Product Version 23.1 September 2023 © 2023 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.

OrCAD X Capture 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. vtkQt, © 2000-2005, Matthias Koenig. 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.

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.

Cadence is committed to using respectful language in our code and communications. We are also active in the removal and replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

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

About this Manual23
<u>Welcome</u>
Where to Find Information in this Manual
Menu Commands
Window descriptions 23
Dialog Box descriptions
Where to Find Additional Information
How to Contact Technical Support
• • • • • • • • • • • • • • • • • • •
2
Project Manager Command Reference27
File Menu
New command
Open command
Close Project command
Save command
Check and Save command3
Save As command
Save Project As
Archive Project command34
Print Preview command
Print command
Print Setup command
Import Design command
Export command
1,2,3,4 command
Change Product command
Exit command
Design menu
New Schematic command
New Schematic Page command

New VHDL File command	 . 39
New Verilog File command	 . 40
New Part command	
New Part from Spreadsheet command	
New Symbol command	
Rename command	
Remove Occurrence Properties command	 . 41
Make Root command	
Replace Cache command	 . 43
Update Cache command	 . 46
Cleanup Cache command	 . 47
Set Password Command	 . 47
Remove Password	 . 47
Change Password	 . 47
Edit menu	 . 48
Cut command	 . 49
Copy command	 . 49
Paste command	 . 51
Lock command	 . 51
UnLock command	 . 53
Project command	 . 53
Properties command	 . 54
Object Properties command	 . 57
Browse command	 . 57
Find command	 . 64
Rename Part Property command	 . 65
Delete Part Property command	 . 65
Replace command	 . 65
Go To command	 . 65
Clear Session Log command	 . 66
<u>View</u>	 . 66
Toolbar command	 . 68
Capture Toolbar command	 . 68
Draw Toolbar command	 . 68
PSpice Toolbar command	 . 70
FPGA Toolbar command	 . 70

	CIS Explorer Toolbar command	70
	Part Manager Toolbar Command	71
SL	Analysis Menu	74
	SI Library Setup command	75
	Auto Assign Discrete SI Models command	75
	Identify DC Nets command	75
	Assign Voltage to Power Nets command	75
	Export Electrical Csets command	75
	Import Electrical Csets command	76
	Remove Electrical Cset Assignments command	76
	Validate Electrical Cset Assignments command	
	Validate SI Model Assignments command	
	SI Model Integrity command	
	Export SI Models Used command	
	Remove SI Model Assignments command	
	View Xnet Signals	
To	ols menu	77
	Annotate command	79
	Associate PSpice Model command	79
	Backannotate command	
	Board simulation command	81
	Update Properties command	81
	Design Rules Check command	
	Create Netlist command	
	Create Differential Pair command	
	Cross Reference command	85
	Intersheet References command	
	Bill of Materials command	87
	Export Properties command	
	Export Placement command	
	Save as HTML command	
	Save as CSV command	
	Import Properties command	
	Generate Part command	
	Export FPGA command	
	Split Part command	

Sync NetGroup command93
Customize command93
Utilities command
<u>PSpice</u>
Bias Points command
New Simulation Profile command102
Edit Simulation Profile command
Run command
View Simulation Results command
View Output File command
Make Active command
Simulate Selected Profile(s) command
<u>Create Netlist command</u>
View Netlist command
Marker List command
Advanced Analysis menu
Sensitivity command
Optimizer command
Monte Carlo command
<u>Smoke command</u>
Export Parameters to Optimizer command
Import Optimizable Parameters command
Accessories menu
Accessories commands
Rotate Aliases command
Library Verification/Correction command
Push Occ. Prop into Instance command
Start Page command
Export Hierarchy command
Export Hierarchy with Parts command114
Mentor Netlist command
Mentor Netlist (to UNIX) command
Mentor Backannotation command
Options menu 115
Autobackup command
Preferences command

	More Preferences command	116
	Design Template command	121
	Design Properties command	
Wi	ndow menu	
	New Window command	123
	Cascade command	123
	Tile Horizontally command	123
	Tile Vertically command	
	Close All Tabs of Active Project command	
	Close All Tabs of Active Project Except Current command	
	Arrange Icons command	
	1,2 command	
	Close All Windows	125
He	 <u>lp menu</u>	
	OrCAD X Capture Help command	
	Known Problems and Solutions command	
	What's New command	
	Learning PSpice command	
	About OrCAD X Capture command	
	Web Resources command	
	Documentation command	
	PSpice Documentation command	
Sh	ortcut Menu	
	Add File command	
	Annotate command	
	Cleanup Cache command	
	Copy command	
	Cut command	
	Delete command	
	Design Properties command	
	Edit Object Properties command	
	Edit Page command	
	Edit selected object properties command	
	Find command	
	Lock command	
	Make Root command	

	New Page command	135
	New Schematic command	
	Open File Location command	
	Part Manager command	
	Paste command	
	Properties command	
	Remove PSpice Resources command	
	Rename command	
	Reports – Cross Reference command	
	Reports – Export Placement command	
	Reports – Export Properties command	
	Reports – InterSheet Reference command	
	Save command	
	Save As command	
	Save Project As command	137
	Schematic Page Properties command	137
	UnLock command	138
<u>3</u>		
S	chematic page editor and part editor command reference	139
File	. •	
	e menu	139
	e menu New command	
	New command	141
	New command	141 143
	New command Open command Close command	141 143 146
	New command Open command Close command	141 143 146 147
	New command Open command Close command Save command	141 143 146 147 147
	New command Open command Close command Save command Export Selection command	141 143 146 147 147
	New command Open command Close command Save command Export Selection command Import Selection command	141 143 146 147 147 147
	New command Open command Close command Save command Export Selection command Import Selection command Export ISCF command	141 143 146 147 147 148 148
	New command Open command Close command Save command Export Selection command Import Selection command Export ISCF command Print Preview command	141 143 146 147 147 147 148 148
	New command Open command Close command Save command Export Selection command Import Selection command Export ISCF command Print Preview command Print command	141 143 146 147 147 148 148 148
	New command Open command Close command Save command Export Selection command Import Selection command Export ISCF command Print Preview command Print Setup command	141 143 146 147 147 148 148 148 149
	New command Open command Close command Save command Export Selection command Import Selection command Export ISCF command Print Preview command Print command Print Setup command Print Area command	141 143 146 147 147 148 148 149 149

<u>1,2,3,4 command</u>	50
Exit command	50
Edit menu	51
Undo command1	53
Redo command1	53
Repeat command1	55
Label State commands	55
Cut command	57
Copy command1	57
Paste command1	59
Delete command1	59
Label command1	61
Select All command1	61
Part command1	63
PSpice Model command1	63
Properties command1	63
Part command1	66
Reset Location command1	67
Mirror command1	69
Rotate command	71
Lock command	71
UnLock command1	73
Find command	73
Global Replace command1	74
Check Verilog syntax command1	74
Check VHDL syntax command1	74
Samples command	75
Align Command	75
View menu	81
Ascend Hierarchy command1	83
Convert command1	83
Descend Hierarchy command1	83
Go To command1	85
Synchronize Up command1	85
Synchronize Down command1	85
Synchronize Across command1	86

•	Previous Part command	6
•	Next Part command	6
	Previous Page command	7
•	Next Page command	7
	Package command	7
	<u>Zoom command</u>	8
	<u>Fisheye command</u> 19 ⁻	1
	Tool Palette command	2
	<u>Toolbar command</u>	3
•	Status Bar command	3
•	Command Window command193	3
	Grid command	3
•	Grid References command195	5
•	Selection Filter command	5
	Invoke UI command	5
Toc	<u>lls menu</u>	6
;	Assign Power Pins command	6
SI A	<u> Analysis menu</u>	6
;	Assign SI Model command	7
	Explore Signal command	7
•	Export Topology command197	7
	Associate Electrical Cset command199	9
,	Validate SI Model Assignments command	9
•	Remove SI Model Assignments command	9
Pla	<u>ce menu</u> 200	0
•	Part command	2
•	Component command	2
	PSpice Part command	3
	Parameterized Part command	6
	NetGroup command 206	6
	Wire command 200	
	Auto Wire Two Points command	6
	Auto Wire Multiple Points command	7
	Auto Wire Connect to Bus	
•	Bus command 208	8
	Junction command	n

	Bus Entry command	210
	Net Alias command	210
	Power command	211
	Ground command	211
	Off-Page Connector command	211
	Hierarchical Block command	213
	Hierarchical Port command	213
	Hierarchical Pin command	214
	No Connect command	214
	Pin command	215
	Pin Array command	215
	Title Block command	215
	Bookmark command	218
	Text command	218
	IEEE Symbol command	218
	<u>Line command</u>	219
	Rectangle command	219
	Ellipse command	219
	Arc command	221
	Elliptical Arc command	221
	Bezier curve command	221
	Polyline command	222
	Picture command	222
	OleObject command	222
<u>PS</u>	Spice/Markers menu	223
	Markers command	224
	Marker List command	224
	Voltage Level command	225
	Voltage Differential command	225
	Current Into Pin command	225
	Power Dissipation command	225
	Advanced command	227
	Plot Window Templates command	
	Show All command	
	Hide All command	227
	Delete All command	228

	Update Design Hierarchy command	241
	Show Footprint command	
	Edit PSpice Model	
	Selection Filter	
	Fisheye View	242
	Zoom In command	
	Zoom Out command	242
	Go To command	242
	Delete command	242
	Tooltip command	243
	Lock command	243
	UnLock command	243
	SI Analysis – Assign SI Model command	243
	SI Analysis – Validate SI Model Assignments command	243
	SI analysis- Remove SI Model Assignments command	
	Create UnNamed NetGroup command	
	Reorder pins for UnNamed NetGroup command	
	Add or Remove Pins on NetGroup Block command	244
	Save As HTML command	244
	Waive DRC	244
	Unwaive DRC	245
<u>4</u>		
S	ession log command reference	247
	e menu	
1 111	New command	
	Open command	
	Save command	
	Save As command	
	Print Preview command	
	Print command	
	Print Setup command	
	Import Design command	
	Exit command	
	1,2,3,4 command	

<u>View menu</u>
Toolbar command
Status Bar command
Command Window command
Edit menu
<u>Copy command</u> 257
Select All command
Find command
Clear Session Log command
<u>Options Menu</u>
Preferences command
Design Template command259
Autobackup command259
<u>Window menu</u>
Cascade command261
Tile Horizontally command
Tile Vertically command
Arrange Icons command261
<u>1,2 command</u>
Close All Windows
<u>Help menu</u>
OrCAD X Capture Help command
Known Problems and Solutions command
What's New command
About OrCAD X Capture command
Web Resources command
Documentation command
<u>5</u>
Command Window command reference 265
Command Window Shortcut Menu
Font command
Background Color command
Text Color command
Save command

Clear All command	266
<u>6</u>	
Window Descriptions	269
Browse window	
Session frame window	
Session Log Window	
Part editor window	273
Property Editor window	273
Project manager window	277
Schematic page editor window	279
Text editor window	279
Browse Spreadsheet Editor	283
Command Window	
Project Manager folders	285
<u>7</u>	
Color Reference	287
8	
Dialog Box Descriptions	291
Add file to Project Folder dialog box	
Add New Property dialog box	291
Add to Project dialog box	
Annotate dialog box	
Packaging tab	
PCB Editor Reuse tab	
Archive Project dialog box	
Attach Implementation dialog box	
Assign Power Pins dialog box	
Associate PSpice Model dialog box	
Advanced Annotation	
Assign Tolerance	
Backannotate dialog box	313

Differential Pair Automatic Setup dialog box	367
Display Properties dialog box	369
Distributions dialog box	371
Design XML dialog box	371
Design Difference dialog box	372
Edit Bookmark dialog box	375
Edit Filled Graphic dialog box	375
Edit Graphic dialog box	375
Edit Hierarchical Port dialog box	376
Edit Net Alias dialog box	376
Edit Off-Page Connector dialog box	377
Edit Part Properties dialog box	377
Edit Text dialog box	379
Edit Wire(s) dialog box	379
Export ECSets from design	380
Export Design dialog box	380
EDIF tab	381
DXF tab	381
Export Properties dialog box	382
Export Selection dialog box	
Export Variant list dialog box	
Extended Preferences Setup	
Export Design View to HTML dialog box	
Edit Comment Text dialog box	
Find and Replace dialog box	395
Font dialog box	395
Find dialog box	396
FPGA Export Dialog Box	398
Reserve Pins Dialog Box	399
FPGA Options Dialog Box	
Generate Part dialog box	403
Go To dialog box	
Location tab	
Grid Reference tab	408
Bookmark tab	
Goto Label State dialog box	

Go To Line dialog box4	109
Hierarchical PSpice Netlist Settings dialog box4	11
Identify DC Nets 4	13
Import Design dialog box 4	13
PSpice tab	14
EDIF tab	14
<u>PDIF tab</u>	14
Import Selection dialog box	15
Intersheet References dialog box 4	15
ISCF Export dialog box	17
XML To DSN dialog box 4	18
XML To OLB dialog box 4	19
Library Management dialog box 4	21
Library Setup (SI Analysis) dialog box 4	21
Library XML dialog box	
Markers dialog box	23
Monte Carlo Worst-Case Output File Options dialog box	23
Multi-level Backup Settings dialog box	
Model Import Wizard	25
Associating PSpice Model to Capture Symbols	25
Associating Parts to a PSpice Model	
Push Occ. properties to instance dialog box	
NC Verilog Simulation dialog box	
NC Verilog Simulation Setup dialog box	
Simulation tab (NC Verilog)4	
Testbench tab (NC Verilog)4	40
Model compilation tab (NC Verilog)4	41
New Property dialog box	
NC Verilog Simulation Setup dialog box	42
Simulation tab (NC Verilog)	
Testbench tab (NC Verilog)4	
Model compilation tab (NC Verilog)4	
NC VHDL Postroute Simulation dialog box	
NC VHDL Postroute Simulation Setup dialog box	
Simulation tab (NC VHDL)	
Testbench tab (NC VHDL)4	

Model compilation tab (NC VHDL)
NC VHDL Preroute Simulation dialog box
NC VHDL Preroute Simulation Setup dialog box
NC VHDL Simulation dialog box
NC VHDL Simulation Setup dialog box
NC VHDL Library Compilation dialog box
New Alias dialog box
New Layout dialog box
New Page in Schematic dialog box
New Part Creation Spreadsheet
New Part Properties dialog box
New Project dialog box
New Property dialog box
New Schematic dialog box
New Simulation dialog box
New Symbol Properties dialog box
New NetGroup / Modify NetGroup dialog box
Rename NetGroup Member dialog box
Open dialog box
Property Sheet Pane
Package Properties
Part Properties
Pin Properties
Text Properties
Basic Attributes
Display Properties
Delete Current Section
Add Convert View
Delete Convert View
Edit Pins
Associate PSpice Model
Paste Options dialog box
Part Aliases dialog box
Part Search dialog box
PCB Project Wizard dialog box
Place and Route Settings dialog box

Place Bookmark dialog box
Place Ground dialog box
Place NetGroup dialog box
Place Off-Page Connector dialog box
Place Hierarchical Block dialog box
Place Hierarchical Pin dialog box
Place Hierarchical Port dialog box
Place IEEE Symbol dialog box
Place Net Alias dialog box
Place Part dialog box
Place Part Pane
Search Part
Place Pin Array dialog box
Place Pin dialog box
Place Power dialog box
Place Text dialog box
Place Title Block dialog box
Preferences dialog box
Color/Print tab
Grid Display tab
Miscellaneous tab
Pan and Zoom tab
<u>Select tab</u>
Text Editor tab
Board Simulation tab
Print dialog box
Print Preview and Print Setup dialog boxes
Print Setup dialog box
Print to File dialog box
Programmable Logic Project Wizard dialog box
Propagation Delay dialog box
Properties dialog box
General tab
<u>Type tab</u>
Project tab
PSpice Part Search dialog box

Options tab	 566
Probe Window tab	 567
Stimulus tab	 567
Specify Part Filter dialog box	 568
Split Part Section Input Spreadsheet	 568
Synthesis Option dialog box	 573
Transient Output File Options dialog box	 574
Update Properties dialog box	 577
<u> Update Old Project wizard</u>	 578
User Properties dialog box	 579
<u>Update Alias dialog box</u>	 580
<u>Update Layout dialog box</u>	 581
Update Schematic dialog box	 583
Validate ECSets in design	 585
VHDL Samples dialog box	 585
View DRC Marker dialog box	 587
Zoom Scale dialog box	589

About this Manual

Welcome

OrCAD® X Capture (referred to as Capture elsewhere) is a schematic design tool set for the Windows environment. With Capture, you can draft schematics and produce connectivity and simulation information for printed circuit boards and programmable logic designs.

OrCAD® X Capture Reference Guide describes menu commands, windows, toolbars and dialog boxes available to users via the Capture user interface. This book serves as both a reference manual and the content source for the Capture context-sensitive Help.

Where to Find Information in this Manual

This guide contains information on both menu commands and dialog boxes available in the Capture environment with convenient cross links.

Menu Commands

The menu commands in Capture are covered across the following chapters (grouped by the section of the environment to which they belong):

- Project Manager Command Reference
- Schematic page editor and part editor command reference
- Session log command reference
- Command Window command reference

Window descriptions

The <u>Window Descriptions</u> chapter provides a brief description of the different windows available in the Capture environment.

About this Manual

Dialog Box descriptions

The dialog box descriptions are covered in a number of chapters of this guide, grouped alphabetically. Each of the descriptions provides details of how to open the dialog and a description of each of the controls on the dialog.

About this Manual

Where to Find Additional Information

To access additional technical documentation from the OrCAD X Capture user interface, display the online Help page by choosing Help - Documentation from the main menu.

How to Contact Technical Support

If you have questions about installing or using OrCAD X Capture, contact the <u>Cadence Online Support</u>

About this Manual

Project Manager Command Reference

This chapter covers:

- File Menu on page 27
- Design menu on page 38
- Edit menu on page 48
- View on page 66
- SI Analysis Menu on page 74
- Tools menu on page 77
- PSpice on page 101
- Advanced Analysis menu on page 107
- Accessories menu on page 111
- Options menu on page 115
- Window menu on page 121
- Help menu on page 125

File Menu

New command on page 29

Open command on page 29

Close Project command on page 31

Save command on page 31

Check and Save command on page 31

Project Manager Command Reference

Save As command on page 32

Save Project As on page 32

Archive Project command on page 34

Print Preview command on page 34

Print command on page 34

Print Setup command on page 36

<u>Import Design command</u> on page 36

Export command on page 36

1,2,3,4 command on page 37

Change Product command on page 37

Exit command on page 37

Project Manager Command Reference

New command

Available from:

File menu

Use this command to create a new project, design, library, or VHDL file. Choose a command from the menu that appears:

- **Project**
- Design
- Library
- **VHDL File**

Function:

Verilog File

The number of open windows you can have is only limited by your available system resources. You can use the Window menu to switch among open windows (see1,2.... command).

You can open an existing project, design, library, or VHDL file using the Open command on the File menu.

Note: If you click the Create document toolbar button from the project manager window, the New Project dialog box appears.

Capture Toolbar:

Shortcuts:

Keyboard:

ALT, F, N

Open command

Available from: File menu

Project Manager Command Reference

Use this command to open an existing project, design, library, VHDL, or Verilog file in a new window. Choose a command from the menu that appears:

- Demo Designs
- Project
- Design
- Library
- **Function:**
- Project
- VHDL File
- Verilog File

The number of open windows you can have is only limited by your available system resources. You can use the Window menu to switch among open windows (see 1,2.... command).

You can create a new design, library, VHDL, or Verilog file using the <u>New command</u> on the File menu.

Note: When you click the Open button on the toolbar, a standard Windows Open dialog box appears, in which you can choose the type of file you want to open in the Files of type drop-down list. Among the listed choices are SDT Schematic (*.SCH) and SDT Library (*.LIB).

Capture Toolbar:



Shortcuts:

Keyboard:

■ ALT, F, O

Project Manager Command Reference

Close Project command

Available from: F

File menu

Function:

Use this command to close the active window. If necessary, you are

prompted to save your changes.

Shortcuts:

Keyboard: ALT, F, C

Save command

Available from:

File menu

Use this command to save the active, modified projects, designs, libraries, and VHDL files. You can save a design, library, VHDL file, or session log under a different name using the Save As command on the

File menu.

Function:

Note: When you save a project, Capture automatically creates a backup with a .DBK file extension. When you save a library, Capture automatically creates a backup with a .OBK file extension. If you save only a schematic

page or part, no backup is generated.

Capture Toolbar:

Keyboard:

Shortcuts:

■ CTRL+S

■ ALT, F, S

Check and Save command

Available from:

File menu

Function:

Use this command to execute the design rules check and save the design. The design rules check is executed with the electrical design

rules currently defined in the Design Rules Check dialog box.

Shortcuts:

Keyboard: ALT, F, N

Project Manager Command Reference

Save As command

Available from:

File menu

Use this command to save the active project, design, library, VHDL file, or session log under a different name or to save a new, unnamed project, design, library, VHDL file, or session log. You can save a design, library, schematic page, part, or session log with the <u>Save command</u> on the File menu.

Function:

mona.

The Save As command opens a standard Windows dialog box to save files.

Note: When you save a project, Capture automatically creates a backup with a .DBK file extension. When you save a library, Capture automatically creates a backup with a.OBK file extension. If you save only a schematic page or part, no backup is generated.

Note: When you use the Save As command, you are prompted to choose the file type from the Save As Type list in the Save As dialog box. You can choose to save the file in the current design database schema version or in a schema version that is one version prior to the application version you are currently using.

Shortcuts:

Keyboard: ALT, F, A

Save Project As

Available from:

File menu

Use this command to save the associated files present inside or outside the project directory along with the project at a new location while maintaining their internal and external links. Associated files include referred projects, designs, libraries, simulation profiles, and output files.

Function:

The Save Project As command opens the *Project SaveAs* dialog box. In this dialog box, you can specify a project name that is different from the design name. The Project SaveAs settings can be customized using the Settings tab of the dialog box which gives you the option to copy the design file along with the project and rename it. You can also selectively specify to copy the referred files present within or outside the project folder.

Project Manager Command Reference

Note: Referred files include Projects, Libraries, Output Files, Simulation Files and so on referred from the current project.

/Important

Irrespective of the option selected to copy the referred files, the links to the referred files are always updated for the new saved project. In case of PSpice projects, PSpice files are always copied to the new location, irrespective of the options selected in the settings tab.

Project Manager Command Reference

Archive Project command

Available from:

File menu

Use this command to save the project (*.OPJ) and all the related files (design (*.DSN), library (*.OLB), and referenced projects) in a different directory and create a zip archive (.zip) of this directory for archival

purposes.

Function:

You can also specify any additional files or directories that you may want

to be archived along with your project files.

You can include output files and library files (like *.OLB files in the

Library folder and *.VHD files).

Shortcuts:

Keyboard: ALT, F, H

Print Preview command

Available from:

File menu

Use this command to see how a schematic page or part will look when

printed.

Function:

After setting the options in the Print Preview dialog box, click OK to preview the printed document. You can use the buttons at the top of the

window to view different pages and to zoom in and out.

Note: Be prepared to wait if you attempt to print multiple pages or parts. Depending on the number and size of the pages or parts you are previewing, Capture may require extra time to display the selection.

Shortcuts:

Keyboard: ALT, F, V

Print command

Available from: File menu

Function:

Use this command to print the active schematic page, the active part, or

the selected items in the project manager.

Note: When you print multiple copies, the copies are grouped by page,

not sorted by copy.

Project Manager Command Reference

Capture Toolbar:



Keyboard:

Shortcuts:

- CTRL+P
- ALT, F, P

Project Manager Command Reference

Print Setup command

Available from:

Function:

File menu

Use this command to choose a printer, paper source, and orientation

before printing. The Print Setup command displays the Print Setup dialog box, a standard windows dialog box for configuring your printer or

plotter. For more information on setting up printers and plotters, refer to

the documentation for your configured printer driver.

Tip

Many times, the options for your printer are not available in the standard setup dialog box. If you do not find the options you need, try the printer setup in the Windows Control Panel.

Shortcuts:

Keyboard: ALT, F, R

Import Design command

Available from:

Function:

File menu

Available II OIII.

Use this command to import EDIF, PDIF and PSpice designs. EDIF designs must be graphical EDIF designs, and not EDIF netlists. Not all

imported PDIF parts may be edited in Capture. Such parts won't affect

netlists.

Export command

Available from:

File menu, Export submenu

Function:

Use this command to export EDIF designs and DXF schematic pages. This command saves EDIF designs as graphical EDIF designs, and not EDIF netlists. DXF schematic pages are saved in AutoCAD's V12 file

format.

Project Manager Command Reference

1,2,3,4 command

Available

Function:

File menu

from:

Use the numbers listed at the bottom of the File menu to open one of

the last four projects or files. Choose the file you want to open.

Shortcuts:

Keyboard: ALT, F, N (n = 1, 2, 3, or 4)

Change Product command

Available from:

File menu

Use this command to select a product suite from which to open

Capture without having to exit the tool.

This command is not available if you have a project open in Capture.

Function:

Select the Use as default check box to set the default product suite from which Capture should check out a license each time you start the tool. If this option is selected, then the suite selection dialog box will not appear the next time you start the tool. However, if you want the suite selection dialog box to be displayed again then clear the

Use as default check box in the suite selection dialog box.

Shortcuts:

Keyboard: ALT, F, C

Exit command

Available from: File menu

Use this command to exit the software. If necessary, you are prompted

to save your changes.

Function:

You can also exit the software by choosing the Close command on the

session frame Control menu (ALT, SPACEBAR, C).

Project Manager Command Reference

Keyboard:

■ ALT+F4

Shortcuts:

- ALT, SPACEBAR, C
- ALT, F, X

Design menu

New Schematic command on page 39

New Schematic Page command on page 39

New VHDL File command on page 39

New Verilog File command on page 40

New Part command on page 40

New Part from Spreadsheet command on page 40

New Symbol command on page 41

Rename command on page 41

Remove Occurrence Properties command on page 41

Make Root command on page 43

Replace Cache command on page 43

<u>Update Cache command</u> on page 46

Cleanup Cache command on page 47

Set Password Command on page 47

Remove Password on page 47

Change Password on page 47

Project Manager Command Reference

New Schematic command

Available from: Design menu

Use this command to create a schematic folder in the active project.

Function: You can add a new schematic page to the selected schematic folder

using the New Schematic Page command on the Design menu.

Note: Schematic folder, schematic page, part, part alias, and symbol names are completely case sensitive. It is possible to have a part named "XYZ" and another one named "xyz," and Capture's tools will treat the two

separately.

Shortcuts: Keyboard: ALT, D, S

New Schematic Page command

Available from: Design menu

Use this command to add a new schematic page to the selected

schematic folder.

Function:

You can add a new schematic folder to the active project using the New

Schematic command on the Design menu.

Note: Schematic folder, schematic page, part, part alias, and symbol names are completely case sensitive. It is possible to have a part named "XYZ" and another one named "xyz," and Capture's tools will treat the two

separately.

Shortcuts: Keyboard: ALT, D, P

New VHDL File command

Available from: Design menu

Function: Use this command to create a new VHDL file in the active project.

Shortcuts: Keyboard: ALT, D, V

Project Manager Command Reference

New Verilog File command

Available from: D

Design menu

Function:

Use this command to create a new Verilog file in the active project.

Shortcuts:

Keyboard: ALT, D, V

New Part command

Available from:

Design menu

Function:

Use this command to create a part in the active library. Part aliases are created at the same time as the original part and show up in the library independently from the original part, but are represented in the project manager by a part icon with a horizontal line through the center. You can add part aliases to a library after the original part is created using the *Property Sheet* pane.

Note: Schematic folder, schematic page, part, part alias, and symbol names are completely case sensitive. It is possible to have a part named "XYZ" and another one named "xyz," and Capture's tools will treat the two separately.



You can use an existing part as a model for a new part by moving a copy of the part to a second library and then editing the copy. If you wish to have the new part in the original library, rename the new part, then move it to the original library.

Shortcuts:

Keyboard: ALT, D, T

New Part from Spreadsheet command

Available from:

Design menu

Function:

Use this command to create a new part in the active library.

Shortcuts:

Keyboard: ALT, D, H

Project Manager Command Reference

New Symbol command

Available from: Design menu

Function: Use this command to create a symbol in the active library.

Note: Schematic folder, schematic page, part, part alias, and symbol names are completely case sensitive. It is possible to have a part named "XYZ" and another one named "xyz," and Capture's tools will treat the two

separately.

Shortcuts: Keyboard: ALT, D, L

Rename command

Available from: Design menu

Use this command to change the name of the selected schematic folder,

Function: schematic page, or part. If you rename a power or ground symbol using

this command, the name is limited to 31 characters.

Shortcuts: Keyboard: ALT, D, R

Remove Occurrence Properties command

Available from: Design menu

Use this command to remove all the unique annotations you have placed

on parts if your schematic design part references are no longer

Function: synchronized with your PCB project. To correct this problem,

backannotate your PCB in PCB Editor to create a swap (.SWP) file.

Then, in Capture, use Backannotate.

Project Manager Command Reference

Note:	The following changes will be lost using this command:	
	inherent properties on occurrences	
	user-define properties on occurrences	
	gate or pin swaps unique to the part occurrences	
	occurrence properties on title blocks, pins, and nets	
You can use the property editor to find occurrence properties on objects in your design.		

Shortcuts: Keyboard: ALT, D, O

Project Manager Command Reference

Make Root command

Available from: Design menu

Use this command to designate the selected schematic folder as the root

schematic folder of the hierarchy.

Function: Important

If you haven't specified a root for your design, you cannot generate reports. Also, when folders are copied to a new design, the ROOT designation is lost and must be reestablished in the

design.

Shortcuts: Keyboard: ALT, D, M

Replace Cache command

Available from: Design menu

Function: Use this command to replace the selected part in the design cache, based on its current definition in any library. You can also use this command to replace a selected part in the cache with a different part.

/ Important

> When you replace a selected part in the design cache, make sure the part library is correct before you change the part name to the library part name. If more than one part is selected in the design cache, you can use the Replace Cache command to replace them, but the Part name field will not be available.

> When you replace a part in the design cache, you replace all the parts in the active design that share the same part value and library. If the replaced part displays on an open schematic page, the parts on that page do not change until the page is closed and then reopened.

Project Manager Command Reference

Note: If you select all the parts in the design cache and execute the Replace Cache command to replace parts from a new library in which one of the parts doesn't exist, the execution will quit when it gets to that part. Parts below it will not be replaced in the cache.

Using the Replace Cache dialog box, you can choose to preserve schematic part properties or replace schematic part properties.

If you select the action to **Preserve schematic part properties**, Capture brings in the graphics, pins, package properties, and part properties from the library while retaining all instance and occurrence properties of the schematic part in the design. The part reference changes to an unannotated reference (?), and the value becomes the new part name. If you look at the property editor Parts tab, you will also see that the package property value appears in the PCB Footprint property column. You will lose any changes made to the pin properties after the part was placed, including those made by the Backannotate or the Annotate tools.

Note: Using the Preserve schematic part properties option with the Replace Cache command causes the same behavior as the default Replace Cache command in Capture Release 9 and earlier.

If you select the action to **Replace schematic part properties**, Capture brings in graphics, pins, package properties, and user properties from the library, totally replacing the schematic part in the design. Any value, PCB footprint, part reference, and instance or occurrence property defined on the schematic page will not change. Remember that instance and occurrence properties always obscure library definitions.

Note: The *Replace Cache* and *Update Cache* commands are quite similar. However, there are a couple of significant differences between the two commands. You can modify a part's link to the library (part name, path, and library) with Replace Cache, but not with Update Cache. Update cache only brings in new data when the path has changed. Another difference is that if the path and library names do not change, Replace Cache reloads the part definition into the design. However, if Update Cache finds that the part name and the library names are the same, it does not bring in part changes.

Project Manager Command Reference



If you need to know a part's library of origin, you can select the part in the project manager, then select Replace Cache from the Design menu. The part name and the library and path are listed in the dialog box that appears. Click Cancel to return to the project manager.

You can discover the library of origin for multiple parts by <u>Creating a cross reference report</u>.

Shortcuts:

Keyboard: ALT, D, C

Update Cache command

Available from:

Design menu

Use this command to update the selected parts in the design cache, based on their current definitions in their original libraries. This command works when one or more parts are selected. You can select all the parts also.

Function:

When you change a part in the design cache, you change all the parts in the active design that share the same part value and library. Part properties are retained, but pin properties are not. If the modified part appears on an open schematic page, the parts on that page do not change until the page is closed and reopened.

If you copy pages from one design or library to another, parts displayed on the copied pages may appear different due to differences in each design or library cache. If a part is not already in the destination design cache, Capture will copy it from the source design's cache. Otherwise, it will use the part already present in the destination design's cache.

Note: The Replace Cache and Update Cache commands are quite similar. However, there are a couple of significant differences between the two commands. You can modify a part's link to the library (part name, path, and library) with Replace Cache, but not with Update Cache. Update cache only brings in new data when the path has changed. Another difference is that if the path and library names do not change, Replace Cache reloads the part definition into the design. However, if Update Cache finds that the part name and the library names are the same, it does not bring in part changes.



If you need to know a part's library of origin, you can select the part in the project manager, then select Replace Cache from the Design menu. The part name and the library and path are listed in the dialog box that appears. Click Cancel to return to the project manager.

You can discover the library of origin for multiple parts by <u>Creating a cross reference report</u>.

Shortcuts:

Keyboard: ALT, D, U

Project Manager Command Reference

Cleanup Cache command

Available from: Design menu

Function: Use this command to cleanup the design cache. This command removes

nonexistent parts from the cache.

Shortcuts: Keyboard: ALT, D, N

Set Password Command

Available from: Design menu

Use this command to set a password to a Capture design. Once you

click on the command, you will asked to enter and confirm the password.

Function:

Note: Once the password is set to the design, there is no way to recover

the design without the correct password.

Shortcuts:

Remove Password

Available from: Design menu

Function: Use this command to remove the already set password to a Capture

design.

Shortcuts:

Change Password

Available from: Design menu

Use this command to modify or update the already set password to a

Function: Capture design. Once you click the command, you will asked to enter

and confirm the password.

Shortcuts:

Project Manager Command Reference

Edit menu

Cut command on page 49

Copy command on page 49

Paste command on page 51

Project command on page 53

Properties command on page 54

Object Properties command on page 57

Browse command on page 57

Power Pins command on page 63

Rename Part Property command on page 65

Go To command on page 65

Clear Session Log command on page 66

Delete Part Property command on page 65

Replace command on page 65

Project Manager Command Reference

Cut command

Available from: Edit menu

Use this command to remove the selected object from the active window and put it on the Clipboard. This command is only available when an object is selected

object is selected.

