



DATASHEET

SCHEMATIC EDITOR

Schematic Editor, part of the eCADSTAR product, is an efficient and intuitive schematic capture application that may be used to define part connectivity, variation and design constraints for use in the eCADSTAR PCB Editor.

Define connections quicker than ever, simply draw over pins to have them autoconnect or drop the symbol on existing lines to have it automatically rewire in series. Generate net and bus labels instantly based on connected pin names, with support for multiple bit components.

With concurrent multi-user access, secure backup and sheet locking mechanisms spread the workload within your design team. The automated design rule checking and comprehensive report outputs allow review and approval of errors for complete project management.

www.ecadstar.com

©2019 Zuken Inc. All rights reserved.

Simplify and reuse existing circuits between designs by creating design blocks and hierarchical designs.

Stand out from the crowd by applying your corporate styles to your schematic designs, set design defaults that include fonts, sheet frames, connector symbols and colours for instant reuse and consistent output. Place schematic symbols with filled shapes and transparency instantly from your library with localised editing and scaling to meet your design requirements.

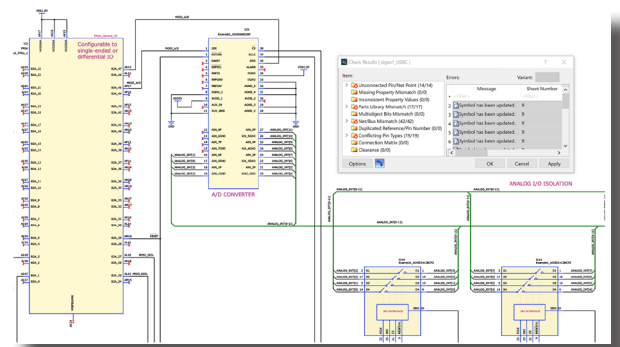
CONCURRENT DESIGN

Break large circuits into condensed multi-sheet designs and collaborate with your colleagues, concurrently. Within the Schematic Editor, individual sheets can be locked and edited by multiple users simultaneously, from anywhere in the world. Convert circuits into block symbols or block areas, creating hierarchical structures and facilitating reuse of common circuitry, even between designs.

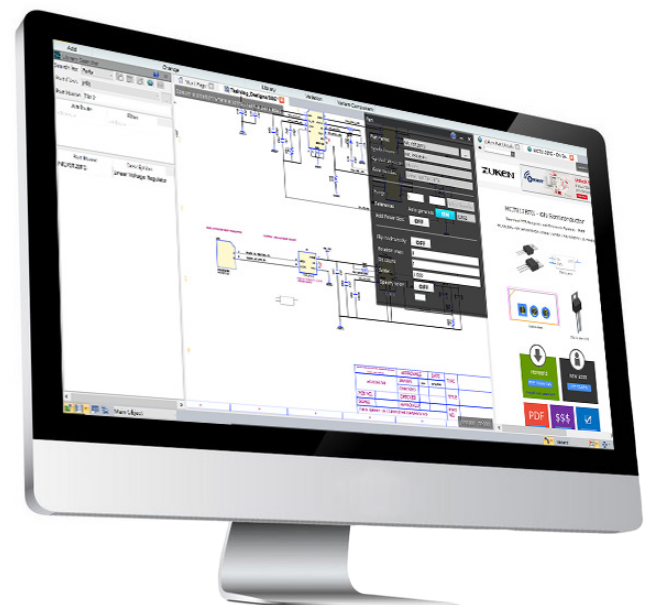
Reuse your existing IP by importing sheets and blocks from previous designs. Navigate blocks and sheets easily using the design tree, which provides a high-level view of all design content.

DRC

To ensure your schematic is easily readable, Schematic Editor includes configurable real-time checks which prevent you from creating overlapping nets and mismatching net labels. The net drawing tools will automatically resolve to prevent cross connections. Unconnected pins and dangling nets are annotated on the canvas for easy identification.



The easily interpretable and interactive report output allows each error to be navigated by type and cross probed for investigation. With commenting and approval, errors can be marked for further investigation or rectification by other users and either resolved or approved with full traceability.



LIBRARY SEARCHER WEB LOOKUP AND DOWNLOAD

Add parts directly from your unified central library, within Schematic Editor, using the Library Searcher panel. Use the parametric search and filter functionality to identify the part you require, preview its properties, footprint, and symbol (including gates and alternates).

