

Cadence OrCAD 16.6 Release

Usability, performance, design flows, Tcl scripting, and more

The Cadence® OrCAD® 16.6 release continues to address our customers' toughest PCB design challenges. Delivering on four key goals—improved product usability, increased product performance, new design flows, and expanded Tcl scripting integration—the 16.6 release will benefit every user of OrCAD technology.

OrCAD 16.6

To boost overall PCB design productivity, OrCAD Capture, OrCAD Capture CIS, PSpice® A/D, and PSpice Advanced Analysis include numerous new features and enhancements that increase each tool's usability, configurability, and performance.

OrCAD Capture has been integrated with OrCAD PCB SI to deliver a new bi-directional schematic entry/signal integrity flow for electrical engineers. The 16.6 release also extends the Tcl programming capability and apps to PSpice, allowing users to customize their simulations and environment.

OrCAD Capture and Capture CIS

Capture – PCB SI integration and flow: With product integration comes a new bi-directional schematic entry/signal integrity flow that allows electrical engineers to explore circuit topologies, develop constraints, and analyze signal integrity.

Quick-place for common components: A new menu, Place > PSpice Component, enables quick-place for commonly used schematic or simulation components. The menu items list of components is user-configurable and has been pre-populated with PSpice simulation devices (passive, discrete, sources, digital).

User-configurable menus and toolbars: Menus, toolbars, and icons in OrCAD Capture, PSpice Advanced Analysis, and Model Editor can now be customized. This makes it possible to run any Tcl method or script from the menus.

Enhancements to the Find function: The Find function now allows searches for parts by value of a given property

Highlights

- New schematic entry/signal integrity flow
- Significant performance increase for CIS operations with large databases
- Tcl-based customization for CIS Explorer, design rule checks, menus, and toolbars
- Multi-core capability for significant PSpice performance increase
- Upgraded model encryption with 256-bit support
- Tcl-based customization for Advanced Analysis, simulation, and .dat file access
- Enhanced IBIS support (5.0)
- Advanced convergence and simulation control options

(e.g. Property Name=Value) or use of a regular expression as the search string. For example, to search for components with designators starting with C or R and followed by any number between 2 and 9, use the search string Part Reference=(C|R)[2-9].

NetGroup enhancements: The NetGroup use model is now aligned with the Bus use model for intuitiveness and consistency. Enhancements areas include: assign a NetGroup to a Bus, reorder pins in an unnamed NetGroup, add and remove pins from a NetGroup, visible NetGroup references, and find NetGroup references.

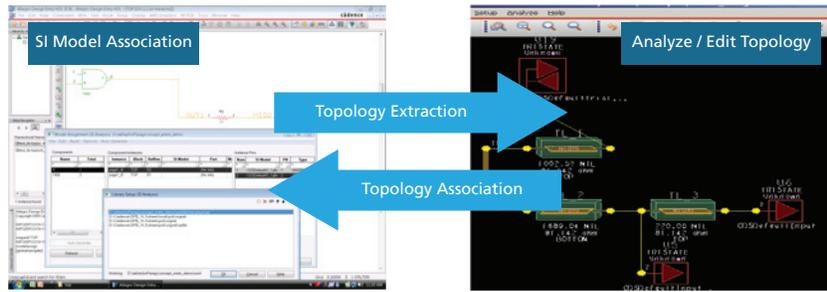


Figure 1: Extract nets from OrCAD Capture into OrCAD PCB SI to perform signal integrity analysis

CIS performance increase: The overall performance for CIS operations, especially when dealing with very large databases or queries, has been significantly improved.

Tcl customization for CIS Explorer: CIS Explorer can be custom-configured with user-definable actions and capabilities. (For example, customized part placement checks can disable placement of an EOL part or provide a warning if part procurement has a long lead time). Query result rows can also be customized. (For example, rows can be highlighted blue for recommended parts or red for parts not recommended or allowed.)

PSpice A/D and Advanced Analysis

Advanced control options: Numerous advanced convergence and simulation control options/parameters have been added or exposed, giving users greater control over simulation and convergence. These options include: bias-point convergence, voltage limiting, worst-case deviations, max-time step control, pseudo transient, and relative tolerance.

Probe .dat upgrade to 64-bit precision: PSpice now generates 64-bit data precision in the .dat file output. This ensures higher precision compared to the 32-bit .dat file data from previous releases. (As an example, in previous releases, when a very small amplitude voltage is superimposed on a large voltage, the resulting voltage lost its resolution in a 32-bit .dat file.)