Function: Cutting objects to the Clipboard replaces any objects previously stored there. Use the <u>Paste command</u> to copy objects to another page or part, or to another Windows application that supports pasting from the

Clipboard.

Note: The Cut and <u>Copy command</u> are unavailable in the part editor when you have one or more pins selected with other objects (such as an

arcs and lines).

Capture Toolbar: 🧩

Keyboard:

Shortcuts: ■ CTRL+X

■ ALT, E, T

Shortcut menu: Cut

Copy command

Available from: Edit menu

Use this command to copy a selected object to the Clipboard without removing it from the active window. This command is available only if an

object is selected.

Function: Copying objects to the Clipboard replaces any objects previously stored

there. Use the Paste command to copy objects to another page or part,

or to another Windows application that supports pasting from the

Clipboard.

Note: The Cut and Copy commands are unavailable in the part editor when you have one or more pins selected with other objects (such as arcs

and lines).

Project Manager Command Reference

Capture Toolbar:



Keyboard:

Shortcuts:

CTRL+C

ALT, E, C

shortcut menu: Copy

Project Manager Command Reference

Paste command

Available from:

Edit menu

Use this command to place any objects stored on the Clipboard into the active window. This command is unavailable if the Clipboard is empty.

Function:

Pasting objects from the Clipboard does not affect the Clipboard's contents. Use Paste to copy objects to another page or part, or to another Windows application that supports pasting from the Clipboard. You can only paste text into text boxes.

Note: If you copy a part into the Clipboard and then paste it onto a schematic page, Capture will automatically assign a unique reference designator to the pasted part when two conditions are met:

- **1.** The Auto Reference option on the <u>Miscellaneous tab</u> of the Preferences dialog box is selected.
- **2.** The pasted part has a reference designator assigned to it when it is copied to the Clipboard.

Capture assigns the reference designator, updated to the next available value (one greater than the highest value used on the schematic at that point.) If the pasted part has a default reference (for example, R?) Capture does not assign a unique reference designator to it.



In part editor, you can copy objects and paste them in the part editor only. Similarly, in symbol editor, you can copy objects and paste them in the symbol editor only.

Capture Toolbar:



Keyboard:

Shortcuts:

■ CTRL+V

■ ALT, E, P

shortcut menu: Paste

Lock command

Available from: Edit menu

Project Manager Command Reference

Use this command to lock a design, schematic folder, or a page in the project manager. A lock icon appears for design, schematic folder, or page you have locked. You cannot delete, move, rename, change a locked object. When you lock a page all the parts in the page are also locked.

Function:

Locking an item locks the sub-items. For example, if you lock a design, the schematic folders and the pages in these folders are also locked.

Note: Save the project after locking to ensure the lock state is saved.

Keyboard:

Shortcuts: ■ ALT, E, L

shortcut menu: Lock

Project Manager Command Reference

UnLock command

Available from: Edit menu

Use this command to unlock a design, schematic folder, or a page in the project manager. The lock icon disappears for design, schematic folder,

Function: or page you have unlocked.

Locking an item locks the sub-items. For example, if you lock a design, the schematic folders and the pages in these folders are also locked.

Note: Save the project after locking to ensure the lock state is saved.

Keyboard:

Shortcuts: ■ ALT, E, N

shortcut menu: UnLock

Project command

Function:

Available from: Edit menu

Use this command to add resources to your project. When you choose this command, the <u>Add file to Project Folder dialog box</u> appears, in which you can locate and select files to add to your project. The files are added to the folder that is currently selected in the project manager window.

This command is only available when the File tab is active in the project

manager window.

Shortcuts: Keyboard: ALT, E, R

Project Manager Command Reference

Properties command

Available from: ^L

Edit menu

In the project manager, use this command to view properties about the selected document. Using the <u>Properties dialog box</u>, you can access information general, type, and project about the file that is currently selected in the project manager window. You can also change the file's type. A file or project must be selected in the project manager window to access the

Properties command.

Function:

In the schematic page editor, use this command to open the property editor, where you can edit properties and other data for the selected objects.

The properties you can edit depend on the selected objects. The following lists the inherent properties you can edit and the dialog boxes in which you edit them:

Objects	Dialog Box
Arcs	Edit Graphic dialog box
Images (pictures)	Not applicable
Bookmarks	Edit Bookmark dialog box
Buses Property editor	Property editor
Bus entries	User Properties dialog box
DRC markers	View DRC Marker dialog box
Ellipses	Edit Filled Graphic dialog box
Hierarchical blocks	Property editor
Hierarchical pins	Property editor
Hierarchical ports	Property editor
IEEE symbols	Place IEEE Symbol dialog box
Junctions	Not applicable
Lines	Edit Graphic dialog box
Multiple objects	Property editor or Browse spreadsheet editor
Nets (wires and buses	Property editor

Project Manager Command Reference

Objects	Dialog Box
Net aliases	Property editor
No connects	Not applicable
Off-page connectors	Edit Off-Page Connector dialog box
Parts	Property editor
Pictures (images)	Not applicable
Part body borders	Not applicable
Pins (part editor)	Pin Properties dialog box (part editor)
Pins (schematic page editor)	Property editor
Polygons	Edit Filled Graphic dialog box
Polylines	Edit Graphic dialog box
Power, ground	Property editor
Rectangles	Edit Filled Graphic dialog box
Text	Place Text dialog box
Title blocks	Property editor
Wires	Property editor

Note: You can edit homogeneous sets of the following objects in the spreadsheet editor:

- Bookmarks
- DRC markers
- Hierarchical ports
- Nets
- Off-page connectors
- Parts
- Pins

Project Manager Command Reference

Keyboard:

■ CTRL+E

Shortcuts:

■ ALT, E, I

Mouse: Double-click on a part

shortcut menu: Edit Properties

Project Manager Command Reference

Object Properties command

Available from:

Edit menu

Use this command to open the Property Editor window for a design. You can use the property editor window to edit part, net, pin, title block, global, port, and alias properties.

This command is available only when you have selected any one of the following items in the Project manager window:

Function:

- Design file (.dsn)
- Schematic folder
- Schematic page

You can access the Edit Object Properties command from the shortcut menu.

Shortcuts:

Keyboard: ALT, E, O

Browse command

Available from: Edit menu

Project Manager Command Reference

Use this command in the project manager to specify which items to search for and how to sort the results. Choose a command from the menu that appears:

- Parts command
- Nets command
- Hierarchical Ports command
- Off-page Connectors command

Function:

- Titleblocks command
- Bookmarks command
- DRC Markers command
- Flat Netlist command
- Power Pins command

You can open a schematic page or part by double-clicking on the selected item. You can also choose properties to edit and change one or more items.

Shortcuts:

Keyboard: ALT, E, B

Project Manager Command Reference

Parts command

Edit menu, Browse command Available from:

> Use this command to list the parts of the selected schematic pages in the browse window. The Browse Properties dialog box gives you the

option to use instances or occurrences.

You can sort the parts in the browser spreadsheet by clicking the button **Function:**

at the top of each column in the browse window. Capture only displays

the occurrences of the parts in the browse window.

You can select single or multiple parts and edit them with the Properties command. When you edit one part, a part editor window appears. When

you select multiple parts, the Browse spreadsheet editor appears.

Keyboard: ALT, E, B, P **Shortcuts:**

Nets command

Edit menu, Browse command Available from:

> Use this command to list the nets of the selected schematic pages in the browse window. Capture only displays the occurrences of the nets in the

browse window.

Function: You can select single or multiple nets and edit them with the Properties

command. When you edit nets, the property editor window appears.

Double-click on a net to view it in the schematic page editor.

Keyboard: ALT, E, B, N **Shortcuts:**

Hierarchical Ports command

Available from: Edit menu, Browse command

Project Manager Command Reference

Use this command to list the hierarchical ports of the selected schematic pages in the browse window. You can use this command to follow ports through a hierarchy when tracing problems in a hierarchical design.

Function:

You can select single or multiple hierarchical ports and edit them with the Properties command. When you edit one hierarchical port, the Edit Hierarchical Port dialog box appears. When you select multiple hierarchical ports, the Edit Properties dialog box appears.

Double-click on a hierarchical port to view it in the schematic page editor.

Shortcuts:

Keyboard: ALT, E, B, H

Project Manager Command Reference

Off-page Connectors command

Available from: Edit menu, Browse command

Use this command to list the off-page connectors of the selected schematic pages in the browse window. You can use this command to follow nets as they travel through off-page connectors to other pages.

You can select single or multiple off-page connectors and edit them with

the <u>Properties command</u>. When you edit one off-page connector, the <u>Edit Off-Page Connector dialog box</u> appears. When you select multiple

off-page connectors, the Browse spreadsheet editor appears.

Double-click on an off-page connector to view it in the schematic page

editor.

Shortcuts: Keyboard: ALT, E, B, O

Titleblocks command

Function:

Function:

Available from: Edit menu, Browse command

Use this command to list the title blocks of the selected schematic pages

in the browse window. When you edit a title block, the <u>User Properties</u>

dialog box appears. Double-click on a title block to view it in the

schematic page editor.

Shortcuts: Keyboard: ALT, E, B, T

Bookmarks command

Available from: Edit menu, Browse command

Use this command to list the bookmarks of the selected schematic pages in the browse window. Bookmarks are useful for marking a

particular spot in your design.

You can select single or multiple bookmarks and edit them with the

<u>Properties command</u>. When you edit one bookmark, the <u>Edit Bookmark</u> dialog box appears. When you select multiple bookmarks, the Edit Part

Properties dialog box appears.

Double-click on a bookmark to view it in the schematic page editor.

OrCAD X Capture CIS Reference Guide Project Manager Command Reference

Keyboard: ALT, E, B, B **Shortcuts:**

Project Manager Command Reference

DRC Markers command

Available from: Edit menu, Browse command

Use this command to list the DRC markers of the selected schematic pages in the browse window. DRC markers are placed on pages by the Design Rules Check tool. They are useful when troubleshooting your

design before creating a netlist.

Function: You can select single or multiple DRC markers and edit them with the

Properties command. When you edit one DRC marker, the <u>View DRC Marker dialog box</u> appears. When you select multiple DRC markers, the

Browse spreadsheet editor appears.

Double-click on an DRC marker to view it in the schematic page editor.

Shortcuts: Keyboard: ALT, E, B, D

Flat Netlist command

Available from: Edit menu, Browse command

Use this command to list the nets of the design, as they will appear in a

netlist.

Function: You can select single or multiple nets and edit them with the Properties

command. When you edit nets, the property editor window appears.

Double-click on a net to view it in the schematic page editor.

Shortcuts: Keyboard: ALT, E, B, F

Power Pins command

Available from: Edit menu, Browse command

Use this command to list the power pins in the design.

Function: You can choose the view mode as occurrences or instances from the

Browse Properties dialog.

Shortcuts: Keyboard: ALT, E, B, W

63

Project Manager Command Reference

Find command

Available from:

Edit menu

Use this command to locate an object or string of text in a design.

In the schematic page editor and project manager, the Find command finds all instances of the specified text search string, parts, part pins, DRC markers and constraints. The Find command supports wildcard searches. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.

In the session log, the Find command will find the next occurrence of the specified text search string from the current position. However, it does not support wildcard searches.

The Find window in part editor can search only pin and text, and it supports wildcard searches.

Function:

In the design variant schematic page, the Find command will find parts that are not present or parts with different property values attached to it.

This command can be used to search at different levels of a design folder hierarchy in the Project manager.

The search command can be executed at any of the following levels or selections of the design folder hierarchy:

- Design Level
- Folder level For any number of selected folders within the design
- Page Level For any number of selected pages within the design. This includes pages selected from different folders.

Note: The command also allows multi-level selection. This means you can simultaneously select folder and pages.

The search command can also be executed on an open schematic page.

Keyboard:

Shortcuts:

- CTRL+F
- ALT, E, F

Project Manager Command Reference

Rename Part Property command

Available from: Edit menu

Use this command to rename a part property for every placed part that

includes the property for an entire design.

Function:

Note: This command is available only when you have selected the design

(.DSN) or the schematic page(s) in the Project Manager window.

Shortcuts: Keyboard: ALT, E, Y

Delete Part Property command

Available from: Edit menu

Use this command to delete a part property from every placed part in a

design.

Function:
Note: This command is available only when you have selected the design

(.DSN) or the schematic page(s) in the Project Manager window.

Shortcuts: Keyboard: ALT, E, D

Replace command

Available from: Edit menu

Use this command to find and replace a particular string in a text editor

Function: window. When you choose Replace from the Edit menu, the Replace

dialog box appears.

Shortcuts: Keyboard: ALT, E, E

Go To command

Available from: Edit menu

Project Manager Command Reference

Use this command to center the view on a specific <u>location</u>, <u>grid</u> <u>references</u>, or bookmark. By clicking the Go To command, the <u>Go To Line dialog box</u> dialog box appears.

Function:

Note: The Go To command is always available on the right mouse button context-sensitive menus in the part editor and schematic page editor. The Go To command, with the Relative option selected, is particularly useful for precise placement and spacing.

Keyboard:

Shortcuts:

■ CTRL+G

■ ALT, E, G

Clear Session Log command

Available from: Edit menu

Use this command to clear the Session log.

Function:

Note: This command is only available if the Session log is selected.

Keyboard:

Shortcuts:

CTRL+DEL

ALT, E, S

View

Toolbar command on page 68

Capture Toolbar command on page 68

Draw Toolbar command on page 68

PSpice Toolbar command on page 70

FPGA Toolbar command on page 70

CIS Explorer Toolbar command on page 70

Part Manager Toolbar Command on page 71

Project Manager Command Reference

Command Window Command on page 71

Session Log Window Command on page 71

DRCs Command on page 72

Online DRCs Command on page 72

Workspace - File Manager Command on page 72

Workspace - Configuration Command on page 73

Project Manager Command Reference

Toolbar command

Function:

Function:

Available from: View menu

Use this command to show or hide the toolbars. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbars anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats on the

screen wherever it is released.

If the toolbar is floating, you can hide it by choosing Toolbar from the

View menu.

Shortcuts: Keyboard: ALT, V, T

Capture Toolbar command

Available from: View - Toolbar menu

Use this command to show or hide the Capture toolbar. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbar anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats on the

screen wherever it is released.

If the toolbar is floating, you can hide it by choosing Toolbar from the View menu.

Draw Toolbar command

Available from: View - Toolbar menu

Project Manager Command Reference

Function:

Use this command to show or hide the Draw toolbar. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbar anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats on the screen wherever it is released.

If the toolbar is floating, you can hide it by choosing Toolbar from the View menu.

Project Manager Command Reference

PSpice Toolbar command

Function:

Available from: View - Toolbar menu

Use this command to show or hide the PSpice toolbar. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbar anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats on the

screen wherever it is released.

If the toolbar is floating, you can hide it by choosing Toolbar from the

View menu.

FPGA Toolbar command

Available from: View - Toolbar menu

Use this command to show or hide the FPGA toolbar. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbar anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats on the screen wherever it is released.

Function:

If the toolbar is floating, you can hide it by choosing Toolbar from the View menu.

CIS Explorer Toolbar command

Available from: View - Toolbar menu

Project Manager Command Reference

Use this command to show or hide the CIS Explorer toolbar. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbar anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats on the

screen wherever it is released.

If the toolbar is floating, you can hide it by choosing Toolbar from the View menu.

Part Manager Toolbar Command

Function:

Function:

View - Toolbar menu Available from:

> Use this command to show or hide the Part Manager toolbar. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbar anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats

on the screen wherever it is released.

If the toolbar is floating, you can hide it by choosing Toolbar from the View menu.

Command Window Command

Available from: View menu

Use this command to open the TCL Command window in the OrCAD X Capture interface. You can use this command as a toggle to show or

hide Command Window.

For details see the Command Window.

Session Log Window Command

Function:

Project Manager Command Reference

Available from: View menu

Use this command as a toggle to show or hide Command Window.

Session Log Window contains a record of events that occur during the

current session of Capture. See Session Log Window for more

information.

DRCs Command

Function:

Available from: View - Others menu

Use this command as a toggle to show or hide the DRCs window. The

DRCs window in the output area displays a list of electrical DRC errors

and warnings generated from the running Design Rules Checks. It also **Function:**

displays the DRC markers on the schematic design.

Online DRCs Command

View - Others menu Available from:

Use this command as a toggle to show or hide the Online DRCs window.

The Online DRCs window checks and lists the violations of design rules **Function:**

as you are creating or updating the schematic design. Double-click a

violation in this window to move the cursor to the location in the

schematic design where the violation occurred.

Workspace – File Manager Command

Available from: View - Workspace menu

Project Manager Command Reference

Use this command to display the File Manager interface. You access

Cloud workspaces using the *File Manager* user interface. File Manager displays the local workspace and all the workspaces shared by or with

you. The project folder structure in the workspaces is displayed just the

way it appears for a project on a local disk in the project manager.

Workspace – Configuration Command

Function:

Function:

Available from: View - Workspace menu

Use this command to display the Workspace Configuration dialog

box. Use this dialog box to add new workspaces and members. You provide the members access on the workspaces by assigning them

roles with pre-defined access permissions to the workspaces.

Project Manager Command Reference

SI Analysis Menu

SI Library Setup command on page 75

Auto Assign Discrete SI Models command on page 75

Identify DC Nets command on page 75

Export Electrical Csets command on page 75

Import Electrical Csets command on page 76

Remove Electrical Cset Assignments command on page 76

Validate Electrical Cset Assignments command on page 76

Validate SI Model Assignments command on page 76

SI Model Integrity command on page 76

Export SI Models Used command on page 77

Remove SI Model Assignments command on page 77

View Xnet Signals on page 77

Project Manager Command Reference

SI Library Setup command

Available from: SI Analysis menu

Use this command to open the Library Setup (SI Analysis) window and

set the DML libraries.

Auto Assign Discrete SI Models command

Available from: SI Analysis menu

Function: Use this command to assign discrete SI models to the design parts.

Identify DC Nets command

Available from: SI Analysis menu

Use this command to open the Identify DC Nets window, where you can

assign voltage for power and ground nets.

Assign Voltage to Power Nets command

Available from: SI Analysis menu

Use this command to open the Assign Voltage to Power Nets window,

where you can assign voltage to power and ground nets.

Function:
You can specify default values for ground and power nets, and also

specify the naming convention to identify ground nets.

Export Electrical Csets command

Available from: SI Analysis menu

Use this command to open the Export ECSets from design window. You

Function: can export a topology file that can then be updated using SigXplorer

offline in a distributed design environment.

Project Manager Command Reference

Import Electrical Csets command

SI Analysis menu Available from:

Use this command to open the Select Directory window and specify the

directory from where you want to import an ECSet.

Function: This allows you to import topology files that have been updated offline in

a distributed design environment.

Remove Electrical Cset Assignments command

Available from: SI Analysis menu

Use this command to open the Remove ECSets from design window **Function:**

and selectively remove existing ECSets from the design.

Validate Electrical Cset Assignments command

SI Analysis menu Available from:

Use this command to open the Validate ECSets in design window and **Function:**

selectively validate ECSets in the design.

Validate SI Model Assignments command

SI Analysis menu Available from:

Use this command to validate the SI model assignments. **Function:**

SI Model Integrity command

Available from: SI Analysis menu

Use this command to launch Model Integrity with the DML file for the **Function:**

devices.

Project Manager Command Reference

Export SI Models Used command

Available from: SI Analysis menu

Function: Use this command to export the SI models used in the design into a

DML library file.

Remove SI Model Assignments command

Available from: SI Analysis menu

Function: Use this command to remove SI model assignments.

View Xnet Signals

Available from: SI Analysis menu

Use this command to view Xnets that are associated with ECSets.

Function: Note: If you have modified the Xnets in the design after applying the

ECsets, this command will display the updated list of Xnet signals.

Tools menu

Annotate command on page 79

Associate PSpice Model command on page 79

Backannotate command on page 79

Board simulation command on page 81

<u>Update Properties command</u> on page 81

Design Rules Check command on page 82

Create Netlist command on page 84

Project Manager Command Reference

Create Differential Pair command on page 85

Cross Reference command on page 85

Intersheet References command on page 85

Bill of Materials command on page 87

Export Properties command on page 87

Import Properties command on page 89

Generate Part command on page 91

Split Part command on page 93

Sync NetGroup command on page 93

Customize command on page 93

<u>Utilities command</u> on page 95

Project Manager Command Reference

Annotate command

Available from: Tools menu

Use this command to update part references in the active design. You specify the scope and parameters of the update in the Annotate dialog

box.

Function: Annotate will update references for primitive hierarchical blocks. For

> example, you could specify the reference to be "Halfadd?" when you place a hierarchical block. Then, when you run Annotate, the hierarchical

block's reference is updated along with other parts.

Capture Toolbar:

Keyboard: **Shortcuts:**

ALT, T, O

Associate PSpice Model command

Available from: Tools menu

Use this command to associate a PSpice model with an existing Capture Function:

symbol.

Keyboard: ALT, T, M **Shortcuts:**

Backannotate command

Available from: Tools menu

Use this command to swap parts in a package, part references, and pins in the active design based on the contents of a swap file created by you or your PC board layout software. Capture swap files use a . SWP file extension. The purpose of backannotation is to ensure that the physical

information in the board layout is consistent with the logical information

in the schematic design.

Function:

Project Manager Command Reference

Note: Swap files are not true transaction swap files. If your a swap file contains the following lines:

SWAP UI U2 SWAP U2 U3 SWAP U3 U4

the original U1 is changed to U2. It does not change to U3 or U4, as it would in a true transaction system.

Capture Toolbar:



Shortcuts:

Keyboard:

ALT, T, A

Project Manager Command Reference

Board simulation command

Available from: Tools menu

Initiates the board simulation process. This process provides a method for you to simulate your PCB designs (which may include FPGA designs,

Function: as components). You can use Verilog or VHDL to simulation your PCB

designs by choosing the appropriate option in the Board Simulation tab

of the Preferences dialog box.

Update Properties command

Available from: Too

Tools menu

Use this command to update properties based on an update file. This command constructs a combined property string for a part or net. Then, if that string matches a string in the update file, it replaces the specified properties of the combined property string with the update string properties. Capture update files use a .UPD file extension.

Function:

If you are updating net properties, Capture will II of the nets in the schematic folder even if only one schematic page is selected. Capture updates all of the nets in the schematic folder because a single net can appear on more than one schematic page within the schematic folder. Capture only updates the selected schematic folders and schematic pages when updating part properties.



Capture report files are text files, and can be opened in any text editor. You may want to use the tab alignment capability of your word processor to line reports up correctly. Spreadsheets will automatically align the columns of Capture-generated report files.

Project Manager Command Reference

Note: When backannotating from PCB Editor, an update value of "/IGNORE/" is interpreted as a property that is not to be updated. When "/IGNORE/" is found, the property in the schematic unchanged from it's previous value. For example, in the following line, the "TOL" property will not be updated for the part with a reference of "U1".

"{Part Reference}" "TOL"

"R1" "10%"

"U1" "/IGNORE/"

You can update properties of parts in libraries as well as update properties of parts in designs.

Shortcuts:

Keyboard: ALT, T, U

Design Rules Check command

Available from:

PCB menu

Function:

Use this command to check a design for violations of design rules. Capture places DRC error markers on schematic pages as needed. You can search for the markers by using the Browse DRC Markers command on the Edit menu.

Note: Generally, you should run Design Rules Check to verify your design before you generate a netlist. This allows for more efficient netlist creation, and you can concentrate on netlist-specific problems if they should occur during the Create Netlist process. Design Rules Check warns you if certain conditions exist in your design. The severity of the specific problem may prevent completion of the design. Other conditions are subject to your judgment, and may be of no consequence. Once you are satisfied with the results of design tests like Design Rules Check, then proceed with the creation of a netlist.

Note: Design Rules Check uses the decision matrix located in the ERC Matrix tab in the Design Rules Check dialog box. It also uses a set of predetermined rules, which are part of the executable code.

Note: Use Design Rules Check as a guide to verify the integrity of your design. It is only a guide. It is possible to generate a valid netlist even if Design Rules Check reports errors.

Project Manager Command Reference



If you run a Design Rules Check on a single schematic page, Capture checks the entire schematic folder the schematic page is in. This ensures that all nets on the schematic page are valid.

If you select the Check hierarchical port connections option, Capture also checks the attached schematic folders.



Capture report files are text files, and can be opened in any text editor. You may want to use the tab alignment capability of your word processor to line reports up correctly. Spreadsheets will automatically align the columns of Capture-generated report files.

Capture Toolbar:



Shortcuts:

Keyboard:

ALT, T, D

Project Manager Command Reference

Create Netlist command

Available from:

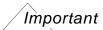
Tools menu

Use this command to create a netlist from the selected design. This command displays the Create Netlist dialog box, a tabbed dialog box that you use to choose a netlist format.

Function:

If you have translated a design with multiple schematic folders, use Annotate (and check for duplicate references) before you create a netlist.

Note: Note that screws, washer, and other hardware appear in a bill of materials, but not in a netlist. Netlists include only objects with pins.



Generally, you should run Design Rules Check to verify your design before you generate a netlist. This allows for more efficient netlist creation, and you can concentrate on netlist-specific problems if they should occur during the Create Netlist process. Design Rules Check warns you if certain conditions exist in your design. The severity of the specific problem may prevent completion of the design. Other conditions are subject to your judgment, and may be of no consequence. Once you are satisfied with the results of design tests like Design Rules Check, then proceed with the creation of a netlist.

Note: The value, if any, you create for the PCB footprint depends on the particular netlist format you want to produce. Different applications require netlists with different types of PCB footprints. If you do not specify this property, the PCB footprint will be set to the part value.



Capture report files are text files, and can be opened in any text editor. You may want to use the tab alignment capability of your word processor to line reports up correctly. Spreadsheets will automatically align the columns of Capture-generated report files.

Capture Toolbar:



Shortcuts:

Keyboard:

ALT, T, N

Project Manager Command Reference

Create Differential Pair command

Available from: Tools menu

Function:

Use this command to create a differential pair between two flat nets in

your design. This command displays the <u>Create Differential Pair dialog</u> box where you select the nets to be associated with a differential pair.

You can also use the Create Differential Pair dialog box to modify or

delete a differential pair.

Note: You can access this command only if you select a design file

(.DSN) or a schematic page (.sch) in the project manager.

Shortcuts: Keyboard: ALT, T, F

Cross Reference command

Available from: Tools menu

Use this command to create a cross-reference listing telling you where

each part is located, and the library it comes from.



Capture report files are text files, and can be opened in any text editor. You may want to use the tab alignment capability of your word processor to line reports up correctly. Spreadsheets will automatically align the columns of Capture-generated report files.

Capture Toolbar:



Shortcuts: Keyboard:

■ ALT, T, C

Intersheet References command

Available from: Tools menu

Use this command to create the intersheet references for the signals on

Function: a design. In addition, you can also generate a report for all the signals on

the design.

OrCAD X Capture CIS Reference Guide Project Manager Command Reference

Keyboard: ALT, T, T **Shortcuts:**

Project Manager Command Reference

Bill of Materials command

Available from:

Tools menu

Function:

Use this command to create a summary list of all parts used in the design. You can also use an include file to add information to the bill of materials. By default, Capture include files use a .INC file extension.



Reference designators should not exceed 24 characters. When Bill of Materials encounters a reference designator that is longer than 24 characters, an error occurs and the bill of materials isn't generated.



Capture report files are text files, and can be opened in any text editor. You may want to use the tab alignment capability of your word processor to line reports up correctly. Spreadsheets will automatically align the columns of Capture-generated report files.

Capture Toolbar:



Shortcuts:

Keyboard:

ALT, T, B

Export Properties command

Available from: Tools menu

Use this command to write the properties of the selected documents to an ASCII text file. Properties are delimited by tabs so the file is suitable for manipulation by spreadsheet or database software. You can export properties from a design or library.

Function:

For more information on property files see **Exporting part and pin**

properties.

Project Manager Command Reference

Important

If you add, delete, or reorder lines in a project's property file, the file cannot be imported.

If you move a PART line in a design property file created in a library property file, be sure to move all the PIN lines associated with it and keep them in the same order. Otherwise, importing the file may fail or cause unwanted changes to your project or library.

In every case, it is much safer to refrain from adding, deleting, or reordering the lines in a property file.

Note: It is a good idea to update part references before you export properties.

Because various popular spreadsheet and database applications behave differently, Capture can import properties with or without enclosing quotation marks around each field in the property file. The fields must be tab-delimited, though—all other characters, including commas and leading and trailing spaces, are treated as part of a field's text. Be sure your spreadsheet or database program can save in this format.

Shortcuts:

Keyboard: ALT, T, E

Project Manager Command Reference

Export Placement command

Project manager shortcut menu Available from:

Use this command to generate a placement report for the selected levels

of the current design hierarchy.

Function: You can generate the report at any level of the hierarchy, page, folder or

design. You can also generate the report at different level by

multi-selecting the level in the Project manager.

The report is saved in the design directory with the name <design

name>-placements.csv.

Save as HTML command

Find window shortcut menu Available from:

Use this command to generate a report, in HTML format, for the results **Function:**

of a Find command.

The report is saved in the design directory with the name

Search_Result<design name>.html.

Save as CSV command

Find window shortcut menu Available from:

Use this command to generate a report, in CSV format, for the results of **Function:**

a Find command of a specific object type.

The report is saved in the design directory with the name <design

name>-<object type>-Report.csv.

Import Properties command

Available from: Tools menu

Project Manager Command Reference

Use this command to import the contents of a tab-delimited property file. The imported properties may add to or supersede existing properties. The property file must be in the format used by Capture when you choose the Export Properties command from the Tools menu. You can import properties to a design or library.

Function:

For more information on property files see Exporting part and pin properties.

The Import Properties command opens a standard Windows dialog box for opening files.

Important

If you add, delete, or reorder lines in a project's property file, the file cannot be imported.

If you move a PART line in a design property file created in a library property file, be sure to move all the PIN lines associated with it and keep them in the same order; otherwise, importing the file may fail or cause unwanted changes to your project or library.

In every case, it is much safer to refrain from adding, deleting, or reordering the lines in a property file.

Note: It is a good idea to update part references before you export properties.

Because various popular spreadsheet and database applications behave differently, Capture can import properties with or without enclosing quotation marks around each field in the property file. The fields must be tab-delimited, though—all other characters, including commas and leading and trailing spaces, are treated as part of a field's text. Be sure your spreadsheet or database program can save in this format.

Note: If Capture finds errors in the property file, the project or library remains unchanged. There is no risk that some parts will be changed and others not.

Shortcuts:

Keyboard: ALT, T, I

Project Manager Command Reference

Generate Part command

Available from:

Tools menu

Use this command to create a library and an associated part that represents your design. You can use a schematic, a library containing schematics, or a netlist to generate a symbol for the project.

You can use the symbol you generate with the Generate Part command to represent the actual component (such as an FPGA or CPLD) in schematic designs or other projects. When you use the Generate Part command, Capture creates a library file (.OLB) and part based on the pins defined in the report file or schematic and references it in the Outputs folder of the project manager. If the library already exists, the new part is appended to the existing library. If the part already exists in the library, the new part replaces it.

Function:

Note: If the schematic from which you generate a part has one or more Param symbols, the entries in that symbol are placed on the resulting part as user properties. You can then overwrite default values for those properties on a specific instance or occurrence.

The Generate Part dialog box offers a number of netlist and source file types that are used to generate a part or symbol. The Capture Schematic/Design source file type uses a library or a single schematic to create the new .OLB containing the new part and a copy of the schematic for easy portability and design reuse. You can use this source type to create a design reuse module.

Capture reads a variety of PLD vendor pin reports to create library parts for the Capture schematic system. Most PLD vendor pin reports describe the pin number, signal name, and direction (or mode) of a package pin programmed by the place-and-route process. Pins are sorted alphabetically by name, with input type pins located on the left-hand side, and output or bidirectional pins on the right-hand side.

Generate part can create new parts or update the pin numbers of an existing library part with the Update pins on existing part in library option, which allows for engineering change orders (ECOs) from a programmable logic project to update the part symbol of the system schematic.

Note: To create a pin on a symbol using the Generate Part utility, the pin must have a pin to port mapping in the pin file.

OrCAD X Capture CIS Reference Guide Project Manager Command Reference

Keyboard: ALT, T, G **Shortcuts:**

Project Manager Command Reference

Export FPGA command

Available from: Tools menu

Use this command export the selected (or all) part to export as an FPGA

oart.

Function:

Note: You can select a vendor family and if you want to generate if a

VHDL or Verilog wrapper file.

Shortcuts: Keyboard: ALT, T, X

Split Part command

Available from: Tools menu

Use this command to divide a part into multiple sections.

Function: Note: You need to select a single-sectioned part from a library. You can

split a multi-sectioned part only when it has been split already using this

command.

Shortcuts: Keyboard: ALT, T, L

Sync NetGroup command

Available from: Tools menu

If a hierarchical block pin in connected to a NetGroup and the width

(number of contained signals) of the NetGroup is modified, use this

command to synchronize the number of signals in the hierarchical pin

and the NetGroup.

Customize command

Available from: Tools menu

Function:

Project Manager Command Reference

Use this command to customize the OrCAD X Capture menus.

You can:

Function:

- choose the menus to show and hide.
- add or remove buttons (commands) from the menus.
- create a custom menu and add buttons (commands) to the menu.

Project Manager Command Reference

Utilities command

Available from: Tools menu

Use this command to use different utilities available in the Utilities command.

Using Utilities, you can:

- start and stop a communication server
- replace library path in the design cache

Function:

- see opened libraries and designs in the current session
- customize the Schematic Page using TCL
- correct the library path
- find and replace text, such as comment text, alias text, global name text, off-page connector name text, and port name text.

Communication Server

Available from: Tools - Utilities menu

Project Manager Command Reference

Use this command for a TCL-based communication server-client framework. It enables Capture as a communication server. You can write your own server and client-side TCL methods and use this framework to establish a communication channel between Capture (server) and other applications (client).

You can enter a port number and start the server using the *Start* button. You can stop the server using the *Stop* button.

You can invoke an application, which is a client, to invoke TCL commands on Capture, which acts as server. For example, use Command Window to source a TCL script for Capture (server).

