

OrCAD Lite Products Reference

Product Version 16.5
May 2011

© 1991 – 2011 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Product PSpice contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Patents: Cadence product PSpice, described in this document, is protected by U.S. Patents 5,481,695; 5,510,998; 5,550,748; 5,590,049; 5,625,565; 5,715,408; 6,516,447; 6,594,799; 6,851,094; 7,017,137; 7,143,341; 7,168,041; 7,398,500; 7,464,358; 7,536,665; 7,562,330; 7,562,331; 7,574,686; 7,620,922; 7,761,836; 7,793,249.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

<u>1</u>	
<u>Lite Version of OrCAD Products</u>	5
<u>Limits of the Lite Version of OrCAD Products</u>	5
<u>Getting Started with OrCAD Lite Products</u>	9
<u>Accessing OrCAD Documentation</u>	10
<u>Accessing Online Help</u>	10
<u>Accessing Online Books</u>	10
<u>Accessing OrCAD Tutorials</u>	11
<u>Accessing OrCAD Web Sites</u>	11

OrCAD Lite Products Reference

Lite Version of OrCAD Products

The Lite version of Cadence OrCAD personal productivity tools have certain limitations in the size and complexity of the designs you can use with these products. You might use the lite versions of Cadence OrCAD personal productivity tools because:

- You downloaded and installed a Lite release version of OrCAD 16.5: Depending upon the lite release version you have downloaded, you will install lite versions of different sets of productivity tools.
- You do not have license to installed OrCAD products: OrCAD Products start in the lite mode if license for these products are not available. You can change to production mode by making the license available.

This document is designed to so that you can quickly find information to get started with the Lite products.

Limits of the Lite Version of OrCAD Products

The lite version of OrCAD products have the following limits with design size and complexity. If your design exceeds these limits, you will not be able to save your work or take your design through the flow.

OrCAD Lite Products Reference

Lite Version of OrCAD Products

Product

Limits in Lite Version

OrCAD Capture
CIS Lite

- You cannot save designs that have more than 75 nets, including the hierarchical blocks in the design. You can still view or create larger designs.
- You cannot save a design with more than 60 parts, including the hierarchical blocks in the design. You can still view or create larger designs.
- You cannot have more than 1000 parts in the Capture CIS database.
- The Internet Component Assistant (ICA) tab in the CIS Explorer window opens the About ActiveParts page (www.activeparts.com) and not the component search page.
- You cannot create parts with more than 100 pins.
- The Capture FPGA flow is not available.

OrCAD Lite Products Reference

Lite Version of OrCAD Products

Product

PSpice A/D
Lite

Limits in Lite Version

- Circuit simulation limited to circuits with up to 75 nodes, 20 transistors, no sub-circuit limits but 65 digital primitive devices, and 10 transmission lines (ideal or non-ideal) with not more than four pairwise coupled lines.
- Device characterization and parameterized part creation using the PSpice Model Editor limited to diodes.
- No limit to stimulus generation using Stimulus Editor.
- Sample model library named eval.lib (containing analog and digital parts) and evalp.lib (containing parameterized parts) are provided.
- The library nomd.lib is configured for simulations. The nomd.lib file references the set of libraries that can be used with the lite version.
- You cannot simulate parameterized parts that are not from the evalp.lib library. This library consists of parametrized resistor, source, and diode.
- You cannot use Level 3 of Core model (Tabrizi), MOSFET BSIM 3.2, or MOSFET BSIM 4 models.
- Displays only simulation data created using the lite version of the simulator.
- Magnetic Parts Editor allows you to design power transformers only. The database shipped with Magnetic Parts Editor cannot be edited and contains a single core.
- The Model Import Wizard supports parts and simulation models that have a maximum of two pins or two terminals, respectively.
- The maximum nodes in a digital circuit can be equal to or less than 250.
- The non-ideal Tline is limited to 4.

OrCAD Lite Products Reference

Lite Version of OrCAD Products

Product	Limits in Lite Version
PSpice Advanced Analysis Lite	<ul style="list-style-type: none">■ Smoke Analysis : Can run only on diodes, resistors, transistors, and capacitors.■ PSpice Advanced Analysis Optimizer<ul style="list-style-type: none">□ Only Random and MLSQ engines can be used.□ Values of up to two component parameters can be optimized.□ A maximum of one measurement specification and one curvefit specification are supported.□ Only one error calculation method supported for optimizing the curve-fit specification.■ Parametric Plotter<ul style="list-style-type: none">□ Can sweep the values of only two design and/or model parameters.□ Only Linear sweep in supported.□ A maximum of 10 sweeps allowed.□ Can evaluate the influence of changing parameter values only on one measurement expression or a trace.■ Monte Carlo/Worst Case Analysis<ul style="list-style-type: none">□ Only one measurement specification is allowed.□ A maximum of three devices with tolerance are supported.□ Maximum of 20 Monte Carlo runs are supported.■ Sensitivity Analysis<ul style="list-style-type: none">□ Only one measurement specification is allowed.□ A maximum of three devices with tolerance are supported.□ Maximum of 20 runs are supported.■ Encrypted parameterized models cannot be simulated.■ The Optimizer Random Engine can make a maximum of 5 runs.

