🖂 Share   🗐 Subscribe   🗐 Contact   🖶 Print

OrCAD Downloads

Note: To download the files we recommend using the Internet Explorer or Firefox web browsers.
All OrCAD downloads require a valid email address.

High-speed PCB layout, routing, and manufacturing output

The latest product downloads are available free to all registered Cadence® customers. In some cases, you'll be asked to complete a brief user profile. All information supplied is used only to help us develop future products and technology.

- OrCAD PCB Designer Lite DVD (All Products)
- OrCAD PCB Designer Lite DVD (Capture & PSpice only)
- CIS Admin Tool 10.x
- CIS Admin Tool 16.2
- OrCAD Capture/OrCAD Capture CIS ViewReader
- OrCAD CIS Wizard
- PSpice Schematics Installer
- Third-party translator
- PSpice models
- Allegro/OrCAD Starter Library

Top

OrCAD PCB Designer Lite DVD

OrCAD 16.5 Demo Software (All Products)

Designers around the world rely on the powerful yet intuitive Cadence® OrCAD® personal productivity tools. OrCAD has a long history of providing individuals and teams with a complete set of technologies that offer unprecedented productivity, seamless tool integration, and exceptional value—the OrCAD 16.5 release continues with that tradition.

The OrCAD 16.5 demo software will let you experience all the features and functionality of the actual software*. So go ahead, discover how easy it is to use these state-of-the-art OrCAD technologies.

* Limitations are in the size and complexity of the design.

The OrCAD PCB Designer Lite DVD includes demo versions of the following tools: OrCAD Capture, OrCAD Capture CIS Option, PSpice® A/D, PSpice A/A, OrCAD PCB Editor and SPECCTRA® for OrCAD.

Download FREE - OrCAD 16.5 demo software (851MB) includes OrCAD Flow Tutorial with the example design files **(You must use the path/folder option in your zip tool when extracting this archive.)**

Submit request to receive a hard copy of the OrCAD 16.5 PCB Designer Lite DVD.

Top

OrCAD PCB Designer Lite DVD (Capture & PSpice only)

OrCAD 16.5 Demo Software (Capture and PSpice only)

Designers around the world rely on the powerful yet intuitive Cadence® OrCAD® personal productivity tools. OrCAD has a long history of providing individuals and teams with a complete set of technologies that offer unprecedented productivity, seamless tool integration, and exceptional value—the OrCAD 16.5 release continues with that tradition.

The OrCAD 16.5 demo software will let you experience all the features and functionality of the actual software*.

Highlights

- Webinar: PCB Library Development – Build OrCAD Symbols and IPC-7351 compliant footprints in a fraction of the time with EDABuilder
- What's Good About OrCAD Apps? You Can Try Them for Free!
- FPGA-PCB codesign; a 21st Century approach to integrating fpgas into the pcb design process

Recent Blog Posts
- Altera 28 Gbps Stratix V IBIS-AMI Models Now Blazing Channels with Allegro PCB SI
- What's Good about OrCAD Apps? Symbol and Footprint Creation Just Got a Lot Easier!

Visit the Design Community »

Related Info

- Technical papers
- PCB Service Bureaus
- Cadence Channel Partners (VARs) for OrCAD products
- OrCAD downloads
- Allegro PCB Design Solutions

I want to learn more!
For OrCAD product sales, support, and training, contact a Cadence Channel Partner near you!
More info ▶

So go ahead, discover how easy it is to use these state-of-the-art OrCAD technologies.

* Limitations are in the size and complexity of the design.

The OrCAD 16.5 PCB Designer Lite includes demo versions of the following tools: OrCAD Capture, OrCAD Capture CIS Option, PSpice® A/D, PSpice A/A

Download FREE - OrCAD 16.5 demo software (495MB) includes OrCAD Flow Tutorial with the example design files **(You must use the path/folder option in your zip tool when extracting this archive.)**

Submit request to receive a hard copy of the OrCAD 16.5 PCB Designer Lite DVD.