Function:

```
source {<Installation
Path>\tools\capture\tclscripts\capCustomSamples\capCommServerMe
thods.tcl}
```

Similarly, you can use the TCL shell (tclsh) as client to do any operation on a Capture object in Capture. For example, use the following TCL script from TCL shell to select a Capture object in the Full Adder design in Capture.

```
source {<Installation
Path>\tools\capture\tclscripts\capCustomSamples\capCommClientMe
thods.tcl}
::capCommClientMethod::SelectObject 4.79 2.49
```

Project Manager Command Reference

Open Library and Design

Available from: Tools - Utilities menu

Use this command to display the following in the current session of Capture:

Function:

- list of design files (.dsn)
- list of library files (.olb) added by user in the current session

Page Customization

Available from: Tools - Utilities menu

Using this command you can customize the process of page creation and page size change by specifying different TCL procedures that are called automatically at the following instances:

- before the page is created (On Page Pre-Create)
- after the page is created (On Page Post-Create)
- when the page size changes (On Page Size Change)

You need to specify your TCL script path along with the Callback procedures in the Page Customization dialog box. One or more than one of the callback procedure field can be left empty if specific handle any of these callbacks is not required. Once done, these options are saved in the Capture.ini file.

Function:

For your reference, the following procedures from the sample TCL script are added by default in *On Page Pre-Create*, *On Page Post-Create*, and *On Page Size Change* fields:

■ ::capCustomizePage::onPagePreCreate

■ ::capCustomizePage::onPagePostCreate

■ ::capCustomizePage::onPageSizeChange

The sample TCL script file that is used in the Page Customization window is *<installation*

path>/tools/capture/tclscripts/capCustomSamples/capCustomize Page.tcl.

Project Manager Command Reference

Check/Correct Corrupt Library

Available from: Tools - Utilities menu

Use this command to check or correct the corrupted Capture library (.lib):

Function: Specify the corrupted Capture library path

Specify the log file path

Find and Replace Text

Available from: Tools - Utilities menu

Use this command to find and replace the one or more of the following text type in a Capture design:

Comment Text

Function:

Global Name

Off-page Connector Name

Port Name

Replace Path in Design Cache

Available from: Tools - Utilities menu

Project Manager Command Reference

Use this command to replace the library path and design path in the design cache of a Capture Design. To use the command, do the following:

- Specify the old library path that is used in the design cache
- Specify the new library path that you want to use
- If you want to replace the design path, select the *Replace Also.DSN Path* check box.
- In the Report section, you will see the following columns:
 - Index: Specifying row number

Function:

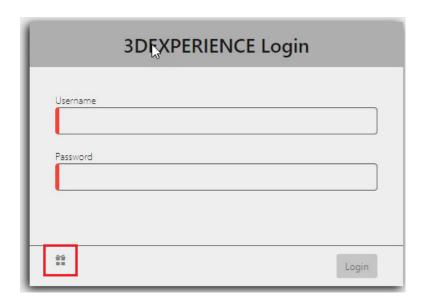
- □ Lib Object: Specifying Library Object
- Status: Specifying if replaced or not
- □ Old Library: Specifying Old Library path
- New Library: Specifying New Library path
- Type: Specifying type of the library part
- □ Reason: Specifying success or failure message
- In the Logs section, you will see message related to how many instances have been replaced in the design.

Publish for Manufacturing

Available from: Tools menu. Tools – Publish for Manufacturing.

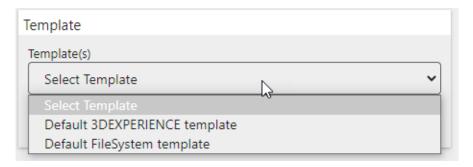
Project Manager Command Reference

Use this command to run the Publish for Manufacturing application. When the server connection is established, the 3D EXPERIENCE Login dialog box prompts for a username and password to publish data to the 3D EXPERIENCE PLM Server using a Cadence-defined template. The template generates a dataset and publishes a Cadence-defined product structure using the 3DEXPERIENCE data types.



Function:

Select the *Switch Template* icon to display the *Template* selection box, from where you can select another option to publish data:



Selecting the *Default FileSystem template* option, publishes data to a folder on the file system. When clients are connected to a Pulse server, the Location value specified for the connector refers to a folder in the Pulse server.

For more information on Publish for Manufacturing, see <u>Publish for Manufacturing User</u> <u>Guide</u>.

Project Manager Command Reference

PSpice

Bias Points command on page 102

New Simulation Profile command on page 102

Edit Simulation Profile command on page 102

Run command on page 103

View Simulation Results command on page 103

View Output File command on page 105

Make Active command on page 105

Simulate Selected Profile(s) command on page 105

Create Netlist command on page 106

View Netlist command on page 106

Marker List command on page 106

Advanced Analysis menu on page 107

Project Manager Command Reference

Bias Points command

Available from: PSpice menu, Bias Points submenu

From a schematic page, point to Markers on the PSpice menu.

Function: Use the commands in the Bias Points submenu to enable and disable

bias point display, toggle selected bias points and to set preferences for

displaying and printing bias points.

Shortcuts: Keyboard: ALT, S, B, E

New Simulation Profile command

Available from: PSpice menu

You must create a simulation profile (or edit an existing one) before you can set up a PSpice simulation. Use this command to create a new

simulation profile. A simulation profile (*.SIM) saves your simulation

Function: settings for an analysis type so you can reuse them easily.

After creating a new profile, you can edit the settings with the Edit

Simulation Settings command.

PSpice Toolbar:

Shortcuts: Keyboard:

■ ALT, S, N

Edit Simulation Profile command

Available from: PSpice menu

Use this command to edit an existing PSpice simulation profile.

Simulation profiles can be edited in Capture and PSpice.

PSpice Toolbar:

%

Shortcuts: Keyboard:

■ ALT, S, E

Project Manager Command Reference

Run command

Available from:

PSpice menu

After setting all the simulation parameters you need, choose Run to perform the simulation. This command automatically performs the following steps:

- checks design rules for your design.
- creates a simulation netlist for PSpice.
- opens PSpice using the netlist created from your design.

Function:

PSpice creates an output file (.OUT) as the simulation progresses. It contains bias point information, model parameter values, error messages, and so on. If the simulation fails, you can view the output file to see the error messages.

If the simulation completes successfully, PSpice produces a data file (.DAT). This is the file PSpice uses to display the simulation results.

To see marker simulation results, the schematic must be open.

Note: You can run PSpice simulations from the Capture environment by pressing the F11 function key.

PSpice Toolbar:



Shortcuts:

Keyboard:

ALT, S, R

View Simulation Results command

Available from: PSpice menu

Use this command to view the most recent simulation results, for the active simulation profile. To see marker simulation results, the schematic must be open.

Function:

Note: You can view the simulation results from the currently active profile by pressing the F12 function key.

Project Manager Command Reference

PSpice Toolbar:



Keyboard: **Shortcuts:**

ALT, S, V

Project Manager Command Reference

View Output File command

Available from: PS

PSpice menu

Function:

Use this command to view the most recent output file for the current

design.

Shortcuts:

Keyboard: ALT, S, W

Make Active command

Available from:

PSpice menu

Function:

Use this command, with the project manager active, to make the

selected simulation profile the active profile. In the project manager, the

simulation profile folder is in the PSpice Resources folder.

Simulate Selected Profile(s) command

Available from:

PSpice menu

Use this command, with the project manager active, to simulate selected profiles.

Simulate Selected Profile(s) automatically performs the following steps:

- Checks design rules for your design.
- Creates a simulation netlist for PSpice.

Function:

Opens PSpice using the netlist created from your design.

You can select one profile or multiple profiles to be simulated or viewed. If you select only one profile for simulation, it is handled as though you chose the Run command. If you select one file for viewing, it is handled as though you chose the View Simulation Results command.

If you select multiple profiles, simulations for all selected profiles are performed using the simulation queue. You must then open the .DAT files to view the results.

Shortcuts:

Keyboard: ALT, P, A

Project Manager Command Reference

Create Netlist command

Available from: PSpice menu

Function:

Use this command to create a simulation netlist for the current design. The netlist is generated for all levels of hierarchy, starting from the top, regardless of whether you are pushed into any level of the hierarchy.

In OrCAD X Capture, the TEMPLATE property specifies the primitive parts' contributions to the netlist. In the process of creating the netlist,

buses, connectors, and so on, are resolved. Only parts with a

TEMPLATE property are included in the simulation.

During the netlist process, Capture creates several files with different extensions: The .NET file contains the netlist; the .CIR file contains simulation commands; and the .ALS file contains alias information.

Shortcuts: Keyboard: ALT, S, C

View Netlist command

Available from: PSpice menu

Function: Use this command to view the most recent simulation netlist for a

selected or the current design.

Shortcuts: Keyboard: ALT, S, I

Marker List command

Available from: PSpice menu

From the Project Manager, choose Marker List from the PSpice menu.

Use this command to display or hide markers in the design. The Markers

Function:

dialog box displays markers that exist in the profile. An empty check box

dialog box displays markers that exist in the profile. An empty check box

beside a marker indicates that the marker is hidden. If a marker is hidden, it will not appear in Capture but it will still exist in the profile.

Shortcuts: Keyboard: ALT, S, L

Project Manager Command Reference

Advanced Analysis menu

Sensitivity command on page 108

Optimizer command on page 108

Monte Carlo command on page 108

Smoke command on page 110

Export Parameters to Optimizer command on page 110

Import Optimizable Parameters command on page 110

Project Manager Command Reference

Sensitivity command

Available from: PSpice menu, Advanced Analysis submenu

If you have installed PSpice Advanced Analysis, use this command to

run Sensitivity Analysis.

Function: Sensitivity analysis identifies which components have parameters critical

to the measurement goals of your circuit design. See the PSpice

Advanced Analysis User Guide, for more information on the Sensitivity

Analysis tool.

Shortcuts: Keyboard: ALT, S, D, S

Optimizer command

Available from: PSpice menu, Advanced Analysis submenu

If you have installed PSpice Advanced Analysis or PSpice Optimizer, use

this command to optimize your design.

Optimizer is a design tool for optimizing analog circuits and their behavior. It helps you modify and optimize analog designs to meet your

performance goals. For more information on the Optimizer tool, see:

Function:

PSpice Advanced Analysis User Guide, if you have installed PSpice

Advanced Analysis

PSpice Optimizer User Guide (Capture version), if you have installed

PSpice Advanced Analysis

Shortcuts: Keyboard: ALT, S, D, O

Monte Carlo command

Available from: PSpice menu, Advanced Analysis submenu

Project Manager Command Reference

If you have installed PSpice Advanced Analysis, use this command to run Monte Carlo analysis.

Function:

Monte Carlo analysis predicts the behavior of a circuit statistically when part values are varied within their tolerance range. Monte Carlo analysis also calculates yield, which can be used for mass manufacturing predictions. See the PSpice Advanced Analysis User Guide, for more information on the Monte Carlo analysis tool.

Shortcuts:

Keyboard: ALT, S, D, M

Project Manager Command Reference

Smoke command

Available from: PSpice menu, Advanced Analysis submenu

If you have installed PSpice Advanced Analysis, use this command to

run Smoke analysis.

Function: Smoke analysis determines whether components in your design are

operating within their safe operating limits. See the PSpice Advanced Analysis User Guide, for more information on the Smoke analysis tool.

Shortcuts: Keyboard: ALT, S, D, K

Export Parameters to Optimizer command

Available from: PSpice menu, Advanced Analysis submenu

If you have installed PSpice Advanced Analysis or PSpice Optimizer, use this command to export device-level parameters to Optimizer.

Select a component on the schematic and use this command to export its device-level parameters to the Optimizer tool. The component and its parameters are added in the Parameters table.

Note: This feature of exporting device-level parameters to Optimizer is available only if the selected component is based on PSpice-provided templates.

Function:

For more information on the Optimizer tool, see:

- PSpice Advanced Analysis User Guide, if you have installed PSpice Advanced Analysis
- PSpice Optimizer User Guide (Capture version), if you have installed PSpice Advanced Analysis

Shortcuts:

Keyboard: ALT, S, D, E

Import Optimizable Parameters command

Available from: PSpice menu, Advanced Analysis submenu

Project Manager Command Reference

If you have installed PSpice Advanced Analysis or PSpice Optimizer, use this command to import optimizable parameters from the component's model library as instance properties.

Select a component on the schematic and use this command to import optimizable parameters from the component's model library. The parameter names and their default values are displayed on the component instance in the schematic editor.

Function:

This feature of importing optimizable parameters is available only if the selected component is based on PSpice-provided templates.

For more information on the Optimizer tool, see:

- PSpice Advanced Analysis User Guide, if you have installed PSpice Advanced Analysis
- PSpice Optimizer User Guide (Capture version), if you have installed PSpice Advanced Analysis

Shortcuts:

Keyboard: ALT, S, D, I

Accessories menu

Accessories commands on page 112

Rotate Aliases command on page 112

Library Verification/Correction command on page 112

Push Occ. Prop into Instance command on page 112

Export Hierarchy command on page 114

Export Hierarchy with Parts command on page 114

Mentor Netlist command on page 114

Mentor Netlist (to UNIX) command on page 114

Mentor Backannotation command on page 115

Project Manager Command Reference

Accessories commands

Available from: Accessories menu

You can use extensions to the OrCAD X-supplied functionality of Capture if you purchase software developed by associates of OrCAD X.

These associates create .DLL files that address specific Capture

Function: functionality, such as customized netlisting. The associates configure

their .DLL files so that they are listed as menu choices in the

Accessories menu, which is available in either the project manager

window or the schematic page editor window.

Rotate Aliases command

Available from: Accessories menu, AliasRot submenu

Use this command to rotate 270 degree aliases, ports, and off page

connector names to 90 degrees.

Library Verification/Correction command

Available from: Accessories menu, Library Correction Utility submenu

Use this command to:

- Verify and correct all the parts with missing pin numbers or duplicate pin names in a library.
- Change all lowercase pin numbers and pin names to uppercase in a library.
- Make all Power pins visible in a library.

This command launches the Library Correction Utility dialog box where you specify the name of the library that you want to correct and select the correction to be done.

Push Occ. Prop into Instance command

Available from: Accessories menu, Transfer Occ. Prop. to Instance submenu

Function:

Project Manager Command Reference

Suppose that you copied a circuit or part of a circuit from design A and pasted it in design B. You might see occurrence and instance level properties with different values on the pasted parts/nets in design B. For example, the reference designators of the occurrences and instances may be different. To avoid confusion in the future you have to ensure that each part has only one reference designator by replacing the instance value of the part reference property with the occurrence value of the part reference property for each part.

Function:

Use this command to automatically:

- transfer occurrence property values of the part reference and PCB footprint properties as instance level property values.
 - remove all occurrence properties from the design, and set Capture to update instance properties when the design is backannotated.
- transfer occurrence property values of flat nets to schematic nets.

This command opens the <u>Push Occ. properties to instance dialog box</u>.

Project Manager Command Reference

Start Page command

Function:

Available from: Select Help - Start Page

Use this command to go to the OrCAD X Capture Start Page.

Export Hierarchy command

Available from: Accessories menu, Hierarchy Report submenu

Use this command to generate a report that depicts the hierarchical

structure of a design. This command generates a .txt file that lists all the **Function:**

schematics used in the design and stores the file under the Outputs

folder in project manager.

Export Hierarchy with Parts command

Available from: Accessories menu, Hierarchy Report submenu

Use this command to generate a report that depicts the complete

hierarchical structure of a design listing all the parts that are included in

Function: the design. This command generates a .txt file that lists all the

schematics along with all the parts used in the design and stores the file

under the Outputs folder in project manager.

Mentor Netlist command

Available from: Accessories menu, Mentor submenu

Function: Use this command to create a netlist in Mentor Graphics format.

Mentor Netlist (to UNIX) command

Available from: Accessories menu, Mentor submenu

____ Use this command to create a netlist in Mentor Graphics format for

Function: UNIX.

Project Manager Command Reference

Mentor Backannotation command

Available from: Accessories menu, Mentor submenu

Use this command to backannotate Mentor board information to Capture

design

Options menu

Function:

Autobackup command on page 116

Preferences command on page 116

Design Template command on page 121

Design Properties command on page 121

Project Manager Command Reference

Autobackup command

Available from: Options menu

Determines the frequency, location, and the number of copies of

autobackup done by Capture.

The Multi-level backup Settings dialog box appears when you choose

Autobackup option from the Options menu.

Enter the values for the following fields to determine the duration,

number of backups, and its storage.

Function: Backup time (in minutes) - Enables you to determine the time after

which Capture will perform automatic backup.

No of backups to keep - Enables you to determine the total number of

backups that will be stored.

Directory for backup - Enables you to determine the storage location

for the backup

Shortcuts: Keyboard: ALT, O, B

Preferences command

Available from: Options menu

Use this command to set your environment preferences for the current

project (and all future projects) on your system. The options you specify

affect the behavior of the software, and are saved in the .INI file.

Shortcuts: Keyboard: ALT, O, P

More Preferences command

Available from: Options menu, Preferences submenu

Function:

Project Manager Command Reference

Use this command to set extended environment preferences for the current project (and all future projects) on your system. The options you specify affect the behavior of the software, and are saved in the .INI file.

Command Shell

These commands are related to the TCL command window in Capture.

- **Journaling**: Select the option to enable journaling of the various Capture commands, including TCL commands.
- Flush Commands: Select the option to print the journaling commands in a text file. By default, the text file is saved in the TEMP folder (<%TEMP%>\CAPTURELOG\<%DATE%>\
 OrCaptureLogFile.captcl)
- Display Commands: Select the option to enable display of Capture commands in Command Window.

Design and Libraries

Function:

These commands are related to Capture's designs and libraries.

- Content text instance properties: Select the option to rotate the instance part properties along with the instance part on the same axis.
- **Draw arrow on part input pins**: Select the option to draw arrow on part's input pins.
- Enable communication with legacy tools: Select the option to enable message-based communication with the legacy tools, such as PCB Editor.

Note: By default, message-based communication is not enabled in OrCAD X Capture. Enable this option if you want OrCAD X Capture to communicate with the legacy tool as it impacts OrCAD X Capture's performance.

■ Perform read only check on tab switch: Select the option to enable Capture to check the library and design files' permissions on every switch of tab.

Note: Deselecting the option improves user experience as Capture ignores the permissions check on the design and library files.

Project Manager Command Reference

- Backannotate pin numbers only: Select the option to ignore pin name changes during back annotation.
- Save design name as UPPERCASE: Select the option to save the design file name (.dsn) in uppercase letters.
- Enable global net ITC: Select the option to start cross probing of global nets.
- Convert images to BMP format: Select the option to convert all format images to .bmp internally.
- Path lookup timeout (in seconds): Specify the time out time in seconds if the design or library files are found at the configured locations.
- Net Naming Options (requires application restart): The drop down menu options provides different options to select how Capture generates flat-net names for complex hierarchy designs. By default, flatname has a hierarchy block path. Following are the drop down options:

Functions

- Append hierarchy on collision: Appends hierarchical path to the conflicting flat nets only (for global nets, numeric ID is used). The flat net that is closest to the root design and is the only flat net at that level is not renamed.
- Always append hierarchy: Appends hierarchical path to all the flat nets except those in root schematic. For global nets, numeric ID is used as they do not have any hierarchical path.
- □ Append ID on collision: Appends numeric IDs to all the conflicting flat nets only. The flat net that is closest to the root schematic and is the only flat net at that level, is not renamed.
- □ **Always append ID**: Appends numeric IDs to all the flat nets except those in the root schematic.

Design Cache

■ **Update Cache**: Select the Default or Forced option to update cache in Capture. Selecting Forced will update the selected library even if it is older than the original library file in the Design Cache.

Project Manager Command Reference

DRC

Select the *Display Waived DRC* check box to display all the waived DRCs on a schematic page.

CIS

These commands are related to CIS operations in Capture.

■ Query All Configured Tables

Select this option to query all the tables that are configured in the DBC configuration.

■ Disable Regional Setting

Select this option to disable regional setting.

Quoted (") refdes in variant list

Select this option to quote (") part references in the variant export report.

Functions:

NetGroup

The following options, which are related to NetGroups, will be set only for the current active design:

- Never
- Always
- Only when mismatch (definition name mismatched instance name)

Netlist

This command is related to Netlisting in Capture.

■ Apply Allegro Character Limits on All Projects

Select this option to set the character limit globally for all the projects.

Project Manager Command Reference

Schematic

These commands are related to Schematic page in Capture.

Schematic Descend

The following three modes are used for schematic descend:

- Default: Opens the default page of child schematic
- ☐ **First**: Opens the first page of child schematic (alphabetical-wise)
- □ Ask: Asks user to select one of the schematic page from the list of schematic pages

Junction Mode

The following two modes are used for create junctions in Capture:

□ Default

Select this option to place a junction at a straight wire break point.

Junction on multiple connections on wire end

Select this option to create a junction when there are multiple connections on a wire end.

Note: It is recommended that in order to create a junction automatically on placing a wire, you should first select the Junction on multiple connections on wire end option and then place the wire.

Display underscore (_) on User Assigned Part References on Schematic Page

Select this option to display underscore (_) symbol on user-assigned part references on a schematic page.

■ Display underscore (_) on User Assigned Part References in Page Print

Select this option to display underscore (_) symbol on user-assigned part references in page or design print.

■ Distribute in a fixed area (may cause uneven distribution)

Select this option to distribute the selected objects in a fixed area.

Project Manager Command Reference

Design Template command

Available from: Options menu

Function:

Use this command to specify default settings for new projects, designs,

and schematic pages. The values specified in this dialog box do not

affect existing projects or designs.

Note: To change the properties of an active design, use the <u>Design</u> <u>Properties command</u>. To change the properties of an active schematic page, use the <u>Schematic Page Properties command</u>. You cannot change

the default title block of an active schematic page.

Shortcuts: Keyboard: ALT, O, D

Design Properties command

Available from: Options menu

Use this command in the project manager to globally set design related

options throughout a design.

Note: To change the properties for objects in new designs, use the

Design Template command.

Shortcuts: Keyboard: ALT, O, R

Window menu

New Window command on page 123

Cascade command on page 123

Tile Horizontally command on page 123

Tile Vertically command on page 123

Close All Tabs of Active Project command on page 124

Close All Tabs of Active Project Except Current command on page 124

Arrange Icons command on page 125

1,2.... command on page 125

OrCAD X Capture CIS Reference Guide Project Manager Command Reference

Close All Windows on page 125

Project Manager Command Reference

New Window command

Available from: Window menu

Use this command to create a new window, which is a copy of the

Function: currently active window. This new window is another *view* on the same

data, and you can scroll the two windows to different positions.

Shortcuts: Keyboard: ALT, W, N

Cascade command

Available from: Window menu

Use this command to *stack* all open Capture windows so that just their

title bars are visible. The active window stays on top.

Shortcuts: Keyboard: ALT, W, C

Tile Horizontally command

Available from: Window menu

Use this command to arrange open Capture windows, one above

another, so that all are visible.

Shortcuts: Keyboard: ALT, W, H

Tile Vertically command

Available from: Window menu

Use this command to arrange open Capture windows, one beside

another, so that all are visible.

Shortcuts: Keyboard: ALT, W, V

Project Manager Command Reference

Close All Tabs of Active Project command

Available from: Window menu

Function: Use this command to close all tabs of the active project.

Shortcuts: Keyboard: ALT, W, C

Close All Tabs of Active Project Except Current command

Available from: Window menu

Use this command to close all tabs of the active project except the tab

Function: that is currently active. This option is not available for the Project

Manager tab.

Shortcuts: Keyboard: ALT, W, C

Project Manager Command Reference

Arrange Icons command

Available from: Window menu

Function: Use this command to arrange the icons for minimized windows across

the bottom of the session frame.

Shortcuts: Keyboard: ALT, W, A

1,2.... command

Function:

Available from: Window menu

Use the numbers listed at the bottom of the Window menu to view which windows are currently open, and to determine which window is active.

(The active window is indicated by a check mark.) When you choose a

window from this list, Capture restores that window if it was in icon form,

pops it to the front of the Capture session, and makes it the active

window.

Shortcuts: Keyboard: ALT, W, n (n = 1, 2, ...)

Close All Windows

Available from: Window menu

Function: Use this command to close all open windows.

Help menu

OrCAD X Capture Help command on page 127

Known Problems and Solutions command on page 127

What's New command on page 127

Learning PSpice command on page 127

About OrCAD X Capture command on page 129

Project Manager Command Reference

Web Resources command on page 129

Documentation command on page 131

PSpice Documentation command on page 131

Note: To use the context-sensitive menu commands, select one or more items, then press the right mouse button. The contents of the menu differ depending on the objects selected.

Project Manager Command Reference

OrCAD X Capture Help command

Available from: Help menu

Function: Use this command to display the Capture Help window.

Capture Toolbar:

Shortcuts: Keyboard: ALT, H, H or F1

Known Problems and Solutions command

Available from: Help menu

Use this command to display a document listing the known problems in

Function: this release of OrCAD X Capture and tells you how to solve or work

around these problems.

Shortcuts: Keyboard: ALT, H, K

What's New command

Available from: Help menu

Use this command to display a document describing the new features

and enhancements in this release.

Shortcuts: Keyboard: ALT, H, K

Learning PSpice command

Available from: Help menu

Project Manager Command Reference

Use this command to run the interactive learning material that uses designs created in Capture to explain several diverse topics, ranging from basic theorems to some of the advanced topics in the field of Electrical & Electronics Engineering. It also introduces basic Electronics

Function:

Design Automation (EDA) concepts.

The Learning PSpice tab has a browsing pane on the left that lists the different topics and subtopics. Click on any topic to open the content in the right pane. Each circuit is accompanied by an icon on the right that you can click to open the circuit in OrCAD X Capture.

Shortcuts:

Keyboard: ALT, H, R

Project Manager Command Reference

About OrCAD X Capture command

Available from: Help menu

Function: Use this command to get the software version number, copyright

information, registration number, and license information.

Shortcuts: Keyboard: ALT, H, A

Web Resources command

Available from: Project manager Help menu, or schematic page editor Help menu

Function: Use this command to link to Capture resources on the web

Shortcuts: Keyboard: ALT, H, W

Note: You can add other web resources to the displayed list by modifying your CAPTURE.INI file. Note, however, that the first resource in the list, appears as the last name in the menu. Therefore, if you want to add a web resource to the top of the list, include it in the CAPTURE.INI file as the

last entry.

Cadence Online Support command

Available from: Help menu, Web Resources command

Use this command to launch your web browser and visit the Cadence

Customer Support web site.

Shortcuts: Keyboard: ALT, H, W, S

Training command

Available from: Help menu, Web Resources command

Function: Use this command to launch your web browser and visit the OrCAD X

web site.

Shortcuts: Keyboard: ALT, H, W, E

Project Manager Command Reference

Web Collaboration command

Available from: Help menu, Web Resources command

Function: Use this command to launch your web browser and visit the Webex

page.

Shortcuts: Keyboard: ALT, H, W, W

OrCAD X Community Site command

Available from: Help menu, Web Resources command

Use this command to launch your web browser and visit the OrCAD X

Function: Community web page. This page provides usable technical solutions for

Windows™ based PCB design.

Shortcuts: Keyboard: ALT, H, W, O

Software Updates command

Available from: Help menu, Web Resources command

Use this command to launch your web browser and visit the Cadence

Downloads web page.

Shortcuts: Keyboard: ALT, H, W, U

PSpice Model Download command

Available from: Help menu, Web Resources command

Use this command to launch your web browser and visit the Downloads

Web page on OrCAD.com site.

Shortcuts: Keyboard: ALT, H, W, M

Project Manager Command Reference

Documentation command

In the project manager or schematic page editor, from the Help menu, Available from:

choose Documentation.

Use this command to launch the HTML page, which contains links to all

the documentation types (manuals and online help), product tutorial, and **Function:**

multimedia demonstrations shipped with this product release.

Keyboard: ALT, H, D Shortcuts:

PSpice Documentation command

In the project manager or schematic page editor, from the Help menu, **Available from:**

choose PSpice Documentation.

Use this command to launch an HTML page in Cadence Help, which

contains links to all the PSpice documentation (manuals and online help)

and product tutorial that are shipped with this product release. Function:

Note: This command is only enabled when you have PSpice simulation

profile opened in your design.

Keyboard: ALT, H, T **Shortcuts:**

Shortcut Menu

This section covers:

Add File command on page 133

Annotate command on page 133

Cleanup Cache command on page 133

Cleanup Cache command on page 133

Copy command on page 133

Cut command on page 133

Delete command on page 133

Project Manager Command Reference

Design Properties command on page 134

Edit Object Properties command on page 134

Edit Page command on page 134

Edit selected object properties command on page 134

Find command on page 135

Lock command on page 135

Make Root command on page 135

New Page command on page 135

New Schematic command on page 135

Open File Location command on page 135

Part Manager command on page 136

Paste command on page 136

Properties command on page 136

Rename command on page 136

Reports – Cross Reference command on page 137

Reports – Export Placement command on page 137

Reports – Export Properties command on page 137

Reports – InterSheet Reference command on page 137

Save command on page 137

Save As command on page 137

Save Project As command on page 137

Schematic Page Properties command on page 137

UnLock command on page 138

Project Manager Command Reference

Add File command

See Project command on page 53

Annotate command

See Annotate command on page 79

Cleanup Cache command

See Cleanup Cache command on page 47

Copy command

See Copy command on page 49

Cut command

See <u>Cut command</u> on page 49

Delete command

Available from: In the project manager, from the shortcut menu

Use this command to delete the selected schematic folders, schematic pages, parts, and symbols that are listed in the project manager window.

Function:

Caution
>
Deleting schematic folders, schematic pages, parts, and

symbols is permanent. You cannot use the Undo command to bring back deleted items from the project

manager.

Project Manager Command Reference

DELETE

Shortcuts BACKSPACE

DEL

Design Properties command

See Design Properties command on page 121

Edit Object Properties command

Available from: In the project manager, from the shortcut menu

Use this command to edit the properties of project objects in the project

manager.

The command is available for at the:

Function:

☐ design level

folder level

page level

Edit Page command

Available from: In the project manager, from the shortcut menu for schematic pages

Function: Use this command to open a page for edit

Edit selected object properties command

Available from: In the project manager, from the shortcut menu for schematic pages

Project Manager Command Reference

Function: Use this command to property editor for selected components

Find command

See Find command on page 64

Lock command

See Lock command on page 51

Make Root command

See Make Root command on page 43

New Page command

See New Schematic Page command on page 39

New Schematic command

See New Schematic command on page 39

Open File Location command

Available from: In the project manager, from the shortcut menu for DSN folder

Function: Use this command to open the location of the design

Project Manager Command Reference

Part Manager command

In the project manager, from the shortcut menu Available from:

Use this command to open the part manager window that summarizes

the status of all the parts in your design and provides a graphical **Function:**

interface for creating bill of materials variants

Paste command

See Paste command on page 51

Properties command

See Properties command on page 54

Remove PSpice Resources command

In the project manager, from the shortcut menu for Design Resources Available from:

or the .DSN file.

Use this command to remove PSpice resources from a project. This command is enabled only for a non PSpice-enabled project—where the Enable PSpice Simulation option is not selected during new project

creation.

If you create a project without enabling PSpice and try to create a new **Function:**

simulation profile, you can see that simulation profile in the *PSpice* Resources – Simulation Profiles folder in the project manager window. Running this command removes the profile and closes the project. On reopening the project, the simulation profile is removed from

this folder.

Rename command

See Rename command on page 41

Project Manager Command Reference

Reports - Cross Reference command

See Cross Reference command on page 85

Reports – Export Placement command

See Export Placement command on page 89

Reports – Export Properties command

See Export Properties command on page 87

Reports - InterSheet Reference command

See Intersheet References command on page 85

Save command

See Save command on page 31

Save As command

See Save As command on page 32

Save Project As command

See Save Project As on page 32

Schematic Page Properties command

Available from: In the project manager, from the shortcut menu for schematic pages

Function: Use this command to open the <u>Schematic Page Properties dialog box</u>

Project Manager Command Reference

UnLock command

See <u>UnLock command</u> on page 53

Schematic page editor and part editor command reference

This chapter covers:

- File menu on page 139
- Edit menu on page 151
- <u>View menu</u> on page 181
- SI Analysis menu on page 196
- Place menu on page 200
- PSpice/Markers menu on page 223
- Accessories menu on page 228
- Options menu on page 228
- Window menu on page 230
- Help menu on page 233
- Shortcut menu on page 236

File menu

This section covers:

New command on page 141

Open command on page 143

Close command on page 146

Save command on page 147

Schematic page editor and part editor command reference

Export Selection command on page 147

Import Selection command on page 147

Print Preview command on page 148

Print command on page 148

Print Setup command on page 149

Print Area command on page 149

Import Design command on page 150

Export Design command on page 150

1,2,3,4 command on page 150

Exit command on page 150

Schematic page editor and part editor command reference

New command

Available from:

File menu

Use this command to create a new project, design, or library. Choose a command from the menu that appears:

Project

Function:

- Design
- Library
- **VHDL File**
- Verilog File

The number of open windows you can have is only limited by your available system resources. You can use the Window menu to switch among open windows (see 1,2.... command).

You can open an existing project, design, library, or VHDL file using the Open command on the File menu.

If you click the Create document toolbar button from a schematic page editor, the New Page in Schematic dialog box appears.

If you click the Create document toolbar button from the part editor, the New Part Properties dialog box appears.

Shortcuts:

Keyboard: ALT, F, N

New Project command

Available from:

File menu, New command

Function:

Use this command to create a new project in Capture.

Keyboard:

Shortcuts:

ALT, F, N, P

New Design command

Available from: File menu, New command

Schematic page editor and part editor command reference

Use this command to create one schematic folder with one schematic **Function:**

page, which Capture opens in the schematic page editor.

Keyboard:

Shortcuts:

ALT, F, N, D

Schematic page editor and part editor command reference

New Library command

Available from: File menu, New command

Use this command to create a new library (containing no parts or

symbols) with a library cache folder.

Shortcuts: Keyboard: ALT, F, N, L

New VHDL File command

Available from: File menu, New command

Use this command to create a new VHDL File, opened in Capture's text

editor.

Shortcuts: Keyboard: ALT, F, N, V

New Verilog File command

Available from: File menu, New command

Use this command to create a new Verilog File, opened in Capture's text

editor.

Shortcuts: Keyboard: ALT, F, N, V

New Text File command

Available from: File menu, New command

Function: Use this command to create a new text file, opened in Capture's text

editor.

Shortcuts: Keyboard: ALT, F, N, T

Open command

Available from: File menu

Schematic page editor and part editor command reference

Use this command to open an existing project, design, library, VHDL, or Verilog file in a new window. Choose a command from the menu that appears:

- Design
- **Function:**
- Library
- Project
- VHDL File
- Verilog File

The number of open windows you can have is only limited by your available system resources. You can use the Window menu to switch among open windows (see 1,2.... command).

Note: When you choose the Open button on the toolbar, a standard Windows Open dialog box appears, in which you can choose the type of file you want to open in the Files of type drop-down list. Among the listed choices are SDT Schematic (*.SCH) and SDT Library (*.LIB).

Capture Toolbar:



Shortcuts:

Keyboard: ALT, F, O

Schematic page editor and part editor command reference

Open Project command

Available from: File menu, Open command

Function: Use this command to open an existing project.