OrCAD Lite Products Reference

Lite Version of OrCAD Products

Product	Limits in Lite Version
OrCAD PCB Editor Lite	<ul style="list-style-type: none">■ You cannot save boards that have more than 50 components and/or 100 nets. However, you can still view larger designs.■ The tutorial and tutorial design files will work with lite limitations.■ A limited sample library is provided.
OrCAD PCB Router Lite	<ul style="list-style-type: none">■ You cannot save the results of a route session.

Note: OrCAD SigXplorer is not available with the OrCAD Lite release.

Getting Started with OrCAD Lite Products

For an overview of the basic process for designing a PCB using OrCAD 16.5, refer to the OrCAD Flow Tutorial. This flow guide shows you the design cycle for an electronic design, starting with capturing the electronic circuit in OrCAD Capture, simulating the design with PSpice, through the PCB layout stages in OrCAD PCB Editor Lite and OrCAD PCB Router Lite, and finishing with the processing of the manufacturing output and maintaining the design.

Note: You can use the FULLADD design shipped as `flowtut.zip` located in the `<installation>\flowtut\tutorial_example\` folder in your installation directory to work through the design flow as described in this flow guide.

To help you quickly get started with the lite products, the table below lists the products and the online documents.

Product	Online Documents
OrCAD Capture CIS	<ul style="list-style-type: none">■ OrCAD Capture User Guide■ OrCAD Component Information System Users Guide■ Chapter 2, Creating a schematic design of OrCAD Flow Tutorial
PSpice	<ul style="list-style-type: none">■ PSpice User's Guide■ Chapter 3, Simulating a design of OrCAD Flow Tutorial

OrCAD Lite Products Reference

Lite Version of OrCAD Products

Product	Online Documents
OrCAD PCB Editor	<ul style="list-style-type: none">■ Allegro PCB Editor User Guide■ Chapter 5, Board design using OrCAD PCB Editor of OrCAD Flow Tutorial■ Allegro PCB Editor Tutorial
OrCAD PCB Router	<ul style="list-style-type: none">■ Allegro PCB Router User Guide■ Allegro PCB Router Tutorial■ Chapter 5, Board design using OrCAD PCB Editor of OrCAD Flow Tutorial

Accessing OrCAD Documentation

The user documentation for OrCAD 16.5 is in the form of online help, online books, and tutorials. For more information on OrCAD products, see [Accessing OrCAD Web Sites](#).

See the following topics for more information:

- [Accessing Online Help](#)
- [Accessing Online Books](#)
- [Accessing OrCAD Tutorials](#)
- [Accessing OrCAD Web Sites](#)

Accessing Online Help

You can access online help from the Help menu in a product, by clicking the Help button in a dialog box, or by pressing the F1 key.

Accessing Online Books

You can access the online books for your product using Cadence Help and the documentation page.

To access online books using Cadence Help:

OrCAD Lite Products Reference

Lite Version of OrCAD Products

1. From the Windows Start menu, choose the *Cadence release* and then the *Cadence Help* command.

The Cadence Help window appears.

2. Double-click the product name to display the documents for the product in the Cadence Help window.
3. Double-click a document title to open it in the Cadence Help window.

To access online books using the documentation

1. From the product *Help* menu, choose the *Documentation* command.

The product documentation page opens in the Cadence Help window.

2. Choose the appropriate tab and click the hyperlink for a document to view the document in the Cadence Help window.

Accessing OrCAD Tutorials

OrCAD products have self-paced interactive tutorials that you can use to quickly get started with the products. For information on accessing the tutorials, see [Getting Started with OrCAD Lite Products](#).

Note: You may not be able to complete some of the steps in the tutorials because of the limits to the capabilities of the lite version of OrCAD products. These limits are described in [Limits of the Lite Version of OrCAD Products](#).

Accessing OrCAD Web Sites

The following web sites provide extensive product and technical information on OrCAD products.

www.cadence.com/orcad The main website for OrCAD products. This site contains technical information on OrCAD products. It provides product updates, demos, models, tools, and utilities. There are also technical articles and application notes from OrCAD engineers and users.

www.cdnusers.org The community site where high-speed designers meet to share, contribute, and exchange information on how to best use OrCAD PCB Router.

OrCAD Lite Products Reference
Lite Version of OrCAD Products