Top

CIS Admin Tool for OrCAD Capture CIS

CIS Administration Tool for OrCAD Capture CIS Version 10.3, 10.5

If your OrCAD® Capture CIS designers work in a multi-user networked environment, you can leverage CIS functionality for easier management of your group's part and footprint libraries and files you can browse. Placing libraries and files in a designated location forces all engineers to pull information from a central source. This not only promotes data integrity, since all users are getting part information from a common source, but also eases the burden of administrating libraries.

Download FREE CIS Admin Tool 10.x - 175 KB

Top

CIS Admin Tool for OrCAD Capture CIS 16.2

CIS Administration Tool for OrCAD Capture CIS Version 16.2

If your OrCAD® Capture CIS designers work in a multi-user networked environment, you can leverage CIS functionality for easier management of your group's part and footprint libraries and files you can browse. Placing libraries and files in a designated location forces all engineers to pull information from a central source. This not only promotes data integrity, since all users are getting part information from a common source, but also eases the burden of administrating libraries.

Download FREE CIS Admin Tool 16.2 - 175 KB

Top

OrCAD Capture/OrCAD Capture CIS ViewReader

You can use the Cadence® OrCAD® ViewReader to convert Viewlogic ViewDraw designs in OrCAD Capture or OrCAD Capture CIS versions 7.2, v9, v9.1, v9.2, v9.2.1, 10.X and 15.7.

If you're an OrCAD Capture 9.2 user, please remember to do the following:

- Select the OrCAD Capture v9.10 option on the first installation dialog box. The ViewReader is the same for Release 9.1, 9.2, 9.2.1, 10.X and 15.7
- Do not install the ViewReader in your OrCAD Capture directory. The ViewReader contains files that will overwrite files you need to use OrCAD Capture

Download FREE OrCAD ViewReader - 2 MB

Top

CIS Wizard

The CIS Wizard—Database creation made easy

The CIS Wizard has been created for users of OrCAD® Capture CIS 10.0 (and later) to simplify and assist in the creation of the Engineering Part Database. Following the installation of the CIS Database Wizard, you can create a new database or update an existing database from your schematic design files.

The CIS Database Wizard will install and run from within your OrCAD Capture CIS 10.0 (and later) installation. Taking an existing schematic design you can generate a new database in one of three different formats: Microsoft Access, Microsoft Excel, or comma-delimited text. The CIS Database Wizard has the ability to update an existing database using the Microsoft Access format only.

System Requirements:

- OrCAD Capture v10.0 or later
- Windows 2000, Windows XP, Windows XP Home

[Download FREE CIS Wizard](#) - 4.2 MB

PSpice Schematics Installer

The development and support for the Cadence® PSpice® Schematics product has been discontinued with the OrCAD® 10.0 release. It is imperative that Cadence® continually evaluate its product offerings to guarantee that we are maintaining our focus and providing you with the most effective and efficient products.

You are strongly urged to fulfill your schematic entry needs with OrCAD Capture ,Allegro® Design Entry CIS, Allegro® Design Entry HDL.

We will continue distribution of PSpice Schematics as a download from www.cadence.com as an aid to our customers who wish to continue using it. Cadence may discontinue subsequent versions of PSpice Schematics at any time.

Please [contact](#) your local Cadence Channel Partner for information on moving to OrCAD Capture or Allegro Design Entry HDL.

[Download the PSpice Schematic v16.5 Executable](#) - 7.6MB
[Download the PSpice Schematic v16.3 Executable](#) - 5.6MB
[Download the PSpice Schematic v16.2 Executable](#) - 3.2MB
[Download the PSpice Schematic v16.0 Executable](#) - 3.19MB
[Download the PSpice Schematic V15.7 Executable](#) - 2.67MB

Third-party translator

Use MOS Level 49 and other models in libraries supplied by foundries in a third-party format with PSpice®.

System Requirements:

- OrCAD® Capture v10.0 or later
- Windows 2000, Windows XP, Windows XP Home

[Download Third-party Library Translator](#) - 250 KB

PSpice models

PSpice® application engineers have scoured the web for vendor models. Do you know of other vendors who supply models via the web? [Send us](#) the URL and we'll add it to this list.