Shortcuts: Keyboard: ALT, F, O, P

Open Design command

Available from: File menu, Open command

Use this command to open one schematic folder with one schematic

page, which Capture opens in the schematic page editor.

Shortcuts: Keyboard: ALT, F, N, D

Open Library command

Available from: File menu, Open command

Use this command to open a library (containing no parts or symbols)

with a library cache folder.

Shortcuts: Keyboard: ALT, F, O, L

Open VHDL File command

Available from: File menu, Open command

Function: Use this command to open a VHDL File, opened in Capture's text editor.

Shortcuts: Keyboard: ALT, F, O, V

Open Verilog File command

Available from: File menu, Open command

Schematic page editor and part editor command reference

Use this command to open a Verilog File, opened in Capture's text

editor.

Shortcuts: Keyboard: ALT, F, O, E

Open Text File command

Function:

Available from: File menu, Open command

Function: Use this command to open an existing text file in a text editor

Shortcuts: Keyboard: ALT, F, O, T

Close command

Available from: File menu

Function: Use this command to close the active window. If necessary, you are

prompted to save your changes.

Shortcuts: Keyboard: ALT, F, C

If you open a part editor via the Part command on the Edit menu, modify the part, and then close it, Capture asks if you want to update the current part only, Il parts of this type in the design, discard your changes, or cancel the Close command.

Schematic page editor and part editor command reference

Save command

Function:

File menu Available from:

Use this command to save the active, modified projects, designs,

libraries, and VHDL files. You can save a design, library, VHDL file, or session log under a different name using the Save As command on the

File menu.

Note: When you save a project, Capture automatically creates a backup with a .DBK file extension. When you save a library, Capture automatically creates a backup with a OBK file extension. If you save only a schematic

page or part, no backup is generated.

Capture Toolbar:

Keyboard:

Shortcuts: CTRL+S

ALT, F, S

Export Selection command

File menu Available from:

Use this command to export the selected objects on a schematic page to

a design or library. You can later import them onto a schematic page

using the Import Selection command on the File menu. **Function:**

This is useful if you have portions of a schematic page that you want to

use on different schematic pages.

Keyboard: ALT, F, E Shortcuts:

Import Selection command

Available from: File menu

Use this command to import the contents of a file created with the **Export Function:**

Selection command on the File menu to the active schematic page.

Keyboard: ALT, F, I **Shortcuts:**

Schematic page editor and part editor command reference

Export ISCF command

Available from: File menu, Export Command

Use this command to export the contents of selected design to Inter

Schematic Export Format (ISCF).

Shortcuts: Keyboard: ALT, F, I

Print Preview command

Available from: File menu

Use this command to see how a schematic page or part will look when

printed.

Function: After setting the options in the Print Preview dialog box, click OK to

preview the printed document. You can use the buttons at the top of the

window to view different pages and to zoom in and out.

Note: Be prepared to wait if you attempt to print multiple pages or parts.

Depending on the number and size of the pages or parts you are previewing, Capture may require extra time to display the selection.

Shortcuts: Keyboard: ALT, F, V

Print command

Available from: File menu

Use this command to print the active schematic page, the active part, or

the selected items in the project manager.

Note: When you print multiple copies, the copies are grouped by page,

not sorted by copy.

Capture Toolbar:

Keyboard: Shortcuts:

■ CTRL+P

■ ALT, F, P

Schematic page editor and part editor command reference

Print Setup command

Available from:

File menu

Function:

Use this command to choose a printer, paper source, and orientation before printing. The Print Setup command displays the <u>Print Setup</u> <u>dialog box</u>, a standard windows dialog box for configuring your printer or plotter. For more information on setting up printers and plotters, refer to the documentation for your configured printer driver.



Many times, the options for your printer are not available in the standard setup dialog box. If you do not find the options you need, try the printer setup in the Windows Control Panel.

Shortcuts:

Keyboard: ALT, F, R

Print Area command

Available from:

File menu

Use this command in the schematic page editor to select and set or clear a specific area to print on your schematic page. Choose a command from the menu that appears:

Function:

■ Set

■ Clear

Shortcuts:

Keyboard: ALT, F, A

Clear command

Available from: File menu

Use this command to clear a print area setting from the active schematic page. The print area setting also clears automatically when you close the

schematic page.

Shortcuts: Keyboard: ALT, F, A, C

Function:

Schematic page editor and part editor command reference

Import Design command

Available from: File menu

Use this command to import EDIF, PDIF and PSpice designs. EDIF

Function: designs must be graphical EDIF designs, and not EDIF netlists. Not all imported PDIF parts may be edited in Capture. Such parts won't affect

netlists.

Export Design command

Available from: File menu

Use this command to export EDIF designs and DXF schematic pages.

This command saves EDIF designs as graphical EDIF designs, and not EDIF notices. DVF schematic pages are saved in AutoCAD's V12 file.

EDIF netlists. DXF schematic pages are saved in AutoCAD's V12 file

format.

1,2,3,4 command

Available

Function:

File menu

from:

Use the numbers listed at the bottom of the File menu to open one of

the last four projects or files. Choose the file you want to open.

Shortcuts: Keyboard: ALT, F, n (n = 1, 2, 3, or 4)

Exit command

Available from: File menu

Use this command to exit the software. If necessary, you are prompted

to save your changes.

Function:
You can also exit the software by choosing the Close command on the

session frame Control menu (ALT, SPACEBAR, C).

Schematic page editor and part editor command reference

Keyboard:

Shortcuts:

- ALT, SPACEBAR, C
- ALT, F, X

ALT+F4

Edit menu

This section covers:

<u>Undo command</u> on page 153

Redo command on page 153

Repeat command on page 155

<u>Label State commands</u> on page 155

Cut command on page 157

Copy command on page 157

Paste command on page 159

Delete command on page 159

Label command on page 161

Select All command on page 161

Part command on page 163

PSpice Model command on page 163

Properties command on page 163

Part command on page 166

Mirror command on page 169

Rotate command on page 171

Lock command on page 171

Schematic page editor and part editor command reference

UnLock command on page 173

Find command on page 173

Global Replace command on page 174

Align Command on page 175

Schematic page editor and part editor command reference

Undo command

Edit menu Available from:

Use this command to reverse the effect of the last operation, if possible.

You can perform the undo and redo operation multiple times to return **Function:**

your design to any particular state, as described in <u>Undoing and</u>

repeating.

The Undo command applies to the following actions:

Placing

Deleting

Moving

Resizing

Rotating

Mirroring

Pasting

Capture Toolbar:



Keyboard:

Shortcuts:

Function:

CTRL+Z

ALT, E, U

Redo command

Edit menu Available from:

Use this command to reverse the effect of the most recent Undo

command. You can undo and redo multiple commands to return your

design to any particular state, as described in <u>Undoing and repeating</u>.

Schematic page editor and part editor command reference

The Redo command applies to the following actions:

- **Placing**
- Deleting
- Moving
- Resizing
- Rotating
- Mirroring

Capture Toolbar:



Keyboard:

Shortcuts:

- CTRL+Y
- ALT, E, E

Schematic page editor and part editor command reference

Repeat command

Available from:

Edit menu

Function:

Use this command to repeat the last operation on the currently selected object, when the last operation can be repeated. The name of the command changes, depending on what the last repeatable operation was—for example, Repeat Rotate or Repeat Paste. This command is most useful for placing objects and creating arrays of objects quickly.

In the schematic page editor, the Redo command applies to the following actions:

- Placing
- Rotating
- Mirroring
- Moving
- Resizing
- Pasting

In the part editor, the Redo command applies to the following actions:

- Placing
- Rotating
- Mirroring
- Pasting

Keyboard:

Shortcuts:

- ∎ F4
- ALT, E, R

Label State commands

Set

Available from: Edit menu

Schematic page editor and part editor command reference

Use this command to specify a label for the current state of the active **Function:**

schematic page.

Schematic page editor and part editor command reference

Go To

Available from: Edit menu

Function: Use this command to specify the label of the schematic state to which

you want to return.

Delete

Available from: Edit menu

Function: Use this command to specify a label state to delete.

Cut command

Available from: Edit menu

Use this command to remove the selected object from the active window and put it on the Clipboard. This command is only available when an

object is selected.

Function: Cutting objects to the Clipboard replaces any objects previously stored

there. Use the Paste command to copy objects to another page or part,

or to another Windows application that supports pasting from the

Clipboard.

Capture Toolbar:

✂

Keyboard:

Shortcuts: ■ CTRL+X

■ ALT, E, T

Shortcut menu: Cut

Copy command

Available from: Edit menu

Schematic page editor and part editor command reference

Use this command to copy a selected object to the Clipboard without removing it from the active window. This command is available only if an object is selected.

Function:

Copying objects to the Clipboard replaces any objects previously stored there. Use the Paste command to copy objects to another page or part, or to another Windows application that supports pasting from the Clipboard.

Capture Toolbar:



Keyboard:

Shortcuts:

CTRL+C

■ ALT, E, C

shortcut menu: Copy

Schematic page editor and part editor command reference

Paste command

Available from:

Edit menu

Use this command to place any objects stored on the Clipboard into the active window. This command is unavailable if the Clipboard is empty.

Function:

Pasting objects from the Clipboard does not affect the Clipboard's contents. Use Paste to copy objects to another page or part, or to another Windows application that supports pasting from the Clipboard. You can only paste text into text boxes.

Note: If you copy a part into the Clipboard and then paste it onto a schematic page, Capture will automatically assign a unique reference designator to the pasted part when two conditions are met:

- 1. The Auto Reference option on the Miscellaneous tab of the Preferences dialog box is selected.
- 2. The pasted part has a reference designator assigned to it when it is copied to the Clipboard.

Capture assigns the reference designator, updated to the next available value (one greater than the highest value used on the schematic at that point.) If the pasted part has a default reference (for example, R?) Capture does not assign a unique reference designator to it.



In part editor, you can copy objects and paste them in the part editor only. Similarly, in symbol editor, you can copy objects and paste them in the symbol editor only.

Capture Toolbar:



Keyboard:

Shortcuts:

CTRL+V

ALT, E, P

Shortcut menu: Paste

Delete command

Available from: Edit menu

Schematic page editor and part editor command reference

Use this command to remove the selected object from the active window without putting it on the Clipboard. This command is available only if an object is selected.

Deleting objects does not affect the Clipboard's contents.

Keyboard:

- DEL
- DELETE
- **Shortcuts:**

Function:

- BACKSPACE
- ALT, E, D

Shortcut menu: Delete

Schematic page editor and part editor command reference

Label command

Available from:

Edit menu

Use this command to tag the schematic designs at different stages of development. This command is useful when you want to quickly jump to a particular state of schematic design. For example, consider the situation while developing a schematic:

- 1. Place a 7400 part.
- 2. Add the ATOD library to the design.
- 3. Place the 5962-8700 part on the schematic
- **4.** Join the components.

Function:

- **5.** Add the Discrete library to the schematic.
- **6.** Place the 126ANS part on the schematic.

At this point you realize that instead of 5962-8700 part, the 5962-87786 part should have been used. Now, to do so, you can either undo the last three steps. Alternately, you could have used the label state option to label the schematic at different stages of design. For example, you could have set a label, say stage2, after step 2. In such a scenario, you could have used the label and jumped straight to the state of the schematic as it was after step 2, thus saving yourself a lot of effort.

Note: A label can have a maximum of 31 characters.

Keyboard:

Set: Ctrl + Shift + F2

Shortcuts:

■ Goto: Ctrl + Shift + F5

Delete: Ctrl + Shift+ F7

Select All command

Available from:

Edit menu

Function:

Use this command to select all items in the active window.

Schematic page editor and part editor command reference

Keyboard:

Shortcuts: ■ CTRL+A

■ ALT, E, L

Schematic page editor and part editor command reference

Part command

Available from: Edit menu, Browse command

Use this command to list the parts of the selected schematic pages in the browse window. The Browse Properties dialog box gives you the

option to use instances or occurrences.

You can sort the parts in the browser spreadsheet by clicking the button at the top of each column in the browse window. Capture only displays

the occurrences of the parts in the browse window.

You can select single or multiple parts and edit them with the Properties command. When you edit one part, a part editor window appears. When

you select multiple parts, the Browse spreadsheet editor appears.

Shortcuts: Keyboard: ALT, B, P

PSpice Model command

Available from: Edit menu, PSpice Model command

Function: Use this command to open and edit a PSpice model in Model Editor.

Properties command

Available from: Edit menu

In the project manager, use this command to view properties about the selected document. Using the <u>Properties dialog box</u>, you can access information general, type, and project about the file that is currently selected in the project manager window. You can also change the file's type. A file or project must be selected in the project manager window to access the

Properties command.

In the schematic page editor, use this command to open the property editor, where you can edit properties and other data for the selected objects.

The properties you can edit depend on the selected objects. The following lists the inherent properties you can edit and the dialog boxes in which you

edit them:

Schematic page editor and part editor command reference

Objects	Dialog box	
■ Arcs	■ Edit Graphic dialog box	
■ Images (pictures)	■ Not applicable	
■ Bookmarks	■ Edit Bookmark dialog box	
■ Buses Property editor	■ Property editor	
■ Bus entries	■ User Properties dialog box	
■ DRC markers	■ View DRC Marker dialog box	
■ Ellipses	■ Edit Filled Graphic dialog box	
■ Hierarchical blocks	■ Property editor	
■ Hierarchical pins	■ Property editor	
■ Hierarchical ports	■ Property editor	
■ IEEE symbols	■ Place IEEE Symbol dialog box	
■ Junctions	■ Not applicable	
■ Lines	■ Edit Graphic dialog box	
■ Multiple objects	■ Property editor or Browse spreadsheet	
■ Nets (wires and buses)	editor	
Net aliases	Property editor	
No connects	Property editor	
■ Off-page connectors	■ Not applicable	

Parts

Pictures (images)

Edit Off-Page Connector dialog box

Property editor

Not applicable

Schematic page editor and part editor command reference

- Part body borders
- Pins (part editor)
- Pins (schematic page editor)
- Polygons
- Polylines
- Power, ground
- Rectangles
- Text
- Title blocks
- Wires

- Not applicable
- Pin Properties dialog box (part editor)
- Property editor
- Edit Filled Graphic dialog box
- Edit Graphic dialog box
- Property editor
- Edit Filled Graphic dialog box
- Place Text dialog box
- Property editor
- Property editor

Note: You can edit homogeneous sets of the following objects in the spreadsheet editor:

- Bookmarks
- DRC markers
- Hierarchical ports
- Nets
- Off-page connectors
- Parts
- Pins

Keyboard:

- CTRL+E
- **Shortcuts:**
- ALT, E, I

Mouse: Double-click on a part Shortcut menu: Edit Properties

Schematic page editor and part editor command reference

Part command

Function:

Available from: Edit menu

Use this command to open the selected part in a part editor window.

The part command edits the part in the design cache. After saving the part, you have the option to apply your changes to just one part or all parts with the same part value in the design. If you edit the one part only,

a new part is created in the cache and all other parts with the same part value are left unchanged. Otherwise, the changes are applied to the part in the cache. To replace a part in the cache with another part, use the

Replace Cache command.

Keyboard: ALT, E, A

Shortcuts: Shortcut menu: Edit Part

Schematic page editor and part editor command reference

Reset Location command

Available from: Edit menu

Select the pin name and pin number text you had moved in the part editor and use this command to reset the pin name or pin number text movement.

Choose a command from the menu that appears:

Function: To reset a pin name movement, choose Pin Name

■ To reset a pin number movement, choose Pin Number

This command also resets the display properties such as *Bold*, *Italic*, *Color*, *Rotation*, *Justification*, *Font*, and *Font Size* related to Pin

Name and Pin Number.

Shortcut Menu: Reset Location

Pin Name command

Available from: Edit menu, Reset Location command

Select the pin name text you had moved in the part editor and use this

command to reset the pin name movement.

Function: This command also resets the display properties such as *Bold*, *Italic*,

Color, Rotation, Justification, Font, and Font Size related to Pin

Name.

Shortcut Shortcut menu: Reset Location

Pin Number command

Available from: Edit menu, Reset Location command

Select the pin number text you had moved in the part editor and use this

command to reset the pin number movement.

Function: This command also resets the display properties such as *Bold*, *Italic*,

Color, Rotation, Justification, Font, and Font Size related to Pin

Number.

Schematic page editor and part editor command reference

Shortcut menu: Reset Location

Schematic page editor and part editor command reference

Mirror command

Available from: Edit menu

Use this command to mirror selected items in the schematic page editor or the part editor. Choose a command from the menu that appears:

Function: Horizontally

Vertically

■ Both

Note: Multiple selected objects are mirrored and rotated as a group. They

do not mirror or rotate around their individual axes.

Note: Title blocks and text cannot be mirrored or rotated.

Shortcuts: Keyboard: ALT, E, M

Horizontally command

Available from: Edit menu, Mirror command

Function: Use this command to mirror selected objects from side to side (across

the Y axis).

Note: Multiple selected objects are mirrored and rotated as a group. They

do not mirror or rotate around their individual axes.

Title blocks and text cannot be mirrored or rotated.

Keyboard:

■ H

Shortcuts: ■ ALT, E, M, H

Shortcut menu: Mirror Horizontally

Vertically command

Available from: Edit menu, Mirror command

Schematic page editor and part editor command reference

Function:

Use this command to mirror selected objects from top to bottom and from bottom to top (across the X axis).

Note: Multiple selected objects are mirrored and rotated as a group. They do not mirror or rotate around their individual axes.

Note: Title blocks and text cannot be mirrored or rotated.

Keyboard:

Shortcuts:

V

■ ALT, E, M, V

Shortcut menu: Mirror Vertically

Schematic page editor and part editor command reference

Both command

Available from: Edit menu, Mirror command

Function: Use this command to mirror selected objects both horizontally and vertically. This is equivalent to rotating the objects by 180 degrees.

Note: Multiple selected objects are mirrored and rotated as a group. They

do not mirror or rotate around their individual axes.

Note: Title blocks and text cannot be mirrored or rotated.

Shortcuts: Keyboard: ALT, E, M, B

Rotate command

Available from: Edit menu

Use this command to rotate selected objects counterclockwise in

Function: 90-degree increments. Selected objects rotate as a set, not as individual

objects rotating in place.

Note: Multiple selected objects are mirrored and rotated as a group. They

do not mirror or rotate around their individual axes.

Note: Title blocks and Non-TrueType fonts cannot be mirrored or rotated.

Keyboard:

■ R

Shortcuts: ■ ALT, E, O

Shortcut menu: Rotate

Lock command

Schematic page editor shortcut menu, when you select the schematic

Available from: part.

Note: This command is disabled if the object is currently locked.

Schematic page editor and part editor command reference

Use this command to lock an object on a schematic page.

Function: Note: You can lock one or more objects on a schematic page or you can

lock all the objects in design, folder or page.

Shortcuts: Shortcut menu: Lock

Schematic page editor and part editor command reference

UnLock command

part.

Schematic page editor shortcut menu, when you select the schematic

Available from:

Note: This command is disabled if the object is currently unlocked.

Use this command to unlock an object on a schematic page.

Function: Note: You can unlock one or more objects on a schematic page or you

can unlock all the objects in design, folder or page.

Keyboard: ALT, E, N

Shortcuts:

Shortcut menu: UnLock

Find command

Available from: Edit menu

Use this command to locate an object or string of text in the active window.

In the schematic page editor and project manager, the Find command finds all instances of the specified text search string, parts, part pins, DRC markers and constraints. The Find command supports wildcard searches. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.

Function:

In the session log, the Find command will find the next occurrence of the specified text search string from the current position. However, it does not support wildcard searches.

The Find window in part editor can search only pin and text, and it supports wildcard searches.

In the design variant schematic page, the Find command will find parts that are not present or parts with different property values attached to it.

Keyboard:

Shortcuts: ■ CTRL+F

■ ALT+E+F

Schematic page editor and part editor command reference

Global Replace command

Available from: Edit menu

Use this command to locate and replace an object or a string of text in

the schematic editor.

Function: The Global Replace command does not support wildcard searches.

In the schematic page editor, the Global Replace command will find all

instances of the specified text search string.

Keyboard:

Shortcuts:

Function:

ALT+E+B

Check Verilog syntax command

Available from: Edit menu

Use this command to check the syntax of Verilog files for errors.

When you choose Check Verilog Syntax from the Edit menu (or Check Syntax from the popup menu), the Check Syntax tool finds the first error in the file and highlights it so that you can fix it. Once you fix the error, you can continue to check the file by choosing Check Syntax from the

Edit menu again.

If the project manager window is active, you can check all selected Verilog files. Or, you can check an open, active Verilog file.

Note: If you check an open file that has been modified but not saved, note that you are checking the version of the file that is open, not the version of the file saved to disk.

Note: You must have a project open to use the Check Syntax command. Error reporting for the tool requires some project resources that are not available unless a project is open.

Shortcuts: Shortcut menu: Check Syntax

Check VHDL syntax command

Available from: Edit menu

Schematic page editor and part editor command reference

Use this command to check the syntax of VHDL files for errors.

Function:

When you choose Check VHDL Syntax from the Edit menu (or Check Syntax from the popup menu), the Check Syntax tool finds the first error in the file and highlights it so that you can fix it. Once you fix the error, you can continue to check the file by choosing Check Syntax from the Edit menu again.

If the project manager window is active, you can check all selected VHDL files. Or, you can check an open, active VHDL file.

Note: If you check an open file that has been modified but not saved, note that you are checking the version of the file that is open, not the version of the file saved to disk.

Note: You must have a project open to use the Check Syntax command. Error reporting for the tool requires some project resources that are not available unless a project is open.

Shortcuts:

Shortcut menu: Check Syntax

Samples command

Available from: Edit menu

Use this command to display VHDL file samples in the VHDL Samples dialog box or Verilog file samples in the Verilog Samples dialog box.

Function:

Note: The VHDL samples dialog box appears if you open a VHDL file in Capture and then use this command. The Verilog Samples dialog box appears if you open a Verilog file in Capture and then use this command.

When you select a sample in the upper box, the associated sample lines appear in the lower box. Double-click on the sample type in the upper box, or select it and click OK to copy the sample into the text editor.

Shortcuts:

Keyboard: ALT, E, A

Align Command



As you align or distribute objects, remember the following points:

Schematic page editor and part editor command reference

	If the text is justified, align command has no impact on the justified text.	
□ Align command is not supported for the following objects:		gn command is not supported for the following objects:
	О	PSpice Markers
	О	DRC and ERC Markers
	О	Bookmarks
	О	Hierarchical Pins
	O	Part Pins
	O	Biaspoints
	О	Bus
	О	Bus Entry
	О	Wire
	О	NetGroup
	О	Locked Objects
	You need to select at least two objects to perform alignment or distribution.	
	You can select different object types, such as parts, net aliases, and property texts, at same time to perform alignment or distribution.	
	The following objects may connect during alignment or distribution, if the Drag Connected Object option is OFF:	
	О	Global
	О	OPC
	О	Ports
	О	Power/GND
	Objects that have their left side on a grid may not align properly. Set the Snap To Grid option OFF to align these objects properly.	
	In Vertical Distribution, the objects are always vertically distributed in top-to-bottom direction. Similarly, in Horizontal Distribution, the objects are always horizontally distributed in left-to-right direction.	
	By default, the <i>Distribute in a fixed area</i> check box in the Schematic Group section of the Extended Preferences Setup window is unchecked. To open this	

Schematic page editor and part editor command reference

window, select *Options – Extended Preferences*. If the check box is unchecked, the objects in the virtual selection bounding box are evenly distributed and the bounding box might be compressed or expanded. If you check the box, the objects are distributed in a fixed area, but the distribution might be uneven.



When two parallel parts have one of the their pins on same grid and are connected through a net having net alias, any alignment task performed on these two parts may break the connectivity between them.

Schematic page editor and part editor command reference

Align Left

Available from: Edit menu

Use this command to left align the selected objects with respect to the

leftmost selected object.

Function: In Mouse Mode:

Use this command to left align the selected objects with respect to the

mouse click.

Align Center

Available from: Edit menu

Use this command to center align the selected objects with respect to the virtual selection bounding box. The virtual selection bounding box is

formed with respect to the leftmost and rightmost of the selected objects.

Function:

In Mouse Mode:

Use this command to center align the selected objects with respect to

the mouse click.

Align Right

Available from: Edit menu

Use this command to right align the selected objects with respect to the

rightmost selected object.

Function: In Mouse Mode:

Use this command to right align the selected objects with respect to the

mouse click.

Schematic page editor and part editor command reference

Align Top

Available from: Edit menu

Use this command to top align the selected objects with respect to the topmost selected object.

topinost selected object

Function: In Mouse Mode:

Use this command to top align the selected objects with respect to the

mouse click.

Align Middle

Available from: Edit menu

Use this command to middle align selected objects with respect to the virtual selection bounding box. The virtual selection bounding box is formed with respect to the topmost object and bottommost object of the

selected objects.

Function:

In Mouse Mode:

Use this command to middle align the selected objects with respect to

the mouse click.

Align Bottom

Available from: Edit menu

Use this command to bottom align the selected objects with respect to

the bottommost selected object.

Function: In Mouse Mode:

Use this command to bottom align the selected objects with respect to

the mouse click.

Schematic page editor and part editor command reference

Distribute Horizontal

Available from: Edit menu

Use this command to horizontally distribute selected objects with equal spacing within the virtual selection bounding box. The virtual selection bounding box is formed with respect to the leftmost object and the rightmost object of the selected objects.

Function:

In Mouse Mode:

Use this command to horizontally distribute the selected objects with equal spacing between the two selected points on a schematic page.

Distribute Vertical

Available from: Edit menu

Use this command to vertically distribute selected capture objects with equal spacing within the virtual selection bounding box. The virtual selection bounding box is formed with respect to the topmost object and the bottommost object of the selected objects.

Function:

In Mouse Mode:

Use this command to vertically distribute the selected objects with equal spacing between two selected points on a schematic page.

Mouse Mode

Available from: Edit menu

Schematic page editor and part editor command reference

Use this command to enable or disable Mouse Mode. When Mouse Mode is enabled, the point where mouse is clicked on schematic page is taken as reference for alignment or distribution task.

Alignment using Mouse Mode:

- 1. Choose Edit Align Mouse Mode to set Mouse Mode ON.
- **2.** Select the objects you want to align.
- Select the alignment type (top, middle, bottom, left, center, right). As you select the alignment or distribution type, the shape of mouse pointer changes.
- **4.** Click on the schematic page to align selected objects with respect to mouse click.

Function:

Distribution using Mouse Mode:

- 1. Choose Edit Align Mouse Mode to set Mouse Mode ON.
- 2. Select the objects you want to distribute.
- **3.** Select the distribution type horizontal or vertical. As you select the distribution type, the shape of mouse pointer changes.
- **4.** Select the two points on the schematic between which the distribution is to performed.
 - a. Click to set the first point.
 - **b.** Click to set the second point.

As you set the second point on a schematic page, the distribution on selected objects is performed between the two selected points.

View menu

This section covers:

Ascend Hierarchy command on page 183

Convert command on page 183

Schematic page editor and part editor command reference

<u>Descend Hierarchy command</u> on page 183

Go To command on page 185

Synchronize Up command on page 185

Synchronize Down command on page 185

Synchronize Across command on page 186

Previous Part command on page 186

Next Part command on page 186

Package command on page 187

Zoom command on page 188

Fisheye command on page 191

<u>Tool Palette command</u> on page 192

Toolbar command on page 193

Status Bar command on page 193

Command Window command on page 193

Grid command on page 193

Grid References command on page 195

Selection Filter command on page 195

Invoke UI command on page 195

Schematic page editor and part editor command reference

Ascend Hierarchy command

Available from: View menu

Use this command to view the parent of the active schematic page.

If the parent schematic page is open in another window, that window becomes active; otherwise, it opens in a new schematic page editor

window.

You can view and traverse the hierarchy in the project manager.

Keyboard:

SHIFT+A

Shortcuts:

Function:

ALT, V, A

Shortcut: Click right mouse button and choose Ascend Hierarchy

Convert command

Available from: Convert tab in the part editor window

Function:

To display the convert view in the part editor window, select the *Convert* tab next to Normal tab in the bottom left of the canvas. This tab is available only when the parts in the package have a convert view.

Descend Hierarchy command

Available from: View menu

Use this command to view the schematic page. This command is available only when the selected part or hierarchical block has an attached schematic folder or file. If the attached schematic folder has not yet been created, this command creates a new page. If the child schematic folder is open in another window, that window becomes

Function:

active. Otherwise, it opens in a new schematic page editor window.

You can view and traverse the hierarchy in the project manager.

Schematic page editor and part editor command reference



Once you have attached a file and associated a text editor with it, you can use the Descend Hierarchy command to open that file. If you have an attached schematic folder as well as an attached file, Descend Hierarchy opens the schematic folder and not the file.

Keyboard:

■ SHIFT+D

Shortcuts:

■ ALT, V, D

Shortcut menu: Descend Hierarchy

Schematic page editor and part editor command reference

Go To command

Available from: View menu

Function: Use this command to center the view on a specific location, grid

reference, or bookmark.

Tip

The *Go To* command is always available on the right mouse button context-sensitive menus in the part editor and schematic page editor. The *Go To* command, with the Relative option selected, is particularly useful for precise placement and spacing.

Keyboard:

■ CTRL+G

Shortcuts: ■ ALT, V, G

Shortcut menu: Go To

Synchronize Up command

Available from: View menu / Shortcut menu

Updates the hierarchical blocks with all the changes made to the

Function: hierarchical ports of the underlying schematic of the selected

hierarchical block.

Keyboard: Shift + U

Shortcuts: Shortcut menu: Synchronize Up

Synchronize Down command

Available from: View menu / Shortcut menu

Updates the underlying schematic with all the changes made to the biographical pine of the selected biographical block

hierarchical pins of the selected hierarchical block.

Keyboard: Shift + O

Shortcuts: Shortcut menu: Synchronize Down

Schematic page editor and part editor command reference

Synchronize Across command

Available from: View menu / Shortcut menu

Updates all the instances of hierarchical block at the same level of

hierarchy with the changes on the hierarchical pin of the selected

hierarchical block.

Keyboard: Shift + C

Shortcuts:

Shortcut menu: Synchronize Across

Previous Part command

Available from: View menu

Use this command to view the previous part in the package. In Part view, this command displays the previous part of the package in the part

Function: editor. In Package view, this command selects the previous part in the

package.

Keyboard:

■ ALT, V, V

Shortcuts:

Function:

■ CTRL+B

■ SHIFT+TAB (PACKAGE VIEW ONLY)

Next Part command

Available from: View menu

Use this command to view the next part in the package. In Part view, this command displays the next part of the package in the part editor. In

Package view, this command selects the next part in the package.

Keyboard:

■ ALT, V, X

Shortcuts: ■ CTRL+N

■ TAB (PACKAGE VIEW ONLY)

Schematic page editor and part editor command reference

Previous Page command

Available from: View menu

Function: Use this command to open the previous page in the current schematic

folder. The option is disabled for the first page in the folder.

Shortcuts: Keyboard: Shift+F10

Next Page command

Available from: View menu

Use this command to open the next page in the current schematic folder.

The antiquid disabled for the least reason in the folder.

The option is disabled for the last page in the folder.

Shortcuts: Keyboard: F10

Package command

Available from: View menu

Use this command in the part editor to view all the parts in a package in

Function: a new tab. Parts cannot be edited in package view.

You can zoom in and zoom out the parts using the shortcuts I and O.

Shortcuts: Keyboard: ALT, V, K

Schematic page editor and part editor command reference

Zoom command

Available from: View menu

Use this command to change your view of the schematic page. Choose one of the commands listed:

- In
- Out

Function:

- Scale
- Area
- ΑII
- Selection
- Redraw

Shortcuts:

Keyboard: ALT, V, Z

In command

Available from: View menu, Zoom command

Use this command to zoom in on the schematic page or part. The zoom scale is multiplied by the current zoom factor.

Capture uses the following order to determine where the view of the zoom centers:

Function:

- On the pointer location
- On the selected item or items
- In the center of the window (not the center of the schematic page or part)

Capture Toolbar:



Keyboard:

Shortcuts:

- ALT, V, Z, I

Shortcut menu: Zoom In

Schematic page editor and part editor command reference

Out command

View menu, Zoom command Available from:

Use this command to zoom out from the schematic page or part. The **Function:**

zoom scale is divided by the current zoom factor.

Capture Toolbar:

Keyboard:

0 Shortcuts:

ALT, V, Z, O

Shortcut menu: Zoom Out

Scale command

Available from: View menu, Zoom command

In the schematic page editor, use this command to zoom to a preset or **Function:**

user-defined scale. The new view centers on the selected objects, the

pointer location, or the center of the previous view.

Keyboard: ALT, V, Z, S **Shortcuts:**

Area command

Available from: View menu, Zoom command

Use this command to make a specific area of the document as large as

will fit in the window. You define the area by dragging a rectangle around **Function:**

it.

Capture Toolbar:

Keyboard: ALT, V, Z, A

All command

Shortcuts:

Available from: View menu, Zoom command

Schematic page editor and part editor command reference

Use this command to view the entire document in the active window.

This command uses the size of the work area, not the limits of the **Function:**

objects.

Capture Toolbar:

Shortcuts: Keyboard: ALT, V, Z, L

Schematic page editor and part editor command reference

Selection command

Available from: View menu, Zoom command

Function:

In the schematic page editor, use this command to view all selected

objects.

Shortcuts: Keyboard: ALT, V, Z, E

Redraw command

Available from: View menu, Zoom command

Function: In the schematic page editor, use this command to refresh the display.

Keyboard:

Shortcuts: ■ F5

■ ALT, V, Z, R

Fisheye command

Available from: View menu

Use this command to work in a page in a non-linear mode. Choose one of the following commands listed:

Fisheye view

Function: Fisheye Dynamic Focus Mode

Set Fisheye Focus

Reset Fisheye Focus

Fisheye view command

Available from: View menu, Fisheye command

Use this command to set the Fisheye mode that allows yo to zoom into

only specific objects on your schematic.

Schematic page editor and part editor command reference

Fisheye Dynamic Focus Mode command

Available from: View menu, Fisheye command

Use this command to shift the focus of the page as you move the mouse pointer across the page. As the mouse pointer hovers over a part of the

Function: page, only that part of the page comes into focus. The focus area is

magnified while the rest of the viewable area loses relative

magnification.

Shortcuts: Q

Set Fisheye Focus command

Available from: View menu, Fisheye command

Use this command to set the Fisheye focus to selected objects on your

Function: schematic, causing only these objects to zoom while the rest of the

viewable area remain in view but is zoomed out.

