ZUKEN ®

CADSTAR ™

Intuitive Schematic and PCB Design - CADSTAR Lite

TOP FEATURES AND BENEFITS

- Shorter time-to-market with the intuitive Fluent™ GUI, configurable menus, toolbars and macros.

- A scalable design solution that offers total flexibility to design simple or complex PCBs.

- Faster, more efficient design with block reuse and intelligent copy/paste to extract the maximum value from your design archive.

- Can be integrated with your corporate MRP/ERP via the ODBC-compliant database parts library.

- Unparalleled interactive and automatic placement and routing with online DRC.

- Outstanding performance, completion and manufacturability.
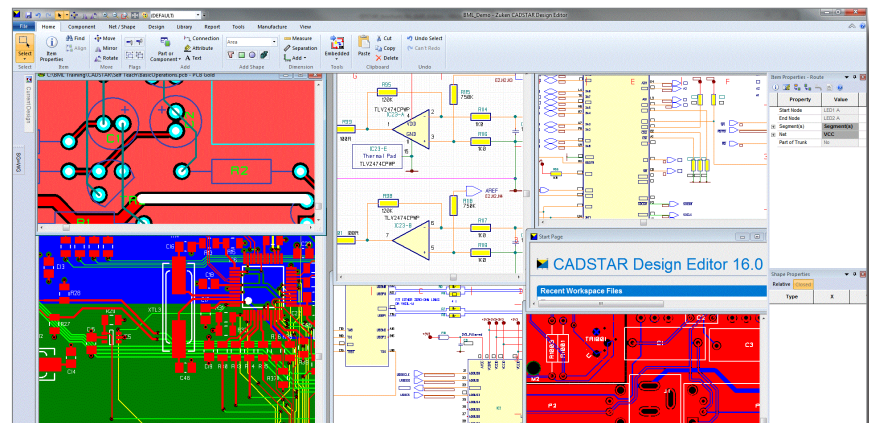
- Complete range of manufacturing formats.

Introduction

CADSTAR Lite is Zuken's affordable integrated schematic and PCB design solution.
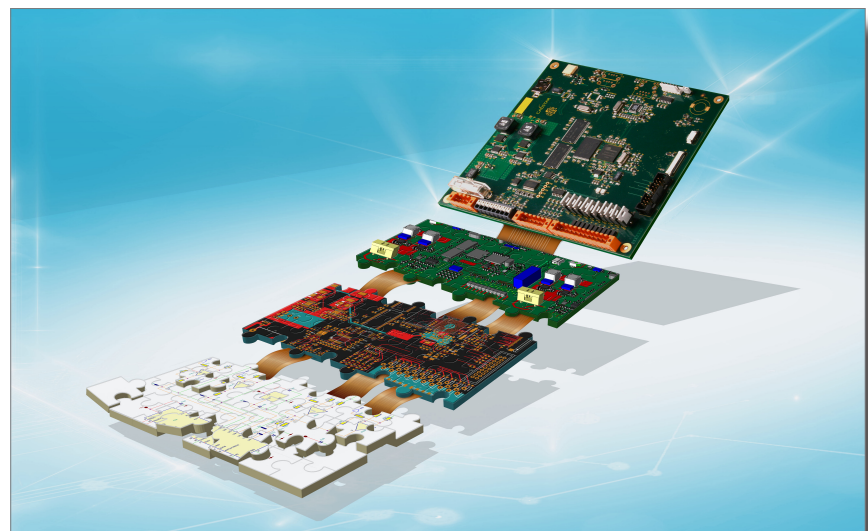
CADSTAR is an intuitive Windows-based program that is easy-to-use, fast and reduces design errors, helping you deliver effective designs in less time.

With industrial-strength technology from Zuken, one of the longest established suppliers in EDA, it has the breadth of capability and power you need to address today's demanding design challenges.

CADSTAR provides extensive functionality and performance at an affordable price. One third of the world's PCBs are designed using Zuken tools - why not join them?

