OrCAD 10.5: What's New

Product Version 10.5
July 2005

© 2005 Cadence Design Systems, Inc. All rights reserved.

Basic Scripting Engine provided by Cypress Software Inc. Copyright 1993 - 1998, all rights reserved.

Printed in the United States of America.

Cadence Design Systems, Inc., 555 River Oaks Parkway, San Jose, CA 95134, USA

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) 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.

All other trademarks are the property of their respective holders.

Restricted Print Permission: This publication is protected by copyright and any unauthorized use of this publication may violate copyright, trademark, and other laws. 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. This statement grants you permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used solely for personal, informational, and noncommercial purposes;
2. The publication may not be modified in any way;
3. Any copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement; and
4. Cadence reserves the right to revoke this authorization at any time, and any such use shall be discontinued immediately upon written notice from Cadence.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. The information contained herein is the proprietary and confidential information of Cadence or its licensors, and is supplied subject to, and may be used only by Cadence's customer in accordance with, a written agreement between Cadence and its customer. 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

What's New in OrCAD Capture 10.5 5
New Features and Enhancements 6
Split Part Symbol Generation Enhancement 6
Archiving Enhancement 7
Differential Pairs Between Flat Nets 8
New Part Creation from Spreadsheet 9
Replacement of Manuals Command With Documentation Command in the Help Menu 10
Multimedia Demonstrations 10
List of Important PCRs Fixed 10
What's New in OrCAD Capture CIS 10.5 13
New Features and Enhancements 14
Setting Preferences for Linking a Placed Part to a Database Part 14
Viewing Design Variants Information 14
List of Important PCRs Fixed 15
What's New in PSpice 10.5 17
New Features 18
Enhanced Behavioral support in PSpice engine 18
Support for Philips Mextram 504 model 19
Model Import Wizard 19
Transformer Designer 19
Support for using European notation 20
Behavioral Libraries 21
New Vendor models 21
Fixed PCRs 22
What's New in OrCAD Layout 10.5 23
What's New in OrCAD Layout 10.5 24
Stackup Editor 24
Enhanced selection mechanism 24
Handling good DRCs 25
Using scroll-mouse button in the Layout window 25

Support for GerbTool Version 14 25
Support for CADopia IntelliCAD Version 4 26
Product Change Requests (PCRs) addressed in Layout 10.5 27
For More Information 28
OrCAD PCB Designer (OrCAD PCB Designer with PSpice) 29
OrCAD PCB EDITOR 30
OrCAD Capture 31
SPECCTRA for OrCAD 32
PSpice A/D 33

1

What’s New in OrCAD Capture 10.5

This chapter contains the following sections describing the OrCAD Capture 10.5 release.

- New Features and Enhancements on page 6
- List of Important PCRs Fixed on page 10

New Features and Enhancements

OrCAD Capture 10.5 release brings you the following new features and enhancements:

