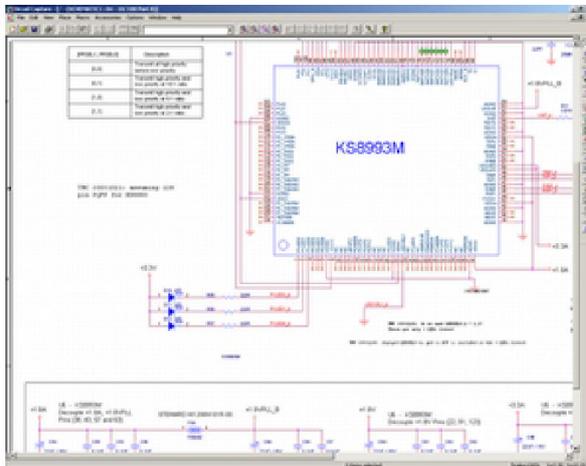


OrCAD

This article **contains content that is written like an advertisement**. Please help [improve it](#) by removing [promotional content](#) and inappropriate [external links](#), and by adding encyclopedic content written from a [neutral point of view](#). *(July 2018)* ([Learn how and when to remove this template message](#))

"PSPICE" redirects here. For the spice, see [Pumpkin pie spice](#). For the seasonal beverage, see [Pumpkin Spice Latte](#).

OrCAD



OrCAD Schematic Capture Program

<u>Original author(s)</u>	OrCAD Systems Corporation
<u>Developer(s)</u>	Cadence Design Systems
Initial release	1985
<u>Stable release</u>	17.4 / November 2019
Written in	C/C++
<u>Operating system</u>	Windows (originally DOS)
<u>Type</u>	Electronic design automation
<u>License</u>	Proprietary
Website	www.orcad.com

OrCAD Systems Corporation was a software company that made **OrCAD**, a proprietary software tool suite used primarily for [electronic design automation](#) (EDA). The software is used mainly by electronic design engineers and electronic technicians to create electronic schematics, perform mixed-signal simulation and electronic prints for manufacturing printed circuit boards.

OrCAD was taken over by [Cadence Design Systems](#) in 1999 and was integrated with [Cadence Allegro](#) since 2005.

The name OrCAD is a portmanteau, reflecting the company and its software's origins: *Oregon* + [CAD](#).



Contents

- [1 Company](#)
- [2 Products](#)
 - [2.1 OrCAD Capture](#)
 - [2.2 OrCAD EE PSpice](#)
 - [2.2.1 History](#)
 - [2.2.2 Analyses](#)
 - [2.3 OrCAD PCB Designer](#)
- [3 See also](#)
- [4 References](#)
- [5 External links](#)

Company

Founded in 1985 by John Durbetaki, Ken and Keith Seymour as "OrCAD Systems Corporation" in Hillsboro, Oregon, the company became a supplier of desktop electronic design automation (EDA) software. In 1984 Durbetaki began designing an expansion chassis for the [IBM PC](#). Durbetaki, who had left [Intel Corp.](#) after five years as an engineer and project manager, decided, along with brothers Keith and Ken Seymour, to start his own company to develop add-on instrumentation for the PC.^[1] Durbetaki began creating his own schematic capture tool for his use in the PC expansion chassis project; but eventually shelved the hardware project entirely in favor of developing low-cost, PC-based CAD software. The company's first product was SDT (Schematic Design Tools) for [DOS](#), which shipped first in late 1985.

In 1986, OrCAD hired Peter LoCascio to develop sales and co-founder Ken Seymour left the company. The flagship SDT product was soon followed with a digital simulator, VST (Verification and Simulation Tools) and printed circuit board (PCB) layout tools.^[2]

Over time, OrCAD's product line broadened to include [Windows](#)-based software products to assist electronics designers in developing [field-programmable gate arrays](#) (FPGAs), including [complex programmable logic devices](#) (CPLDs). In 1991, Durbetaki, then CEO and head of R&D, left the company. He was succeeded as CEO by Michael Bosworth.

In June 1995, OrCAD acquired Massteck Ltd.,^{[3][4]} a small company that offered a printed circuit board layout tool and a sophisticated autorouter,^[5] and [Intelligent Systems Japan, KK](#), OrCAD's distributor in Japan. In 1996, OrCAD made a public offering.^{[6][7]}

In late 1997 and early 1998, OrCAD and MicroSim Corporation merged, a business combination that ultimately proved to be disappointing. MicroSim has been a supplier of PC-based analog and mixed-signal simulation software for designing printed circuit board systems ([PSpice](#)).^{[8][9]}

On 16 July 1999, the company and its products were acquired by former competitor [Cadence Design Systems](#).^{[10][11][12]}

OrCAD Layout has been discontinued. The latest iteration of OrCAD CIS schematic capture software has the ability to maintain a database of available integrated circuits. This database may be updated by the user by downloading packages from component manufacturers, such as [Analog Devices](#)^[13] and others. Another announcement was that [ST Microelectronics](#) will offer OrCAD PSpice models for all the power and logic semiconductors, since PSpice is the most used circuit simulator.^[14] [Intel](#) offers reference PCBs designed with Cadence PCB Tools in the OrCAD Capture format for embedded and personal computers.

Products

OrCAD is a suite of products for PCB Design and analysis that includes a schematic editor ([Capture](#)), an analog/mixed-signal circuit simulator ([PSpice](#)) and a PCB board layout solution (PCB Designer Professional).