CADSTAR Lite - Intuitive integrated schematic and PCB design solution

CADSTAR - Integrated schematic, PCB design, 3D MCAD/ECAD and Manufacturing outputs

zuken.com/cadstar

A Familiar, Customisable, Powerful G.U.I.

Founded on the Microsoft® Office Fluent™ user interface familiar to millions of PC users worldwide, the CADSTAR G.U.I. is both powerful and intuitive, with configurable menus, toolbars, macros and shortcuts to make it your own.

The Multiple Document Interface lets you display any combination of application windows, making it easy to compare designs or copy data from one to the next. Dockable feature windows, such as libraries and dynamic properties, automatically minimise when not in use to ensure you retain the maximum screen area for your design data.
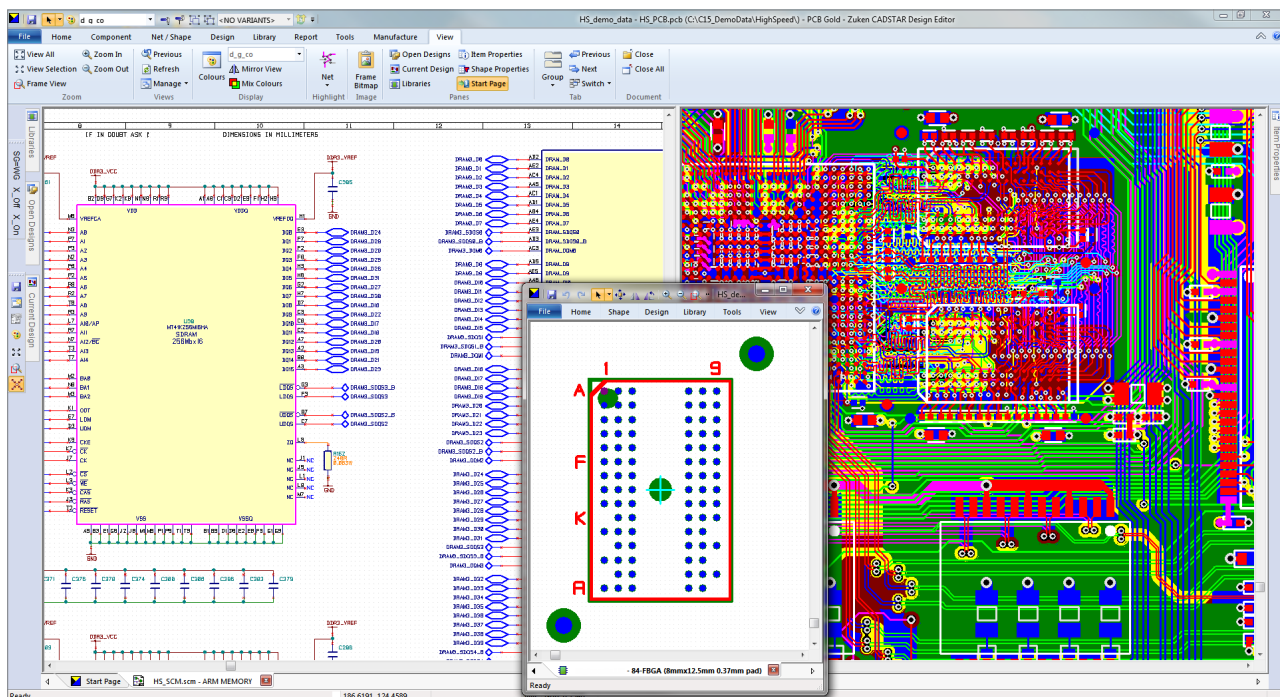
Save unnecessary key-clicks with strokes, a series of intuitive mouse actions that enhance pan and zoom capability, to speed up navigation around your design and rapidly realise your design goals.

Integrated System Design

CADSTAR's true connective data structure ensures that copy and paste intelligently reassigns net names and reference designators to maintain electrical integrity of the design, when copying individual items or sets of data.