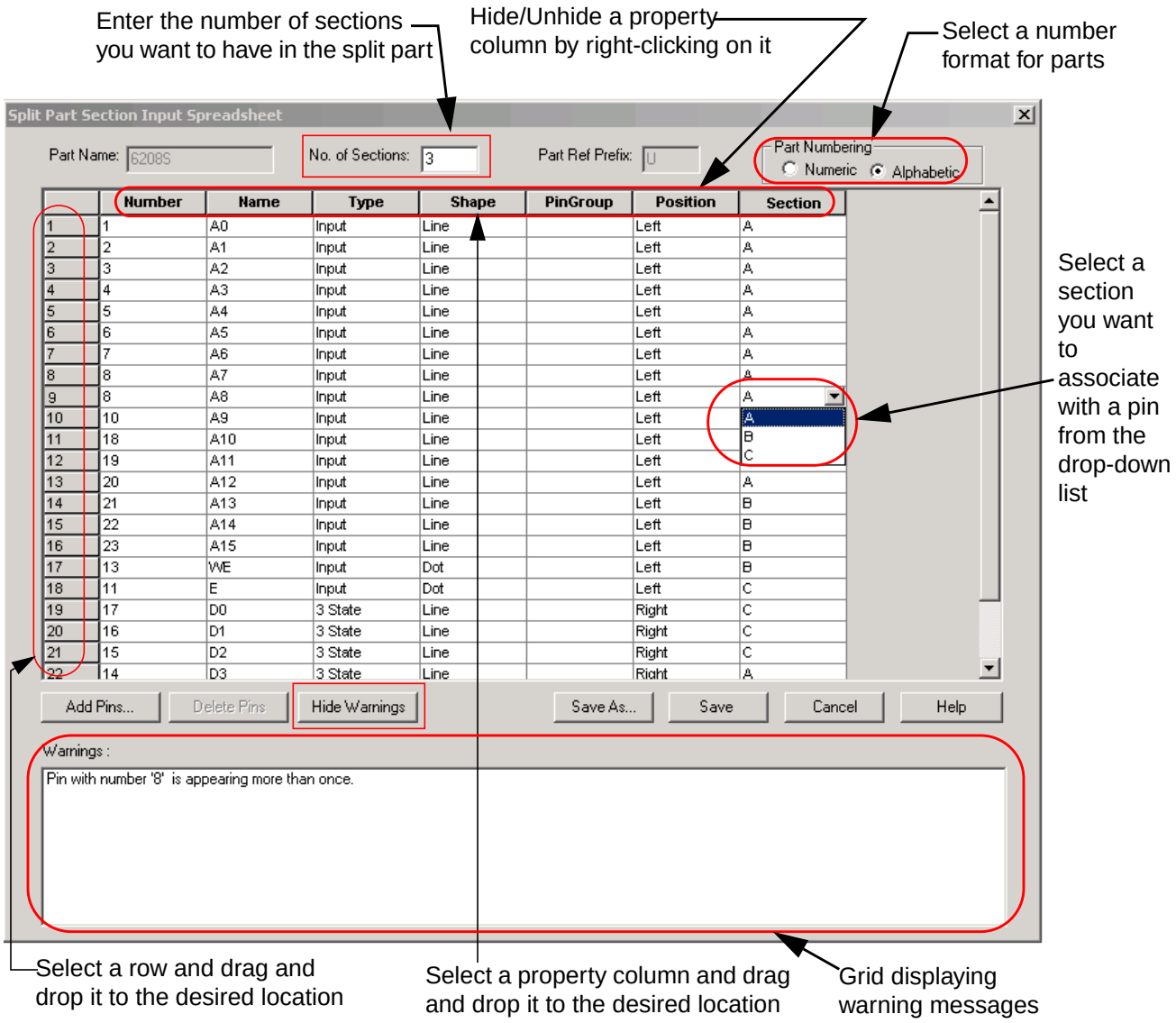
- Split Part Symbol Generation Enhancement
- Archiving Enhancement
- Differential Pairs Between Flat Nets
- New Part Creation from Spreadsheet
- Replacement of Manuals Command With Documentation Command in the Help Menu
- Multimedia Demonstrations

Split Part Symbol Generation Enhancement

The Split Part Section Input Spreadsheet in Capture 10.5 allows you to:

- Specify the number of sections into which you want to split the part.
- Choose the numbering format that should be used for numbering sections in the split part. For example, you can split a part into multiple sections that are alphabetically numbered.
- Enter or select a section from a drop-down list you want to associate with a pin.
- Add new pins to the Split Part Section Input Spreadsheet.
- Delete pins from the Split Part Section Input Spreadsheet.
- Save the selected part as a new part with a new name in the same library with changed property values.

To access the Split Part Section Input Spreadsheet, select *Tools - Split Part* or use the shortcut keys, Alt, T, L.