Can't find the part in your Library? Not a problem, perform a web lookup using the contents of your library searcher, from your preferred online part vendor. View live part information including stock and pricing, download parts directly to your library and have the symbol appear on your cursor, within Schematic Editor, with just one click.

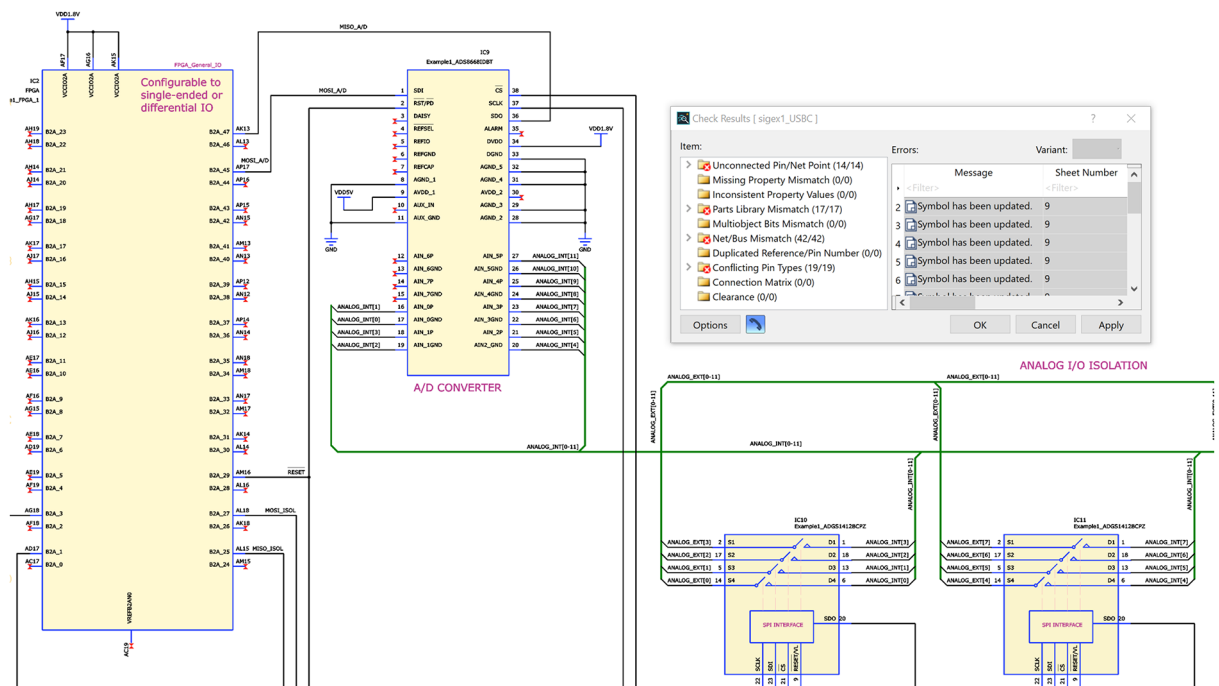
VARIATION & COMPONENT BROWSER

eCADSTAR users can define simple and advanced design variants within Schematic Editor, which may be annotated to the PCB design. Creating variant designs allows components, or groups of components, to have properties such as fitted status and part to be changed within the same design.

The Component Browser is a global dialog in Schematic Editor which provides an abstract view of all components within your schematic at a design or sheet level. Apply searches and filters to distinguish between components and modify a range of properties including variants either singularly or on mass, then cross probe to a component sheet and board position instantly.

CROSS PROBE

eCADSTAR is fully connected, cross probe your selection between dialogs for rapid design navigation, switch contextually between schematic sheets, blocks and symbols applying selection highlight and zoom or to identify changes or target objects. Connect beyond your Schematic, cross probe between your Schematic design and PCB design, in both the Viewer and Editor applications.



FEATURES & BENEFITS

- Familiar and intuitive, the Schematic Editor features a design environment and toolset that is common to all eCADSTAR applications.
- Don't wait for parts to be created, download part data natively within the Schematic Editor from your preferred online part vendor, for instant use within your design.
- Keep your designs valid and up to date by reloading and synchronising parts with your unified central library.
- Design faster and smarter by reusing design blocks and implementing hierarchical design structures.
- Achieve your electrical performance characteristics by defining high speed constraints at the start of your design process and have them propagate through to your PCB.