Company standards can be built in to user-defined templates: drawing borders, colour schemes, layer stacks, board finishes, manufacturer notes, etc., can all be incorporated within a design template to ensure consistency and repeatability. The suite of drawing and dimensioning tools, together with 'autosnap' functionality, is shared by all applications to simplify shape creation and editing of any design element.

Since CADSTAR is an integrated application, there is no need to manage netlists or parts lists to link schematic and PCB layout – just one click and CADSTAR manages the binary transfer for you. Bi-directional cross-probing works as a placement aid and supports diagnostics on a completed design. This enables selection of any electrical item(s) in one window to automatically select and highlight the corresponding item in all other windows, with support for multiple monitors.

Integrated system design

Powerful Schematic Capture

CADSTAR seamlessly manages design connectivity across all levels of hierarchy and all sheets of a schematic, supporting any combination of sheets and blocks to organise and bring structure to even the most complex design. Combined with intelligent design reuse, CADSTAR dramatically reduces the time to implement multi-channel designs and efficiently reuse proven circuit blocks.
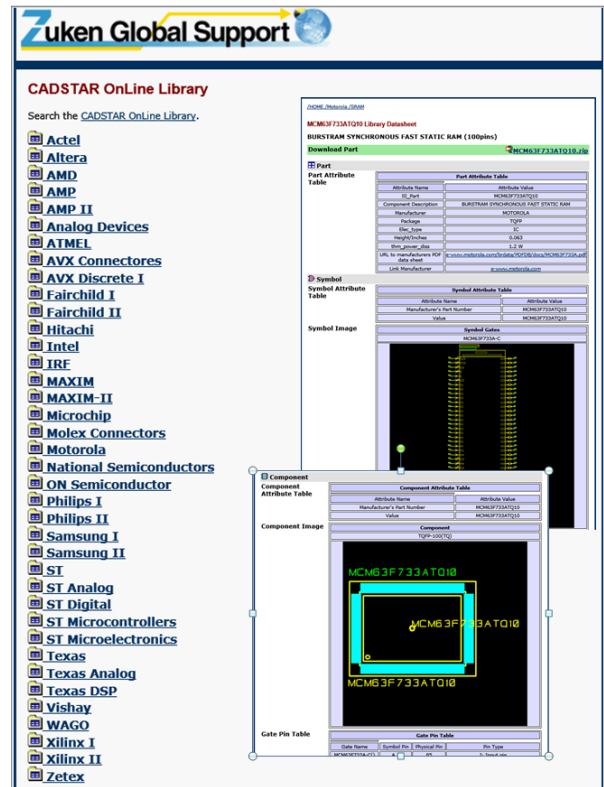
Sheet Zones simplify navigation around your schematic, with automated hyperlinks to track signal references through the schematic.