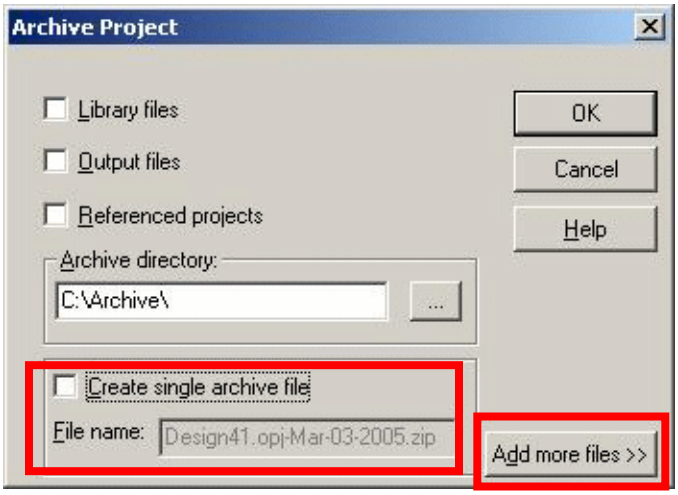
For details, see section *To split a part* in the OrCAD Capture User Guide

Archiving Enhancement

The Archive Project dialog box in Capture 10.5 allows you to save the project (.OPJ) and all the related files (design, library, and referenced projects) in a different directory and create a zip file of this directory for archival purposes. Also, you can specify any additional files or folders that you may want to be archived along with your project files.

To access the Archive Project dialog box, select *File > Archive Project* or use the shortcut keys, Alt, F, H.

To create a zip archive for you project, select the Create single archive file check box and specify the zip archive name in the File name text box. For details, see section *To create a zip archive for the project* in the OrCAD Capture User Guide.



You can also add more files and directories to your archive by clicking the Add more files >> button in the Archive Project dialog box. For details, see section *To add external files and directories to the archive* in the OrCAD Capture User Guide.

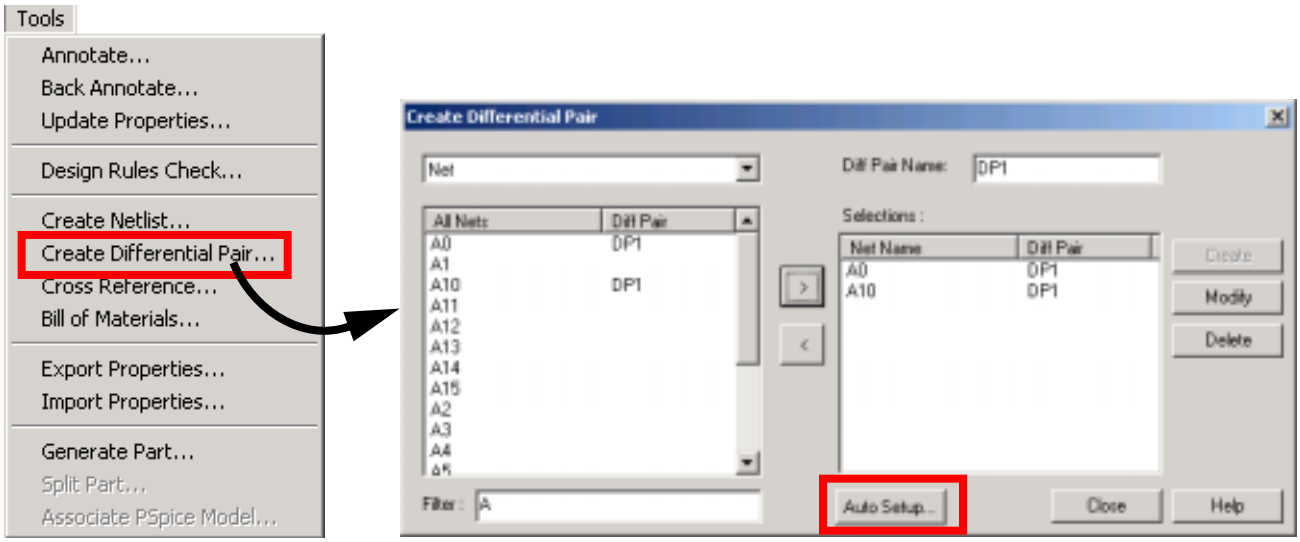
Differential Pairs Between Flat Nets

Capture 10.5 allows you to create differential pairs between two flat nets for the Capture - to-Allegro flow. In Capture, you define a differential pair between two flat nets by:

- Specifying the DIFFERENTIAL_PAIR property in the property editor.For more information, see section *Specifying the DIFFERENTIAL_PAIR property* in OrCAD Capture User Guide.
- Using the Create Differential Pair dialog box.

