraries for Cadence OrCAD PCB Editor

- Over 6,000 common prebuilt surface mount parts
- Includes LP Wizard Suite

PCB Libraries for Cadence OrCAD Layout

- Over 6,000 common prebuilt surface mount parts
- Includes LP Wizard Suite

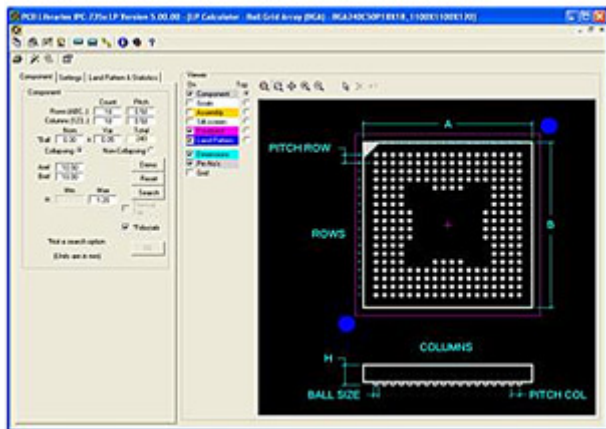
LP Wizard Suite

The IPC-7351A PCB Libraries LP Wizard Suite is a series of powerful library maintenance tools that use IPC and industry-approved mathematical algorithms and component orientations to calculate footprint (land pattern) dimension data from any manufacturer's component dimensional data. Build standard SMT library parts in just minutes!

The LP Wizard Suite is included with both PCB Libraries for Cadence® Allegro®, Cadence OrCAD® PCB Editor and Cadence OrCAD Layout. This tool suite will revolutionize the way you work, allowing you to quickly and easily browse your PCB footprint (land pattern) library to locate, view, construct and verify the mechanics of your PCB library design.

The IPC-7351A LP Software Suite is especially beneficial if you:

- Spend more time than necessary to match up PCB components with footprints
- Consider the current name of your footprint to be all the documentation you need to have,
- Are tired of laying parts on a 1:1 plot of your printed circuit board to verify that the footprints you've selected are the right ones (if you see the difference between 0.1mm and 0.2 mm)
- Design your product with its operating environment in mind
- Want to improve the bottom line by making PCB design more efficient



This suite provides you with extensive flexibility and control. For example, when creating complex ball grid arrays (BGAs), the software gives you the ability to control individual pins. You can add or remove individual pins as well as ranges of pins—in any combination.

Advanced features include:

- Graphic display of component features aligned with calculated footprint
- Compensation variables for fabrication and assembly processes
- Preset operating environments or user-defined goals for footprint toe, heel and side protrusions
- Solder joint analysis calculations
- DRC protection from overlapping lands
- Automatic footprint name generation
- Save library documentation for later reference/retrieval
- Edit part attribute names and values
- Easily group parts by project
- Export footprints directly to Allegro or OrCAD

The LP Wizard Suite includes the following tools:

LP Viewer

With the LP Viewer you can easily search the library for the footprints you need, and view PDF and Web-linked datasheets. The viewer provides on-screen graphic display for most standard SMD components and lets you quickly browse library contents. The on-screen display includes component, description, footprint name, and links to the component manufacturer websites.

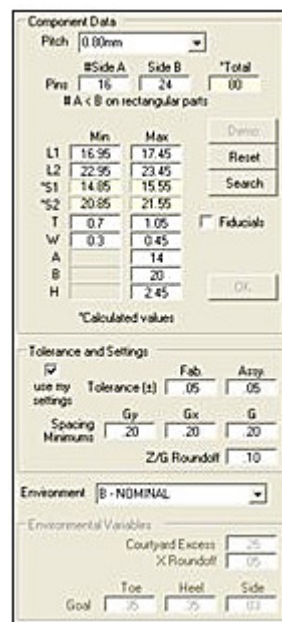
LP Calculator

- Calculates footprint geometries directly from component data with phenomenal precision
- Builds standard parts in minutes
- Reduces risk of human error
- Uses the default IPC settings or custom settings configured to match your specific manufacturing tolerances

The LP Calculator allows you to design PCB footprints by simply entering the manufacturer's component data; you can see the results of your design immediately in the viewer window and evaluate solder joint analysis.

With the calculator you can define your own CAD footprints if the correct CAD library part is not already available, and export to your CAD tool; it automatically builds CAD library decals in their native CAD format. You can change the manufacturing tolerances that determine the pad size, placement courtyard tolerance, PAD width round-off, component tolerance, footprint name, and the solder joint analysis. Alternatively, you can simply use the default IPC settings.

This product provides the pick & place zero component rotation for accurate assembly data. The JEDEC JEP95 publication and the EIA / ANSI 481 specification for tape and reel packaging was used to determine the exact component zero rotation on the assembly tape and reel or tube. You can build all your CAD library parts in the same zero rotation and provide your vendors with accurate assembly data.



LP Librarian

The LP Librarian allows you to save, store and reference component data to eliminate redundant work. Saved data can be shared with others using the LP Viewer.

The library documentation edit feature can be used to create libraries from scratch, modify individual attribute names and change IPC-7351A LP software-generated values based on your unique project needs. This feature provides

unprecedented flexibility in library management as you can now completely control what component data is stored in individual data files.

With the ability to group all parts of a project together in one data file, you can more easily keep your projects organized. It will also save engineers time because they don't have to sort through numerous data files to find all the parts of a given project. Simply group the parts you created for any project into a single data file, and share it with engineers. The data files can be browsed with the LP Viewer by anyone in your company!

If you build parts in one library and at some point decide certain parts should be copied to a different library, the software can easily transfer individual parts or groups of parts as you see most appropriate, making library management easier than ever.

The LP Librarian allows you to export data to Allegro, Altium Designer/Protel 99SE, Board Station, CADSTAR, CR5000, Expedition, OrCAD Layout, OrCAD PCB Editor, Pantheon, PADS, PADS ASCII, P-CAD, and Pulsonix.

System Requirements

MS-Windows XP
Microsoft .NET Framework (a free download)
1024 X 768 resolution or higher
11MB of available hard drive space

NOTE: the LP Wizard for Allegro requires your version of Allegro to have SKILL enabled, and will only work with Allegro v14.0 through v15.2.

Single license or site licenses available

SALES AND TECHNICAL SUPPORT

EMA Design Automation • 225 Tech Park Drive, Rochester NY 14623
Tel: 877.362.3321 • eMail: info@ema-eda.com • web: www.ema-eda.com