Shortcuts: Shift+F11

Reset Fisheye Focus command

Available from: View menu, Fisheye command

Function: Use this command to remove Fisheye focus.

Shortcuts: Shift+Ctrl+F11

Tool Palette command

Available from: View menu

Use this command to show or hide the tool palette. This setting is stored

Function: in your CAPTURE.INI file and thus affects the visibility of the palette in

subsequent sessions.

Shortcuts: Keyboard: ALT, V, P

Schematic page editor and part editor command reference

Toolbar command

Function:

Available from: View menu

Use this command to show or hide the toolbar. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbar anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats on the screen wherever

it is released.

If the toolbar is floating, you can hide it by choosing Toolbar from the

View menu.

Shortcuts: Keyboard: ALT, V, T

Status Bar command

Available from: View menu

Use this command to show or hide the status bar. This setting is stored

Function: in your .INI file and thus affects the visibility of the status bar in

subsequent sessions.

Shortcuts: Keyboard: ALT, V, S

Command Window command

Available from: View - Toolbar menu

Use this command to open the TCL Command window in the OrCAD X

Function: Capture workspace.

For details see the Command Window.

Grid command

Available from: View menu

Schematic page editor and part editor command reference

Use this command to show or hide the grid dots. You can show or hide

the grid independently in each schematic page and part you have open.

You can also set the grid dots to show or hide in the Preferences dialog

box.

Shortcuts: Keyboard: ALT, V, I

Function:

Schematic page editor and part editor command reference

Grid References command

Available from: View menu

Use this command to show or hide the grid references.

Function: You can also set the grid references to show or hide in the Design

Template dialog box and the Schematic Page Properties dialog box.

Keyboard: ALT, V, R **Shortcuts:**

Selection Filter command

View menu

Available from:

Note: You can access this command only if a schematic page is open.

Function:

Use this command to control the selection of objects in a schematic

page during a block-select operation.

Keyboard:

CTRL+I

Shortcuts:

ALT, V, F

This command is also available from the schematic page editor Shortcut menu.

Invoke UI command

Available from: View menu

Opens the Propagation Delay dialog box or the Relative Propagation Delay dialog box while editing these properties in the Property Editor

window.

This command is available only when you have selected the grid corresponding to the Propagation Delay/Relative Propagation Delay

property in the Property Editor window.

You can also access the Invoke UI command by right-clicking the grid corresponding to the Propagation Delay/Relative Propagation Delay

property in the Property Editor window.

Function:

Schematic page editor and part editor command reference

Keyboard:

Shortcuts: ■ CTRL+U

■ ALT, E, V

Tools menu

Assign Power Pins command on page 196

Assign Power Pins command

Available from: Tools menu

Use this command to specify the invisible powers on the selected object function:

(design, schematic folder, schematic page or schematic part) as NC Pins.

Shortcuts: Keyboard: ALT, T, W

SI Analysis menu

Assign SI Model command on page 197

Explore Signal command on page 197

Export Topology command on page 197

Associate Electrical Cset command on page 199

Validate SI Model Assignments command on page 199

Remove SI Model Assignments command on page 199

Schematic page editor and part editor command reference

Assign SI Model command

SI Analysis menu

Available from: Note: You can access this command only if a model is selected in the

schematic page.

Use this command to open the SI Model Assignment window where you

Function: can assign models from the listed libraries. You can also auto-generate

models.

Keyboard:

Shortcuts:

This command is also available from the schematic page editor Signal

Integrity Shortcut menu.

Explore Signal command

SI Analysis menu

Available from: Note: You can access this command only if a flat net is selected in the

schematic page.

Function: Use this command to open SigXplorer to explore the signal.

Keyboard:

Shortcuts:

This command is also available from the schematic page editor Signal

Integrity Shortcut menu.

Export Topology command

SI Analysis menu

Available from: Note: You can access this command only if a flat net is selected in the

schematic page.

Function: Use this command to export the associated topology files.

Schematic page editor and part editor command reference

Keyboard:

Shortcuts:

■ ALT, N, T

This command is also available from the schematic page editor Signal Integrity shortcut menu.

Schematic page editor and part editor command reference

Associate Electrical Cset command

SI Analysis menu

Available from: Note: You can access this command only if a flat net is selected in the

schematic page.

Use this command to associate an Electrical Constraint set using a **Function:**

topology file.

Keyboard:

ALT, N, C Shortcuts:

This command is also available from the schematic page editor Signal

Integrity shortcut menu.

Validate SI Model Assignments command

SI Analysis menu

Available from: Note: You can access this command only if a part with SI model

assignment is selected in the schematic page.

Use this command to remove SI models assigned to a part. **Function:**

Keyboard:

ALT, N, D Shortcuts:

This command is also available from the schematic page editor Signal

Integrity shortcut menu.

Remove SI Model Assignments command

SI Analysis menu

Available from: Note: You can access this command only if a part with SI model

assignment is selected in the schematic page.

Use this command to validate SI models assigned to a part. **Function:**

Schematic page editor and part editor command reference

Keyboard:

Shortcuts:

■ ALT, N, G

This command is also available from the schematic page editor Signal Integrity shortcut menu.

Place menu

This section covers:

Part command on page 202

PSpice Part command on page 203

Parameterized Part command on page 206

NetGroup command on page 206

Wire command on page 206

Bus command on page 208

Junction command on page 210

Bus Entry command on page 210

Net Alias command on page 210

Power command on page 211

Ground command on page 211

Off-Page Connector command on page 211

Hierarchical Block command on page 213

<u>Hierarchical Port command</u> on page 213

Hierarchical Pin command on page 214

No Connect command on page 214

Title Block command on page 215

Schematic page editor and part editor command reference

Bookmark command on page 218

Text command on page 218

Line command on page 219

Rectangle command on page 219

Ellipse command on page 219

Arc command on page 221

Elliptical Arc command on page 221

Bezier curve command on page 221

Polyline command on page 222

Picture command on page 222

OleObject command on page 222

Schematic page editor and part editor command reference

Part command

Place menu Available from:

Use this command to place a part you select in the Place Part dialog

Function: In the Place Part dialog box, you choose a part by selecting libraries to

> view. You may view parts from both Capture and SDT libraries. If you choose a part from an SDT library, Capture translates the library for you.

Draw Toolbar:

Keyboard:

Shortcuts:

Р

ALT, P, P

Component command

Available from: Place menu

Use this command to open the *Component Explorer* interface from where you can place a component on the schematic from various sources including Cadence-supplied libraries and content providers,

such as SamacSys, SnapEDA, and Ultra Librarian. **Function:**

In the Component Explorer interface, you choose a component from the

PSpice libraries or content providers' database by selecting the

corresponding node from the Categories tree.

Schematic page editor and part editor command reference

PSpice Part command

Available from: Place menu

Schematic page editor and part editor command reference

Use this command to place a part with ideal SPICE model from the generic PSpice library on your schematic. The submenu items and the parts you can access are listed below:

- PSpice Ground
- Capacitor
- Diode
- Inductor
- Resistor
- Digital:
 - Gates: AND, OR, NAND, NOR, XOR, INV
 - □ Flip Flop: D, JK, RS, T
 - □ ADC: 8Bit, 10Bit, 12Bit
 - □ DAC: 8Bit, 10Bit, 12Bit
 - □ Memory:
 - O RAM: 8kx1, 8Kx8
 - ROM: 32Kx1
- Discrete: Diode, NPN, PNP, NPN Darlington, PNP Darlington
- Passive: R, C, L, Potentiometer, Coupling, Tline Ideal, Tline Lossy
- Source:
 - Controlled sources: VCVS, VCCS, CCVS, CCCS
 - □ Current Sources: AC, DC, Pulse, Sine, Exponential, FM Sine
 - □ Voltage Sources: AC, DC, Pulse, sine, Exponential, FM Sine
- Search
- Modeling Application:
 - Circuit Protection: TVS
 - Diodes: Zener, LED
 - Passives: Capacitor, Inductor

Function:

Schematic page editor and part editor command reference

- □ Sources: Independent Sources, PWL Sources
- □ System Modules: Switch, Transformer, VCO

Shortcuts:

Keyboard:

■ ALT, P, S

Schematic page editor and part editor command reference

Parameterized Part command

Available from: Place menu

Function: Use this command to place a parameterized part you select in the Place

Part dialog box.

Shortcuts: Keyboard: ALT, P, D

NetGroup command

Available from: Place menu

Use this command to place a NetGroup on a schematic page.

Function: Note: You also use this dialog to add or modify associated NetGroup

definitions.

Shortcuts: Keyboard: U

Wire command

Available from: Place menu

Use this command to place a part you select in the Place Part dialog

box.

Function: In the Place Part dialog box, you choose a part by selecting libraries to

view. You may view parts from both Capture and SDT libraries. If you choose a part from an SDT library, Capture translates the library for you.

Draw Toolbar:

Ţ

Keyboard:

Shortcuts:

W

■ ALT, P, W

Auto Wire Two Points command

Available from: Place menu

Schematic page editor and part editor command reference

Use this command to auto-wire two points (pins or wires) on a schematic

page.

Shortcuts: Draw Toolbar: 5

Auto Wire Multiple Points command

Available from: Place menu

Function:

Function: Use this command to auto-wire multiple points (pins or wires) on a

schematic page.

Shortcuts: Draw Toolbar:

Auto Wire Connect to Bus

f

Available from: Place menu

Function: Use this command to auto-wire any number of points on a schematic

page to a bus on the page.

Shortcuts: Draw Toolbar:

Schematic page editor and part editor command reference

Bus command

Available from:

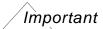
Place menu

Use this command to place a bus. When placing a bus, you click the left mouse button to start the bus. Click the left mouse button to change the bus's direction or create a junction with another bus. Double-click the left mouse button or press ESC to end the bus, and place another bus. Press ESC again to exit the bus tool.

Function:

When placing buses, you are constrained to 90-degree angles. If you want to draw non-orthogonal buses, hold the SHIFT key down while placing the bus.

You may also use the keys B and E to start and end buses.



As you place buses and wires, remember the following points:

- □ A bus and a wire can be connected only by name.
- If you begin or end a bus segment on a segment of a wire, a vertex is added to the wire, but no junction appears—the bus and wire are not connected.
- If you begin or end a wire segment on a segment of a bus, a vertex is added to the bus, but no junction appears—the wire and bus are not connected.
- ☐ Two buses or two wires can be connected physically.
- ☐ If you begin or end a bus segment on a segment of another bus, a vertex is added to the second bus, and a junction appears—the buses are connected.
- If you begin or end a wire segment on a segment of another wire, a vertex is added to the second wire, and a junction appears—the wires are connected.

Schematic page editor and part editor command reference

Note: Bus names and aliases have the form X[m..n].

- X represents the "basename" (how you think of the bus, perhaps)
- m..n represents the range of signals carried by the bus.

Note that m may be less than or greater than n: both A[0..3] and A[3..0] are valid bus aliases. You can use two periods (..), a colon (:), or a dash (-) to separate m and n.

Capture ignores any spaces between the basename and the left bracket ([). For example, ADDR[0..31], ADDR[0:31], and ADDR[0-31] represent the same bus.

Draw Toolbar:



Keyboard:

Shortcuts:

- B
- ALT, P, B

Schematic page editor and part editor command reference

Junction command

Function:

Place menu Available from:

> Use this command to place a junction to connect two nets, or remove a junction connecting two nets. Junctions can only connect wires to wires,

> or buses to buses. Junctions cannot connect wires to buses or buses to

wires.

Draw Toolbar: 🌉

Keyboard:

Shortcuts:

ALT, P, J

Bus Entry command

Available from: Place menu

Use this command to place a bus entry. **Function:**

Draw Toolbar:

Keyboard:

Shortcuts: Ε

ALT, P, E

Net Alias command

Place menu Available from:

Use this command to place a net alias on the selected object. To quit **Function:**

placing net aliases, press ESC or choose the selection tool.

Draw Toolbar: 📆

Keyboard:

Shortcuts:

ALT, P, N

Schematic page editor and part editor command reference

Power command

Available from: Place menu

Function: Use this command to place a power symbol.

Draw Toolbar:

Keyboard:

Shortcuts:

■ F

■ ALT, P, O

Ground command

Available from: Place menu

Function: Use this command to place a ground symbol.

Draw Toolbar:

Ť

Keyboard:

Shortcuts:

■ G

■ ALT, P, G

Off-Page Connector command

Available from: Place menu

Use this command to place an off-page connector, which connects to another page in the schematic folder, and to isolate power to a schematic folder. Net aliases of the same name connect wires to these

nets.

Note: If you have single pin nets connecting to internal signals then use

off-page connectors for making connections. The use of ports is

recommended when you have multiple pin nets.

Draw Toolbar:

1

Shortcuts:

Function:

Keyboard: ALT, P, F

Schematic page editor and part editor command reference

Schematic page editor and part editor command reference

Hierarchical Block command

Available from:

Place menu

Use this command to place a hierarchical block. A hierarchical block

maps to a schematic folder, not a schematic page.

Function:

If you attach an existing schematic folder to a hierarchical block, Capture automatically creates the hierarchical pins that correspond with the schematic folder's hierarchical ports. If you descend hierarchy on a hierarchical block whose schematic folder doesn't yet exist, then Capture automatically creates the hierarchical ports that correspond with the

hierarchical pins of the hierarchical block.

Draw Toolbar:

Shortcuts:

Keyboard: ALT, P, H

Hierarchical Port command

Available from:

Place menu

Use this command to place a hierarchical port. A hierarchical port is electrically connected by name to a hierarchical pin or signal "above" the schematic page. Hierarchical ports can connect laterally to other

hierarchical ports.

Function:

If you attach an existing schematic folder to a hierarchical block, Capture automatically creates the hierarchical pins that correspond with the schematic folder's hierarchical ports. If you descend hierarchy on a hierarchical block whose schematic folder doesn't yet exist, then Capture automatically creates the hierarchical ports that correspond with the hierarchical pins of the hierarchical block.

Note: If you have multiple pin nets, use hierarchical ports for making connections. The use of off-page connectors is recommended when you

have single pin nets connecting to internal signals.

Draw Toolbar: 🚾

Shortcuts:

Keyboard: ALT, P, I

Schematic page editor and part editor command reference

Hierarchical Pin command

Available from:

Place menu

Use this command to place a hierarchical pin. You must select a hierarchical block to use this command. Hierarchical pins can only be placed inside hierarchical blocks. A hierarchical pin is electrically connected by name to a port or signal in the schematic folder attached to

the pin's hierarchical block.

Function: If you attach an existing schematic folder to a hierarchical block, Capture

automatically creates the hierarchical pins that correspond with the schematic folder's hierarchical ports. If you descend hierarchy on a hierarchical block whose schematic folder doesn't yet exist, then Capture automatically creates the hierarchical ports that correspond with the

hierarchical pins of the hierarchical block.

Draw Toolbar:

Shortcuts:

Keyboard: ALT, P, R

No Connect command

Available from:

Place menu

Use this command to place a no connect symbol. This object causes unused pins to be ignored by reports that show unconnected pins. Design rule checks and netlists won't report errors for pins with no connects placed on them. No connects do not affect connected pins, even if the Is No Connect property is set to TRUE. If the No Connect property is set to TRUE and the pin is unconnected, an X appears on the pin.

Function:

No connects can also be placed by setting the Is No Connect property of

a pin to TRUE. No connects cannot be deleted using the Delete

command. You must either set the pin property to FALSE, or connect a

wire to the pin.

Draw Toolbar:

____*

Keyboard:

Shortcuts:

■ X

■ ALT, P, C

Schematic page editor and part editor command reference

Pin command

Available from: Place menu

Use this command to place one or more pins on a part. A pin is placed

with each click of the left mouse button. Press ESC or choose the

Function: selection tool to stop placing pins.

When you are placing the pin in the place mode, to edit the pin before

placing it, use SHIFT+G.

Draw Toolbar:

Keyboard:

Shortcuts:

■ ALT, P, P

■ SHIFT+G

Pin Array command

Available from: Place menu

Function: Use this command to place an array of pins.

Draw Toolbar:

Keyboard:

Shortcuts:

■ ALT, P, I

■ SHIFT+J

Title Block command

Available from: Place menu

Use this command to place optional title blocks.

You can set title block visibility in the Design Template and Schematic

Function: Page Properties dialog boxes.

There are a number of default title block properties. You can set the

values for these properties with the property editor. They are:

Schematic page editor and part editor command reference

- Cage Code: Specifies the Cage Code.
- Design Create Date: Specifies the date of creation for the design.
- Design Create Time: Specifies the time of creation for the design.
- Design File Name: Specifies the path and file name of the design file.
- Design Modify Date: Specifies the date of the last modification to the design.
- Design Modify Time: Specifies the time of the last modification to the design.
- Design Name: Specifies the name of the design.
- Doc: Specifies the document number.
- Name: Specifies the name of the title block.
- OrgAddr1: Specifies the first line of the organization's address.
- OrgAddr2: Specifies the second line of the organization's address.
- OrgAddr3: Specifies the third line of the organization's address.
- OrgAddr4: Specifies the fourth line of the organization's address.
- OrgName: Specifies the organization name.
- Page Count: Specifies the number of schematic pages in the design.
- Page Create Date: Specifies the date of creation for the schematic page.
- Page Create Time: Specifies the time of creation for the schematic page.
- Page Modify Date: Specifies the date of the last modification to the schematic page.
- Page Modify Time: Specifies the time of the last modification to the schematic page.
- Page Number: Specifies the number of the schematic page. The page number determines when it will be printed in relation to the other schematic pages of the design.

Schematic page editor and part editor command reference

- Page Size: Specifies the page size of the schematic page, as was set at creation time.
- RevCode: Specifies the revision.
- Schematic Create Date: Specifies the date of creation for the schematic folder.
- Schematic Create Time: Specifies the time of creation for the schematic folder.
- Schematic Modify Date: Specifies the date of the last modification to the schematic folder.
- Schematic Modify Time: Specifies the time of the last modification to the schematic folder.
- Source Library: Specifies the path and file name of the library from where the title block was placed.
- Schematic Page Count: Specifies the number of schematic pages in the given schematic folder.
- Schematic Page Number: Specifies the order of the schematic page within the schematic.
- Symbol Library: Specifies the name of the symbol for the title block in the Source Library.
- Title: Specifies the title.

You can add the following property to display system generated information:

Path Name: Specifies the hierarchical blocks leading from the root to the child using the Name Property for each hierarchical block in the path.

You can use the property editor to add the following property to display the hierarchical path of the schematic on an instance of a title block:

Schematic Path: Displays the full hierarchical path to the schematic visible and printable on the page.

Note: Title blocks and text cannot be mirrored or rotated.

Shortcuts: Keyboard: ALT, P, K

Schematic page editor and part editor command reference

Bookmark command

Available from: Place menu

Use this command to place a bookmark. A bookmark is a reference point

on a schematic page for finding a location.

Shortcuts: Keyboard: ALT, P, M

Text command

Available from: Place menu

Function: Use this command to place comment text. To quit placing text, press

ESC or choose the selection tool.

Draw Toolbar:

Keyboard:

Shortcuts:

■ T

■ ALT, P, T

IEEE Symbol command

Available from: Place menu

Use this command to place an IEEE symbol. When you select this

command, the <u>Place IEEE Symbol dialog box</u> appears.

Click the left mouse button to place an IEEE symbol once you have selected a symbol. Press ESC, or click on the selection tool, to quit placing the selected symbol. You can choose the Properties command

from the Edit menu, or the Edit Properties command from the right mouse button shortcut menu, to change the IEEE symbol without having

to quit the IEEE symbol tool.

Keyboard:

Shortcuts:

■ ALT, P, E

■ SHIFT+X

Draw Toolbar: 🔕

Function:

Schematic page editor and part editor command reference

Line command

Available from: Place menu

Use this command to draw a line. To place a line, you press the left mouse button to start the line. Without releasing the left mouse button,

Function: drag the pointer to the other end point for the line. Release the left

mouse button.

You may also use the keys B and E to start and end lines.

Draw Toolbar:

Keyboard:

Shortcuts:

■ ALT, P, L

■ SHIFT+L

Place menu

Rectangle command

Available from:

Use this command to draw a rectangle. Press the left mouse button and

drag the pointer to define the rectangle. To quit drawing rectangles,

Function: press ESC, or click on the selection tool.

To draw a square, hold down the SHIFT key while drawing.

Draw Toolbar: Keyboard:

Shortcuts: ■ ALT, P, R

■ SHIFT+R

Ellipse command

Available from: Place menu

Use this command to draw an ellipse. Press the left mouse button and

drag the pointer to define the ellipse. To quit drawing ellipses, press

Function: ESC, or click on the selection tool.

To draw a circle, hold down the SHIFT key while drawing.

Schematic page editor and part editor command reference

Draw Toolbar:



Keyboard:

Shortcuts:

- ALT, P, S, S
- SHIFT+F

Schematic page editor and part editor command reference

Arc command

Available from: Place menu

Function: Use this command to draw an arc. To quit drawing arcs, press ESC, or

click the selection tool.

Draw Toolbar: 🎧

Shortcuts: Keyboard: SHIFT+T

Elliptical Arc command

Available from: Place menu

Use this command to draw an elliptical arc. To quit drawing arcs, press

ESC, or click on the selection tool.

Draw Toolbar:

Shortcuts: Keyboard: SHIFT+T

Bezier curve command

Available from: Place menu

Use this command to draw a Bezier curve. To quit drawing arcs, press

ESC, or click on the selection tool.

When drawing a Bezier curve, click the left mouse button to define the start point of the curve. The click the left mouse button to define the first

start point of the curve. The click the left mouse button to define the first control point, again the left mouse button to define the second control point of the curve. Finally, click the left mouse button to define the end

point of the curve.

Draw Toolbar:

Keyboard:

Shortcuts: ■ ALT, P, Z

■ SHIFT+Q

Schematic page editor and part editor command reference

Polyline command

Available from: Place menu

Use this command to draw a polyline or polygon. To quit drawing polylines or polygons, press ESC, or click the selection tool. Click the left mouse button once to place one segment of the line and start another. Double-click the left mouse button to end the line when drawing polylines, or single-click the left mouse button to end the line while

Function: drawing polygons.

When placing polylines in the schematic page editor, you are constrained to 90-degree angles. To place a non-orthogonal polyline, hold the SHIFT key down while placing the polyline. You may also use the keys B and E to start and end polylines.

Draw Toolbar:

Keyboard:

Shortcuts:

■ Y

■ ALT, P, Y

Picture command

Available from: Place menu

Function: Use this command to place images on the schematic or in part editor.

Shortcuts: Keyboard: ALT, P, U

OleObject command

Available from: Place menu

Use this command to place images on the schematic. This command displays the standard Insert Object Windows dialog box, in which you choose to create a new object or create an Ole reference for an existing

Function: object.

After you select an object, the cursor switches to the draw mode where

you drag and draw an rectangular area to fit the object.

Schematic page editor and part editor command reference

PSpice/Markers menu

This section covers:

Markers command on page 224

Marker List command on page 224

Voltage Level command on page 225

Voltage Differential command on page 225

Current Into Pin command on page 225

Power Dissipation command on page 225

Advanced command on page 227

Plot Window Templates command on page 227

Show All command on page 227

Hide All command on page 227

Delete All command on page 228

Schematic page editor and part editor command reference

Markers command

Available from:

PSpice menu

Use this command to place markers in the design. You can place markers in your design to indicate the points for which you want to see simulation waveforms displayed in PSpice.

You can place markers:

- before simulation to limit results written to the waveform data file, and automatically display those traces in the active Probe window.
- during or after simulation, to automatically display traces in the active Probe window.

Function:

The color of the marker in Capture and its corresponding trace in the Probe window are the same. If you change the color of one or the other, its counterpart also changes.

To view the markers in the simulation results, the schematic must be open.

Marker types on the Advanced command submenu are only available after defining a simulation profile for an AC Sweep/Noise analysis.

You can also choose to show all, hide all, delete all, or list markers using this command.

Shortcuts:

Keyboard: ALT, T, G

Marker List command

Available from:

PSpice menu

From the Project Manager, choose Marker List from the PSpice menu.

Function:

Use this command to display or hide markers in the design. The <u>Markers dialog box</u> displays markers that exist in the profile. An empty check box beside a marker indicates that the marker is hidden. If a marker is hidden, it will not appear in Capture but it will still exist in the profile.

Shortcuts:

Keyboard: ALT, S, L

Schematic page editor and part editor command reference

Voltage Level command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function: Use this command to place voltage level markers on the schematic, in a

location of your choice.

Shortcuts: Keyboard: ALT, S, M, V

Voltage Differential command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function: Use this command to place voltage differential markers on the

schematic, in a location of your choice.

Shortcuts: Keyboard: ALT, S, M, D

Current Into Pin command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function: Use this command to place current into pin markers on the schematic, in

a location of your choice.

Shortcuts: Keyboard: ALT, S, M, C

Power Dissipation command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function: Use this command to place power dissipation markers on the schematic,

in a location of your choice.

Schematic page editor and part editor command reference

Shortcuts: Keyboard: ALT, S, M, P

Schematic page editor and part editor command reference

Advanced command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function:Use this command to place markers for AC Sweep/Noise analysis.

Note: Marker types on the Advanced command submenu are only available after defining a simulation profile for an AC Sweep/Noise

analysis.

Plot Window Templates command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function: Use this command to place a plot window template marker on the

schematic. The marker will restore the associated template when you

run the simulation in PSpice.

Shortcuts: Keyboard: ALT, S, M, T

Show All command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function:
Use this command to display all the markers on the schematic.

Shortcuts: Keyboard: ALT, S, M, S

Hide All command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function:

Use this command to hide all the markers on the schematic

Schematic page editor and part editor command reference

Shortcuts: Keyboard: ALT, S, M, H

Delete All command

Available from: PSpice menu, Markers submenu

From a schematic page, point to Markers on the PSpice menu.

Function:

Use this command to delete all the markers on the schematic.

Shortcuts: Keyboard: ALT, S, M, E

Accessories menu

This section covers:

"Accessories command" on page 228

Accessories command

Available from: Accessories menu

You can use extensions to the Cadence-supplied functionality of Capture if you purchase software developed by associates of Cadence. These associates create .DLL files that address specific Capture functionality, such as customized netlisting. The associates configure their .DLL files

so that they are listed as menu choices in the Accessories menu, which is available in either the project manager window or the schematic page

editor window.

Options menu

This section covers:

Function:

Autobackup command on page 229

Preferences command on page 230

Design Template command on page 230

Schematic page editor and part editor command reference

Schematic Page Properties command on page 230

Autobackup command

Available from: Options menu

Determines the frequency, location, and the number of copies of

autobackup done by Capture.

The Multi-level backup Settings dialog box appears when you choose

Autobackup option from the Options menu.

Enter the values for the following fields to determine the duration,

number of backups, and its storage.

Function: Backup time (in minutes) - Enables you to determine the time after

which Capture will perform automatic backup.

No of backups to keep - Enables you to determine the total number of

backups that will be stored.

Directory for backup - Enables you to determine the storage location

for the backup

Shortcuts: Keyboard: ALT, O, B

Schematic page editor and part editor command reference

Preferences command

Available from: Options menu

Use this command to set your environment preferences for the current

Function: project (and all future projects) on your system. The options you specify

affect the behavior of the software, and are saved in the .INI file.

Shortcuts: Keyboard: ALT, O, P

Design Template command

Available from: Options menu

Use this command to specify default settings for new projects, designs,

Function: and schematic pages. The values specified in this dialog box do not

affect existing projects or designs.

Note: To change the properties of an active design, use the <u>Design</u> <u>Properties command</u>. To change the properties of an active schematic page, use the <u>Schematic Page Properties command</u>. You cannot change

the default title block of an active schematic page.

Shortcuts: Keyboard: ALT, O, D

Schematic Page Properties command

Available from: Options menu

Function: Use this command in the schematic page editor to set

schematic-page-related options.

Shortcuts: Keyboard: ALT, O, R

Window menu

New Window command on page 232

Schematic page editor and part editor command reference

Cascade command on page 232

Tile Horizontally command on page 232

Tile Vertically command on page 232

Arrange Icons command on page 233

1,2.... command on page 233

Schematic page editor and part editor command reference

New Window command

Available from: Window menu

Use this command to create a new window, which is a copy of the

Function: currently active window. This new window is another "view" on the same

data, and you can scroll the two windows to different positions.

Shortcuts: Keyboard: ALT, W, N

Cascade command

Available from: Window menu

Use this command to "stack" all open Capture windows so that just their

title bars are visible. The active window stays on top.

Shortcuts: Keyboard: ALT, W, C

Tile Horizontally command

Available from: Window menu

Use this command to arrange open Capture windows, one above

another, so that all are visible.

Shortcuts: Keyboard: ALT, W, H

Tile Vertically command

Available from: Window menu

Use this command to arrange open Capture windows, one beside

another, so that all are visible.

Shortcuts: Keyboard: ALT, W, V

Schematic page editor and part editor command reference

Arrange Icons command

Available from: Window menu

Function: Use this command to arrange the icons for minimized windows across

the bottom of the session frame.

Shortcuts: Keyboard: ALT, W, A

1,2.... command

Available from: Window menu

Use the numbers listed at the bottom of the Window menu to view which windows are currently open, and to determine which window is active

windows are currently open, and to determine which window is active.

(The active window is indicated by a check mark.) When you choose a window from this list, Capture restores that window if it was in icon form,

pops it to the front of the Capture session, and makes it the active

window.

Shortcuts: Keyboard: ALT, W, n (n = 1, 2, ...)

Help menu

Function:

This section covers:

OrCAD X Capture Help command on page 234

Known Problems and Solutions command on page 234

What's New command on page 234

Learning PSpice command on page 234

About OrCAD X Capture command on page 235

Web Resources command on page 235

<u>Documentation command</u> on page 236

Note: To use the context-sensitive menu commands, select one or more items, then press the right mouse button. The contents of the menu differ depending on the objects selected.

Schematic page editor and part editor command reference

OrCAD X Capture Help command

Help menu Available from:

Use this command to display the Help window. **Function:**

Draw Toolbar:

Keyboard:

Shortcuts:

F1

ALT, H, H

Known Problems and Solutions command

Available from: Help menu

Use this command to display a document listing the known problems in

this release of OrCAD X Capture and tells you how to solve or work **Function:**

around these problems.

Keyboard: ALT, H, K Shortcuts:

What's New command

Available from: Help menu

Use this command to display a document describing the new features **Function:**

and enhancements in this release.

Keyboard: ALT, H, K **Shortcuts:**

Learning PSpice command

Available from: Help menu

Schematic page editor and part editor command reference

Use this command to run the interactive learning material that uses designs created in Capture to explain several diverse topics, ranging from basic theorems to some of the advanced topics in the field of Electrical & Electronics Engineering. It also introduces basic Electronics

Function: Design Automation (EDA) concepts.

The Learning PSpice tab has a browsing pane on the left that lists the different topics and subtopics. Click on any topic to open the content in the right pane. Each circuit is accompanied by an icon on the right that

you can click to open the circuit in OrCAD X Capture.

Shortcuts: Keyboard: ALT, H, R

About OrCAD X Capture command

Available from: Help menu

Use this command to get the software version number, copyright

information, registration number, and license information.

Shortcuts: Keyboard: ALT, H, A

Web Resources command

Available from: Project manager Help menu, or schematic page editor Help menu

Function: Use this command to link to Capture resources on the web

Shortcuts: Keyboard: ALT, H, W

Note: You can add other web resources to the displayed list by modifying your CAPTURE.INI file. Note, however, that the first resource in the list, appears as the last name in the menu. Therefore, if you want to add a web resource to the top of the list, include it in the CAPTURE.INI file as the last entry.

Schematic page editor and part editor command reference

Documentation command

In the project manager or schematic page editor, from the Help menu, **Available from:**

choose Documentation.

Use this command to launch the HTML page, which contains links to all

the documentation types (manuals and online help), product tutorial, and **Function:**

multimedia demonstrations shipped with this product release.

Keyboard: ALT, H, D **Shortcuts:**

Shortcut menu

This section covers:

Mirror Horizontally command on page 237

Mirror Vertically command on page 237

Rotate command on page 238

Edit Properties command on page 238

Edit Part command on page 238

Select Entire Net command on page 239

Descend Hierarchy command on page 239

Synchronize Up command on page 239

Synchronize Down command on page 240

Synchronize Across command on page 240

Connect to Bus command on page 240

<u>User Assigned Reference – Set command</u> on page 241

<u>Update Design Hierarchy command</u> on page 241

Show Footprint command on page 241

Edit PSpice Model on page 242

Schematic page editor and part editor command reference

Zoom In command on page 242

Zoom Out command on page 242

Go To command on page 242

Delete command on page 242

Tooltip command on page 243

Lock command on page 243

<u>UnLock command</u> on page 243

SI Analysis - Assign SI Model command on page 243

SI Analysis – Validate SI Model Assignments command on page 243

SI analysis- Remove SI Model Assignments command on page 243

Create UnNamed NetGroup command on page 244

Reorder pins for UnNamed NetGroup command on page 244

Save As HTML command on page 244

Waive DRC on page 244

Unwaive DRC on page 245

Note: To use the context-sensitive menu commands, select one or more items, then press the right mouse button. The contents of the menu differ depending on the objects selected.

Mirror Horizontally command

See <u>Horizontally command</u> on page 169

Mirror Vertically command

See Vertically command on page 169

Schematic page editor and part editor command reference

Rotate command

See Rotate command on page 171

Edit Properties command

See Properties command on page 163

Edit Part command

See Part command on page 166

Schematic page editor and part editor command reference

Select Entire Net command

Available from: Popup menu

Use this command to select the entire net associated with the selected

wire or bus. To select an entire net, you must first select a single wire or bus. The Select Entire Net command only works on the active schematic

page.

Note: The Select Entire Net command is restricted to the active

schematic page—it doesn't follow hierarchical blocks, hierarchical ports, or off-page connectors across schematic folders or schematic pages. For

more information, see Tracing a net.

Remember that nets on a schematic page are electrically connected by name, by alias, or by connection to a named hierarchical port or off-page

connector.

Keyboard: CTRL+SHIFT+W

Shortcuts:

Function:

Shortcut menu: Select Entire Net

Edit Wire Properties command

Available from: Popup menu

Use this command to open the Edit Wire(s) dialog box to specify line

width and style and color for the selected wire.