To access the Create Differential Pair dialog box, select *Tools > Create Differential Pair* or use the shortcut keys, ALT, T, F.

In the Create Differential Pair dialog box you can view all the nets defined in your design and create a differential pair between those nets. For more information, see section *Creating differential pairs between flat nets* in the OrCAD Capture User Guide.



You can also create multiple differential pairs between multiple nets simultaneously using the Differential Pair Automatic Setup dialog box. To access the Differential Pair Automatic Setup dialog box, click Auto Setup. For more information, see section *Creating differential pairs between flat nets* in the OrCAD Capture User Guide.

New Part Creation from Spreadsheet

Capture 10.5 has simplified the process of creating a new part in the active library. You can use the New Part Creation Spreadsheet to create new parts. The New Part Creation Spreadsheet has a spreadsheet-like interface that allows you to paste contents from a part datasheet to the spreadsheet.

To access the New Part Creation Spreadsheet, select *Design > New Part from Spreadsheet* or use the shortcut keys, Alt, D, H.

For more information about how to use the New Part Creation Spreadsheet to create new parts, see section *Creating a new part from spreadsheet* in the OrCAD Capture User Guide.

Replacement of Manuals Command With Documentation Command in the Help Menu

In Capture 10.5, the *Help* > *Manuals* command is replaced with the *Help* > *Documentaton* command.

Clicking the Documentation command will launch an HTML page containing links to all the documentation types (manuals and online help), product tutorial, and multimedia demonstrations shipped with this product release.

Multimedia Demonstrations

In Capture 10.5, you can watch multimedia demonstrations for the new features and enhancement to the existing features.

Multimedia demonstrations is available for the following new features and enhancements:

- Creating a zip archive
- Creating a differential pair between two flat nets
- Creating differential pairs between multiple pairs of flat nets simultaneously
- Creating a new part from spreadsheet

To access the multimedia demonstrations, point to the Help menu and choose the Documentation command. The Capture Help System - Documentation page opens. Click the Learning Aids tab to view all the multimedia demonstratons.

These demonstrations run as stand-alone files from the browser window and require the one-time installation of a Flash plugin.

List of Important PCRs Fixed

Some of the important PCRs addressed in this release are listed below:

PCR No.	Description
561740	Double the zoom level in Library editor. This will help in designing graphic objects. The zoom level has been increased from 504% to 1008% in the Library/Part editor.

PCR No.	Description
650279	PXL-Lite error message with no explanation.
	If this error occurs, the following error message will be displayed in the Session Log: Error [ALG0050] Duplicate Pin Name "VDD" found on Package PIC16F874 , U1 Pin Number 32: SCHEMATIC1, PAGE1 (4.50, 0.70). Please renumber one of these.
702252	Update Cache option to preserve the Reference Designator and Value properties.
706629	DRC does not report stray net connected to a BUS.
	Use Report Misleading Tap Connections check box in the Design Rule Check dialog box.
755278	Export property should have a report PIN_NUMBER and netname.
756553	Export/Import Properties> Flat Nets.
	Added a Note in the Capture User Guide and Online Help under section "Editing property files". The note text is "Do not make changes in the ID, Net Name, and Net ID fields in the property file (*.exp). The changes will not reflect in your design when you use the Import Properties command."
761532	Unable to edit a part once it as been split.
	Capture 10.5 allows you to split an already split part.
761812	Pin Shape/Type option should be in Split Part Spreadsheet.

2

What’s New in OrCAD Capture CIS 10.5

This chapter contains the following sections describing the OrCAD Capture CIS 10.5 release.