You are strongly urged to fulfill your schematic entry needs with OrCAD® Capture or Allegro® Design Entry HDL.

- [STMicroelectronics Simulation Models:](#) Amplifier and Comparators ICs, Diodes, Memories, Thyristors and AC Switches, Transistors. **NEW!**
- [Microchip Technology, Microchip, op-amps, mosfets](#) **NEW!**
- [Comlinear (now National) SPICE Macromodels](#): Buffers, Op-amps (BJTs, FET), Op-amp (CMOS)
- [Infineon Technologies AG (formerly Siemens):](#) Mosfets, Diodes, IGBTs, MOVs, Inductors
- [Intersil Corporation (formerly Harris Semiconductor)](#): Intelligent Power, Analog Signal Processing, Microcontroller Development, Digital Signal Processing
- [International Rectifier-PSpice Models](#): Power Mosfets, Rectifiers, IGBTs, Hexfreds, Schottky diodes
- [Maxim](#): Buffers, Op-amps (BJTs, FET), Op-amp (CMOS)
- [PolyFet PSpice Models](#): RF Power Mosfets
- [Shindengen Semiconductor](#): Diodes and Diode Bridges
- [Zetex Semiconductors PSpice Models](#): BJTs, Power BJTs, Darlington BJTs, Diodes, Power Mosfets. On the website, choose 'Product' on the top navigation menu, and then choose 'Applications' and 'Spice Models' from the left navigatio menu

Other useful sites:

Cadence OrCAD Downloads

- Microchip Op- Amp spice Macro Models-Application note **NEW!**
- Microchip's Power MOSFET Driver Simulation Models **NEW!**
- Using Texas Instruments SPICE Models in PSpice

Vendor-contributed Models
These models are made available exactly as submitted by the vendors. They have not been reviewed or verified by OrCAD engineers.

Vendor	PSpice Model	Description
Advanced Linear Devices	adv_lin.lib	Library of op-amps
Advanced Linear Devices	adv_lin.olb	Op-amp Capture symbols
Analog Devices Inc.	anlg_dev.lib	Library of op-amps, transistor arrays, analog aultipliers, buffer, switches, voltage references
Analog Devices Inc.	anlg_dev.olb	Op-amp, transistor array, analog aultiplier, buffer, switch, voltage reference Capture symbols
Apex Microtechnology Corp.	apex.lib	Library of power op-amps
Apex Microtechnology Corp.	apex.olb	Power op-amp Capture symbols
Burr-Brown Corp.	burr_brn.lib	Library of op-amps
Burr-Brown Corp.	burr_brn.olb	Op-amp Capture symbols
California Eastern Laboratories	cel.lib	Library of CEL/NEC RF devices
California Eastern Laboratories	cel.olb	CEL/NEC RF device Capture symbols
Comlinear	comlinr.lib	Library of op-amps
Comlinear	comlinr.olb	Op-amp Capture symbols
Coilcraft, Inc **NEW!**	Ccrfind.lib	Coilcraft RF Inductor PSpice models
Coilcraft, Inc **NEW!**	Ccrfind.olb	Coilcraft RF Inductor Capture Symbols
Elantec Inc.	elantec.lib	Library of op-amps, transistor arrays
Elantec Inc.	elantec.olb	Op-amp, transistor array Capture symbols
Epcos	epcos.lib	Library of varsistors(MOV), inductors and NTC thermistors
Epcos	epcos.olb	Varsistor(MOV), inductor and NTC thermistor Capture symbols
Fairchild	fairchild.lib	Library of power MOSFETs
Fairchild	fairchild.olb	Power MOSFET Capture symbols
F. W. Bell	fwbell.lib	Library of generator models
F. W. Bell	fwbell.olb	Generator model Capture symbols
Harris Semiconductor	harris.lib	Library of op-amps, transistor arrays, power MOSFETs, MCTs, half-bridge drivers
Harris Semiconductor	harris.olb	Op-amp, transistor array, power MOSFET, MCT, half-bridge driver Capture symbols
Infineon	infineon.lib	Library of semiconductors
Infineon	infineon.olb	Semiconductor Capture symbols
IXYS	ixys.lib	Library of IGBTs and power MOSFETs
IXYS	ixys.olb	IGBT and power MOSFET Capture symbols
Linear Technology Corp.	lin_tech.lib	Library of op-amps
Linear Technology Corp.	lin_tech.olb	Op-amp Capture symbols
Maxim Integrated Circuits	maxim.lib	Library of op-amps and comparators

