DATASHEET

cadence

OrCAD X PCB Design Platform

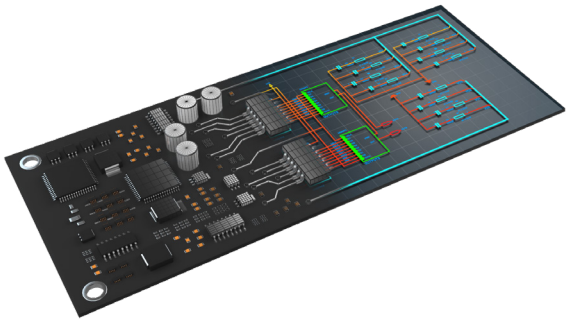
Design Fast, Design Correct, Design Connected

Cadence® OrCAD® X is a comprehensive PCB Design platform that offers a rich set of features to streamline your entire electronic design process and take your designs from concept to production. It has been designed to be intuitive and efficient to use, enabling seamless schematic capture, mixed-signal circuit simulation, and PCB layout. With significant improvements in performance, automation, data management, and cloud-enabled collaboration, OrCAD X is a scalable solution that caters to a wide range of market segments, spanning smart-home controllers to pacemakers and flight control systems to powertrain control modules. So whether you are tackling simple or complex designs with higher densities, stricter requirements, mixed-signal circuits, and/or standards-based interfaces, it has everything your team needs to improve productivity and reduce time-to-market.

Applications

Schematic Design

OrCAD X has the most widely used schematic capture solution in the industry with years of refinement when it comes to useability, functionality, and customization. Electrical engineers can easily capture design intent using a variety of circuit reuse capabilities, as it supports both flat and hierarchical design configurations. Your design's bill-of-materials (BOM) can be optimized based on cost, lead time, inventory, life cycle, and material compliance.



The Live BOM feature in OrCAD X provides supply chain intelligence and insight powered by Sourcengine™, where users have access to over 1 billion components across more than 3,600 suppliers, distributors, and manufacturers. Additionally, physical and electrical constraints can be defined in the schematic using the time-tested OrCAD constraint manager found in the platform. Tight coupling of PCB Layout allows bi-directional communication of constraints, pin/gate swaps, reference designator (refdes) updates, part changes, and connectivity. Part grouping and cross-probe features simplify component placement and design navigation between the two editing domains.

SPICE Simulation

PSpice® is the industry leader when it comes to commercial-grade analog and mixed-signal circuit simulation, solving virtually any design challenge, from high-frequency systems to low-power IC designs. Its native integration into OrCAD X schematic capitalizes on the same ease of use while preventing the redraw of the circuit for simulation and layout.

With over 35,000 parameterized models provided as part of the installation, getting started is simple. Its large ecosystem of silicon manufacturers makes obtaining new models easy. Analysis types like sensitivity, temperature, parametric, Monte Carlo, noise, electrical overstress (EOS), circuit optimization, and electromechanical system analysis with Mathworks® MATLAB® and Simulink® maximizes design performance, yield, cost-effectiveness, and reliability. Multicore CPU utilization parallelizes simulation sweeps separating its performance from other solutions on the market.

PCB Layout

With the ability to learn in a week with little to no training, OrCAD X PCB Layout is ideal for the novice designer, electrical engineer, or layout designer focused on quick-turn PCB. The new layout environment consists of simplified menus, toolbars, nested commands, and streamlined use models, helping simplify tasks like routing, shape editing, and navigation by reducing the number of user interfaces and mouse clicks.

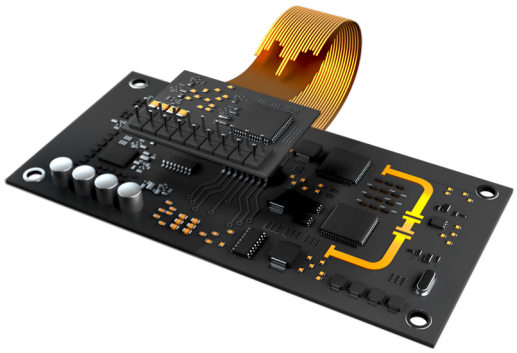
Designed for optimal productivity, it contains the following features – interactive and automatic routing, rigid-flex, fanout, pin and gate swapping, constraint management with design for manufacturing (DFM) checks, high-performance 3D, and more. Effortlessly tackle high-speed design with single-ended and different pair phase control, along with real-time feedback with modeless impedance and coupling analysis. Several built-in features, such as commenting, allow designers to markup the layout, reducing the time it takes to review a design, while native Altium Designer®, Siemens® PADS™, and Autodesk® Eagle® translators allow the preservation of existing design IP. Moreover, ISO26262 certification qualifies the editor for automotive functional safety requirements. The OrCAD X PCB Layout database is fully compatible Allegro X PCB Editor, ensuring a continuous workflow in a mixed environment.

Key Features

Rigid-Flex

The layer stack manager within OrCAD X PCB Layout supports multiple stackup zones, allowing you to specify any rigid, rigid-flex, and flex configurations. Stackup includes materials like masks, surface finishes, and stiffeners to ensure proper fabrication.

