



OrCAD Library Builder

Advanced, Automated Schematic Symbol and PCB Footprint Generation

Overview

OrCAD® Library Builder provides advanced and highly automated capabilities for quickly creating part symbols and footprints for OrCAD PCB schematics and layouts. It virtually eliminates errors caused by manual processes, improving design quality and reducing rework.

Building PCB libraries of symbols and footprints has traditionally entailed a significant time investment using manual methods, which are prone to errors. As design cycles shrink and device pin count and complexity grow, manual methods place an unsustainable burden on the engineering team. Automated tools can help reduce PCB library development time and improve quality, allowing PCB design teams to focus on product differentiation.

OrCAD Library Builder delivers the most advanced PDF datasheet extraction, schematic symbol creation, and PCB footprint/land pattern automation. With OrCAD Library Builder, you can eliminate manual processes and work with your design team to create fully validated component libraries in a fraction of the time.

Accelerating Your Workflow

OrCAD Library Builder reduces down to just minutes the symbol and footprint creation processes that previously required hours, if not days, to complete. Purpose-built tools provide a single environment for extracting PDF data, managing and formatting pins, generating symbols, creating footprints, and validating outputs. Because the library creation process is automated, you get a dramatically accelerated workflow that delivers consistent, repeatable results.

Transforming a historically difficult, repetitive, and error-prone task into an intelligent, automated process requires advanced technology. OrCAD Library Builder contains industry-leading PDF data extraction technology that can pull component data from any PDF datasheet. OrCAD Library Builder can intelligently

Highlights

- Eliminate countless hours of tedious, manual entry with the most advanced PDF datasheet extraction technology
- Extract data from pin tables and diagrams to quickly build and manage complex symbols with thousands of pins
- Easily build complex footprints using a simple calculator-style interface
- Promote ECAD/MCAD collaboration with automated 3D model development
- Eliminate library part errors with comprehensive checking and verification tools

extract pin data from common datasheet elements such as: BGA maps, SOIC diagrams, QFP packages, as well as standard pin tables. With this capability, you no longer have to spend time manually copying and pasting content.

Correct by Construction

OrCAD Library Builder ensures your libraries are built correctly the first time by automating error-prone tasks and providing a complete set of error-checking and reporting utilities. With tight integration between the symbol and footprint builders, you can run comprehensive checks to ensure that the symbols and footprints match. The tool identifies errors and inconsistencies immediately, saving time and reducing costly rework. You can also create templates with OrCAD Library Builder. By applying these templates to the library creation process, you can easily ensure that all symbols and footprints developed adhere to common standards.

Symbol Creation

Saving time on symbol creation starts with advanced datasheet extraction. OrCAD Library Builder contains symbol generation capabilities which allow you to quickly and accurately extract datasheet specifications, format and check the data, and export this to the schematic tool.

Symbol formatting and checking is efficiently managed with tools for quickly inputting column assignments, removing unwanted data, and defining partitions. You are no longer required to partition the symbol set before building the symbols. You can make assignments of pins to interface groups, and assign pins at a higher level of abstraction. You can also run reports to check accuracy and completeness before generating the symbol for OrCAD Capture.

Footprint Creation

For PCB footprint generation, OrCAD Library Builder provides a comprehensive set of land pattern calculators which allow you to input dimensions directly from a datasheet. The calculators will then generate the resulting pads and pin geometries according to the IPC-7351 specification. A comprehensive list of component package calculators are available, including BGA, CHIP, CHIPARRAY, DIP, LCC, LGA, PLCC, QFN, QFP, SOJ, and SOP. Using this method, you can ensure your footprints will be consistent and always match the industry-standard IPC spec.

OrCAD Library Builder provides the ultimate placement control and editing of pads. You can select from D-shape, oblong, rectangular, or custom pad shapes, including rounding and chamfering specifications. Padstacks are user-configurable and can be assigned to unique locations, including specific corner pads or row and column locations. A comprehensive GUI allows you to see pad shape and placement, leads, dimensions, and more, so you can observe the exact effects of changing parameters.

3D Model Generation

Ensuring proper fit and clearance is an important part of the design process. However, acquiring realistic 3D models to perform model-accurate fit analysis can be a significant challenge. OrCAD Library Builder solves this problem with its 3D model generator. The dynamically generated 3D models are based on the exact dimensions of your footprints, allowing for fast and accurate 3D analysis in either your ECAD or MCAD design environments.

For the latest product or release information, visit us at www.orcad.com or contact your local Cadence Channel Partner.

Sales, Technical Support, and Training

The OrCAD product line is owned by Cadence Design Systems, Inc., and is supported by a worldwide network of Cadence® Channel Partners (VARs). For sales, technical support, or training, contact your local channel partner. For a complete list of authorized channel partners, visit www.orcad.com/CCP-Listing.