

What's New in Cadence OrCAD 16.0 Products

A flexible and scalable solution that adapts to your needs

To stay competitive in today's market, engineers must take a design from engineering through manufacturing with shorter design cycles and faster time to market. To be successful, you need a set of powerful, intuitive, and integrated tools that work seamlessly from start to finish.

Cadence® OrCAD® personal productivity tools (including Cadence PSpice®) have a long history of addressing these demands. Designed to boost productivity for smaller design teams and individual PCB designers, OrCAD PCB design suites grow with your needs and technology challenges. The powerful, tightly integrated PCB design suites include design capture, librarian tools, a PCB editor, an auto/interactive router, and optional analog and mixed-signal simulator. The affordable, high-performance OrCAD product line is easily scalable with the full complement of Cadence Allegro® PCB design technologies.

All OrCAD 16.0 products are now supported in Windows Vista™ Enterprise.

New Cadence OrCAD and PSpice Bundle!

Cadence OrCAD EE Designer Plus

OrCAD EE Designer Plus provides powerful schematic entry and simulation in one premium package. This competitively priced, complete front-end design solution includes:

- OrCAD Capture schematic entry, the world's leading schematic capture tool
- PSpice A/D for analog and mixed-signal simulation
- PSpice advanced analysis capabilities—sensitivity analysis, optimization, Smoke (stress analysis), and Monte Carlo (yield analysis)—to automatically maximize the performance of circuits

Cadence OrCAD Capture

- Improved support for complex hierarchical designs, including better support for occurrence properties and externally referenced designs.
- Support for placing and moving text and drawing objects on fine grid independent of connectivity objects
- Enhancements to archiving adds an option to include PSpice models in the archive libraries which improves the performance of archived simulation profiles
- Improved error messages and DRC engine
- Cadence Help Online Documentation System replaces CDSDoc, providing faster invocation time, advanced search capabilities, and built-in viewing window
- Usability enhancements and implementation of over 100 CCRs for improved quality and performance

Cadence OrCAD Capture CIS Option

- Support for including mechanical parts and assemblies in standard component information system (CIS) Bill of Materials (BOM)
- Better support for Japanese characters in CIS

- Better BOM generation for non-English operating systems

Cadence PSpice A/D

- Improved speed for power electronics designs, especially those using ABM if-then expressions, which will simulate up to 50x faster
- CheckPoint Restart feature allows you to save the state of a transient simulation at specific times as CheckPoints. Prior to restarting, you can change component values, so if you're not getting the results you want, you can tune your circuit without having to restart from the beginning. This is particularly useful for circuits that have a long startup time like a switch mode power supply. You can simulate once to reach steady state, tune the circuit to get the final result you want, and then simulate from the beginning to verify the full simulation. In this situation, CheckPoint Restart can be a significant time saver
- Autoconvergence feature allows PSpice to automatically modify convergence options as necessary. This reduces the need for user interaction for problem circuits
- Improved transient convergence. Adaptive iteration, improvements in the step time algorithm, and an improved switch model allow PSpice to converge on certain types of problem circuits
- Run in resume mode accessible from graphical user interface (GUI)

Minimum step size recalculation

- Usability enhancements
- Tasks such as adding a trace, adding a plot, zooming the view, or adding a text label are now available using the right mouse button to display a context-sensitive menu in the Probe window
- Probe window cursors now display X and Y coordinates on the status bar of the Probe window
- Improved simulation speed for E and G device-based circuits
- PSpice libraries have 120 new power devices. Updated vendor libraries include 290 new LED models
- Cadence Help Online Documentation System replaces CDSDoc, providing faster invocation time, advanced search capabilities, and built-in viewing window

Cadence PSpice AA

PSpice Advanced Analysis support for legacy PSpice models. Traditional tolerance specifications can now be used by the Monte Carlo and Sensitivity tools. This includes the use of PDF custom distributions and device and lot tolerances

Cadence OrCAD PCB Designer (including OrCAD PCB Editor)

- OrCAD Layout translator user interface opens from within OrCAD PCB Editor
- Cadence Help Online Documentation System replaces CDSDoc, providing faster invocation time, advanced search capabilities, and built-in viewing window
- Usability enhancements made within the display canvas and to the command structure
- Redesigned color / visibility GUI
- Enhancements made to the underlying graphics system, based on OpenGL graphics engine
- Physical and spacing constraints incorporated in the Allegro constraint management system
- Interactive and automatic controls for component fanout (pin escaping)

- Context-sensitive editing paradigm built on selecting database objects first, followed by action command

Cadence OrCAD Layout

- Cadence Help Online Documentation System replaces CDSDoc, providing faster invocation time, advanced search capabilities, and built-in viewing window
- Support for GerbTool Version 15.0
- Implementation of numerous CCRs for improved quality and performance