UNDO support for captured netlists: Netlisting to PSpice now preserves UNDO, making it easier to make iterations and modify parameters, components, and connectivity.

Enhanced IBIS support: The PSpice IBIS translator now supports IBIS models up to version 5.0 (not including AMI) for use in simulations.

Multi-core engine support: Enhancements to multi-core support and I/O read-write provide significant performance improvements. Focused performance enhancements, especially for large designs or designs with complex model instances (MOSFETS, BJT), also boost performance.

Encryption enhancements: Upgraded model encryption now includes 256-bit (AES) encryption support.

Tcl-based customization: Advanced Analysis, simulation, and .dat file access can be custom-configured with user-definable actions and capabilities. This enables an environment that can be customized to specific flows and needs, and allows users to leverage enhanced features and design capabilities

For More Information

For more information on the OrCAD 16.6 release, please contact your local Cadence channel partner: http://www.cadence.com/Alliances/channel_partner/pages/default.aspx.

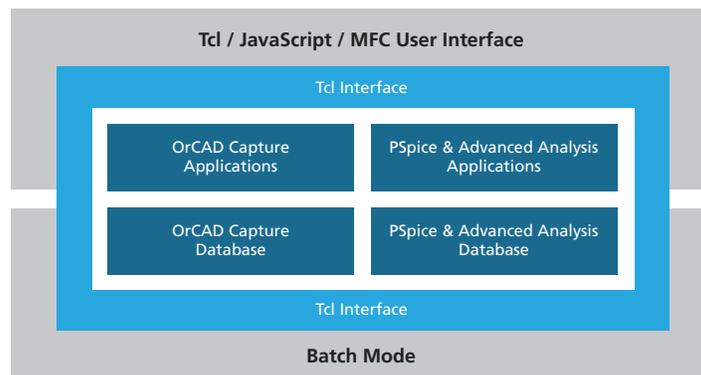
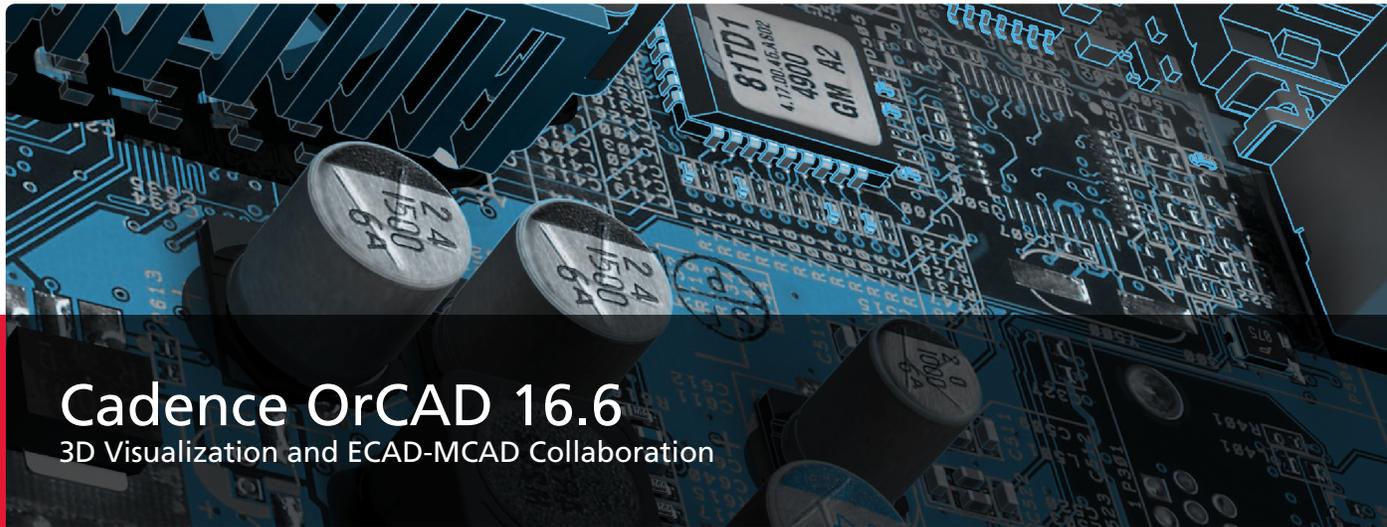


Figure 2: OrCAD 16.6 extends the Tcl programming capability and apps methodology originally in Capture to PSpice



Cadence is transforming the global electronics industry through a vision called EDA360. With an application-driven approach to design, our software, hardware, IP, and services help customers realize silicon, SoCs, and complete systems efficiently and profitably. www.cadence.com



Cadence OrCAD 16.6

3D Visualization and ECAD-MCAD Collaboration

3D and ECAD-MCAD in OrCAD 16.6

Cadence® OrCAD® 16.6 has added full 3D visualization and incremental MCAD-ECAD collaboration capabilities that utilize two common industry data formats; STEP and IDX.