Maxim Integrated Circuits	maxim.olb	Op-amp and comparator Capture symbols
Motorola	motor_rf.lib	Library of RF bipolar transistors
Motorola	motor_rf.olb	RF bipolar transistor Capture symbols
Motorola	motorsen.lib	Library of pressure sensors
Motorola	motorsen.olb	Pressure sensor Capture symbols
National Semiconductor	nat_semi.lib	Library of op-amps
National Semiconductor	nat_semi.olb	Op-amp Capture symbols
NEC	nec_mos.lib	Library of power MOSFETs
NEC	nec_mos.olb	Power MOSFET Symbols
ON Semiconductor	on_amp.lib	Library of op-amps
ON Semiconductor	on_amp.olb	Op-amp Capture symbols
ON Semiconductor	on_bjt.lib	Library of BJTs
ON Semiconductor	on_bjt.olb	BJT Capture symbols
ON Semiconductor	on_diode.lib	Library of diodes
ON Semiconductor	on_diode.olb	Diode Capture symbols
ON Semiconductor	on_mos.lib	Library of power MOSFETs
ON Semiconductor	on_mos.olb	Power MOSFET Capture symbols
ON Semiconductor	on_pwm.lib	Library of PWMs
ON Semiconductor	on_pwm.olb	PWM Capture symbols
Philips	phil_bjt.lib	Library of small signal transistors
Philips	phil_bjt.olb	Small signal transistor Capture symbols
Philips	phil_fet.lib	Library of field effect transistors
Philips	phil_fet.olb	Field effect transistor Capture symbols
Philips	phil_rf.lib	Library of RF bipolar transistors
Philips	phil_rf.olb	RF bipolar transistor Capture symbols
Polyfet RF Devices	polyfet.lib	Library of RF device MOSFETs
Polyfet RF Devices	polyfet.olb	RF device MOSFETs Capture symbols
Shindengen	shindngn.lib	Library of diodes and diode bridges
Shindengen	shindngn.olb	Diode and diode bridge Capture symbols
Texas Instruments	tex_inst.lib	Library of op-amps, voltage comparators
Texas Instruments	tex_inst.olb	Op-amp and voltage comparator Capture symbols
Zetex Inc.	zetex.lib	Library of PLC Diodes, BJTs, Darlingtons and MOSFETs
Zetex Inc.	zetex.olb	PLC Diode, BJT, Darlington and MOSFET Capture symbols

Top

Allegro / OrCAD Starter Library

The Allegro® /OrCAD® Starter Library 1.0 is a free library that includes Allegro Design Entry HDL, Allegro Design Entry CIS, and OrCAD Capture schematic symbols along with Allegro / OrCAD PCB Editor footprints and the necessary component properties. It is designed for new customers who are evaluating or implementing a Cadence PCB flow or wanting to build a fully compatible library for use with the Allegro or OrCAD family of PCB design tools. A documented catalog of the Allegro / OrCAD Starter Library contents, as well as sample specifications used in the development of the library is available in Adobe .pdf format.

- Allegro / OrCAD Starter Library Installation README Download

Cadence OrCAD Downloads

- Allegro / OrCAD Starter Library Catalog Download
- Allegro / OrCAD DE-HDL Schematic Symbol - Sample Specification Download
- Allegro / OrCAD PCB Editor Footprint - Sample Specification Download
- Allegro / OrCAD Starter / Evaluation Library Download

This "starter" or baseline library is provided free of charge for use with Allegro Design Entry HDL, OrCAD Capture, Allegro Design Entry CIS and Allegro / OrCAD PCB Editor. It can be used 'out-of-the-box' or as a template from which customized parts can be created. The library is provided as a zipped archive, with installation and usage instructions included.