- New Features and Enhancements on page 14
- List of Important PCRs Fixed on page 15

New Features and Enhancements

OrCAD Capture CIS 10.5 brings you the following new features and enhancements:

- Setting Preferences for Linking a Placed Part to a Database Part
- Viewing Design Variants Information

Setting Preferences for Linking a Placed Part to a Database Part

Capture CIS 10.5 now allows you to set preferences for linking a placed part to a database part. Use the Extended CIS Linking dialog box to specify whether you want to:

- replace the symbol, reference designator, and all the properties of the placed part with that of the database part.
- replace the reference designator and all the other properties of the placed part with that of the database part, but retain the placed part symbol.
- replace the symbol and all the other properties of the placed part with that of the database part, but retain the placed part reference designator.
- replace all the properties of the placed part with that of the database part, but retain the placed part symbol and the reference designator.

To access the Extended CIS Linking dialog box, select *Options > CIS Preferences > Extended CIS Linking*. For more information, see the section *Setting linking preferences* in the *OrCAD Component Information System* User Guide.

Now you can also link part occurrence (s) to a database part. You can select whether you want to link all the occurrences of a selected part (s) or only the selected occurrences (s) of a part to a database part. You specify these options in the Occurrence Level Settings dialog box, which appears only when the selected part (s) has more than one occurrence in the design. For more information, see the section *Linking part occurrence (s) to a database part* in the *OrCAD Component Information System* User Guide.

Viewing Design Variants Information

Capture CIS 10.5 now allows you to view variant information for all the design variants defined in your project. The variant information includes different property values for common components or different or not present components for identical footprints on a schematic page. You can use the Variant View Mode command to display the variant information on a schematic page. You can also view the properties for the variants in the Variant property viewer. You can access the Variant View Mode command from the View menu. For more

information, see the section *Viewing design variants on a schematic page* in the *OrCAD Component Information* User Guide.

You can also now search for a specific design variant component on a schematic page using the Find or the Browse command. For more information, see the section *Searching for variant information on a schematic page* in the *OrCAD Component Information* User Guide.

Capture CIS 10.5 allows you to customize how you want to display the design variant information on a design variant schematic page. You can choose from different color combinations to display the variant information. The options are available in the Preferences dialog box. For more information, see the section *Customizing variant information display on a schematic page* in the *OrCAD Component Information* User Guide.

You can now preview and print design variant schematic pages. For more information, see the section *Previewing or printing design variant schematic pages* in the *OrCAD Component Information* User Guide.

List of Important PCRs Fixed

Some of the important PCRs addressed in OrCAD 10.5 are listed below:

PCR No	Description
719507	Link opens up in existing IE window only and not in new one.
551111	Design variants should be visible in the design.
554654	Reference Designator changes when linking a part in hierarchical design.
609891	Derive Database part does not follow allowed part reference.
717681	Printing design variants is not easy.
726860	FootPrint not visible in CIS Explorer Window.
748504	Layout fooprint - wrong and poor preview in CIS.

3

What’s New in PSpice 10.5

This chapter describes the new features in the PSpice 10.5 release.

- New Features on page 18
- Fixed PCRs on page 22

New Features

PSpice 10.5 brings you the following new features and enhancements:

- Enhanced Behavioral support in PSpice engine
- Support for Philips Mextram 504 model
- Model Import Wizard
- Transformer Designer
- Support for using European notation
- Behavioral Libraries
- New Vendor models

Enhanced Behavioral support in PSpice engine

In this release, the capability of the PSpice engine to support behavioral devices has been enhanced. The PSpice engine now supports:

- error and warning conditions

  You can now use the Error and Warning conditions to implement assertions in PSpice model.

- charge and flux sources

  From this release, you can model flux source using a E device, with voltage as the output. The charge source is modeled using a G device. The output of this charge source is a current source.

- math function, such as ceil and floor

- state functions, such as delta, state, and time

  State functions provide the history of Time, Delta, and node voltages during simulation. You can use these functions to find get the information about last three simulation points.

To know more about these enhancements, see *PSpice A/D Reference Guide*.