OrCAD Capture

OrCAD Capture is a [schematic capture](#) application, and part of the OrCAD circuit design suite.^[15]

Unlike [NI Multisim](#), Capture does not contain in-built simulation features, but exports [netlist](#) data to the simulator, OrCAD EE. Capture can also export a hardware description of the circuit schematic to [Verilog](#) or [VHDL](#), and netlists to circuit board designers such as OrCAD Layout, Allegro, and others.^[16]

Capture includes a component information system (CIS), that links component package footprint data or simulation behavior data, with the circuit symbol in the schematic.^[16]

Capture includes a [Tcl/Tk](#) scripting functionality that allows users to write scripts, that allow customization and automation. Any task performed via the GUI may be automated by scripts.^[16]

The OrCAD Capture Marketplace enables customers to share and sell add-ons and design resources. Such add-ons can customize the design environment and add features and capabilities.^[16]

Capture can interface with any database which complies with [Microsoft's ODBC](#) standard etc. Data in an [MRP](#), [ERP](#), or [PDM](#) system can be directly accessed for use during component decision-making process.

OrCAD EE PSpice

OrCAD EE PSpice is a [SPICE circuit simulator](#) application for simulation and verification of analog and mixed-signal circuits.^[17] PSpice is an acronym for *Personal Simulation Program with Integrated Circuit Emphasis*.

OrCAD EE typically runs simulations for circuits defined in OrCAD Capture, and can optionally integrate with [MATLAB/Simulink](#), using the *Simulink to PSpice Interface* (SLPS).^[18] OrCAD Capture and PSpice Designer^[19] together provide a complete circuit simulation and verification solution with schematic entry, native analog, mixed signal, and analysis engines.

PSpice was a modified version of the academically developed SPICE, and was commercialized by MicroSim in 1984. MicroSim was purchased by OrCAD a decade later in 1998.

OrCAD PSpice Designer is available in two options: PSpice Designer and PSpice Designer Plus.

OrCAD PSpice Designer includes OrCAD Capture and OrCAD PSpice solution. An upgrade option to PSpice Designer Plus provides the PSpice Advanced Analysis^[20] simulation engine for functional simulation and improvement in design yield and reliability.

The PSpice Advanced Analysis simulation capabilities covers various analyses- Sensitivity, Monte Carlo, Smoke (Stress), Optimizer, and Parametric Plotter providing in depth understanding of circuit performance beyond basic validation.

The OrCAD PSpice Simulink - PSpice Integration(SLPS)^[21] provides co-simulation and helps verify system level behavior.

A circuit to be analyzed using PSpice is described by a circuit description file, which is processed by PSpice and executed as a simulation. PSpice creates an output file to store the simulation results, and such results are also graphically displayed within the OrCAD EE interface.

OrCAD EE is an upgraded version of the PSpice simulator, and includes automatic circuit optimization and support for waveform recording, viewing, analysis, curve-fitting, and post-processing.^{[17][22]} OrCAD EE contains an extensive library of models for physical components, including around 33,000 analog and mixed-signal devices and mathematical functions.^[17] OrCAD EE also includes a model editor, support for parameterized models, auto-convergence and checkpoint restart, several internal [solvers](#) and a magnetic part editor.

History

SPICE was first developed at the University of California, Berkeley, in the early 1970s. Subsequently an improved version SPICE 2 was available in the mid-1970s especially to support computer aided design.

PSpice was released in January 1984, and was the first version of UC Berkeley SPICE available on an IBM Personal Computer. PSpice later included a waveform viewer and analyser program called Probe. Subsequent versions improved on performance and moved to [DEC/VAX](#)

[minicomputers](#), Sun workstations, Apple Macintosh, and Microsoft Windows. Version 3.06 was released in 1988, and had a "Student Version" available which would allow a maximum of up to ten transistors to be inserted. PSpice (even the student version) increases the students' abilities to understand the behavior of electronic components and circuits.^{[23][24]}

Analyses

Main article: [PSpice circuit file](#)

The type of simulation performed by PSpice depends on the source specifications and control statements. PSpice supports the following types of analyses:

- DC Analysis - for circuits with time-invariant sources (e.g. steady-state DC sources). It calculates all nodal voltages and branch currents over a range of values. Supported types include Linear sweep, Logarithmic sweep, and Sweep over List of values.
- Transient Analysis - for circuits with time variant sources (e.g., sinusoidal sources/switched DC sources). It calculates all nodal voltages and branch currents over a time interval and their instantaneous values are the outputs.
- AC Analysis - for small signal analysis of circuits with sources of varying frequencies. It calculates the magnitudes and phase angles of all nodal voltages and branch currents over a range of frequencies.

The operating temperature of an analysis can be set to any desired value, and nodal parameters are assumed to be measured at a nominal temperature, by default 27 °C.

PSpice User Community

[PSpice.com](#) is a PSpice User Community, an open platform dedicated to PSpice Spice circuit simulation discussions. It is a web portal with access to resources for all things related to PSpice circuit simulator. Users can find datasheets, application notes, tutorials, videos, and also information about regional PSpice training events and webinars. PSpice web portal provides extensive model library of more than 33,000 PSpice models which are also easily available with the PSpice Lite Download.

PSpice Lite version, which can be used by students comes with full functionality of the software, limited only by size and complexity.

OrCAD PCB Designer

OrCAD PCB Designer is a [printed circuit board](#) designer application, and part of the OrCAD circuit design suite.^[25] PCB Designer includes various automation features for PCB design, board-level analysis and design rule checks (DRC).

The PCB design may be accomplished by manually tracing PCB tracks, or using the Auto-Router provided. Such designs may include curved PCB tracks, geometric shapes, and ground planes.^[26]

PCB Designer integrates with OrCAD Capture, using the component information system (CIS) to store information about a certain circuit symbol and its matching PCB footprint. [\[16\]\[25\]](#)