Libraries

Download a part from Zuken's extensive online library (with more than 250,000 parts from over 30 major manufacturers, created to IPC7351 standards) then simply drag and drop parts from your library onto your design, making schematic generation fast and intuitive.
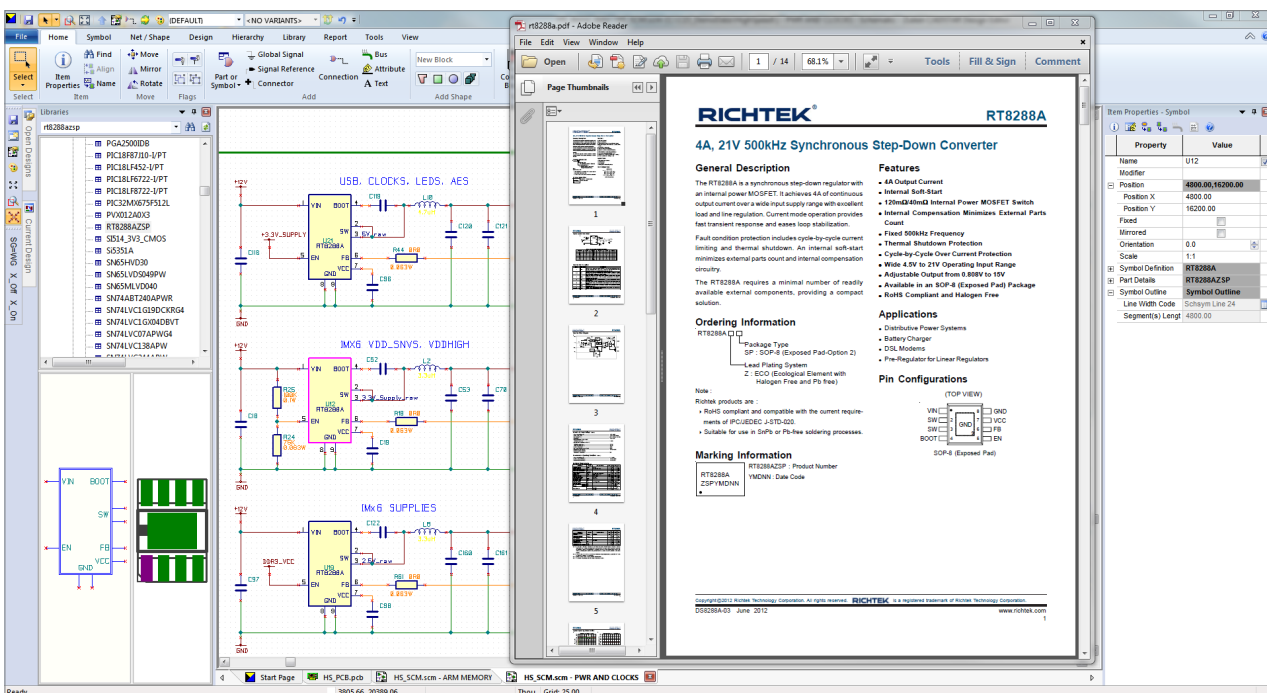
If you need to make your own parts follow the step-by-step design wizards to create your own schematic symbols and PCB components (footprints), then create alternate representations for different standards, regions, documentation or assembly requirements to give your engineers more choice and add power and flexibility to your design process.

Build intelligence into your component library to prevent routes or vias being placed under components to avoid sensitive areas.

Zuken Online Library

You can also add any number of hyperlinks for documents, images, web pages, manufacturer's data sheets, etc. to parts within the library that propagate throughout the design process, giving engineers instant access to additional reference data.

Hyperlinks to external data sources

CADSTAR automatically keeps track of the part, symbol and component versions used in your design, allowing you to confirm the validity of your project at any stage prior to manufacture. Store the parts library data in an ODBC-compliant database to allow real-time updates of attributes from your engineering or corporate database, then reload the library data to synchronise your design

Connectivity

Add point-to-point nets with automatic avoidance of obstacles or connect via busses and signal references, with implicit net connectivity throughout the design.

Capture design intent and rules within CADSTAR Schematic that are automatically annotated to the PCB design through forward engineering control. Route widths, component groups, spacing classes and a wide range of other attributes can be assigned by the design engineer, with the reassurance that they will automatically be accessible to the PCB layout specialist.

Report and Programming Options

CADSTAR provides a range of standard reports for DRC, electrical rule check, part lists, etc., plus a built-in report