Support for Philips Mextram 504 model

From this release onwards, you can use the Mextram model for simulating a bipolar junction transistor in PSpice. The Mextram model provides improved handling of the features of modern down-scaled transistors, such as temperature scaling, charge storage, quasi saturation, and high injection effects.

To know more about the Mextram model, see *PSpice A/D Reference Guide*.

Model Import Wizard

You can now use the Model Import Wizard, to generate Capture parts either for a single model or for all the models in a library in an interactive mode.

The Model Import Wizard provides users with the capability of attaching Capture symbols to PSpice models. Using Model Import Wizard, you can attach symbols to the The wizard facilitates the process of providing valid Capture symbols for PSpice models that users may either have created themselves or may have downloaded from the web.

You can use the Model Import wizard, when you have a Capture symbol and want to attach a PSpice model to it.

To know more about importing traces, see *PSpice User's Guide*.

Transformer Designer

This is a new tool added to the PSpice family of products. Using Transformer Designer, you can design different type of transformers. The first version of Transformer Designer allows you to design power transformers, flyback transformers, and transformers for forward converters. Transformer Designer also supports designing of DC Inductors.

Besides designing transformers, Transformer Designer also help you:

- generate PSpice model for the designed component.

  You can associate the PSpice model generated by Transformer Designer with a part symbol and use the symbol in a schematic to simulate the transformer behavior.

- generate manufacturer report.

  At the end of the design process, Transformer Designer generates a report containing all the data required for manufacturing the transformer for a end user.

- maintain a database of components required for designing a transformer.

One of the modules of the Transformer Designer is the Transformer Designer database. This database contains data about commercially available components, such as cores, bobbins, wires and insulation material, that are required for designing a transformer. You can maintain this database by adding new components to it and deleting information about the obsolete parts.

To know more about Transformer Designer, see *Transformer Designer User's Guide*.

Support for using European notation

You can now use the european notation, such as 2k2, to set the value the VALUE property attached to resistors, inductors, and capacitors. Engineers and component distributors use this notation to avoid errors while reading component values from screen or from a print out of the schematic. The table below lists the alphabets that can be used in the 2K2 notation.

Alphabet used..	Stands for..
F(f)	femto
P(p)	pico
N(n)	nano
U(u)	micro
M(m)	milli
K(k)	kilo
MEG	mega
G(g)	giga
T(t)	tera

Some examples and their explanations are listed in the table below.

Notations	Equivalent to...
2M2	2.2 M
2MEG2	2.2 MEG
4L5	inductor of 4.5 henry
2K2	2.2K
12C2	capacitor of 12.2 farads

Notations	Equivalent to...
5R4	resistor of 5.4 ohms
2p2	2.2 p ($2.2*10^{-12}$)

Behavioral Libraries

New behavioral models added in form of two new libraries, `function.lib` and `spice_elem.lib`, provide support for modeling electromechanical devices, such as DC Motor and tachometer.

For all new behavioral models, it is recommended that you use `function.lib`. The `spice_elem` library is available mainly for users migrating from AWBHDL to PSpice.

To know more about the elements in these libraries, see *PSpice A/D Reference Guide*.

New Vendor models

Approximately 1500 vendor models, provided by Infineon and Tyco electronics have been added to the PSpice libraries. These have been added in form of 17 new libraries, which contain models for CoolMOS™, OPTIMOS™, IGBT, FET, Silicon Digital Transistor Array, and Metal Oxide Varistors.

Fixed PCRs

The following lists the problems reported and fixed in PSpice 10.5. For detailed information about a fixed problem, contact your customer support.

