

## The Ultra Librarian 2015

### A single format, CAD tool-neutral library environment for...

- **An enterprise-wide library solution**
- **Access to large CAD neutral vendor part database**
- **Building a CAD/CAE library from scratch**
- **Merging multiple tool libraries in one standard set of libraries**
- **Converting an existing library in one EDA format to another EDA format**
- **Sophisticated capabilities to greatly increase speed, efficiency and accuracy in symbol and footprint generation and creation and maintenance of large libraries**
- **Inputs and outputs to a wide variety of PCB CAD/CAE tool environments**
- **Automated methods of collecting library data**

### Savings with Ultra Librarian:

- Customers have experienced time savings of 6 to 20 times depending on the CAD tool and features that are being used. Very significant time savings in multi-CAD system tool libraries have been achieved.
- Template-based component building allows adherence to Company or Industry library standards - library consistency has company and enterprise-wide benefits for your whole organization.
- New templates (for style parts not currently supported) can be created in under an hour. Most modifications take only minutes. Templates save hours of librarian time in generating new parts.
- Potential for enormous cost savings through a single library for all CAD systems with one set of consistent parts for all designers across the company, reducing duplicate parts building.
- Flexible part-customization for varying manufacturing specifications.
- Manual entry or creation of symbols significantly reduced and replaced with automated creation methods.

### KEY FEATURES OF THE ULTRA LIBRARIAN:

#### Component System

- The **Component Database System** provides access to information on a component or series of components, based on specific electrical parameters (Example: Select a resistor with 10 ohm impedance , 1/4 watt 5% in a 0805 case size). The database also provides a location for collecting information from the Internet and mating the results to correct footprint and symbol combination that contains all appropriate attributes.
- Can be interfaced directly into an existing company database allowing viewing and management of existing and newly created footprints and symbols.
- Automatically links to datasheet information on components, as well as industry websites for current information. Download an entire BOM (Bill of Material) of component spreadsheets without user interface. Select all existing components in your BOM, and report components not in the database.
- Intelligent linking of pre-existing footprints and symbols to new components. Verifications are automatic.
- Built in Revision Control for Components and Footprints.
- Built in user request and status / fulfillment database.

#### Footprint Database System

- The **Footprint Database System** provides access to physical parameters for a component or series of components. Search the database for components based on body or pin parametrics.
- Remotely access our library of over 120,000 different footprints including all JEDEC and IPC definitions.

#### Component Intelligence

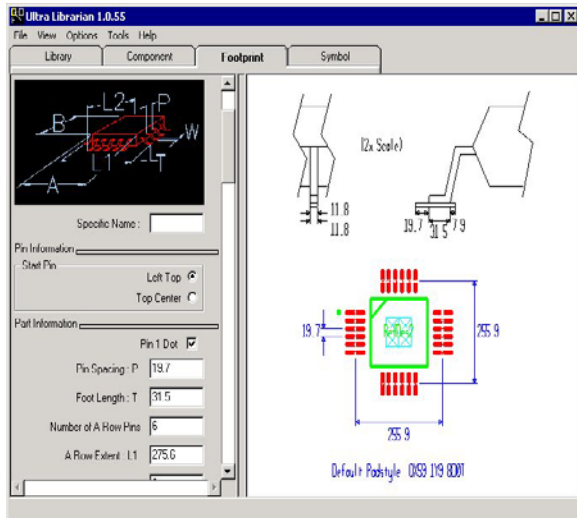
- Use wizards, spreadsheets, Internet search and data scraping tools to quickly create symbols and footprints that are properly linked together, supporting gate and pin swapping.
- Develop or manipulate the intelligence of the component, mapping the correct footprint and schematic symbols, its pin and gate swap capabilities, as well as attributes for describing a component.
- Cross probe and verification of symbol, footprint and mapping information simultaneously.
- Access over 7.2 million components already built from our on-line database of parts, with 82% now featuring dynamically created 3D STEP models.