generator that lets you extract the data you need in a flexible format. 'Active' reports, such as design rule check, include hyperlinks for instant navigation to the selected area of the design.
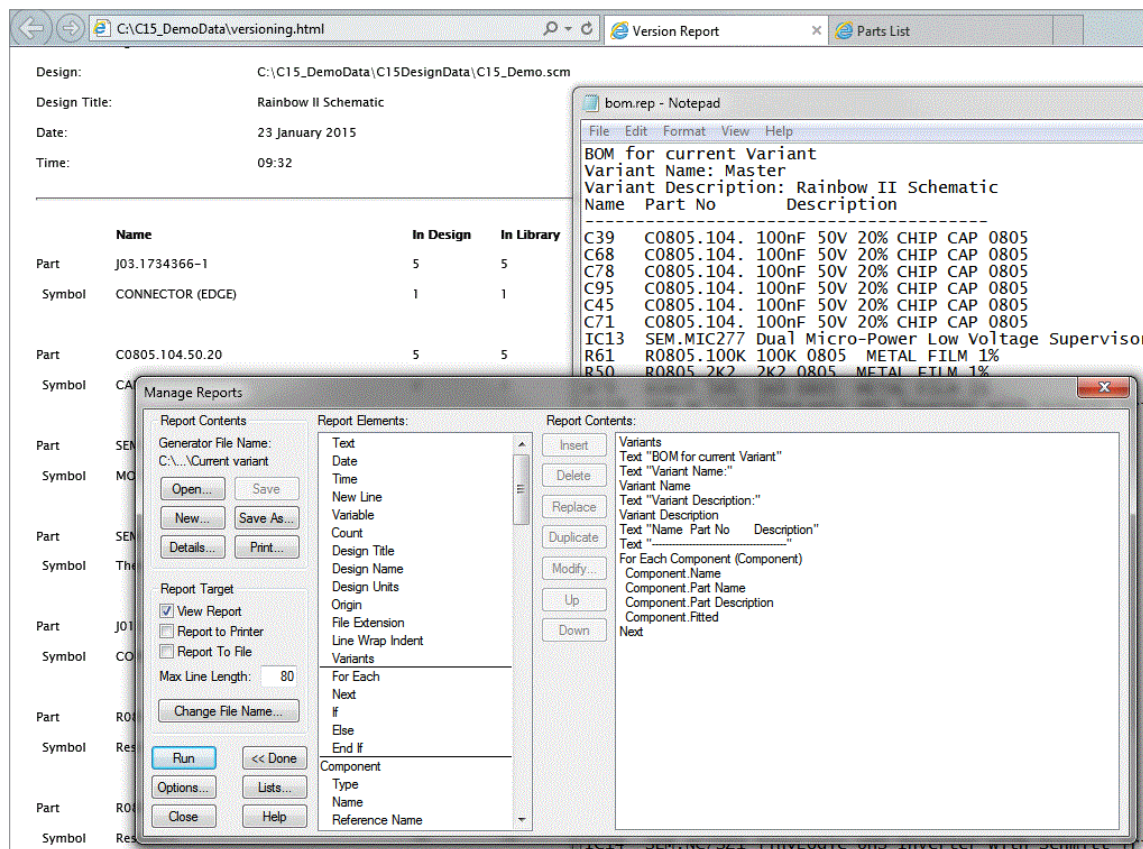
If you need more power or want to customise the capabilities of CADSTAR further, the OLE Automation interface lets you manipulate the design and extend the core functionality by creating your own applications or programing your own functionality.

Design Comparison

Powerful and configurable Design Comparison will compare any combination of PCB to PCB, Schematic to Schematic, or Schematic to PCB designs and report the list of differences.

Import and Export of DXF

Import symbol outlines, board profiles, critical placement and other complex geometry from your mechanical CAD systems using industry-standard DXF format, saving time and minimising rework. Export all or part of your design to DXF format for documentation or transfer to other applications.

Standard and custom report

Placement, Routing and Manufacture

Assisted Placement

Use the Auto Placement routines to logically organise components around the board, check space requirements for both sides of the PCB and quickly evaluate alternative locations for arrays of high pin count devices.

Replicating multiple channels in your designs? Just place the first group and CADSTAR will automatically select the correct components to complete the rest.