Descend Hierarchy command

See <u>Descend Hierarchy command</u> on page 183

Synchronize Up command

See Synchronize Up command on page 185

Schematic page editor and part editor command reference

Synchronize Down command

See Synchronize Down command on page 185

Synchronize Across command

See Synchronize Across command on page 186

Connect to Bus command

See Auto Wire Connect to Bus on page 207

Schematic page editor and part editor command reference

User Assigned Reference – Set command

Shortcut with part selected in schematic page or Reference property in **Available from:**

the Property Editor, User Assigned Reference – Set

Use this command to mark references as user assigned

OrCAD X Capture marks a reference as user assigned if:

Reference is changed in the Property Editor **Function:**

Reference is changed in the Schematic canvas

Reference is changed in board (Backannotation)

Shortcut menu: User Assigned Reference – Set **Shortcuts:**

Update Design Hierarchy command

Shortcut menu from the Hierarchy tab in the Project manager Available from:

Use this command to update the design hierarchy for the current project. **Function:**

Shortcut menu: Update Design Hierarchy Shortcuts:

Show Footprint command

Popup, Show Footprint command Available from:

> Use this command to open the footprint viewer for the selected schematic part.

Note: For the footprint to display in the viewer, you need to ensure that:

a valid PCB Footprint property is defined for the part. **Function:**

> the Allegro Footprints is defined in the capture.ini file.

the pcb env file contains the psmpath, padpath variables. And that both these variables are assigned valid footprint and pad file paths.

Shortcut menu: Show Footprint Shortcuts:

Schematic page editor and part editor command reference

Edit PSpice Model

See PSpice Model command on page 163

Selection Filter

Selection Filter command on page 195

Fisheye View

See Fisheye view command on page 191

Zoom In command

See In command on page 188

Zoom Out command

See Out command on page 189

Go To command

See Go To command on page 185

Delete command

See <u>Delete command</u> on page 159

Schematic page editor and part editor command reference

Tooltip command

Schematic page editor shortcut menu, when you select the schematic

part.

Available from:

Function:

Note: The Tooltip menu item will not appear if you select a component on

the schematic page and then invoke the shortcut menu.

Use this command to toggle the display on tooltips for pins, parts and

nets on the schematic page.

Shortcuts: Shortcut menu: ToolTip

Lock command

See Lock command on page 171

UnLock command

See <u>UnLock command</u> on page 173

SI Analysis – Assign SI Model command

See <u>Assign SI Model command</u> on page 197

SI Analysis – Validate SI Model Assignments command

See Validate SI Model Assignments command on page 199

SI analysis – Remove SI Model Assignments command

See Remove SI Model Assignments command on page 199

Schematic page editor and part editor command reference

Create UnNamed NetGroup command

Schematic page when you select wires. Available from:

Use this command to create and place an unnamed NetGroup based on **Function:**

selected wires.

None. Shortcuts:

Reorder pins for UnNamed NetGroup command

Schematic page when you select an UnNamed NetGroup. Available from:

Use this command to open the Reorder UnNamed NetGroup Pins dialog **Function:**

box, where you can change the order of pins.

Add or Remove Pins on NetGroup Block command

Schematic page when you select a NetGroup Block. **Available from:**

Use this command to open the Add/Remove the Pins on NetGroup Block **Function:**

dialog box, where you can add or remove the pins on a NetGroup Block.

Save As HTML command

Find window, when you select an object in the list. Available from:

Use this command to save the results of a Find operation to an HTML

file. The file is saved to the save location as the current design. Also, a

message, giving the name and path of the file is displayed on this

command.

None. **Shortcuts:**

Waive DRC

Function:

Schematic and Find window, when you select one or more DRC Available from:

markers.

Schematic page editor and part editor command reference

Use this command to waive a DRC. A waived DRC is not shown in the

schematic page and Yes is written under Waived column in the Find

window. Waived DRCs are not included in reports.

If you select Preserve waived DRC in the Design Rules Check dialog

box, the waived state is preserved for the design.

None. **Shortcuts:**

Unwaive DRC

Function:

Function:

Schematic window and Find window, when you select one or more DRC **Available from:**

markers.

Use this command to unwaive DRCs. You can select one or more waived

DRCs in the schematic page or the Find window and unwaive them so

that they are shown in the schematic page and included in reports.

None. **Shortcuts:**

Schematic page editor and part editor command reference

Session log command reference

This chapter covers:

- File menu on page 247
- <u>View menu</u> on page 254
- Edit menu on page 256
- Options Menu on page 258
- Window menu on page 260
- Help menu on page 262

File menu

New command on page 248

Open command on page 248

Save command on page 250

Save As command on page 250

Print Preview command on page 251

Print command on page 251

Print Setup command on page 253

<u>Import Design command</u> on page 253

Exit command on page 253

1,2,3,4 command on page 254

Session log command reference

New command

Available from:

File menu

Use this command to open a new design, or library. Choose a command from the menu that appears:

- Design
- Library
- **Function:**
- VHDL File
- Verilog File

The number of open windows you can have is only limited by your available system resources. You can use the Window menu to switch among open windows (see 1,2.... command).

Toolbar:



Shortcuts:

Keyboard:

■ ALT, F, N

Open command

Available from:

File menu

Use this command to open an existing project, design, library, VHDL, or Verilog file in a new window. Choose a command from the menu that appears:

Design

Library

Function: Project

VHDL File

Verilog File

The number of open windows you can have is only limited by your available system resources. You can use the Window menu to switch among open windows (see 1,2... command).

Session log command reference

Note: When you choose the Open button on the toolbar, a standard Windows Open dialog box appears, in which you can choose the type of file you want to open in the Files of type drop-down list. Among the listed choices are SDT Schematic (*.SCH) and SDT Library (*.LIB).

Toolbar:



Shortcuts:

Keyboard:

■ ALT, F, O

Session log command reference

Save command

Available

Function:

File menu

from:

Use this command to save the active, modified projects, designs, libraries, and VHDL files. You can save a design, library, VHDL file,

or session log under a different name using the Save As command

on the File menu.

Note: When you save a project, Capture automatically creates a backup with a .DBK file extension. When you save a library, Capture automatically creates a backup with a.OBK file extension. If you save only a schematic page or part, no backup is generated.

Toolbar:

Keyboard:

Shortcuts:

CTRL+S

ALT, F, S

Save As command

Available

Function:

File menu

from:

Use this command to save the active project, design, library, VHDL

file, or session log under a different name or to save a new,

unnamed project, design, library, VHDL file, or session log. You can save a design, library, schematic page, part, or session log with the

Open command on the File menu.

The Save As command opens a standard Windows dialog box to

save files.

Session log command reference

Note: When you save a project, Capture automatically creates a backup with a .DBK file extension. When you save a library, Capture automatically creates a backup with a.OBK file extension. If you save only a schematic page or part, no backup is generated.

Note: When you use the Save As command, you are prompted to choose the file type from the Save As Type list in the Save As dialog box. You can choose to save the file in the current design database schema version or in a schema version that is one version prior to the application version you are currently using.

Shortcuts:

Keyboard: ALT, F, A

Print Preview command

Available

File menu

from:

Use this command to see how a schematic page or part will look when printed.

Function:

After setting the options in the Print Preview dialog box, click OK to preview the printed document. You can use the buttons at the top of the window to view different pages and to zoom in and out.

Note: Be prepared to wait if you attempt to print multiple pages or parts. Depending on the number and size of the pages or parts you are previewing, Capture may require extra time to display the selection.

Shortcuts:

Keyboard: ALT, F, V

Print command

Available from:

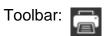
File menu

Function:

Use this command to print the active schematic page, the active part, or the selected items in the project manager.

Note: When you print multiple copies, the copies are grouped by page, not sorted by copy.

Session log command reference



Keyboard:

Shortcuts:

- CTRL+P
- ALT, F, P

Session log command reference

Print Setup command

Available

Function:

File menu

from:

Use this command to choose a printer, paper source, and orientation before printing. The Print Setup command displays the <u>Print Setup dialog box</u>, a standard windows dialog box for configuring your printer or plotter. For more information on setting up printers and plotters, refer to the documentation for your configured printer driver.

Tip

Many times, the options for your printer are not available in the standard setup dialog box. If you do not find the options you need, try the printer setup in the Windows Control Panel.

Shortcuts:

Keyboard: ALT, F, R

Import Design command

Available

Function:

File menu

from:

Use this command to import EDIF and PDIF designs. EDIF designs

must be graphical EDIF designs, and not EDIF netlists. Not all imported PDIF parts may be edited in Capture. Such parts won't

affect netlists.

Exit command

Available

File menu

from:

Use this command to exit the software. If necessary, you are

prompted to save your changes.

Function:

You can also exit the software by choosing the Close command on

the session frame Control menu (ALT, SPACEBAR, C).

Session log command reference

Keyboard:

■ ALT+F4

Shortcuts:

ALT, SPACEBAR, C

■ ALT, F, X

1,2,3,4 command

Available from:

File menu

Function:

Use the numbers listed at the bottom of the File menu to open one of

the last four projects or files. Choose the file you want to open.

Shortcuts:

Keyboard: ALT, F, n (n = 1, 2, 3, or 4)

View menu

This section covers:

Toolbar command on page 255

Status Bar command on page 255

Command Window command on page 255

Session log command reference

Toolbar command

Available

View menu

from:

Use this command to show or hide the toolbars. This setting is stored in your .INI file and thus affects the visibility of the toolbars in subsequent sessions. You can move the toolbars anywhere on the screen by pressing the left mouse button over the toolbar, and then

Function:

moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place.

Otherwise, it floats on the screen wherever it is released.

If the toolbar is floating, you can hide it by choosing Toolbar from the

View menu.

Shortcuts:

Keyboard: ALT, V, T

Status Bar command

Available from: View menu

Use this command to show or hide the status bar. This setting is stored

Function: in your .INI file and thus affects the visibility of the status bar in

subsequent sessions.

Shortcuts: Keyboard: ALT, V, S

Command Window command

Available from: View - Toolbar menu

Use this command to open the TCL Command window in the OrCAD X

Function: Capture interface.

For details see the Command Window.

Session log command reference

Edit menu

This section covers:

Copy command on page 257

Select All command on page 257

Find command on page 257

Clear Session Log command on page 258

Session log command reference

Copy command

Available from:

Edit menu

l la a thia a ann

Use this command to copy a selected object to the Clipboard without removing it from the active window. This command is available only if an object is selected.

Function:

Copying objects to the Clipboard replaces any objects previously stored there. Use the Paste command to copy objects to another page or part, or to another Windows application that supports pasting from the Clipboard.

Note: The Cut and Copy commands are unavailable in the part editor when you have one or more pins selected with other objects (such as arcs and lines).

Toolbar:

厚

Keyboard:

Shortcuts:

■ CTRL+C

■ ALT, E, C

Select All command

Available

Edit menu

from:

Function: Use this command to select all items in the active window.

Shortcuts: Keyboard: Ctrl+A

Find command

Available from:

Edit menu

Session log command reference

Use this command to locate an object or string of text in the active window.

In the schematic page editor and project manager, the Find command finds all instances of the specified text search string, parts, part pins, DRC markers and constraints. The Find command supports wildcard searches. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single

Function:

character.

In the session log, the *Find* command will find the next occurrence of the specified text search string from the current position. However, it does not support wildcard searches.

The Find window in part editor can search only pin and text, and it supports wildcard searches.

Keyboard:

Shortcuts:

■ CTRL+F

■ ALT, E, F

Clear Session Log command

Available

Edit menu

from:

Function:

Use this command to clear the session log.

Keyboard:

Shortcuts:

CTRL+DELETE

Options Menu

- Preferences command on page 259
- Design Template command on page 259
- Autobackup command on page 259

Session log command reference

Preferences command

Available

Function:

Shortcuts:

Options menu

from:

Use this command to set your environment preferences for the

current project (and all future projects) on your system. The options you specify affect the behavior of the software, and are saved in the

.INI file.

Keyboard: ALT, O, P

Design Template command

Available

Options menu

from:

Use this command to specify default settings for new projects,

Function:

designs, and schematic pages. The values specified in this dialog

box do not affect existing projects or designs.

Note: To change the properties of an active design, use the Design

Properties command. To change the properties of an active

schematic page, use the Schematic Page Properties command. You cannot change the default title block of an active schematic page.

Shortcuts:

Keyboard: ALT, O, D

Autobackup command

Available from: Options menu

Session log command reference

Determines the frequency, location, and the number of copies of autobackup done by Capture.

The Multi-level backup Settings dialog box appears when you choose Autobackup option from the Options menu.

Enter the values for the following fields to determine the duration, number of backups, and its storage.

Function:

Backup time (in minutes) - Enables you to determine the time after which Capture will perform automatic backup.

No of backups to keep - Enables you to determine the total number of backups that will be stored.

Directory for backup - Enables you to determine the storage location for the backup

Shortcuts:

Keyboard: ALT, O, B

Window menu

Cascade command on page 261

Tile Horizontally command on page 261

Tile Vertically command on page 261

Arrange Icons command on page 261

1,2.... command on page 262

Close All Windows on page 262

Session log command reference

Cascade command

Available

Function:

Window menu

from:

Use this command to "stack" all open Capture windows so that just

their title bars are visible. The active window stays on top.

Shortcuts: Keyboard: ALT, W, C

Tile Horizontally command

Available

Window menu

from:

Function: Use this command to arrange open Capture windows, one above

another, so that all are visible.

Shortcuts: Keyboard: ALT, W, H

Tile Vertically command

Available

Function:

Window menu

from:

Use this command to arrange open Capture windows, one beside

another, so that all are visible.

Shortcuts: Keyboard: ALT, W, V

Arrange Icons command

Available

Function:

Window menu

from:

Use this command to arrange the icons for minimized windows

across the bottom of the session frame.

Shortcuts: Keyboard: ALT, W, A

Session log command reference

1,2.... command

Available

Window menu

from:

Use the numbers listed at the bottom of the Window menu to view which windows are currently open, and to determine which window is active. (The active window is indicated by a check mark.) When

Function:

is active. (The active window is indicated by a check mark.) When you choose a window from this list, Capture restores that window if it was in icon form, pops it to the front of the Capture session, and

makes it the active window.

Shortcuts:

Keyboard: ALT, W, n (n = 1, 2, ...)

Close All Windows

Available

Window menu

from:

Function: Use this command to close all open windows.

Help menu

OrCAD X Capture Help command on page 263

Known Problems and Solutions command on page 263

What's New command on page 263

About OrCAD X Capture command on page 263

Web Resources command on page 264

Documentation command on page 264

Note: To use the context-sensitive menu commands, select one or more items, then press the right mouse button. The contents of the menu differ depending on the objects selected.

Session log command reference

OrCAD X Capture Help command

Available

Help menu

from:

Use this command to display the Capture Help window. **Function:**

Toolbar:

Keyboard:

Shortcuts:

F1

ALT, H, H

Known Problems and Solutions command

Available

Help menu

from:

Use this command to display a document listing the known problems

Function:

in this release of OrCAD X Capture and tells you how to solve or

work around these problems.

Shortcuts:

Keyboard: ALT, H, K

What's New command

Available

Function:

Help menu

from:

Use this command to display a document describing the new

features and enhancements in this release.

Keyboard: ALT, H, K **Shortcuts:**

About OrCAD X Capture command

Available

Help menu

from:

Session log command reference

Function:

Use this command to get the software version number, copyright information registration number, and license information.

information, registration number, and license information.

Shortcuts: Keyboard: ALT, H, A

Web Resources command

Available

Project manager Help menu, or schematic page editor Help menu

from: Function:

Use this command to link to Capture resources on the web

Shortcuts: Keyboard: ALT, H, W

Note: You can add other web resources to the displayed list by modifying your CAPTURE.INI file. Note, however, that the first resource in the list, appears as the last name in the menu. Therefore, if you want to add a web resource to the top of the list, include it in the CAPTURE.INI file as the last entry.

Documentation command

Available

Function:

from:

In the project manager or schematic page editor, from the Help

menu, choose Documentation.

Use this command to launch the HTML page, which contains links to

all the documentation types (manuals and online help), product

tutorial, and multimedia demonstrations shipped with this product

release.

Shortcuts: Keyboard: ALT, H, D

Command Window command reference

This chapter covers:

■ Command Window Shortcut Menu on page 265

Command Window Shortcut Menu

- Font command on page 266
- Background Color command on page 266
- Text Color command on page 266
- Save command on page 266
- Clear All command on page 266

Command Window command reference

Font command

Available from: Shortcut menu

Function: Use this command to change the font of the Command window

Shortcuts: None

Background Color command

Available from: Shortcut menu

Function: Use this command to change the background color of the Command

window

Shortcuts: None

Text Color command

Available from: Shortcut menu

Function: Use this command to change the text color of the Command window

Shortcuts: None

Save command

Available from: Shortcut menu

Use this command to save the commands in the Command window to a

TCL file.

Shortcuts: None

Clear All command

Available from: Shortcut menu

Command Window command reference

Use this command to clear all the commands from the Command **Function:**

window

None **Shortcuts:**

OrCAD X Capture CIS Reference Guide Command Window command reference

6

Window Descriptions

The following is an exhaustive set of descriptions for the window types you may encounter using Capture. Each description is listed alphabetically, using the window title.

Browse window

The browse window displays the results of a browse of parts, nets, hierarchical ports, off-page connectors, DRC markers, and bookmarks.

When you browse a design or library, you can sort the results using the buttons at the top of the browse window. Each type of object offers a different set of buttons. When you click on one of these buttons, Capture alphabetically sorts the selection by the value of the corresponding property. To view a specific object, double-click on the item in the browse window. To add, delete, or change properties, select objects in the browse window, and then choose the Properties command from the Edit menu.

Parts

Reference Order by the part reference.

Value Order by the part value. If the part has

no alias, this column is identical to

Source Part.

Source Part Order by the source part. If the part is

an alias, this column shows the original

part.

Source Library Order by the source library. This column

shows the path and library where the

part exists.

Page Order by the schematic page the part is

on.

Window Descriptions

Schematic Order by the schematic folder the part is

in.

Nets

Name Order by the net alias name.

Net Name Order by the net name.

Page Order by the schematic page the net is

on.

Schematic Order by the schematic folder the net is

in.

Hierarchical ports

Port Name Order by the hierarchical port name.

Port Type Order by the hierarchical port type.

Page Order by the schematic page the

hierarchical port is on.

Schematic Order by the schematic folder the

hierarchical port is in.

Off-page connectors

Off-Page Name Order by the off-page connector name.

Page Order by the schematic page the

off-page connector is on.

Schematic Order by the schematic folder the

off-page connector is in.

Window Descriptions

Bookmarks

Bookmark Name Order by the bookmark name.

Page Order by the schematic page the

bookmark is on.

Schematic Order by the schematic folder the

bookmark is in.

DRC markers

DRC Error Order by the DRC error message text.

This is the text that appears in the session log, the DRC report, and the

View DRC Marker dialog box.

DRC Detail Order by the object generating the error.

DRC Location Order by the absolute location of the

error.

Page Order by the schematic page the DRC

marker is on.

Session frame window

The <u>session frame</u> contains the following components:

- session log
- project manager
- browse window
- schematic page editor
- part editor

As with other true Windows applications, each of these components can be reduced to an icon (minimized), opened (maximized), and resized. For more information on using Windows applications, see your Windows documentation.

Window Descriptions

Session Log Window

The session log contains a record of events that occur during the current session of Capture. This window has a ruler with adjustable tabs, so you can format the way the information in the session log appears. This formatting only applies to the session log. It doesn't affect the way reports are formatted in other applications. You can set the session log ruler measurements to appear in U.S. or metric units by using the appropriate setting in the Regional Settings of your Control Panel.

The session log also includes results and messages from Capture utilities found on the Tools menu. If Capture reports an error or warning in the session log, you can get specific help on it by double-clicking on the message. In this case, Capture opens the file that contains the error and places the cursor at the location of the error. These files include netlists, CDS.LIB, HDL.VAR, and VHDL/Verilog models.

The following Capture utilities are found on the *Tools* menu for Session Log:

- Annotate
- Back Annotate
- Update Properties
- Create Netlist
- Cross Reference
- Bill of Materials
- Export Properties
- Import Properties

The session log is replaced every time you start Capture, so it is initially empty. You can clear the session log at any time by choosing Clear Session Log (ALT, E, S) from the Edit menu, or pressing CTRL+DEL.

You can minimize the session log by pressing CTRL+F4, or by choosing the Close button in the upper-right corner of the session log window. To open the session log, choose Session Log (ALT, W, 1) from the Window menu. The session log records utility results and error messages even while it is minimized.

You can save the session log as an ASCII text file, and you can copy text from the session log onto the Clipboard. You cannot load a saved session log into Capture, and you cannot cut or paste text in the session log.

Window Descriptions

Part editor window

You edit parts and symbols in the part editor window. This window has two view splitters. The splitter at the upper right divides the view horizontally. The splitter at the lower left divides the view vertically. Each view has its own scroll bars, so you can view separate areas on the same part.

You can create parts up to 32 by 32 inches.

Part View

You edit parts in this view.

Package View

You see the entire package in this view. You cannot edit parts in this view, but you can select parts to edit. This view has no view splitters.

The part editor tool palette is unavailable in this view.

Property Editor window

The property editor window appears when you select some combination of parts, nets, pins, title blocks, aliases and globals in the schematic page editor, and then choose Properties from the Edit menu or choose Edit Properties from the shortcut menu. You can use the property editor window to edit part, net, pin, title block, global, port, and alias properties. The property editor displays all library definitions, instance properties, and occurrence properties for an object.



Do not manually change the reference designators of heterogeneous parts for a complex hierarchical design. In case you want to change the reference designator for a part placed in the schematic page, delete the part and add it again. This way all the occurrences will get updated correctly.

Window Descriptions

New Property

Displays the Add New Property dialog box, depending on the property editor orientation, to add a new property column or row. To add the property to an object, you must enter a property value for a given object.

Apply

Applies the changes in the property editor to the schematic page. The Apply button does not dismiss the property editor. You can also apply the changes to the schematic page by closing the property editor.

Display

Displays the Display Properties dialog box to set the display option of the selected property and its value. You cannot display properties of an occurrence property using the Display Properties dialog box.

Delete Property

Deletes the editable property from the selected object or objects. (Properties that are not editable appear in italics.) If you select all of a property's cells and click the Delete Property button, the property will be removed from the selected objects but will remain in the filter. This is indicated by the hash marks that appear in the cell.

Filter by

Specifies a filter by which to view the objects. Use the property editor filter to constrain the available properties. For example, the *Capture* filter displays common schematic capture properties available to most parts, while the *Cadence-PCB Editor* filter displays properties needed to send a design to PCB Editor. You can view all the properties available on the objects in the property editor by selecting the *Current properties* filter from the drop-down list.

Parts

Displays the parts of the selected objects. The Parts tab includes hierarchical blocks.

Window Descriptions

Schematic Nets

Displays the schematic nets of the selected objects. This tab includes constituent nets within buses.

Pins

Displays the pins of the selected objects. This tab includes hierarchical pins in hierarchical blocks.

Title Blocks

Displays the title blocks of the selected objects.

With the Title Blocks tab selected, you can add a property to the Title Block instance on a schematic page that will display the full hierarchical path to the schematic.

Globals

Displays selected globals for simultaneous editing of multiple names.

Ports

Displays source symbol, source library, and type of port. Provides for simultaneous editing of multiple ports.

Aliases

Displays color, font, name, and rotation of net aliases. Use the Aliases tab to edit multiple aliases at one time.

Rows and columns

Each row displays an instance or an occurrence of an object. Instance rows appear with a white background. Occurrences appear in yellow below their associated instance row. Occurrence rows automatically appear when one or more of the occurrence property values are different from the instance property values.

Each column is a placeholder that you can use to add properties. The cells in the property editor show the property values for each instance or occurrence. A cell with hash marks in

Window Descriptions

indicates that the property does not exist on the object that the cell represents. You can add a value by clicking inside the cell, typing the value, and pressing ENTER or clicking the Apply button. A property value in italics is a read only property cannot be edited.



Roll the mouse wheel up and down to scroll through vertically in the Property Editor.



Hold down the CTRL key and roll the mouse wheel to zoom in and zoom out.

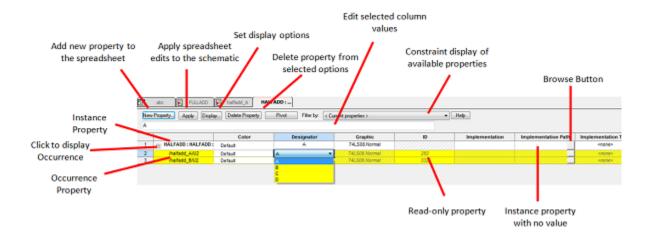


Hold down the SHIFT key and roll the mouse wheel up and down to scroll through horizontally in the Property Editor.



Click the mouse wheel button and drag the mouse wheel:

- □ To the right or left in the Property Editor window to scroll horizontally.
- □ Up or down in the Property Editor window to scroll vertically.



Window Descriptions

Short-cut keys

The following short-cut keys apply to the Property Editor:

Operation/command	Short-cut key
Undo	CTRL+Z
Сору	CTRL+C
Paste	CTRL+V
Cut	CTRL+X
Find	CTRL+F
Move to first cell in column	PageUp/CTRL+ <up-arrow></up-arrow>
Move to last cell in column	PageDown/CTRL+ <down-arrow></down-arrow>
Move to first cell in row	CTRL+ <left-arrow></left-arrow>
Move to last cell in row	CTRL+ <right-arrow></right-arrow>
Undo Edit (within a cell)	Esc
Select	SHIFT+ <arrow key=""></arrow>
Move to top left cell in spreadsheet	CTRL+Home
Move to bottom right cell in spreadsheet	CTRL+End
Select cell contents	CTRL+F2
Close spreadsheet	CTRL+F4

Project manager window

The project manager appears in the Capture session frame whenever you open or create a project. Use the project manager to collect and organize all the resources you need for your project throughout the design flow. These resources include schematic design files, part libraries, netlists, VHDL models, simulation models, timing files, stimulus files, and any other related information.

The project manager provides two views of a project. If you choose the File tab, you see a complete list of all project resources and files, organized in folders. If you choose the Hierarchy tab, you see the hierarchy view, which displays the hierarchical relationship among the various design modules. A design module is a structural block, typically represented as a

Window Descriptions

distinct hierarchical entity, that defines the functionality of a particular portion of your design. A design module in Capture can be either a VHDL model or a schematic folder.

Each project may contain one design. This design may consist of any number of schematic folders, schematic pages, or VHDL models, but must have a single root module. The root module is the module that is defined as the top-level entity for the design. That is, all other modules in the design are referenced within the root module.

Within the project manager, you can expand or collapse the structure you are viewing by clicking on the plus sign or minus sign to the left of a folder. A plus sign indicates that the folder has contents that are not currently visible; a minus sign indicates that the folder is open and its contents are visible, listed below the folder. When you double-click on a schematic folder, Capture displays the schematic pages within that folder. If the folder is a VHDL model, Capture displays each defined entity in that model. When you double-click on a schematic page or a VHDL entity, you open that object in an appropriate editor. For example, double-clicking on a VHDL entity opens the VHDL model file at the location of that entity definition in Capture's VHDL editor.

Each project you open has its own project manager window. You can move or copy folders or files between projects by dragging them from one project manager window to another (as well as from the Windows Explorer). If you close a project manager window, you close the project.

File tab

The file tab shows all the files included in the project. These files may include VHDL models, netlists, schematic pages, simulation models, stimulus files, or any other files that contain information related to the project. The file view is organized in folders, each of which contains certain types of project files.

Hierarchy tab

The Hierarchy tab shows the hierarchical relationship among the various modules of the design.

Each instantiation of a particular module appears in the hierarchy view as part of a hierarchical "tree". The hierarchical view of the design is derived from the files that exist in the Design Resources folder.

Window Descriptions

Schematic page editor window

You edit schematic pages in the schematic page editor window. This window has two view splitters. The splitter at the upper right divides the view horizontally. The splitter at the lower left divides the view vertically. Each view has its own scroll bars, so you can view separate areas on the same page.

Roll the mouse wheel up and down to scroll through vertically.

Hold down the CTRL key and roll the mouse wheel to zoom in and zoom out.

Hold down the SHIFT key and roll the mouse wheel up and down to scroll through horizontally.

Click the mouse wheel button and drag the mouse wheel:

- ☐ To the right or left in the schematic page editor to scroll horizontally.
- □ Up or down in the schematic page editor to scroll vertically.

Text editor window

Use the text editor to create or edit text files such as VHDL or Verilog files and simulation models. You can set syntax for VHDL and Verilog to appear in different colors in the <u>Text</u> <u>Editor tab</u> in the Preferences dialog box.

You can open the text editor by choosing Open from the File menu, by selecting a text file in the project manager and choosing Edit from the shortcut menu, or by dragging the file from the Explorer into the session frame. You can only open ASCII text files using the text editor.

The text editor has the following features:

Window Descriptions

Help

Help Topics F1

Saving and Printing

Save CTRL+S
Print CTRL+P

Editing text

Join Line ALT+J Split Line ALT+S CTRL+C Copy Paste CTRL+V Cut CTRL+X Cut line to clipboard CTRL+Y Undo CTRL+Z Redo CTRL+A Delete **DELETE** Toggle insert/overwrite mode INSERT

Searches

Search Forward CTRL+F

Search Backward CTRL+SHIFT+F

Blocks and marks

Select up one line SHIFT+UP ARROW

Select down one line SHIFT+DOWN ARROW

Select left one character SHIFT+LEFT ARROW

Window Descriptions

Select right one character SHIFT+RIGHT ARROW

Select left one word CTRL+SHIFT+LEFT ARROW

Select right one word CTRL+SHIFT+RIGHT

ARROW

Select to end of line SHIFT+END

Select to end of file CTRL+SHIFT+END

Select to beginning of line SHIFT+HOME

Select to beginning of file CTRL+SHIFT+HOME

Select one page down SHIFT+PAGE DOWN

Select to end of file CTRL+SHIFT+PAGE DOWN

Select one page up SHIFT+PAGE UP

Select to beginning of file CTRL+SHIFT+PAGE UP

Window Descriptions

Cursor control

Move cursor up one line UP ARROW

Move cursor down one line DOWN ARROW

Move cursor left one character LEFT ARROW

Move cursor right one

character

RIGHT ARROW

Move cursor left one word CTRL+LEFT ARROW

Move cursor right one word CTRL+RIGHT ARROW

Move cursor to end of line END

Move cursor to beginning of

line

HOME

Page up PAGE UP

Page down PAGE DOWN

Move cursor to beginning of file CTRL+HOME

Move cursor to beginning of file CTRL+PAGE UP

Move cursor to end of file CTRL+END

Move cursor to end of file CTRL+PAGE DOWN

Shortcut menu

A shortcut menu is available in the text editor window. Click the right mouse button to bring up the shortcut menu. The following commands are available in this menu:

- Cut command
- Copy command
- Paste command
- Delete command
- Select All command
- Undo command
- Find command

Window Descriptions

Browse Spreadsheet Editor

Use the Browse Spreadsheet editor to perform the following tasks:

- Create a new property
- Copy a value from one property to another property
- Remove a user-defined property
- Replace property values

You can open the Browse Spreadsheet editor window from the:

- Project Manager pane
- Schematic Page Editor
- Part Editor
- Find Results window

From the Project Manager

From the project manager, select the schematic design and then select the *Browse* command from the *Edit* menu. You can select a component from the resulting popup menu. To open the Browse Spreadsheet editor window, select the component and click *Properties* from the *Edit* menu. You can change the properties of the following components from the Browse Spreadsheet editor.

- Parts (including hierarchical blocks)
- Nets (including constituent nets within a bus) occurrences
- Flat Netlist
- Hierarchical ports
- Off-page connectors
- Title blocks
- Bookmarks
- DRC markers

Window Descriptions

From the Schematic Page Editor

From the schematic page editor, select the schematic page in the project manager and then select the *Browse* command from the *Edit* menu. You can select a component from the resulting popup menu. To open the Browse Spreadsheet editor window, select the component and click *Properties* from the *Edit* menu. You can change the properties of the following components from the Browse Spreadsheet editor.

- Parts (including hierarchical blocks)
- Nets (including constituent nets within a bus) occurrences
- Hierarchical ports
- Off-page connectors
- Title blocks
- Bookmarks
- DRC markers

From the Part Editor

From the part editor (while in Part View), you can edit the following properties from the Browse Spreadsheet editor:

Pin properties

The Browse Spreadsheet editor browses the entire design for the objects you select, then displays their properties. Each property appears as a column heading in the spreadsheet. Each row is an object located by the editor.

It is important to note that, in the Browse Spreadsheet editor you can edit only occurrences. The only exception being in the part editor, where you can only edit instances. To edit instance properties, you must use the property editor.

From Find Results Window

You can open the Browse Spreadsheet window for selected parts in the Find Results window. To do so:

- 1. Right-click one or more results line items.
- 2. Choose Edit Properties.

Window Descriptions

The Browse Spreadsheet window displays the editable part properties.

Note: You can use the CTRL + C keys to copy a value from a cell and the CTRL + V keys to paste onto another cell in the Browse Spreadsheet editor. Also, you can use the CTRL+ INSERT keys to copy a value from a cell in the Browse Spreadsheet editor and paste it onto a cell in Microsoft Excel worksheet or use the SHIFT+ INSERT keys to paste values copied from Microsoft Excel onto a cell in the Browse Spreadsheet editor.

Command Window

OrCAD X Capture includes a scripting functionality that allows you to execute a Capture command through a command prompt in the Capture command window.

Every user action performed in Capture is logged in the form of a command. This command that logged is registered with a TCL interpreter. When the command is played back, Capture uses the TCL interpreter to retrieve the command and execute it in the resident application. However, this process is completely abstracted from the Capture. This makes logging and replaying of a set of commands an intuitive and simple task.

To execute a command, you type the command at the command prompt and press Enter.

Also, if you perform an operation in the Capture interface, the associated command is registered with the TCL interpreter and the command is logged in the Command window.

Finally, every command that is registered with the interpreter is logged in a captcl file. You can then use this file to re-run a complete set of commands. You can do this from the Capture command window or from the Operating System command prompt by passing the script name (including location) as an argument to capture.

Project Manager folders

The project manager is a tool that allows you to collect and organize all the resources you need for your project throughout the design flow. These resources include schematic pages, part libraries, and netlists, and may also include VHDL models, simulation models, timing files, stimulus files, and other related information.

When the project is first created, the project manager creates a design file with the same name as the project. It also creates a schematic folder within the design file, and a schematic

Window Descriptions

page within the folder. You can create a new design to replace the design created by the project manager

Folder	Description
Design Cache	A local library contained in each project that contains all the parts and symbols used in the design. This folder is present inside the <i>Design Resources</i> folder.
Library	Lists the library files and related files included in the current project. These files include library (*.OLB) files, simulation and synthesis (*.VHD) files, *.STL files, and *.SML files. This folder is present inside the <i>Design Resources</i> folder.
Outputs	Stores output files generated by Capture tools, such as Create Netlist, Design Rules Check, Cross reference reports, Bill of Materials, Export Properties, and Generate Part etc.
Referenced Projects	Stores any projects referenced from within the current project. Typically, referenced projects are FPGA projects that you want to include in your PCB project. This is useful for board simulation that includes the appropriate timing and functionality information for an FPGA that is included in your printed circuit board.
Simulation Resources	Contains simulation resources of your PCB designs which may include FPGA designs, as components. This folder can contain Verilog or VHDL simulation of your PCB designs. This folder is created when you choose the Board Simulation command from the Tools menu.