Eventually, almost every PCB must be housed inside something. Utilizing STEP and IDX, OrCAD PCB allows the import and visualization of physical and mechanical elements, such as enclosures, to help ensure the board is designed correctly with respect to mechanical requirements. Design data—including the ability to collaborate on changes through a managed, bidirectional change-review and approval process—can also be shared between electrical and mechanical teams..

Bidirectional ECAD-MCAD Collaboration

The IDX format and supporting CAD tool interfaces provide capabilities that allow changes between electrical and mechanical teams to be previewed graphically before accepting or rejecting incremental changes. The benefit of this flow is that design teams can more closely collaborate on changes, consider effects on the other, make changes early and as often as needed, and remain in sync throughout the design process.

Accurate 3D Visualization

STEP models for components give designers the ability to see a realistic three-dimensional representation of the design in the context of the enclosure and/or other

physical limitations that must be accounted for. Within the 3D visualization environment, it is easy to perform visual clearance checks to detect conflicts or issues early on and ensure PCB components and mechanical elements have the proper fit prior to manufacture.

“The very first STEP model that I tested (a shield generated by our mechanical department), I found to be placed 180 degrees out,” said Dave Elder, PCB design manager at Tait Radio. “This would have been very difficult to detect without a 3D view. This has allowed us to avoid a potential pitfall, not to mention a sum of money in rework.”

Cadence OrCAD 16.6 Available Now

The IDX interface/flow for OrCAD PCB is available in the 16.6 base release and the STEP capabilities are available in the 16.6 Quarterly Update Release #2.

For more information on the OrCAD 16.6 release, please contact your local Cadence channel partner: www.cadence.com/Alliances/channel_partner/pages/default.aspx.



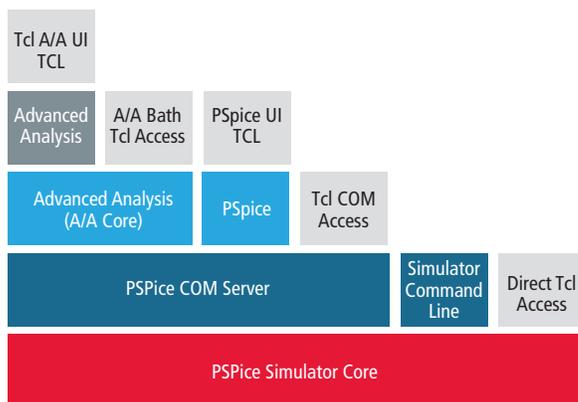
Cadence OrCAD 16.6

Expanded Tcl/HTML5 Capability and New Marketplace Apps

Expanded Tcl/HTML5 Capability

The Cadence® OrCAD® 16.6 release enables deeper access to design data along with greater control over GUI and feature manipulation, providing new levels of customization capabilities for Capture and now PSpice®. The open architecture platform continues to evolve with support for hybrid applications that utilize JavaScript and HTML5. Custom flows and external application integration, such as HTML-based corporate or web services, can greatly extend the OrCAD design environment.

PSpice has also been enhanced with a Tcl/HTML5 programming infrastructure that allows customization of simulations with parameters/algorithms and .DAT file output. This enables you to enhance the simulation environment for specific flows, needs, or control, and allows you to leverage external data or conditions to enhance design simulations.



Tcl/HTML5 customization has also been extended to new user configurable areas of both Capture and PSpice, including Tcl/HTML5-based features, GUI menus and toolbars, and CIS.

User-configurable menus and toolbars: Menus, toolbars, and icons in OrCAD Capture, PSpice Advanced Analysis, and Model Editor can now be customized. This makes it possible to run any Tcl/HTML5 method or script from the menus.

Quick-place for common simulation components: A new menu, Place ->PSpice Component, enables quick-place for commonly used schematic or simulation components. The menu items list of components is user-configurable and has been pre-populated with common PSpice simulation devices (passive, discrete, sources, digital).