Each physical zone can be further enhanced with bend lines, angles, and bend order allowing for a true interactive 3D representation. Zones automatically include associated keep outs and optional constraint regions and rooms. In-design rules coupled with enhanced definitions ensure that the design is correct by construction. Contour routing expedites zone-to-zone connectivity, taking into consideration the unique shapes that flex areas introduce.



Concurrent PCB Design

Unlike the design partitioning methodology, OrCAD X PCB co-design allows two designers to concurrently work on the same PCB database at the same time. Any changes made in the canvas are reflected on the cloud server and instantly seen by the other designer. Concurrent design tasks supported include 3D visualization, shape editing and shape design for power delivery, footprint placement, interactive etch-editing commands, impedance and coupling analysis, silk-screening to label and identify items such as test points, component labels, assembly instructions, report generation and full/window DRC update, and testability.

It also supports global tasks that frequently interrupt the PCB implementation process, such as netlist updates, MCAD changes, constraint modifications, and more. Being cloud-based, there is no special hardware or servers required, reducing any IT overhead.

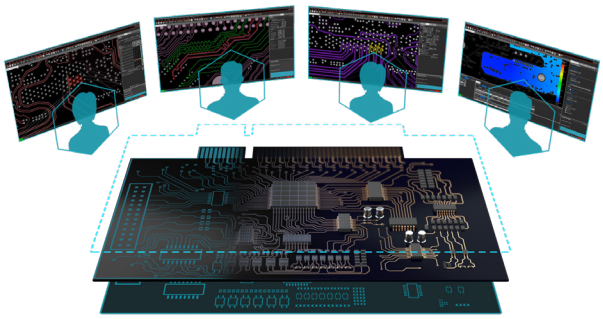
MCAD Collaboration

Dynamic 3D visualization found in PCB layout allows for first-pass mechanical form and fit checks to occur between electrical components and the product enclosure. Changing between 2D and 3D views is as simple as clicking a button and lacks the performance rendering issues that plague competing solutions.

Integration with Autodesk® Fusion™, Dassault Systèmes® SOLIDWORKS and CATIA, Siemens® NX, and PTC® Creo provides a bidirectional pathway for mechanical and electrical changes to occur across design domains. IDX is used as the exchange mechanism providing a collaborative medium to accept, reject, or revert changes. Direct connections to Dassault Systèmes **3D**EXPERIENCE® and Autodesk Fusion cloud platforms provide the first ECAD/MCAD platform to platform connection further enhancing the management of the MCAD collaboration process.

Data Management

Work-in-progress library and design data are managed using version control, which supports check-in, check-out, and rollback capabilities. A revision history is maintained for all managed objects, making changes easily traceable. Private and public workspaces provide a means of organizing and sharing data to both internal and external resources. Data is protected by role-based access control, and any files being edited are automatically locked to prevent simultaneous editing, helping avoid commit conflicts.



Library Management

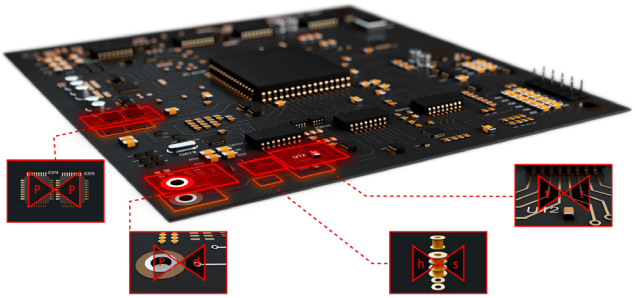
The unified component information system allows for rapid creation and management of part libraries. This includes part metadata along with schematic, PCB layout, mechanical, and analysis models. Integrations with SnapMagic (SnapEDA), SymacSys, and Ultra Librarian® provide millions of free part definitions, simplifying and accelerating the search, creation, and use of new components.

Library data is managed by version control providing complete change history and traceability. Optional cloud implementation removes database and server overhead while providing a platform for distribution and collaboration with both internal and external team members.

Manufacturing

OrCAD X prevents common fabrication issues by shifting design for manufacturing (DFM) analysis into the PCB layout environment. In-design DFM operates like PCB DRC and covers over 100 DFx checks spanning design for assembly (DFA), design for fabrication (DFF), and design for test (DFT). Live DOC automates the fabrication and assembly documentation process utilizing templates to auto-populate documentation views based on the source PCB.

Any change to the PCB design is automatically reflected in Live DOC, removing the need for manual updates and ensuring the documentation always reflects the current PCB version. The batch output generator specifies a list of preconfigured sets of manufacturing outputs, including ODB++, IPC-2581, Gerber, Artwork PDFs, IPC-D-356 netlist, NC Drill, Backdrill Report, and more. The batch list is user-configurable and can be leveraged from project to project providing manufacturing output consistency and repeatability.



cadence®

Cadence is a pivotal leader in electronic design and computational expertise, using its Intelligent System Design strategy to turn design concepts into reality. Cadence customers are the world's most creative and innovative companies, delivering extraordinary electronic products from chips to boards to systems for the most dynamic market applications. **www.cadence.com**