Color Reference

The following list contains a mapping of the Capture Color IDs to their corresponding RGB values.

Capture Color ID	RGB Value	Color Sample
1	255, 255, 128	
2	128, 255, 128	
3	0, 255, 128	
4	128, 255, 255	
5	0, 128, 255	
6	255, 128, 192	
7	255, 128, 255	
8	255, 0, 0	
9	255,255, 0	
10	128,255,0	
11	0, 255, 64	
12	0, 255, 255	

OrCAD X Capture CIS Reference Guide Color Reference

Capture Color ID	RGB Value	Color Sample
13	0, 128, 192	
14	128, 128, 192	
15	255, 0, 255	
16	128, 64, 64	
17	255, 128, 64	
18	0, 255, 0	
19	0, 128, 128	
20	0, 64, 128	
21	128, 128, 255	
22	128, 0, 64	
23	255, 0, 128	
24	128, 0, 0	
25	255, 128, 0	
26	0, 128, 0	
27	0, 128, 64	
28	0, 0, 255	
29	0, 0, 160	
30	128, 0, 128	
29	0, 0, 160	

OrCAD X Capture CIS Reference Guide Color Reference

Capture Color ID	RGB Value	Color Sample
31	128, 0, 255	
32	64, 0, 0	
33	128, 64, 0	
34	0, 64, 0	
35	0, 64, 64	
36	0, 0, 128	
37	0, 0, 64	
38	64, 0, 64	
39	64, 0, 128	
40	0, 0, 0	
41	128, 128, 0	
42	128, 128, 64	
43	128, 128, 128	
44	64, 128, 128	
45	192, 192, 192	
46	64, 0, 64	
47	255, 255, 255	
48	Windows Default Color	L

OrCAD X Capture CIS Reference Guide Color Reference

8

Dialog Box Descriptions

The following is an exhaustive set of descriptions for the dialog boxes you may encounter while using Capture. Each description is listed alphabetically, using the dialog box title.

Add file to Project Folder dialog box

Use this dialog command to add files to a project. You can select multiple files to add using the CTRL key. The title bar of the dialog box displays the folder into which the files will be added (the folder currently selected in the project manager window).

To open this dialog

Choose Project (see Project command) from the Edit menu.

Use this control... To do this...

Look in Browse the hierarchical drive and directory structure for

your system.

File name Select or type the name of the project or file(s) that you

want to add to the Project Folder.

Files of type Filter files by extension.

Add New Property dialog box

Use this dialog to add a new property to the selected object (or objects).

Dialog Box Descriptions

To open this dialog

In the property editor window, click the New Property button.

Use this control	To do this
Name	Displays the name of the new property column or row to add in the property editor.
Value	Displays the value of the new property column or row to add to selected objects and in the property editor. If you enter the name of an existing property in the Name text box, its current value appears in this text box.
Display [ON/OFF]	Select the check box to display the Display Properties dialog box.
Always show this column/row in	Select the check box to save the new column/row in the

this filter

Select the check box to save the new column/row in the current filter. You cannot save a new column or row in the

<Current properties> filter.

Note: You can narrow your selection of objects by selecting one or more object columns/rows in the property editor before opening this dialog box. When you enter a name and value, then click OK or Apply, the property is applied to the selected objects.

Add to Project dialog box

To open this dialog

Choose Library from the File - New menu with the focus away from the currently open schematics or the project manager.

The dialog box prompts you to either add the new library to the currently open projects or to a new project.

Dialog Box Descriptions

Annotate dialog box

To open this dialog

Select a design (.DSN) in the Project manager then

Choose Annotate from the Tools menu.

OR

Choose the Annotate button on the Capture toolbar.

Use this tab... To do this...

<u>Packaging tab</u>
Annotate parts and group parts together that have

common characteristics. Packaging is a key step that should be done before netlisting to a PCB board design

tool such as PCB Editor.

PCB Editor Reuse tab Generate a reuse module or renumber the reference

designators in a reuse module in Capture. Reuse modules may be netlisted then used in PCB Editor or used in Capture as library parts or as hierarchical blocks.

Dialog Box Descriptions

Packaging tab

Use this control...

To do this...

Refdes control required

Check this option, if you want to specify a part reference range for each schematic page or a hierarchical block in the root-level of your design.

/Important

This functionality works independently from the existing annotation behavior of Capture.

/Important

If you are using the Refdes control required option for a project, then the Auto reference placed part option in the Miscellaneous tab of the Preferences dialog box will not honor the range specified in the grid.

For more description about how to use this option to specify a part reference range for a schematic page or a hierarchical block, see <u>Customizing part references in a design</u>.

Scope

Update entire design

Update selection

Schematic Pages

Hierarchical Blocks

Specify whether to update all the part references in the design (or library), or just the selected schematic pages.

Note: These options are available only when you select the Refdes control required check box.

Note: The Scope options changes to Schematic Pages or Hierarchical Blocks depending on whether your design is a flat design or a hierarchical design.

Grid (for specifying part reference range)

Pages or H-Blocks column

Note: This grid appears only when you select the Refdes control required check box.

Displays all the schematic pages or hierarchical blocks in the root schematic folder of your design depending on whether your design is a flat design or a hierarchical design.

Dialog Box Descriptions

Start Value and End Value columns

Specify a numeric value greater than 0 in the cell corresponding to the schematic page name or hierarchical block name.



Use the Tab key to move from the Start Value column to End Value column.

You can also use the Arrow keys to move around in the grid.

You can use the column handle () to resize the rows and columns in the grid.

Note: A valid range must have both the Start and End Values and the End Value must be greater than the Start Value.

Action

Incremental reference update

If checked, Capture incrementally updates parts with a question mark in the part reference. For example, parts with reference designators of U?A will be numbered U1A, U1B, U1C, and so on. Part reference and package information is not updated on existing parts.

Unconditional reference update

If checked, Capture updates all parts in the selected schematic pages. Both part reference and package information may be updated on existing parts. Parts on different schematic pages are not packaged together.

Reset part references to "?"

Specifies to reset all the part references to "U?"

Add intersheet references

Specifies to add intersheet references to the design.

Delete intersheet references

Specifies to remove all intersheet references from the design.

Note: If your design contains multiple heterogeneous parts that are the same part in the package, you will need to update their part references after every time you reset all part references to "U?" or group them using a grouping property.

Dialog Box Descriptions

Annotation Type

Specifies the sequence in which the components on the design are annotated.

The Annotation Sequence list contains three options that you can use to decide the sequence in which the objects on your design are annotated - Default, Left to Right & Top to Bottom.

Mode

Specifies to update either instances or occurrences. Capture automatically sets this option based on the project type. All designs default to use instances. If a PCB or schematic design is complex or has occurrence properties, the default shifts to occurrences. Capture recommends the preferred mode, which you can override.

Physical Packaging

Specifies the properties that must match for Capture to group parts into a single package. Value and Source Library properties are the default property string, but you can use any combination you like.

The value in Additionally From INI is updated using the value specified by you in the Capture.ini file AnnotateAdditionalPropertyString property. For example, to specify PartGroup as the value, set it in Capture.ini as

AnnotateAdditionalPropertyString={PartGroup}. You can also specify more than one value such as {Value}{Source Package}. If you change this value in the field, Capture.ini will be updated accordingly.

Note: Do not use {GROUP} as a property string in combined property strings text box. This may cause problems while annotating your design for a PCB Editor tools, such as PCB Editor or OrCAD X Presto. The GROUP property is used in PCB Editor for a specific purpose.

Dialog Box Descriptions

For example, you might want to use Value and Voltage. Say your design uses both Tantalum capacitors and ceramic disk capacitors. First, you could assign ".01uF" to the part Value property for all the capacitors. Then, you could define a user property called "Voltage" for all capacitors in the design, and assign it the value "100V" or "25V" as appropriate. To annotate your design, type "{Value} {Voltage}" (without the quotation marks) in the Part Value property combine text box.

In this example, Capture groups the parts with "C?" as the part reference, and ".01uF" as the part value, but it separates the 100V Tantalum capacitors from the 25V ceramic disk capacitors.

Additionally From INI

Specifies a combined property string that gets added in the Capture.ini file under the Preferences section, such as

AnnotateAdditionalPropertyString=roperty
name>. For example, adding {ROOM} in the Additionally
From INI field has the following entry under the
Preferences section:

AnnotateAdditionalPropertyString={ROOM}

Once added, any project opened in Capture will have this property.

Reset reference numbers to begin at 1 in each page

Specify whether to number parts within the context of the schematic folder. When this option is selected, Capture begins numbering parts at 1 for every selected page. Otherwise, Capture continues numbering after the highest referenced part in the selected schematic pages.

Annotate as per PM page ordering

Specify whether to perform the annotation on the basis of order of pages/ folders in the Project Manager window. If there are multiple folders and multiple pages in each folder then the root folder is annotated first followed by its pages. Alphabetic order is followed to determine the sequence of pages in a folder.

Annotate as per page ordering in the title blocks

Specify whether to perform annotation according to numbers on the page numbers specified in the title blocks of the schematic pages.

Dialog Box Descriptions

Do not change the page number Check this option if you have chosen to annotate as per

page ordering in the title blocks option, changed the page numbers in the title blocks, but do not want to change the

page ordering on reannotation.

Include non-primitive parts

Specifies whether to annotate non-primitive parts or to

reset non-primitive part references to "?". Select this option to avoid netlisting duplicate reference errors when you want to simulate a design or generate a new part.

Preserve designator Specifies that the designator information of a

homogeneous part during unconditional or reset

annotation is to be preserved.

Preserve User Assigned Valid

References

Specifies that the user assigned reference designator

during annotation is to be preserved.

Note: You can explicitly mark a reference as user assigned by choosing User Assigned Flag – *Set* from the shortcut menu for the Reference property in the Property Editor or from the shortcut menu of a part in the schematic page. Any references changed using Property Editor, Schematic Editor, or during backannotation are marked as changed and are preserved on selecting this and the

previous option.

Auto-package Heterogeneous Part Using First Match

When you have multiple similar symbols or parts from multiple heterogeneous packages, select this option to enable capture to auto package and annotate based on

the first available part in the design.

Dialog Box Descriptions

PCB Editor Reuse tab

Use this control...

To do this...

Function

Generate Reuse modules

Check this to create a reuse design by assigning or re-assigning reuse properties to parts.

- If the design does not contain reuse modules, then a REUSE_ID property is added to all elements of a part and there's a unique REUSE_ID for each part. All parts packaged together into the same part have the same, unique REUSE_ID.
- If the design already contains reuse modules, then REUSE_PID values are generated during netlisting, and these values replace the REUSE_ID values.
- If this option is unchecked and there are parts with REUSE_PID properties, then the values revert to their previous REUSE_ID values.

Renumber design for using reuse modules

Check this to option to annotate the current design for reuse. If only this option is checked, then the design is annotated but no REUSE ID values are generated.

Note: By checking both of these options, you can generate a reuse design and annotate it at the same time. You must check at least one of these two options or Capture generates an error message.

Action

Incremental

If checked, Capture incrementally updates parts with a question mark in the part reference. For example, parts with reference designators or U?1 will be numbered U1A, U1B, U1C, and so on. Part reference and package information is not updated on existing parts.

Dialog Box Descriptions

Unconditional

If checked, Capture updates all parts in the selected schematic pages. Both part reference and package information may be updated on existing parts. Parts on different schematic pages are not packaged together.

Note: If a design references an external design more than once, when annotated the reference designators of the reference designs are updated but keep their assigned packages.

A reuse module root design can be annotated using the <u>Packaging tab</u> and checking the Unconditional reference update option. However, to be assured of avoiding duplicate part references in hierarchical reuse modules, you should check the Unconditional reference update option when you use the *PCB Editor Reuse* tab.

Note: If your design contains multiple heterogeneous parts that are the same part in a package, you will need to update their part references after every time you reset all part references to "U?" or group them using a grouping property.

Physical Packaging/Property Combine String

Specifies the properties that must match for Capture to group parts into a single package. Value and Source Library properties are the default property string, but you can use any combination you like.

Note: Do not use {GROUP} as a property string in combined property strings text box. This may cause problems while annotating your design for a PCB Editor tools, such as PCB Editor or OrCAD X Presto. The GROUP property is used in PCB Editor for a specific purpose.

Dialog Box Descriptions

For example, you might want to use Value and Voltage. Say your design uses both Tantalum capacitors and ceramic disk capacitors. First, you could assign ".01uF" to the part Value property for all the capacitors. Then, you could define a user property called "Voltage" for all capacitors in the design, and assign it the value "100V" or "25V" as appropriate. To annotate your design, type "{Value} {Voltage}" (without the quotation marks) in the Part Value property combine text box.

In this example, Capture groups the parts with "C?" as the part reference, and ".01uF" as the part value, but it separates the 100V Tantalum capacitors from the 25V ceramic disk capacitors.

Do not change the page number Specify whether to renumber the schematic pages as part of the part reference update process.

Select modules to mark for reuse

From this list of all possible reuse modules, check which reuse modules you want to include in your design. These designs represent the root schematics of external designs that are referenced hierarchically. Currently, in Capture, the reuse module of a schematic must be the root of a design, and the design must be referenced externally from the referencing design.

If you are simply storing circuitry in an external design, you do not have to select all the modules. However, if the external design has an accompanying PCB that has been laid out—and you want to reuse this PCB—then you have to make sure that you enable the check box for that design module. Schematics with REUSE NAME properties are checked by default. Any module not checked has its REUSE NAME properties removed.

A module is only listed once, even if it has been referenced multiple times. Checking the box corresponding to the module includes the design in the netlist as a reuse module for every case. When multiple modules are displayed, you must select all the modules for hierarchical renumbering to work successfully.

Dialog Box Descriptions

Reuse annotation works quite differently for design reuse than it does for Capture's standard algorithm. Any parts not contained in a reuse module will first be annotated with Capture's standard annotation tool. Then, any section of the design contained within a checked reused module gets updated.

Part packaging is not determined by the annotation tool. Rather, the packaging in the original design will be used in the referencing design. The "numbering" will likely be different but all parts packaged together also end up packaged together in the referencing design.

Additionally, the parts of multi-part per package devices still fill the same slots in the package. In other words, the suffixes of the reference designators on the original PCB are still valid when used in the new PCB.

Archive Project dialog box

To open this dialog

In the Project manager, choose Archive Project (see <u>Archive Project command</u>) from the File menu.

Use this control... To do this...

Dialog Box Descriptions

Library files

Note: Archive library files and related files located in the Library folder of the project manager. These files include library (*.OLB) files, simulation and synthesis (*.VHD) files, *.STL files, and *.SML files. The PSpice model libraries are archived as follows:

- Profile-level model libraries are archived under their respective profiles and referenced as .\library_name>.lib. For example, when a profile; AC containing a model library diode.lib is archived, the diode.lib is copied under the folder AC and the simulation settings is modified as: .\diode.lib.
- Design-level model libraries are archived under .\<design_name-pspicefiles>\<des ign_name>\library_name>.lib. For example, when a design called histo containing a model library bipolar.lib is archived, the model library bipolar.lib is copied under folder histo-pspicefiles\histo and the simulation settings is modified as: .\histo-pspicefiles\histo\bipola r.lib.
- ☐ In case of global-level model libraries:
 - a copy of model library is created under the existing <design_name>.lib, if it exists
 - O a new <design_name>.lib file is created and a copy of model library is added to the <design_name>lib and the simulation setting is modified as design-level library.

Include TestBench

Output files

Include any testbench which is part of the current project.

Archive output files generated by Capture tools.

For example, cross reference reports (*.XRF files), EDIF netlists (*.EDN files) and PSpice project .DAT files, would be archived.

Referenced projects

Recursively save any projects referenced from within the current project.

Dialog Box Descriptions

Archive directory Specify the drive and directory for the project to be

archived in.

Use the ... button to display the Select Directory dialog

box where you can locate and select a new drive,

directory, or both.

Create single archive file Activate the File name text box for specifying the name of

the compressed archive file.

File name Specify a name for the compressed archive file. The

default name is ctname-current date>.

Add more files Add more files and folders to be archived.

Browse for Specify whether you want to add more files or directories

to your archive.

Select the Directories option to add a directory or the

Files option to add more files to your archive.

Additional Files/Directories Specify files and directories you want to be archived.

Use the ... button to select the files and directories you

want to archive.

Attach Implementation dialog box

To open this dialog

Choose Attach Implementation

■ From the New Part Properties dialog box.

OR

■ From the <u>Edit Part Properties dialog box</u>

Use this control... To do this...

Dialog Box Descriptions

Implementation Type

Specify the type of implementation from one of the following:

Schematic View Indicates that the attached implementation is a schematic. Capture automatically generates the appropriate hierarchical pins for the hierarchical block based on the hierarchical ports.

VHDL Indicates that the attached implementation is a VHDL entity. Capture automatically generates the appropriate hierarchical pins for the hierarchical block based on the port declarations in the VHDL entity.

Verilog Indicates that the attached implementation is a Verilog model. Capture automatically generates the appropriate hierarchical pins for the hierarchical block based on the port declarations in the Verilog model.

EDIF Indicates that the attached implementation is an EDIF netlist. If your design includes EDIF implementations for hierarchical blocks, you must specify the hierarchical pins for the hierarchical block; Capture will not generate them from the EDIF netlist. Also, if your design includes EDIF implementations, you can simulate them, but you cannot compile or build them.

Project Indicates that the attached implementation is a Capture programmable logic project. You must specify the hierarchical pins for the hierarchical block; Capture will not generate them.

Attaching an implementation does not automatically add that file, project, or schematic folder to the project. You must specifically add the implementation to the project with the <u>Project command</u>.

Implementation

Specify the name of the attached object.

Implementation Path

Specify the path and name of the library or file where the

attached object is located.

Assign Power Pins dialog box

To open this dialog

Select a design or schematic folder in the Project manager and choose Assign Power Pins from the Tools menu.

OR

Dialog Box Descriptions

Select an object (or multiple objects) on a schematic page then from the right-click menu choose Assign Power Pins.

Use this control... To do this...

Source Displays the name of the part containing the power pin. This field

is read-only.

Pin No Displays the pin number of the power pin. This field is read-only.

Power Pins Displays the type of the power pin. For example, VCC, VDD,

GND. This field is read-only.

Power Names Displays the names of the power pin. For example, GND for a

ground pin. +15V for a VCC pin.

Power Pin check box Toggle the corresponding power pin as an NC pin

Associate PSpice Model dialog box

Associate PSpice Model dialog box appears when you have finished associating PSpice model to a part placed on the schematic page.

Use this control... To do this...

Update Current Apply the changes only to the selected part instance.

All Apply the changes to all instances of the selected part

that are from same source package having same model implementation. All the instances of the selected part in schematic root present in hierarchical blocks and part's

convert views will also get updated.

Cancel Return to schematic editor.

Advanced Annotation

To open this dialog

Select the Capture design file in the Project Manager, choose *Tools – Annotate*, and click Advanced Annotation.

Dialog Box Descriptions

Note: The Advanced Annotation option appears only in Capture Schematic designs, that is, the Capture designs that have occurrences.

,

Use this control	To do this
------------------	------------

Design Hierarchy

Select the Design Hierarchy option to select a design or

its pages to apply advanced annotation.

Property Block

Select a property from the drop-sown list to apply

advanced annotation on those objects that have the

selected property in their properties.

Reference Range for

Prefix Specify the Part Prefix

Instance Count Specify the number of instances that have the same

prefix

Start Specify the start of the Reference Range

End Specify the end of the Reference Range

Auto Fill Prefix Automatically fills all the prefixes that are present in the

design

Add Row Adds a row to define Prefix, Instance Count, Start, and

End

Delete Row Deletes a row that contains Prefix, Instance Count, Start,

and End

Delete All Delete all the rows that contains Prefix, Instance Count,

Start, and End

Apply Applies the changes made to the selected

Inherited Ranges

Specifies the inherited ranges from the top-level, such as

Capture design

Action

Dialog Box Descriptions

Incremental reference update If checked, Capture incrementally updates parts with a

> question mark in the part reference. For example, parts with reference designators of U?A will be numbered U1A, U1B, U1C, and so on. Part reference and package

information is not updated on existing parts.

Unconditional reference update If checked, Capture updates all parts in the selected

> schematic pages. Both part reference and package information may be updated on existing parts. Parts on different schematic pages are not packaged together.

Reset part references to "?" Specifies to reset all the part references to "U?"

Annotation Type Specifies the sequence in which the components on the

design are annotated.

The Annotation Sequence list contains three options that you can use to decide the sequence in which the objects on your design are annotated - Default, Left to Right &

Top to Bottom.

Annotation Scheme

Specify whether to perform annotation according to Annotate as per page ordering in the title blocks

numbers on the page numbers specified in the title

blocks of the schematic pages.

Dialog Box Descriptions

Combined Property String

Specifies the properties that must match for Capture to group parts into a single package. Value and Source Library properties are the default property string, but you can use any combination you like.

Note: Do not use {GROUP} as a property string in combined property strings text box. This may cause problems while annotating your design for a PCB Editor tools, such as PCB Editor or OrCAD X Presto. The GROUP property is used in PCB Editor for a specific purpose.

For example, you might want to use Value and Voltage. Say your design uses both Tantalum capacitors and ceramic disk capacitors. First, you could assign ".01uF" to the part Value property for all the capacitors. Then, you could define a user property called "Voltage" for all capacitors in the design, and assign it the value "100V" or "25V" as appropriate. To annotate your design, type "{Value} {Voltage}" (without the quotation marks) in the Part Value property combine text box.

In this example, Capture groups the parts with "C?" as the part reference, and ".01uF" as the part value, but it separates the 100V Tantalum capacitors from the 25V ceramic disk capacitors.

Additional from INI

Include Non-Primitive Parts

Specifies whether to annotate non-primitive parts or to reset non-primitive part references to "?". Select this option to avoid netlisting duplicate reference errors when you want to simulate a design or generate a new part.

Preserve Designator

Specifies that the designator information of a homogeneous part during unconditional or reset annotation is to be preserved.

Dialog Box Descriptions

Preserve User Assigned Valid References

Specifies that the user assigned reference designator during annotation is to be preserved.

You can explicitly mark a reference as user assigned by choosing User Assigned Flag – Set from the shortcut menu for the Reference property in the Property Editor or from the shortcut menu of a part in the schematic page. Any references changed using Property Editor, Schematic Editor, or during backannotation are marked as changed and are preserved on selecting this and the previous option.

Assign Tolerance

Using the Assign Tolerance window, you can assign tolerance to PSpice and PSpice Advanced Analysis projects, such as sensitivity analysis and monte carlo analysis, to various devices. The following device types are supported in the Assign Tolerance window:

- Discrete: Resistors, Capacitors, Inductors
- Sources: Voltage, Current DC values
- PSpice primitive model and subcircuit model parameters
- Controlled Sources Output

To open this dialog

Select PSpice - Advanced Analysis - Assign Tolerance in Capture.

Use this control... To do this...

Instance View tab

Dialog Box Descriptions

Global List, you can assign tolerance and

distribution type globally to all the designs. The following tolerance can be added in the global list: R, V, L, C, and I. Once tolerance is applied on the design using the Tolerance Analysis window, it will be saved in the PSpice.ini file in one of the following parameter: GLOBAL_TOLERANCE_R, GLOBAL_TOLERANCE_V, GLOBAL_TOLERANCE_L, GLOBAL_TOLERANCE_C,

and GLOBAL TOLERANCE I.

Instance List Using Instance List, you can assign tolerance and

distribution type to each instance of various devices. For example, if there 3 resistors, 2 capacitors, and 2 diodes, you can see that each instance is listed in the instance

list.

SubcktList Using SubcktList, you can assign tolerance and

distribution type to each of subckt.

Model View tab

In the Model View tab, you can view all the models that are used in the design. The left-hand side of the Assign Tolerance window in the Model View tab displays all the models used in the current design. The right-hand side of the tolerance window displays model parameters.

Edit PSpice Model The Edit PSpice Model allows you to edit the selected

PSpice model using Model Editor. Once changes are done and saved in Model Editor, they get reflected in the

Assign Tolerance window immediately.

Note that only one instance of Model Editor can used at one time. Only tolerance and distribution type can be

added using the Model Editor.

Name Specifies the device's value name or model's

parameters' name.

Value Specifies the device's value or various model's

parameter's value.

PosTol Specify the positive tolerance. The tolerance should be

specified as percents or absolute numbers only, such as

<Positivetolerance>% or <Positivetolerance>.

Dialog Box Descriptions

NegTol Specify the negative tolerance. The tolerance should be

specified as percents or absolute numbers only, such as,

<Negativetolerance>% or <Negativetolerance>.

Distribution Specify the distribution type. The following distribution

types can be specified: FLAT and GAUSS. To specify the distribution type in the model editor, use the following

syntax: DEV/<Distribution Type>. For example,

DEV/GAUSS or DEV/FLAT.

Backannotate dialog box

Use this dialog box to back annotate PCB information from the board layout tool to Capture. Use the PCB Editor tab to backannotate from PCB Editor.

To open this dialog

Select a design (.DSN) in the project manager then choose *Back Annotate* (see <u>Backannotate command</u>) from the *Tools* menu.

Note: To back annotate information from PCB Editor to Capture, refer to the <u>Physical Layout</u> in PCB Editor workflow.

Use this control	To do this
Generate Feedback Files	Select to generate feedback files. Selected by default.
Setup	Click to open the Setup dialog box that lets you specify a configuration (.cfg) file, edit it, and set the numbers of versions for the file. You can also set up the character limit for nets, devices, and pins as well as suppress warning or ignore electrical constraints.
PCB Editor Board File	Specify the board file. By default the board file is allegro\ <dsn name="">.brd.</dsn>
Netlist Directory	Specify the netlist directory. By default, the directory is allegro.
Output File	Specify the name and location of the swap (.SWP) file. By default, the file is allegro $\colon 2000$ name>.swp.
	For more information about swap files, see <u>Designating</u> pins, gates, or packages for swapping.
Back Annotation	
Update Schematic	Select to update the schematic with changes. Selected by default.
View Output (.SWP) File	Select to view the output or swap file. Not selected by default.
Constraints Options	The options in this group are available for the Constraint-enabled flow.

Overwite Select to overwrite all DSN constraints by constraints

from PCB Editor. Not selected by default.

Changes Only Select to overwrite only those DSN constraints that are

changed in PCB Editor. Selected by default.

Show Constraints Difference

Report

Select to show a report of the differences in constraints

between the schematic and layout editors.

Bill of Materials dialog box

To open this dialog

Choose Bill of Materials (see <u>Bill of Materials command</u>) from the Tools menu.

Use this control... To do this...

Scope Select the scope of the bill of materials. The scope can

cover the entire design, or the selected schematic folders

and pages.

Mode Include either instances or occurrences. Capture

automatically sets this option based on the project type. FPGA and PSpice projects default to instances, while PCB and Schematic projects default to occurrences.

Line Item Definition

Header Specify a header that Capture inserts on each page. If

this is left blank, Capture assumes there is no header.

For example, "Item\tQuantity\tPart" creates a header that

displays column entries of "Item", "Quantity", and "Part"

each separated by a tab character.

Combined property string Specify the properties that must match for Capture to

group them in the bill of materials. Typically, this text box should be set to "{Value}" (without the quotation marks).

To insert a tab, use the \t character sequence. For example, "{Reference}\t{Value}" prints a part's reference,

a tab character, and the part's value.

To create separate listings for 100V and 25V .01uF capacitors, for example, set the Part Value combined property string to "{Value} {Voltage}" (without the quotation marks), where Voltage is a user property in which you store the appropriate voltage values.

Place each part entry on a separate line

Specify that each part entry appears on a separate line in the bill of materials report file. When this option is selected, the quantity of parts sharing the same Part Value appear on one line, then each part is listed below. When this option is not selected, all the parts with the same Part Value are listed on one line.

Include File

Merge an include file with report Specify whether to merge an include file with the report.

For more information about include files, see Creating an

update file.

Combined property string Specify a lookup string to match in the include file.

Include file Specify the path and name of the include file.

Report

Report File Specify the bill of material's output file. For an example of

a bill of materials report file, see Creating a bill of

materials.

View Output Open the bill of materials report file in a text editor.

Browse Display a standard Windows dialog box for selecting files.

Browse File dialog box

The Browse File dialog box appears when you choose the Browse button from any dialog box with a Browse button.

Use this control... To do this...

Look in Specifies the drive and directory to locate the file in.

File name Specifies the name of the file to look for.

Files of type Specifies the type of files to look for.

Open as read-only Specifies to open the file as a read-only file.

Color dialog box

The Color dialog box appears when you click on a color in the Colors tab in the <u>Preferences dialog box</u>.

Use this control... To do this...

Basic colors Shows the color of the object selected in the Colors tab.

To change the color, click the left mouse button on a

different color and then click OK.

Custom colors This feature is disabled in Capture.

Define custom colors

This feature is disabled in Capture.

Copy from occurrence

The Copy from occurrence dialog box appears when you copy a part which has multiple occurrences.

Use this control... To do this...

Destination Destination column lists the multiple occurrences of the

block or design where the copied part is placed.

Source Source shows drop-down lists containing the various

occurrences of the copied part.

Create Differential Pair dialog box

To open this dialog

In the Project manager, choose Create Differential Pair (see <u>Create Differential Pair command</u>) from the Tools menu.

Use this control... To do this...

Net/Differential Pair View all the flat nets or differential pairs defined in a

design.

Select the nets between which differential pair needs to All Nets/Diff Pair grid

be created.

Filter Specify the nets you want to view in the All Nets grid.

> **Note:** To view nets of a particular type, specify the initial letters of the net in the Filter text box. All the nets of that particular type will appear in the All Nets grid. For example, if you want to view all nets starting with the letter "A", then enter "A" in the Filter text box. All the nets starting with letter "A" will appear in the All Nets grid.

Move the selected nets from the All Nets grid to the Selections grid.



Double-click the selected net to move it to the Selections grid.



You can use the CTRL or SHIFT keys to move multiple nets to the Selections grid.

Remove the nets from the Selections grid.

Tip

Double-click the selected net to remove it from the Selections grid.

Specify a name for the differential pair.

Note: If the nets forming a differential pair are of the type DP+ and DP-, the name of the differential pair is set to DP. For other pairs of nets, the name of the differential pair is of the type DPn.

View the nets and the differential pair associated with those nets.

Create a differential pair between the nets displayed in the Selections grid.

[>]

[<]

Diff Pair Name

Selections grid

Create

Modify Change the differential pair name selected in the

Selections grid.

Delete the selected differential pair selected from the

design.

Auto Setup Open the <u>Differential Pair Automatic Setup dialog box</u> to

create multiple differential pairs simultaneously.

Close the Create Differential Pair dialog box.

Note: An Auto Differential pair can also be created for a bus. To do so, you need to put _n_ & _p_ as prefix and the Auto command creates differential pairs for all bits in the bus.

Note: In case of Flat Designs, DIFFERENTIAL_PAIR properties added through this dialog will be added on Schematic nets rather than flat nets.

Create Directory dialog box

To open this dialog

Click the Create Dir button in the Select Directory dialog box.

Use this control... To do this...

Current Directory Shows the current directory. The new directory will be a

subdirectory to the current directory.

Name Specify the name of the new directory to be created

below the current directory.

Create Netlist dialog box

To open this dialog

In the project manager, choose *Create Netlist* (see <u>Create Netlist command</u>) from the *Tools* menu.

Use this tab	To do this
PCB	Create the three files associated with PCB Editor netlist. For more information, see <u>PCB tab</u> .
EDIF 2 0 0	Create an EDIF hierarchical netlist. It can include net, part, or pin properties. For more information, see <u>EDIF 2</u> <u>0 0 tab</u> .
INF	Create a netlist for use with OrCAD X Digital Simulation Tools 386+. For more information, see <u>INF tab</u> .
PSpice	Create a PSpice netlist that you want to examine or modify before running a simulation, or to create a subcircuit netlist. For more information, see <u>PSpice tab</u> .
	Important
	Before generating a PSpice netlist make sure that your design contains a PSpice ground symbol (0). Otherwise, you will not be able to use the netlist for running PSpice analog simulation on the design. For information on how to place PSpice ground symbol in your design, see Placing power, ground, and no connect symbols.
SPICE	Create a Spice hierarchical netlist. It can include net, part, or pin properties. For more information, see <u>SPICE</u> <u>tab</u> .
VHDL	Create a 1076-87 or 1076-93 VHDL netlist. For more information, see <u>VHDL tab</u> .
Verilog	Create a Verilog netlist. For more information, see <u>Verilog</u> <u>tab</u> .
Other	Create an EDIF or Spice flat netlist of a simple hierarchy or a netlist using a format not represented on the other tabs. For more information, see Other tab .

PCB tab

Open this dialog box by selecting the .DSN file and choosing Create Netlist from the Tools menu.

Before generating a PCB Editor netlist, you should complete the design by assigning properties, annotating, and running a Design Rules Check (DRC). Assigning appropriate PCB Editor properties, such as PCB Footprint, is a key part of successful netlisting.

The following rules apply to Capture elements you set up for netlisting.

- 1. Net names should not exceed 255 characters and the part name itself should not exceed 255 characters either.
- 2. The part name is made up by combining the values of the Source Package, PCB footprint, and other component definition properties found in the [ComponentDefinitionProps] section of the configuration file. The values are concatenated, separated by an underscore character.

For a legacy designs, if there is DEVICE property on parts, then part name is made of DEVICE property value. For such cases, consider the following rules:

- □ Do not assign the same device value for two components having different component definition properties. Else, an error is generated.
- □ Ensure that there are no conflicts due to power pin visibility.
- Do not assign the DEVICE property value equal to the design name or schematics (root schematic or any schematic) name. Else, netrev will generate the "object not found" error.
- **3.** There are a few illegal characters which the netlister does not allow. The 'character (single quotation mark) is not allowed in net, pin, or part names. Also, the ! (bang) character is not allowed in net names. Similarly, the @ character should not be used while naming library parts used for the PCB Editor.

Where there is an illegal character, it is substituted with an _ (underscore) character. You are warned if the name has been changed for any reason. There are a few exceptions: A! (bang) character in net names is a fatal error. However, the \ (backslash) character in net names is not substituted because it is legal.

Note: Both the backslash (\setminus) and underscore ($_$) characters in net names interfere with cross probing.

