

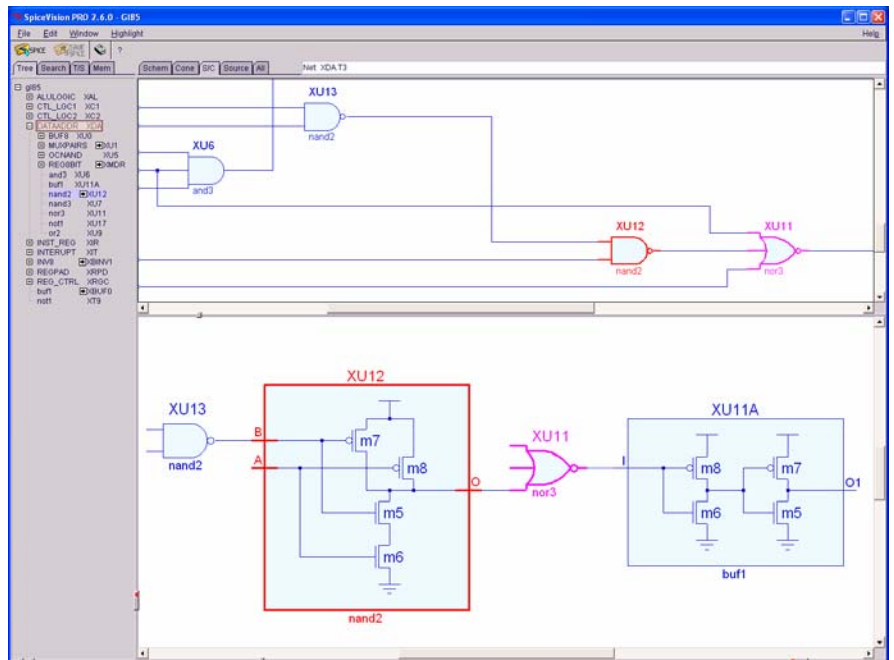


SpiceVision[®] PRO: Customizable Transistor-Level Debugging

SpiceVision PRO takes the complex SPICE descriptions produced by many EDA tools and generates clean, easy-to-read transistor-level schematics and circuit fragments, and design documentation to speed up debugging and project development.

Customization – a Tcl based application program interface (API) allows access to the internal database and graphical user interface (GUI). Users can analyze the design data and generate user-specific design reports and design checks.

Multi-level – the browser displays multiple hierarchy levels, from top level overview to all sub-circuit levels, while multiple windows display the hierarchy tree, source code and schematic diagrams. The user can select areas of interest from a results list, generated by a powerful search engine, to generate circuit diagrams.



- Graphical netlist analyzer – pre-layout and post-layout SPICE
- Tcl based UserWare API – for advanced customization
- 32/64-bit database handles today's largest SoCs and ASICs
- Exports schematics and schematic fragments – Cadence Virtuoso and EDIF 2.0.0
- Cone display – Cone Window displays selected fragments and critical paths
- Cookie-cutting – circuit fragments can be saved as separate SPICE files

Cone Window – the Cone Window, an "intelligent magnifying glass", displays selected fragments, including critical paths calculated by other tools. These can be transferred to a simulator as a separate file, for partial simulation, often running 10 to 100 times faster than a full simulation.

Simplification – the output of place and route tools can be new, larger and more complex Spice files with different critical paths. These paths can be extracted and saved for selective simulation. SpiceVision PRO can also simplify the schematic, merging components in parallel, such as capacitors, or creating a non-parasitic view, showing only functionality.

IP – specific parts of a design can be cookie-cut from the complete design and saved as a Spice text file with an associated circuit diagram, for use as new IP.

Virtuoso – the Virtuoso Schematic Editor environment option, based on the Cadence SKILL language, helps to manage critical paths or sections of a circuit and exports them into the Cadence Virtuoso Schematic Editor environment.

Faster – SpiceVision PRO gives the designer an extra level of understanding of the physical circuits that underlay the abstractions of higher level tools. It is easier to debug problem areas and devices can move into production earlier and with a higher degree of confidence.

At a Glance

FEATURE	BENEFITS
Creates schematics from Spice files	Schematics provide easier and faster debugging for complex circuits. Supported dialects include spice2, spice3, pspice, hspice, calibre and CDL
64-bit support (Solaris, AIX, Linux).	Higher performance and increased capacity, for larger and more complex designs
Powerful GUI	Multiple views, including tree, schematic, cone and source file for increased circuit understanding plus drag and drop between different views
Cone Window	Incremental schematic navigation for big designs
Non-Parasitic view	Displays CMOS function without parasitic structures for comprehension of circuit
Tcl UserWare API	Allows interfacing with tool flow and user customization
Fragment save	Fragments of circuits can be saved as Spice file and schematic for future reuse as IP, or for partial simulation
Schematic export options	Export schematics and schematic fragments into Cadence Virtuoso Schematic Editor and EDIF 2.0.0 schematic files for further optimization and debugging
Predefined symbols	Symbols for components (resistors, capacitors, transistors, current and voltage sources etc) supplied as standard; can link to external symbol libraries

Company Contact

Concept Engineering GmbH · Bötzing Str. 29 · 79111 Freiburg · Germany
 Tel: +49-761- 47094-0 · Fax: +49-761- 47094-29 · Email: info@concept.de · http://www.concept.de