Alternatively, create a library of matching reuse blocks for schematic and PCB, making the best use of existing design elements in future designs. Simply add a circuit block to your schematic, transfer to PCB and load a predefined layout block including pre-placed components, tracks, copper and text. Whether you use it for multiple channel designs, critical analogue or RF circuit blocks, or just to replicate a commonly used layout pattern, design reuse will save you time and ensure repeatability of design.
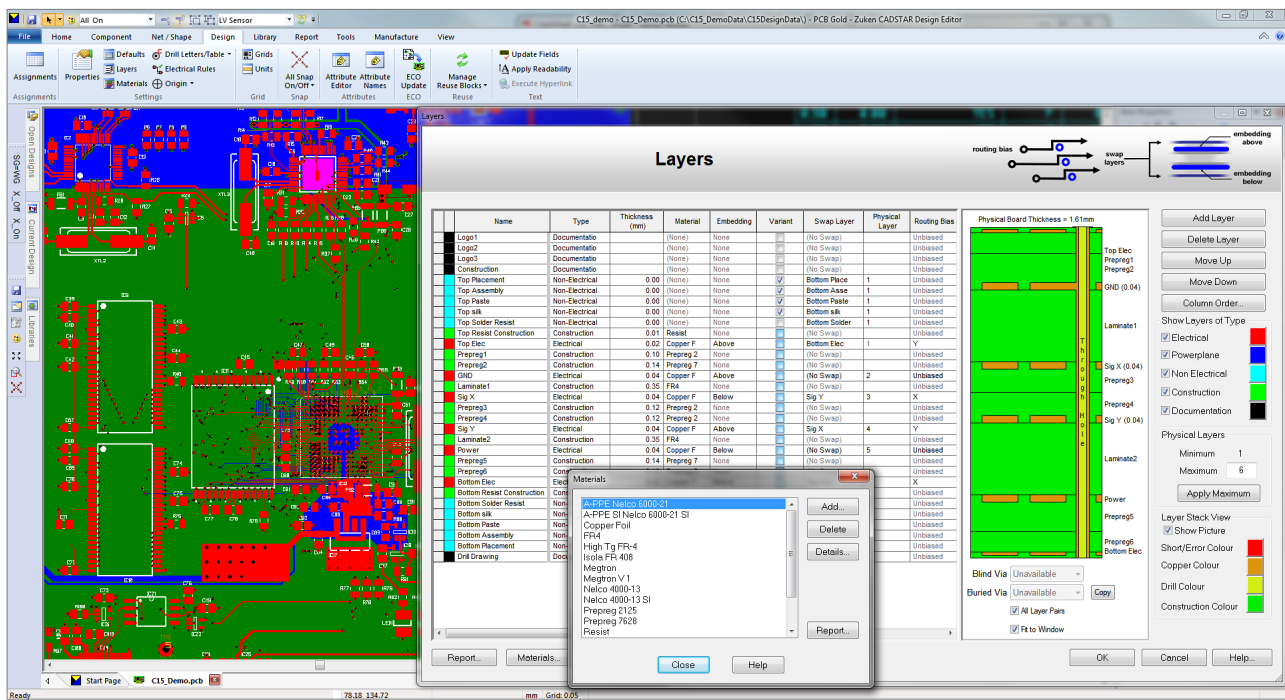
Layer-Stack Editor

Define your PCB layer stack, dielectric materials and spacings with a scaled cross-sectional representation detailing construction requirements, via styles and copper weights, then save the image to document the design.

Manufacturing Options

Suppression of inner layer pads provides more routing area on complex designs by automatically removing unconnected inner layer pad and via lands.

View slots as you would expect to see any standard drill hole. Once you have specified the slot dimensions and orientation the feature will be correctly reflected in the resulting NC drill program.

When joining multiple nets at one location CADSTAR automatically excludes DRC errors, allowing Star Points to be defined that merge disparate supply or ground nets, or allow sense lines to be intelligently defined.

Layer Stack configurator

Place & Route Editor XR2000

The Place & Route Editor offers a powerful interactive, semi-automatic or automatic placement and routing environment that provides outstanding performance, high completion rates and manufacturability of the final design.

Routing modes include Activ45, memory pattern, orthogonal, 45 degree or free angle routing, optimisation (track and via), and a host of other features.

Online DRC (Design Rule Check) ensures that you maximise your productivity while providing assurance of routing an error free layout.

