eCADSTAR™

DATASHEET

LIBRARY EDITOR

Connect to your preferred part vendor directly from within the eCADSTAR Library Editor application to lookup part information such as stock, pricing and datasheets. Download parts, symbols, footprints and 3D models and merge them to your library instantly with automated layer and attribute mapping.

The Library Editor, is a comprehensive unified design resource creation and management tool, put simply – it's more than just a part library.

With native editors for footprints, symbols, parts, pads and padstacks, create manage part data all from within the same application using a single Library Editor license. Create symbols, footprints, pads and padstacks parametrically for fast, accurate and consistent results.

www.ecadstar.com

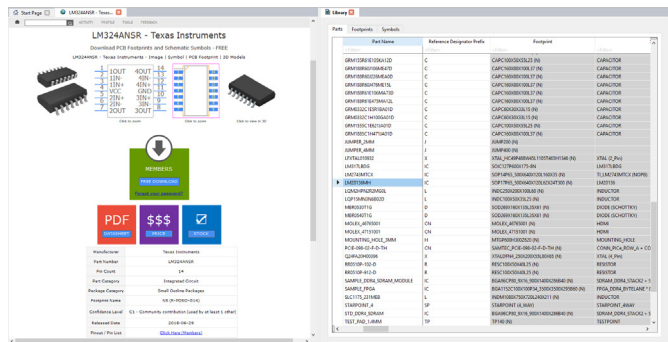Control your design contents by establishing pin equivalence for gate or pin swapping and set part acceptance to have parts made instantly available within the Design Editors. Discrepancies between design and library contents can be easily identified and reloaded to ensure your designs are always in sync and up to date.

WEB LOOKUP & PART DOWNLOAD

eCADSTAR features direct integration to some of the world's largest part vendors and online component libraries. The web lookup functionality allows you to find part information instantly, with direct access to stock, pricing, and datasheets. In addition to part information and datasheets, many of the supported vendors provide eCADSTAR part data that you can download and merge into your library, for free.

Downloading parts is a fast and efficient method of building your library. Providing you with part properties, a schematic symbol, PCB footprint and 3D model, which are ready to use in your designs. Footprints which you download are mapped automatically to your specific library layers and color set, maintaining any 3D model alignment, reducing the need to edit and rework each part.

UNIFIED & CENTRALIZED LIBRARY

Your library is the heart of your design process and as such needs to be easily controlled, supporting collaboration between many users at different design stages – and even on different designs.
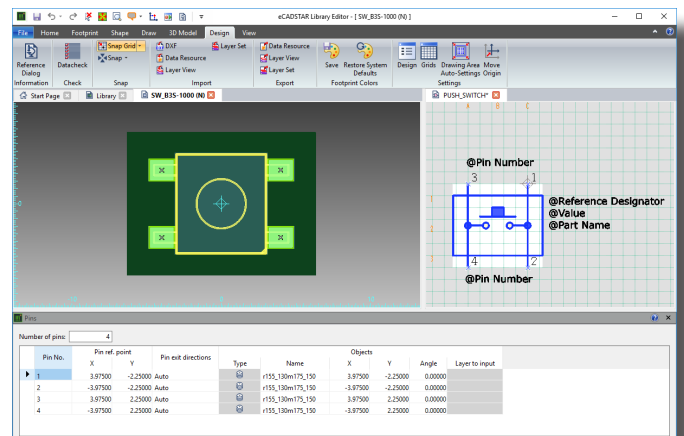
In eCADSTAR we've made this simple, with a single unified library that is referenced by each of the eCADSTAR design editors, all library data can be accessed from any editor using the Library searcher panel. Reload your design to update part information, symbols, and footprints - ensuring your data is always up to date.

A library in eCADSTAR is much more than just footprints, symbols and parts. Define schematic defaults to create consistent schematics that meet your corporate branding. Define board technologies and create complex manufacturing rules for use during PCB creation to ensure your designs are always manufacturable.


SYMBOL & FOOTPRINT EDITORS

The footprint and symbol editors provided within the Library Editor facilitate creation and customization to meet your company standards. Create dazzling schematics by designing symbols using filled shapes and predefine property placement, for consistent readability.

Create precise footprints, quickly, using parametric generation for all common package types or from drawing data using DXF import. Assign and align 3D models, natively, within the design environment or draw block shape data using 3D component areas, switching seamlessly between 2D and 3D views.

PAD & PADSTACK EDITORS

Generate pads and padstacks using the parametric editors, with one to many layer definitions allowing reuse regardless of the number of layers in your PCB design.
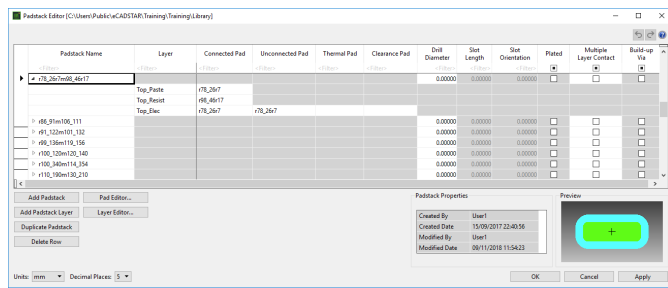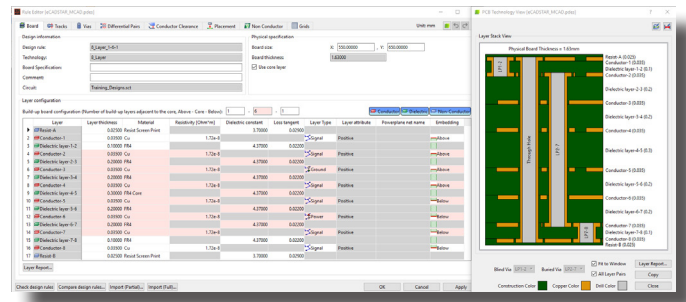
ATTRIBUTE MANAGER

Define part properties or using the attribute manager, create new ones and apply them to all parts within your Library. The attribute manager allows you to define properties which are specific to either PCB or Schematic and even to control data entry using field validation and list boxes.

TECH & RULES

Ensure your designs are always manufacturable, define board technologies and manufacturing rules within the Library Editor for use during PCB layout. Technology and Rules defined in your library are reusable and locally editable during layout, switch seamlessly between board stack-ups as your design evolves, using the one to many layers' definition. Compare and import technologies and rules in your design, with your library, ensuring your designs are always synchronized with your library definitions.

Technologies allow you to define the physical and virtual board stack up, move technologies between libraries by mapping them to your footprints and modify the display order within PCB Editor.

Rules are associated with the technology and define the thickness, material and electrical characteristics of conductor/dielectric layers. Rule definitions are referenced by both real-time and post action Design Rule Checks, preventing design or manufacturing errors from being created. With extensive definitions for track width stacks, via layer spans, placement, and clearance, tools will automatically and contextually adhere to these rules, especially when routing.

FEATURES & BENEFITS

• A connected platform that allows download complete parts with symbols, footprints and 3D models for use instantly within your design – all without switching windows or applications.

• Create, edit and manage all part data (footprints, schematic symbols, 3D models, pin assignments and attributes) in a single application.

• Create accurate footprints and pads quickly using the parametric generation tools.

• Standardize your schematic designs by defining defaults such as sheet sizes, frames, grids, colours and fonts within your Library.

• Define alternate footprints, symbols and allocate pin equivalents for swapping during design.