PCRs	Descriptions
690873	Unable to create a PSpice netlist and simulation profile for a blank project.
781306	Model Editor crashes when you create a new template-based model, hide the model text window, and start modifying any simulation property.
720914 741159	In remote client installation mode, if the client does not have write and execute permissions on the `../tools/pspice/library` folder, running a pspice simulation on a client machine causes OrCAD Capture to crash.
755419	Problem with the MOSFET drain current. R_{out} is not changing with the changing in drain voltage, and source current decreases when drain voltage is beyond saturation.
680296	PSpice throws the error "Error in Opening the Alias file".
739880	Model Editor Comparator model is not implemented correctly
663741	Extend PWL file stimulus to support 1000K points.
696299	Probe Options dialog is blank after changing histogram and unable to change the number of divisions in the Probe.
764999, 765005	Difference between library part and actual model from analog

4

What’s New in OrCAD Layout 10.5

This chapter contains the following sections describing the OrCAD Layout 10.5 release.

- What’s New in OrCAD Layout 10.5
  - Stackup Editor
  - Enhanced selection mechanism
  - Handling good DRCs
  - Using scroll-mouse button in the Layout window
- Product Change Requests (PCRs) addressed in Layout 10.5

What's New in OrCAD Layout 10.5

This section describes the new features and enhancements available in OrCAD Layout 10.5.

- Stackup Editor
- Enhanced selection mechanism
- Handling good DRCs
- Using scroll-mouse button in the Layout window

Stackup Editor

In the 10.5 release, Layout Plus provides you with Stackup Editor which is a simple, intuitive method of defining and editing a stackup. Using Stackup Editor you can map logical stackup of Layout to a physical stackup on a printed circuit board.

You can use a Stackup Editor to:

- customize the stackup entries in a predefined board and use these to create new boards.
- add and modify stackup information to an existing board.

For more information on using Stackup Wizard, see the *Ensuring manufacturability* chapter of the *OrCAD Layout User's Guide*.

Enhanced selection mechanism

In this release, Layout provides you with selection filter, which is an alternate mechanism for selecting objects on a dense Layout board. Selection filter is a mechanism for selecting objects on the graphical editor in Layout using keyboard.

Using selection filter, you can switch between Component selection mode, Obstacle selection mode, Text selection mode, Error selection mode, Edit Segment mode, and Add\Edit Segment mode. If the selection filter is activated, you can use the shortcut keys to select the objects from the current object selection mode that are closest to the current cursor position. This option is very useful for selecting objects on a densely populated layout board, because you use shortcut keys to select objects as well as to switch between various object selection modes, such as component selection mode or obstacle selection mode.

When you use selection filter mechanism, the shortcut keys activated are:

- <Ctrl> and <Y> keys; used simultaneously to select next object from the list in the same mode.
- <Ctrl> and <H> keys; used simultaneously to change the current mode of selecting objects.

For more information on using selection filter, see the *Ensuring manufacturability* chapter of the *OrCAD Layout User's Guide*.

Handling good DRCs

During board design, you may come across conditions that violate one or more design rules, but are acceptable and should not be considered as errors. OrCAD Layout can now handle such known good DRCs.

To ensure that the good DRCs do not appear as errors, select the DRC markers, either in the error report or in the Layout board, right-click and choose Mark as Good DRC.

For more information on handling known good DRCs, see the *Ensuring manufacturability* chapter of the *OrCAD Layout User's Guide*.

Using scroll-mouse button in the Layout window

From this release onwards, you can use the middle mouse button to scroll up and down in your Layout window. You can also use this button for routing purposes.

Note: To use the middle mouse button for scrolling, ensure that the Wheel Button of the mouse is mapped to Middle-click. You can modify the mouse settings through the Control Panel or the Mouse Driver interface supplied by the mouse vendor.

Support for GerbTool Version 14

The version of GerbTool that ships with OrCAD Layout has been upgraded from Version 13 to Version 14. For information on the new features in GerbTool v14, open the WhatsNew.rtf file located in the \tools\Layout_Plus\gtool\ directory.

Support for CADopia IntelliCAD Version 4

The version of IntelliCAD that ships with OrCAD Layout has been upgraded from Version 3 to Version 4. For information on the new features in IntelliCAD 4, see the *What's new in IntelliCAD 4* topic in the IntelliCAD 4 online help.