Action customization for CIS Explorer: CIS Explorer can be custom-configured with user-definable actions and capabilities. (For example, customized part placement checks can disable placement of an EOL part or provide a warning if part procurement has a long lead time). Query result rows can also be customized. (For example, rows can be highlighted blue for recommended parts or red for parts not recommended or not allowed.)

Custom design rule check (DRC): User-specific schematic and circuit checks can be created and added to the Capture DRC routines. Several examples include checks for hanging wires, device pin mismatches, overlapping wires, reference prefix mismatches, port-pin mismatches, and shorted discrete parts.

New Marketplace Apps

Since the launch of Marketplace, the number of OrCAD apps has steadily grown with additions from Cadence and our partners. New productivity apps since the 16.6 release include datasheet display at a component level, logo scaling for PCB, and Capture-cloud connector.

One of the newest productivity apps, PSpice Modeling app, takes advantage of the expanded Tcl capabilities and HTML5 in the 16.6 release; support for rapid simulation modeling development applications.

The new PSpice Modeling app significantly reduces the time and effort to develop components and models needed for simulation during design entry. Models are created with user-input parameters—as from a datasheet—and a schematic component is automatically placed and associated with the newly created model without having to leave the OrCAD Capture canvas. The modeling application also automatically manages the simulation profile configuration without needing library set up for simulation.

The screenshot displays the PSpice Modeling app interface. On the left, a table lists parameters for an exponential wave:

Parameter Name	Param	Value
V1	1	
V2	5	
Rise Delay	1	
Rise Time Constant	0.2	
Fall Delay	2	
Fall Time Constant	0.5	
AC	0	
DC	0	

In the center, an 'EXPONENTIAL Wave' graph shows a rising and falling voltage source. On the right, a properties table for the component is shown:

A	
Example - PAGE1: V	
AC	0
Color	Default
DC	0
Designator	
FT	ExponentialVoltage@B
Graphic	VEXP Normal
ID	
Implementation Path	
Implementation Type	PSpice Model
Location X-Coordinate	460
Location Y-Coordinate	160
Part Reference	V4
PCB Footprint	
Pin Visible	DEFAULT
PSpiceTemplate	V@REFDES %s %s %DCI
Reference	V4
Source Library	D:\SPRIBR\TOOLS\IC4
Source Package	VEXP
Source Part	VEXP Normal
TC1	0.2
TC2	0.5
TD1	1
TD2	2
V1	1
V2	5
Value	VEXP

Below the graph, a schematic component is shown with the following parameters:

V1 = 1
V2 = 5
TD1 = 1
TC1 = 0.2
TD2 = 2
TC2 = 0.5

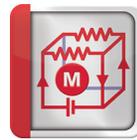
OrCAD Capture Marketplace

Launched in the fall of 2011 as part of the OrCAD 16.5 release, OrCAD Capture Marketplace delivers an integrated, online resource with specialized content and tools for OrCAD Capture and PCB Editor users. Component information, simulation models, educational media, and add-on applications (apps) provide easily assessable design content along with the ability to customize and extend the OrCAD design flow and environment.

Enabling an extensible and customizable design environment, OrCAD's open architecture platform incorporates a highly integrated Tcl/HTML5 programming infrastructure that allows the creation or enhancement of features, functionality, design capabilities, and flows. This development environment is available to customers and partners to quickly develop solutions for their specific needs. Created and delivered as downloadable apps through OrCAD Marketplace, new features, functionality, or design capabilities are available from Cadence or our partners.

Many apps available from Marketplace (www.orcadmarketplace.com/onlinestore) are free or have a free trial option so you can test the app before purchasing.

Popular apps include:



PSpice Modeling App – Free
Build PSpice-ready simulation models from datasheet specs in seconds.



GraserWARE DesignCompare – Free
Easily compare part and connection relationships between two versions of a schematic.



Part Link Powered by Digi-Key – Free
Download component data from Digi-Key directly into Capture schematics.



Capture INI Manager App – Free
Manage and administrate company-wide Capture .ini files and settings.

Cadence OrCAD 16.6 is Available Now.

For more information on the OrCAD 16.6 release, please contact your local Cadence channel partner: http://www.cadence.com/Alliances/channel_partner/pages/default.aspx.



Cadence is transforming the global electronics industry through a vision called EDA360. With an application-driven approach to design, our software, hardware, IP, and services help customers realize silicon, SoCs, and complete systems efficiently and profitably. www.cadence.com