#### Library Standardization

- Librarian definable standards via templates for footprint and symbol generation. Support multiple standards requirements from the same library data.
- Standardize and clean existing libraries. We provide routines for verifying and cleanup of the entire library, both from a physical and electrical viewpoint. Functions such as auto silkscreen removal from pads. Attribute verification and soldermask / solderpaste clearances.

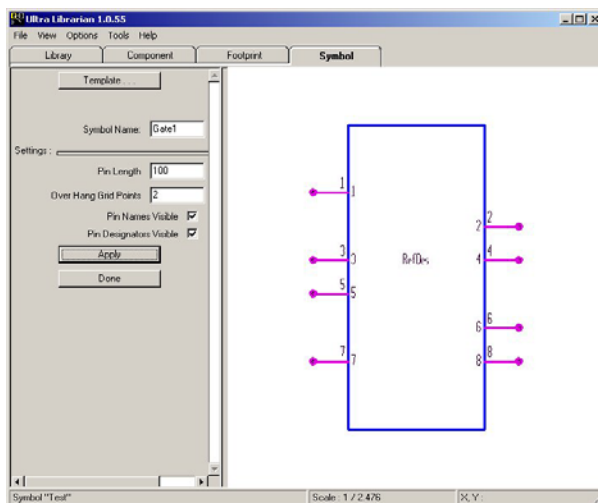
- Design Rule Check capability for the entire library, both physical and electrical properties.
- Convert existing libraries to a templated (and very standardized) library with just a few mouse clicks.

## Automated and Manual Footprint Creation



- Create footprints using a selection of over 550 user-definable templates.
- Use templates that require only the input of the component size, allowing the template to automatically calculate pad sizes and spacing, or use templates that allow you to specify the size of pads etc. As the librarian or designer create their footprints, the pad size and other critical dimensions are documented.
- Templates shipped with the product are based on IPC-SM-7351B.
- Custom-created specialized templates are available.
- Footprints can be created in English or metric units, and the resulting documentation are adjusted to deal with the different units of measure.
- Simple printing of symbols, footprints and related component information.
- Editing of footprints and symbols for "one of a kind" components!

## Automated and Manual Symbol Creation



- Symbols are created based on information entered into a spreadsheet window, including information on the pin function, pin grouping, and side of symbol. This window handles difficult gating information such as multiple non-similar gates. The table is built using a wizard-type interface that helps fill out the information required in an orderly, accurate manner. Verification is always one click away.
- Information regarding the pin types and locations, as well as groupings, is stored, such that busses can be grouped together as well as similar pin types.
- For some template types, power gates for entry on the schematic are also automatically generated.
- Several extract routines for scraping data directly from an Acrobat PDF file into a symbol.
- For many Components a symbol can be created with one keystroke, after creation of the footprint.
- Download of symbols from industry sources!
- Internal editing of auto generated symbols available!

## Library Translations

- Import and Export various CAD tool libraries including: Cadence Concept, Capture, Orcad Layout and Allegro, Mentor DA, Padslogic, PADS\_ Viewdraw/DX Designer, Boardstation and Expedition as well as Cadstar, P-CAD, \_Protel, and others.
- Import entities include the PCB footprints, schematic symbols and the intelligence that ties these together.
- Manage libraries by adding and merging components and their entities from one library to another.
- View the dependencies of each library entity to others.
- Intelligent cleanup routines for moving from one tool to another.
- User definable layer mapping and attribute mapping allow the user to create components in the target tool that has intelligent information.

## Authoring tools and Free Readers

- Ultra Librarian comes in a free reader format that will read data stored at a manufacturer or distributor web site. The reader will not allow users to make any changes to the data, but will allow them to export the collected data to many different CAD tools free of charge. Register for your free copy at <http://www.accelerated-designs.com/registration>.
- Authoring tools to create components to a particular standard, to import existing component, etc. are available in modular form. These tools are offered in a node locked or network licensed version. Pricing and available options and product comparisons are available at [www.accelerated-designs.com/pricing-information](http://www.accelerated-designs.com/pricing-information).
- Try an example of the free reader by downloading a part from Texas Instruments. <http://webench.ti.com/cad/>.