Product Change Requests (PCRs) addressed in Layout 10.5

Some of the important PCRs addressed in this release are listed below:

PCR No.	Description
326737	pinNum out of range error - Import Protel PCB to Layout.
335409	Rotated surface mount components fall off of the grid
440707	On XP Pro, Layout crashes on screen saver or welcome screen.
566250	Incorrect length report for curved tracks.
578075	Design Reused button disabled if the design has fiducials and mounting holes.
654602	Net attribute of the pin gets deleted
684435	Changing component footprint changes side of component.
712346	Changing width during manual routing does not change the width of the segment properly.
719408	Scroll mouse button does not work on 10.0.
722736	Cluster shifts far away from mouse pointer when rotate command is used.
723861	Unable to sort the padstacks.llb and arrange them in logical order.
744113	Update90 no longer processes wild cards
752322	Delete color rules has no user prompt
760313	Footprint search does not allow wildcard in library manager
765705	In Layout, printing with the Print\Plot Current view option, prints with wrong colors
768934	All holes come as plated in ODB++
770078	Wrong pin-selection in Library Manager of Layout.
775606	Selecting a seed vias pass generates an illegal DO file
776718	Add seed vias and Miter corners options to the Layout Specctra setup.

For More Information

To check for product updates, read technical notes from OrCAD engineers and participate in discussions with other OrCAD users visit the OrCAD community site www.orcadpcb.com.

5

OrCAD PCB Designer (OrCAD PCB Designer with PSpice)

OrCAD PCB Designer (OrCAD PCB Designer with PSpice) is a new productivity suite of PCB oriented tools. It includes:

- OrCAD PCB Editor
- OrCAD Capture
- SPECCTRA for OrCAD
- PSpice A/D (included in OrCAD PCB Designer with PSpice)

OrCAD PCB Designer (OrCAD PCB Designer with PSpice) suite is targeted to personal productivity and small design teams.

OrCAD PCB Designer (OrCAD PCB Designer with PSpice) suite is scalable to Allegro platform. You can scale up from OrCAD PCB Designer to Allegro 200 Series or Allegro 600 Series.

The OrCAD PCB Designer (OrCAD PCB Designer with PSpice) suite includes the following tools:

- OrCAD PCB EDITOR on page 30
- OrCAD Capture on page 31
- SPECCTRA for OrCAD on page 32
- PSpice A/D on page 33

OrCAD PCB EDITOR

OrCAD PCB Editor is a constraint-driven PCB layout editing tool derived from the Cadence Allegro PCB Editor. Its extensive feature set addresses a wide range of today's design challenges and manufacturability concerns.

- Multiple drill-in-pad, slot support, and automated legends
- Multiple UNDO/REDO
- HTML-based reports
- Stroke Editor

Note: For more information about OrCAD PCB Editor, see Allegro PCB Editor documentation in CDSDoc.

OrCAD Capture

OrCAD Capture®, the world's leading schematic entry solution. Intuitive, easy to use, with sophisticated part search mechanisms, this is the tool of choice for rapid design capture.

- Signal property flow to Allegro
- Selection Filter for UI Operations
- Split Symbol Support
- Import Latest Xilinx and Altera FPGA Formats
- Retain refdes on replace
- Page Navigation (from UI and keyboard)
- Support for Crystal 10 (CIS)

Note: For more information about What's New in Capture, see What's New in OrCAD Capture 10.5 on page 5.

SPECCTRA for OrCAD

SPECCTRA®, the industry-leading solution for automatic and interactive interconnect routing. It includes a route editor and autorouter designed to speed routing task to completion – increasing productivity and shortening overall design cycle times. New features include:

- Layer Sets

Note: For more information about SPECCTRA for OrCAD, see Allegro PCB Router documentation in CDSDoc.

PSpice A/D

PSpice A/D simulator, the most widely used tool for simulating analog and mixed-signal designs of any size.

- PSpice® Advanced Optimizer Option
- Parametric plotter (part of Advanced Analysis Option)
- Model libraries: PWMs, regulator, Battery model
- Monte Carlo history support
- NLEV support in BSIM3.2
- TOM3 Model Support
- Probe Usability Enhancements