Copper Planes with Auto Re-Pour

Definition of multiple planes and copper areas is fast and easy, with the added benefit of an automatic re-pour mode to make sure copper is regenerated after moving components or editing tracks. This ensures the maximum copper area and optimum plane connectivity is achieved.

Batch Processing

Set up a single batch file for all of your design data requirements, and then all of your manufacturing outputs will be generated with a single mouse click. CADSTAR supports all industry standard formats such as Gerber 274X, NC Drill, ODB++ and IPC-D-356 plus custom reports and Bills of Materials.

Intelligent PDF

The intelligent PDF writer creates a searchable electronic document that contains part attributes from your library. Allow colleagues, customers and design partners to view and navigate around your design without the risk of any unintended changes.

Software Downloads

CADSTAR Express

A free download of CADSTAR Express is a quick and easy way for you to experience the basic features of CADSTAR. It includes all the functionality, limited to 300 pins and 50 components, plus the opportunity to experience Zuken's powerful Place & Route Editor XR 2000. In addition, you can view samples of CADSTAR Exchange, Zuken's online parts library.

CADSTAR Design Viewer

A free download of CADSTAR's design viewer gives easy viewing and sharing of schematic and PCB design data. This allows managers, project leaders, design partners and customers to plan or discuss projects and track and review progress.

For more information on all the tools and solutions available with CADSTAR, please visit www.zuken.com/CADSTAR

Feature List

General

- Native 32 bit Windows application
- Runs on Windows Vista Business, Windows 7, Windows 8.1 and Windows 10 (32 and 64 bit)
- True metric and imperial unit support
- Database resolution 0.001mm
- Rotation to 1/1000 degree
- Embedded or True Type fonts
- True connective data structure
- 2000 pins per design

Interaction

- Multiple undo/redo
- Interactive drag and drop
- User definable toolbars, function keys and keyboard commands
- Comprehensive online help
- Tool tips
- Smart update
- Intelligent cut, copy and paste
- Access to the Zuken online libraries (with Support contract)
- Library Editor with symbol wizards
- Multiple library support, including ODBC-compliant parts database
- Customisable library attributes and intelligent hyperlinks
- Documentation (non-electrical) symbols
- Duplicate, transform and merge shapes
- Macro recorder and OLE automation
- Active dialogs with dynamic properties
- Import of 3rd party Schematics and PCB Layout
- Cross-probing between schematic and PCB

Schematic

- Design templates
- Mixed flat sheet and hierarchical design with user definable sheet sizes
- Zones with off-sheet references
- Parametric and graphical library browsers
- Autoroute connections with crossing avoidance
- Intelligent signal reference and power symbol allocation
- Star-point plane connectors
- Test points
- Binary transfer to PCB layout and back-annotation of changes

PCB Layout

- Layout templates
- Unlimited layers with user-defined layer structure and pad-stacks
- Blind and buried vias
- User definable design origin
- Component areas (keep out, no vias)
- ECO Update from schematic
- Manual and automatic placement
- Positional rename
- Automatic pin and gate swap
- Measurement/dimension/drawing/alignment tools with AutoSnap feature
- Star-point plane connectors
- Test points
- Design Rule Check (spacing and manufacturing)

Routing

- Grid-free routing
- Interactive and semi-automatic routing
- Auto-necking to SMD pads
- Teardrops
- Support for split planes
- Automatic copper pour and re-pour
- Online DRC
- 90, 45 and free angle routing
- Routability analysis and report
- XR2000 L4 router (4 layer autorouter)

Outputs

- Full range of manufacturing export formats
    - Gerber, ODB++, Drill, DXF, IPC-D-356
- 'One Click' batch processing
- Smart PDF output

Optional Modules

- Standalone Schematic Capture
- Constraint Browser for Schematic Capture
- Variant Manager
- EMC Advisor
- IDF 3D-MCAD/ECAD Interface
- Rules by Area
- Place & Route Editor XR5000HS
- Dragon Autorouting platform
- CADSTAR FPGA
- BoardModeler Lite (3D)
- Signal Integrity Verify
- Power Integrity Advance