4. To generate unique net and physical part names, the name is truncated to 255 characters. If the name is not unique, the netlister generates a unique name by

appending _1 (underscore plus the character 1). This digit is incremented until a unique name is formed. The length is always maintained within 31 characters.

- **5.** To exclude PSpice specific parts from a PCB Editor netlisting, you need to set their PSpiceOnly property to TRUE. In this case, no error is thrown for missing or zero pin numbers while netlisting.
- **6.** To exclude a physical part from a PCB Editor netlisting, you need to set NETLIST_IGNORE property to TRUE.

Note: You can check the Capture session log for netlisting details and to verify that netlisting proceeded as you expected.

Note: During netlisting, multi-section, heterogeneous parts are treated as single-section parts.

Note: Both OrCAD X Capture and Allegro PCB router products use .DSN as the extension for their design files. Keep the two different file types in separate directories to avoid the possibility of one file overwriting the other.

Note: Except for occurrence properties, the schematics of externally-referenced libraries and designs should not be edited. You should view them as read-only designs. Trying to edit, then save, these designs from within your schematic can introduce errors such as duplicate reference designators and other problems. Since Capture saves your design before netlisting, you might notice instance properties in externally-referenced designs do not get updates.

When saving schematics with externally-referenced libraries or designs, occurrence properties are saved but altered instance values are not. If you want to change externally-referenced libraries or designs you should first close the referencing design. Then, open the referenced library or design, make the necessary changes, and save and close the referenced library or design. At this point, you can reopen the original design and reference the modified design. To learn more about preparing your design for netlisting, see additional topics on pin swapping and no-connect pins. Here are the options available in the *PCB* tab of the *Create Netlist* dialog box:

Use this control...

To do this...

PCB Footprint

Specify a property name for PCB footprint using combined property string. The default property name is PCB Footprint. You can use the combined property string to pass user-defined properties as PCB Footprint property for PCB Netlist generation. This gives you the flexibility of defining a user-defined PCB Footprint property specifically for the PCB Editor flow. As a result, you can define different PCB Footprint properties for different PCB flows.

Setup button

Click this button to open the Setup dialog box where you can specify, edit, and view a configuration file. This file contains a list of properties available for mapping between Capture and PCB Editor. You can also specify the number of backup versions to be maintained for the PST*.DAT netlist files.

Create PCB Editor Netlist

Select this check box to generate a netlist in PCB Editor format which consist of the PSTCHIP.DAT, PSTXNET.DAT, and PSTXPRT.DAT files. This check box is selected by default. Selecting ensures the three PST*.DAT files are found in the project manager when the netlisting is complete, or in the directory you designate for the Netlist Files Directory. If this check box is cleared, no netlisting takes place and the Options below this check box are unavailable. For a Constraint Manager-enabled design, it creates a zip file, pstdedb.cdsz instead of these three PST files.

Netlist Files Directory

Location where the PST*.DAT files are to be saved. The default location is the netlist directory of the board on which an operation was done last time.

- If this is the first time the design is being netlisted, the default location will be the allegro subfolder in your design directory.
- If the netlist files have been generated previously for the project, the default is last directory used with this dialog box for a design.

View Output

Select this check box to automatically open the three PST*.DAT netlist files to be displayed in separate Capture windows for viewing and editing after netlisting is completed. The default for this option is to leave it cleared.

- If . DAT files are registered to Capture, they will open in Capture. If not, they will open in whatever program they are registered to, such as Notepad or WordPad.
- When this check box is unchecked, the PST*.DAT files will not be opened automatically, but they can be found in the project manager and in the directory specified by Netlist Files Directory.

EDIF 2 0 0 tab

Capture provides two EDIF netlist formats. The first format, provided in this tab, produces either hierarchical or flat netlist output, depending on your design structure and the active mode. The second format produces only flat netlists, and is accessible through the <u>Other tab</u> in the Create Netlist dialog box.

Capture manages the hierarchy by turning pages in the schematic folder into CELLs in the main LIBRARY. These cells can then be referred to by INSTANCE where needed. Because EDIF requires a define-before-use philosophy, the hierarchy appears to be inverted in the netlist (the root schematic page is the last CELL in the main LIBRARY).

Use this control	To do this
Part Value	Specify the value for the Part Value in the netlist, using a combined property string. Most Part Values are specified using the following combined property string:
	{Value}
PCB Footprint	Specify the value for the PCB Footprint in the netlist, using a combined property string. Most PCB Footprints are specified using the following combined property string:
	{PCB Footprint}

Options

Allow non-EDIF characters Create a netlist for PCB 386+ or another EDIF reader

that allows non-EDIF characters.

If you select this option, the EDIF formatter does not

check for legal EDIF characters.

Output pin names (instead of pin

numbers)

Use pin names instead of pin numbers in the netlist. Most EDIF readers expect pin names instead of pin numbers.

Do not select this option to create a netlist for PC Board

Layout Tools 386+.

Do not create "external" library

declaration

Create a netlist file without an EDIF external statement in

the netlist file.

If you do not select this option, Capture uses external statements to identify OrCAD X as the source of the library parts in the netlist, but some EDIF readers do not

accept external statements.

netlist. Some EDIF readers require designator

constructs. Use this option if your reader requires them.

Output net properties Output net properties in addition to the normal netlist.

Output part properties Output part properties in addition to the normal netlist.

Output pin properties Output pin properties in addition to the normal netlist.

Output buses as scalars Output buses as bits in the netlist.

Netlist File Specify the drive and directory for the netlist file.

View Output Display the generated netlist in an editor.

Before you can use this option effectively, you must associate the netlist file type with an editor using the Windows Explorer. For example, if you wanted to view a VHDL netlist in Notepad using this option, you would need to associate *.VHD files with Notepad in the

Windows File Explorer.

INF tab

This format file produces .INF files for use with OrCAD X Digital Simulation Tools 386+. See the *Digital Simulation Tools User Guide* for details.

If you attach a file to any nonprimitive part or hierarchical block, Capture treats the file as a schematic folder external to the design. When this formatter uses such an external file, it won't generate the netlist for the child .INF file. Instead, Capture assumes that this file will be supplied by someone else.

The VST netlist formatter truncates the names of child schematic folders in the hierarchical design. If the schematic folder names are too long (more than eight characters) and they match, the netlist formatter won't descend into the child schematic folder or create the netlist of the part. If you restrict the names of child schematic folders to eight characters, the netlist formatter creates .INF files for each child schematic folder as expected.

Use this control	To do this
Part Value	Specify the value for the Part Value in the netlist, using a combined property string. Most Part Values are specified using the following combined property string:
	{Value}
Netlist File	Specify the drive and directory for the netlist file.
View Output	Display the generated netlist in an editor.
	Before you can use this option effectively, you must associate the netlist file type with an editor using the Windows Explorer. For example, if you wanted to view a VHDL netlist in Notepad using this option, you would need to associate *.VHD files with Notepad in the File Explorer.

Other tab

for this Netlist format	choose this .dll
Accel	oraccel64.dll
Algorex	oralgorex64.dll
AlterADF	orAlteraad64.dll
AppliconBRAVO	orApplbrav64.dll
AppliconLEAP	orApplleap64.dll

Cadnetix orCadnetix64.dll
Calay orCalay64.dll

Calay90 orCalay9064.dll

Case orCase64.dll
CBDS orCbds64.dll

ComputerVision orCompvisn64.dll

Dump orDump64.dll eDIF orEdif64.dll

EEDesigner orEedesign64.dll

FutureNet orFuture64.dll

HiLo orHilo64.dll

IntelADF orInteladf64.dll

Intergraph orIntergra64.dll

MultiWire orMultiwir64.dll

OHDL orOhdlnet64.dll

PADS 2000 orpads2k64.dll

PADS-PCB orPadspcb64.dll

PCAD orPcad64.dll

PCADnlt orPcadnlt64.dll

PCBII orPcbii64.dll

PDUMP orPdump64.dll

PLDnet orPldnet64.dll

PROTEL2 orprotel264.dll

RacalRedac orracalred64.dll

RINF orRinf64.dll

Scicards orScicards64.dll

SPICE orSpice64.dll

Tango orTango64.dll

Telesis orTelesis64.dll

Vectron orVectron64.dll

VST Model orVstmodel64.dll

WinBoard orwinboard64.dll

WireList orWirelist64.dll

Note: Specify a configuration file that contains the properties to be transferred to the <code>.ONL</code> file. The <code>.ONL</code> file is an intermediate file for all netlists in the <code>Other</code> tab. That is, the tool creates a <code>.ONL</code> file and this file is referred while creating the netlists. Note that all properties in the <code>.ONL</code> file might not be present in the netlists because some of the properties might be ignored.

Configuration file (sample format):

[ENABLESWAPDETAILS]

Swap=YES

[LIBRARY]

Part Reference=NO

Implementation Type=NO

Reference=NO

Name=NO

Pin Numbers Visible=NO Pin Names Visible=NO

Pin Names Rotate=NO

Number=NO

Type=NO

Long=NO

Clock=NO

Dot=NO

Order=NO

Is NO Connect=NO

Swap Id=NO

Net Name=NO

SDTSourceLibName=NO

[NET]

Name=NO

Number=NO

Swap Id=NO

Type=NO

Net Name=NO

Is Global=NO

The above format contains the following sections:

- [LIBRARY] Specify the part properties that you want to be transferred to the .ONL file in this section and set the property to YES. The property will be transferred to the .ONL file. If you do not want to transfer the property to the .ONL file, then set the property to NO.
- [NET] Specify the Net properties to be transferred to the .ONL file in the [NET] section and set it to YES.
- [ENABLESWAPDETAILS] This section contains a property named SWAP. The Default value of this property is NO. Setting this property to YES will check the homogeneity of package and add the property [homogeneous = TRUE] to the .ONL file if the package is homogeneous. Also, it will add two occurrence properties Part Reference and timestamp and create the occurrence section. Timestamp is the ID value in octal format.

PSpice tab

When generating a PSpice netlist, you can choose between two types of netlist formats:

Flat netlist

or

Hierarchical netlist

Use the PSpice tab on the *Create Netlist* dialog box to generate a customized PSpice netlist using the options described below.

Use this control... To do this...

Create Hierarchical Format Netlist

Check this box if you want to create a hierarchical netlist; leave the box unchecked for a flat netlist.

A flat netlist is generated for all levels of hierarchy, starting from the top, regardless of whether you are pushed into any level of the hierarchy. Flat netlists are most commonly used as input to PCB layout tools. The flat simulation netlist format for PSpice contains device entries for all parts on a subcircuit (child) schematic multiple times, once for each instance of the hierarchical part or block used.

The hierarchical netlist preserves the hierarchical information in any subcircuit (child) schematics. It contains a single .SUBCKT definition for each child schematic. The devices in the subcircuit are therefore netlisted only once. Each instance of the hierarchical part or block is then netlisted as an instance of that subcircuit (as an "X" device). The subcircuit name corresponds to the name of the subcircuit (child) schematic.

Note: The .SUBCKT arguments nodes, parameters, and optional nodes do not have a maximum limit.

Hierarchical netlists are especially useful to IC designers who want to perform Layout vs. Schematic (LVS) verification because they are more accurate descriptions of the true circuit. You can customize the hierarchical PSpice or LVS netlist by specifying various options in the Hierarchical PSpice Netlist Settings dialog box dialog box. To reach this dialog box, click the Settings button in the PSpice tab of the Create Netlist dialog box.

Settings

Create SubCircuit Format Netlist A subcircuit netlist cannot be simulated directly. Rather, it is a definition of a circuit—a model—that can be called by another circuit being simulated. Use one of the following options to specify how to generate netlists for subcircuits:

- **Descend** generates a definition of a hierarchical design that includes the top level circuit as well as its subcircuits. (This option is only available if Create Subcircuit Format Netlist is enabled.) If the Create Hierarchical Format Netlist is not checked, then this option combination is equivalent to creating a flat netlist.
- Do Not Descend generates a definition of a hierarchical design that includes only the top level circuit, without any of its subcircuits. (This option is only available if Create Hierarchical Format Netlist and Create Subcircuit Format Netlist are enabled.)

Use Template

You can select an alternate template option to define which netlisting template property to use. The template applies to both flat and hierarchical netlists. You can specify a particular netlist template for generating netlists used by other simulation tools or for creating alternate PSpice netlists that contain different part descriptions.

In OrCAD X Capture, the template property specifies how primitive parts are described in the simulation netlist. A template defines the pin order and identifies which part property values to include in the netlist. A part must have a template property to be included in the simulation. (The default template is PSPICETEMPLATE.

Place DRC markers for Errors and Warnings

When selected, this option causes DRC markers to be placed on devices and pins that cause errors in the netlist and would prevent proper simulation.

After creating a PSpice netlist, you can point to Browse on the Edit menu and choose DRC Markers to create a browse spreadsheet. When you double-click a line in the browse spreadsheet, it takes you to the erroneous part in a schematic.

Note: The netlister does not catch all errors. The PSpice Simulator will catch some errors that the netlister misses.

Use the Design Rules Check dialog box to clear the DRC markers for each successive run of the PSpice netlister. Choose *Design Rules Check* on the *PCB* menu and select Delete DRC Markers in the *DRC Action* section on the *Options* tab.

Netlist File

This is the pathname to the file you want to use for storing your netlist.

Compatibility Mode (16.2 and Prior Releases)

In release prior to 16.2, if two nets of the same name are placed on different pages of a design, the nets, in the PSpice netlist, are shorted together.

However, in all subsequent releases, the two nets are assigned unique net names in the PSpice netlist. This causes the two nets not to be shorted together.

Choose this option to create a PSpice netlist with release 16.2 and prior release functionality.

View Output

Select this option if you want the .NET file to open automatically after the netlist is generated. The .NET file is stored in the Outputs folder of the project manager.

/Important

Before generating a PSpice netlist make sure that your design contains a PSpice ground symbol (0). Otherwise, you will not be able to use the netlist for running PSpice analog simulation on the design.

SPICE tab

Use this control... To do this...

Part Value Specify the value for the Part Value in the netlist, using a

combined property string. Most Part Values are specified

using the following combined property string:

{Value}

Options

Include unconnected pins
If you select this option, Capture assigns node numbers

to all unconnected pins. Node numbers for unconnected

pins begin at 32767 and decrease in value.

If you do not select this option and there are

unconnected pins on your schematic page, they are assigned a space character and Capture displays a

warning.

Use net names If you select this option, Capture uses the node names

you placed on the schematic page (via aliases and hierarchical ports) where available. Not all versions of SPICE support alphanumeric node names. Check your SPICE manual for details. If your version of SPICE does not allow alphanumeric node names, you can still give them numeric names such as "17." These numeric names do not interfere with the ones generated by Capture, since the node numbers it generates begin at

10000 (except GND, which is always 0).

PCB Footprint Specifies the value for the PCB Footprint in the netlist,

using a combined property string. Most PCB Footprints are specified using the following combined property

string:

{PCB Footprint}

View Output Specifies to display the generated netlist in an editor.

Before you can use this option effectively, you must associate the netlist file type with an editor using the Windows Explorer. For example, if you wanted to view a VHDL netlist in Notepad using this option, you would need to associate *.VHD file with Notepad in the File

Explorer.

Verilog tab

Use this control... To do this...

Part Value Specify the value for the Part Value in the netlist, using a

combined property string. Most Part Values are specified

using the following combined property string:

{Value}

Timescale Specify the basic time unit (nano- or pico-second) used

for simulation of the netlist. To specify the time unit, use

this syntax:

`timescale X time_unit/Y precision_unit

where

X, Y = 1, 10, or 100 time unit, precision unit = s, ms, us, ns, ps, or fs

So, for example, a setting of `timescale 1ns/10 ps indicates that delays for the netlist are 1 ns duration with

2 decimal points of precision (since 1 ps = .01 ns).

By default, Capture uses a timescale value of 1ns/1ps. If you specifically do not set a timescale for the netlist (that is, if you intentionally leave the field blank), the default timescale for your simulator is used and the timescale

directive does not appear in the netlist.

Net Type Set the default net type of all the wires in the design. By

default, this value is set to "wire" (for standard logic), but you can choose "tri," "tri1," "wand," "triand," "tri0," "wor,"

or "trior."

Text case for Pin/Module names Specify one of three options:

- Lower Case all pin names are converted to lower case in the netlist.
- Upper Case all pin names are converted to upper case in the netlist.

Note: Net names are always converted to upper case in the netlist.

■ User Property for Case - pin name cases are determined via the use of a property, Vlog_Uppercase, which must be assigned to a component. This is the default setting. If Vlog_Uppercase has a value of "TRUE" all pin names are converted to upper case for that component. If the value is "FALSE" all pin names are converted to lower case for that component.

All pin names in your design must have consistent case designations. That is, they must all be either upper case or lower case. If you have mixed cases for pin names (for example, if you have specified Vlog_Uppercase as "TRUE" for some components and "FALSE" for others), Capture will notify you with an error message when you attempt to create a Verilog netlist.

Specify the drive and directory for the netlist file, as well as the netlist file name.

Display the generated netlist in Capture's Verilog editor.

Include all power pins connected to the same named signal in the netlist. By default, only visible power pins appear in the netlist.

Netlist File

View Output

Include Power Pins

VHDL tab

Use this control... To do this...

Part Value

Specify the value for the Part Value in the netlist, using a combined property string. Most Part Values are specified using the following combined property string:

{Value}

VHDL Standard

Choose a standard.

■ The 1076-87 limits legal characters for node names to:

with the following limitations:

- The first character is limited to: A..Z a..z
- The last character is restricted from: _ (underscore)
- The 1076-93 VHDL standard permits special characters, VHDL reserved words, and names that begin with digits. To do so, delimit the name with backslashes (\) and precede any special characters—including "internal" backslashes (not the delimiters)—with a backslash.

Options

Entity Architecture Header

Signal Type

Output net properties

Output part properties

Output buses as scalars

Netlist File

Specify default procedures which appear at the beginning of the netlist output file.

Specify a signal type anywhere a signal needs to be defined with a type.

Output net properties in addition to the normal netlist.

Output part properties in addition to the normal netlist.

Write all buses as individual scalar ports when creating the netlist. This option is useful if you have a mismatch of bus and scalar pins in the hierarchy of your design.

Specify the drive and directory for the netlist file.

View Output Display the generated netlist in an editor.

Before you can use this option effectively, you must associate the netlist file type with an editor using the Windows Explorer. For example, if you wanted to view a VHDL netlist in Notepad using this option, you would need to associate *.VHD files with Notepad in the File

Explorer.

Create Pin Pairs dialog box

Use this dialog box to create a pin-pair for electrical constraints such as PROPAGATION DELAY and RELATIVE ROPAGATION DELAY.

To open this dialog

In the Propagation Delay or Relative Propagation Delay dialog boxes

Click the Add Pin Pair button.

OR

Press ALT+A.

Use this control... To do this...

First pin Select the first pin for the pin-pair.

Second pin Select the second pin for the pin-pair.

Note: You cannot select the same pin in both columns.

Apply Create a pin pair without closing the dialog box.

OK Accept the changes and close the dialog box.



You can use the following methods to select multiple consecutive pins in the Create Pin Pairs dialog box:

- Using SHIFT+Down Arrow keys
- □ Using SHIFT+Left mouse button click

 Dragging the mouse pointer diagonally across the pins appearing in the combo box to select them

Similarly, you can use the CTRL+Left mouse button click to select multiple nonconsecutive pins in the Create Pin Pairs dialog box.

Create PSpice Project dialog box

To open this dialog

Choose the *OK* button on the New Project dialog box after selecting the *Enable PSpice Simulation* check box.

You need to select one of two options in this dialog box.

Use this control...

To do this...

Create based upon an existing project

When you select this option, you indicate that you want to use an existing Capture project file (.OPJ) as an initial starting point for an analog or mixed signal project.

If you select this option, you need to also select a project file, using either the drop-down menu or the Browse button to the right.

After selecting this option and choosing the OK button, a new project appears. This new project is identical to the existing project you previously selected in the following respects:

- It has the same name.
- It contains the same configured libraries and designs.
- It contains renamed copies of simulation profiles, local simulation files, model libraries, include files, and marker files (.MRK).

Create a blank project

By selecting this option and choosing the OK button, you create a new project that is capable of being simulated in PSpice AD.

Cross Reference Parts dialog box

To open this dialog

In the Project manager, choose Cross Reference (see Cross Reference command) from the Tools menu.

Use this control... To do this...

Scope Specify whether to cross-reference the entire design or

just the selected schematic page or pages.

Mode Include either instances or occurrences. Capture

automatically sets this option based on the project type. FPGA and PSpice projects default to instances, while PCB and Schematic projects default to occurrences.

Sorting Specify whether to sort output by part value or reference

designator first.

Report

Report the X and Y coordinates I

of all parts

Include the X and Y coordinates of all parts in the

cross-reference report file.

Report unused parts in multiple

part packages

Include unused parts in multiple-part packages in the

cross-reference report file.

Report file Specify the path and file name for the report. For an

example of a cross-reference report file, see Creating a

cross reference report.

Save as XRF / Save as CSV Specify the output file type XRF (default) or CSV.

View Output Open the cross-reference report file in a text editor.

Browse Display a standard Windows dialog box for selecting files.

Custom Design Rule Check (DRC)

To create Custom DRC in the Rules Setup tab of the Design Rules Check dialog box.

Use this control...

To do this...

Custom DRC in the Rules Setup tab

Lists the custom DRCs included for electrical and physical rules.

To include a DRC in this section, do the following:

- 1. Create a TCL file and add entry in pkgIndex.tcl in any folder under tclscripts (located at <installation_directory>\tools\capture)
- Add the following methods in the TCL file with any desired namespace and modify according to the DRC.

```
namespace eval ::< YOUR NAMESPACE > {
set scriptDir [file dirname [info
script]]
}
proc ::<YOUR NAMESPACE>::<METHOD1>{
args } {
set lScope [lindex $args 0 0]
set lMode [lindex $args 0 1]
set lCreateDrcMarkers [lindex $args 0
21
set lLogFilePath [lindex $args 0 3]
capCustomDRC::capSetCreateMarker
$1CreateDrcMarkers
set lMessage "\ Running < YOUR
NAMESPACE>::<METHOD1> \n\n"
# Setting the Variables for logging
capCustomDRC::capSetLogFilePath
$1LogFilePath
capCustomDRC::capCustomDrcLog
$1Message
```

```
capProcessDRC::capProcessSelection
"<YOUR NAMESPACE>" $1Scope $1Mode
}
proc ::<YOUR NAMESPACE>::<METHOD2>{} {
set lDrcName "<DRC NAME as any text
string>"
set lProc " <YOUR
NAMESPACE>::<METHOD1>"
set lIsExecute
[capCustomDRC::capCustomElectricalDrcF
indExecutableStatus $1Proc]
set 10ptional
[DboTclHelper sMakeStdVector]
DboTclHelper sPushBackToVector
$10ptional "Type"
DboTclHelper sPushBackToVector
$10ptional "Electrical" #valid values =
Electrical/Physical
DboTclHelper sPushBackToVector
$10ptional "Description"
DboTclHelper sPushBackToVector
$10ptional "<Any Description>"
DboTclHelper sPushBackToVector
$10ptional "FilePath"
set lFilePath [file join $::< YOUR
NAMESPACE >::scriptDir <TCL FILE NAME>]
DboTclHelper sPushBackToVector
$10ptional $1FilePath
set lReturn
[CapCustomDRCElectricalAddItem
$1DrcName $1IsExecute $1Proc
$10ptional]
return lReturn
```

3. Now define functions, which are called from capProcessDRC.tcl, according to your requirements. As these functions are called in catch statements, the undefined functions are ignored. For Example:

```
#proc ::<YOUR
NAMESPACE>::capProcess<ObjectType> {
pObject } { # e.g.

proc ::capHangingWires::capProcessWire
{ pWire } {
  set lsearchIndex [lsearch
$::capHangingWires::WireList $pWire]
  if { $lsearchIndex == -1 } {
    capHangingWires::capProcessWireObtaine
  d $pWire
  lappend ::capHangingWires::WireList
$pWire
}
}
```

4. Following are the callback functions which can be defined by the user as per the requirement.

```
<YOUR
NAMESPACE>::capProcessSelectionStart{<
DESIGN >}
# Function will be called once at the start of the process indicates s
<YOUR
NAMESPACE>::capProcessSelectionEnd{<DE
SIGN OBJECT>}
# Function will be called once at the end of the process
```

```
<YOUR YOUR
NAMESPACE>::capProcessPageStart {<PAGE
OBJECT> }
# Function will be called before
processing of a page start.
<YOUR NAMESPACE>::capProcessPageEnd
{ < PAGE OBJECT > }
# Function will be called after
processing of a page gets completed.
<YOUR NAMESPACE>:: capProcessWire {
<WIRE OBJECT> }
# Function will be called when a wire is
encountered while processing page
Objects.
<YOUR NAMESPACE>:: capProcessGlobals
{<GLOBAL OBJECT>}
# Function will be called when a global
is encountered while processing page
Objects.
<YOUR NAMESPACE>::capProcessPorts
{<PORT OBJECT>}
# Function will be called when a port is
encountered while processing page
Objects.
<YOUR NAMESPACE>
:: capProcessOffPageConnector
{<OFFPAGECONNETOTR OBJECT>}
# Function will be called when a offPage
is encountered while processing page
Objects.
<YOUR
NAMESPACE>::capProcessTitleBlocks
{<TITLE BLOCK OBJECT>}
```

Function will be called when a title block is encountered while processing page Objects.

<YOUR NAMESPACE>::capProcessBusEntries
{<BUS ENTRY OBJECT>}

Function will be called when a bus entry is encountered while processing page Objects.

<YOUR NAMESPACE>::capProcessPartInsts
{<PART INST OBJECT>}

Function will be called when a part Inst is encountered while processing page Objects.

<YOUR

NAMESPACE>::capProcessInstOccurenceSta
rt {<INST OCC OBJECT>}

Function will be called before processing of a Inst Occurrence start.

<YOUR

NAMESPACE>::capProcessOffPageOccurence {<OFFPAGEOCC OBJECT>}

#Function will be called when a offPage occurrence is encountered while processing Inst Occ.

<YOUR

NAMESPACE>::capProcessPortOccurence{<P
ORTOCC OBJECT>}

#Function will be called when a port occurrence is encountered while processing Inst Occ.

<YOUR NAMESPACE>::capProcessNetOccurence{<NE T OCC OBJECT> } #Function will be called when a net occurrence is encountered while processing Inst Occ. <YOUR NAMESPACE>::capProcessTitleBlockOccure nce{<ITITLEBLOCKOCC OBJECT>} #Function will be called when a title block occurrence is encountered while processing Inst Occ. <YOUR NAMESPACE>::capProcessDboNet{<DBONET</pre> OBJECT>} #Function will be called when a dbo net is encountered while processing Inst Occ

5. Finally, Add an init file in capAutoLoad folder with the following content.

```
proc ::<METHOD3>{ args } {
    return true
proc ::<METHOD4> { args } {
if { [catch {package require < YOUR</pre>
} else {
eval [concat :: <YOUR</pre>
NAMESPACE>::<METHOD2> $args]
# <METHOD2> is defined in user custom
drc file.
}
proc ::<METHOD5> { args } {
if { [catch {package require < YOUR</pre>
NAMESPACE>}] } {
} else {
eval [concat :: <YOUR NAMESPACE>::<</pre>
METHOD1> $args]
# <METHOD2> is defined in user custom
drc file.
```

6. Add the following line in the TCL file and modify according to DRC:

```
# For Electrical DRC

RegisterAction
"_cdnCapCustomDRCElectricalAddItem" "<
METHOD3>" "" "<METHOD4>" ""

#for Physical DRC

# RegisterAction
"_cdnCapCustomDRCPhysicalAddItem" "<
METHOD3>" "" "<METHOD4>" ""
```

Configure Properties dialog box

To open this dialog

Right-click and select *Configure Properties* in the Find Results window and search browser window.

Use this control	To do this
Left Arrow	Remove the property from the Find window
Right Arrow	Add the property to the Find window
Up Arrow	To move the property to be displayed up in the Configure Properties window
Down Arrow	To move the property to be displayed down in the Configure Properties window

Delete Part Property dialog box

To open this dialog

In the Project manager, select the design (.DSN) a schematic folder, or a schematic page then choose Delete Part Property from Edit menu.

Use this control... To do this...

Property Name Type the name of the property you want to remove.

Design Properties dialog box

To open this dialog box

- **1.** Right-click a design (.DSN) in the project manager.
- **2.** Choose *Design Properties* from the shortcut menu.

Alternatively:

- **1.** Select a design in project manager.
- **2.** Select *Options Design Properties* (see <u>Design Properties command</u>).

Use this tab To d	do this
-------------------	---------

Fonts Change the fonts for objects with text. A standard

Windows Font dialog box appears when you click on the

font display of an item.

Hierarchy Specify default settings of primitive or nonprimitive for

hierarchical blocks and parts. These options affect parts and hierarchical blocks that have the *Primitive* property set to default. When parts are marked as primitive, you cannot descend into them, even if they have attached

schematic folders.

SDT Compatibility

Define the mapping to use when saving designs in SDT format. Capture uses the properties specified in this tab to define the part field lines when it creates an \mathtt{SDT} . CFG during translation.

To make SDT part fields carry over into Capture properties, you need to specify them in the SDT.CFG file.

Also, specify which properties will be reported in the . INF file when creating a netlist using the VST tab in the *Create Netlist* dialog box.

Specify if you want to display the power pins in the design. This tab also displays the following information:

- Design name
- Root schematic name
- Unique ID for design
- Creation time
- Modification time
- Date format

The *Date Format* field shows the format of the modification date and/or time of each schematic page in its title block. This drop-down list also provides a list of date formatting options to select from. To change the format in which the date and time is displayed:

- **a.** Click the down arrow to expand the list.
- **b.** Select a format.
- c. Click OK.

Miscellaneous

Design Rules Check dialog box

To open this dialog

In the project manager, select the design (.DSN), a schematic folder, or a schematic page then choose *Design Rules Check* (see <u>Design Rules Check command</u>) from the *PCB* menu.

Use this tab	To do this
Options tab	Set the scope, mode and type of design rule (electrical and/or physical) test options for the design rules check.
Rules Setup tab	Set the electrical, physical, and custom design rule checks (DRCs).
Reports Setup tab	Select the reports to be generated from the check.
ERC Matrix tab	Set the matrix rules used during the design rules check.
Exception Setup tab	Set any DRC warnings that you do not want to be checked during the DRC check and netlisting.

Options tab

Use this control... Online DRC

To do this...

Select On, if you want to use the Online DRC feature.

This features checks for violations of design rules dynamically (as you are creating the schematic design). With this feature set, you do not have to complete the design and then run Design Rules Check to verify your design before creating the board file.

Select Off, if you do not want to use the *Online DRC* feature. If this option is unset, you will not see any information, warning, and error messages in this tab.

Batch DRC

DRC Action

Select the scope of the design rules check. The scope can cover the entire design, or the selected schematic folder.

If you run Design Rules Check on a single schematic page, Capture checks all pages in the entire schematic folder, which ensures that all nets on the schematic page are valid.

Using this drop-down list, you can also specify if the DRC markers are to be deleted from the design or from the selection.

Use Properties (Mode)

Specify to check either instances or occurrences. Capture automatically sets this option based on the project type. All designs default to use instances. If a PCB or schematic design is complex or has occurrence properties, the default shifts to occurrences. Capture recommends the preferred mode, which you can override.

Warning

Specifies whether or no to create the DRC markers for warning.

Note: The DRC markers are automatically deleted when you run a subsequent design rules check.

Waived DRC

You can select the Preserve option to retain the DRC waiver settings for a design. When you waive a DRC, it is not included in the DRC report. Setting this option ensures that the waived DRCs are not shown when you run DRC on the design.

Show DRC Outputs

Specifies where to display the design rules check report file. It has the following options:

- DRC Window
- Reports
- Both
- None

If DRC Window is selected in the Show DRC Output field, then all the design rule violations are listed in the DRCs window in the output area. You can browse the DRCs using this list displayed in this tab. This list is created and displayed whether or not Create DRC Markers is selected in the Warning field.

Report

Specify the path and file name for the design rules check report.

Click the Browse button that displays a standard Windows dialog box for selecting files.

Rules Setup tab

Electrical Rules Use this check box to specify if you want to run electrical

type of rules.

Use this check box... To do this...

Check single node nets Check if the design contains any nets with only one

connection.

type conflicts

Check no driving source and Pin Check if the net has any driving signal.

Check if the pin connections are according to the

configuration specified in the ERC matrix.

Check duplicate net names Check if the design contains any duplicate net names.

Check off-page connector

connections

Verify that off-page connector nets on a schematic page

match those on other schematic pages.

Check hierarchical port

connections

Verify that hierarchical pins in a hierarchical block match hierarchical ports in the child schematic folder or folders.

Errors are generated if the number of hierarchical ports and hierarchical pins differ between the parent and child schematic folders. Also generates errors if the types of hierarchical ports are not identical between the parent

and child schematic folders.

Check unconnected bus nets Check for and reports all unconnected bus nets. This

check will run for all unconnected bus nets across

schematics in a design.

Check unconnected pins Check for any pins on the design that are unconnected or

do not have no-connect attached.

Physical Rules Use this check box to specify if you want to run physical

type of rules.

Use this check box... To do this...

Check power pin visibility Check if the visibility property of a power pin on one

section of multi-section part is different from the

corresponding power pin on another section of the part.

Check missing/illegal PCB

Footprint property

Check if the PCB footprint property on a part is missing

or the property defined is illegal.

This check is only valid for batch DRC.

Check PCB footprint in Check if the PCB footprint property on a part is missing configured path or the property defined is illegal. In addition, this also checks if the footprint is available in the psmpath (physical location on the system). This check is valid only for online DRC. Check normal convert view sync Check if the pin numbers on the normal view of a part are different from the pin numbers on the convert view. Check incorrect Pin Group Check if all pins in same pin group in a part are of the assignment same type. Check high speed props syntax Check the syntax of the high speed properties of the nets in the design. Check missing pin numbers Check if any part on the design has missing pin numbers. Check device with zero pins Check if any part on the design has no pin on the part. Check power ground short Check if the type of power pin name inside a part is connected to a net on the schematic with a different name. Check name prop consistency Check if the occurrences of a hierarchical block have the same "Name" property. **Custom DRC** Select this check box if you want to use the Custom Design Rule Checker to run custom TCL DRCs. Use this check box... To do this... Device Pin Mismatch Checks for mismatch of physical pin group because of Part reference designator used in occurrence **Hanging Wires** Checks for any unconnected wire ends **Overlapping Wires** Checks if two wires are overlapped with each other Part Reference Prefix Mismatch Reference Designator Prefix is checked against the definition in library Port Pin Mismatch Checks hierarchical pin and port definitions for any mismatch

Shorted Discrete Part

Invalid Pin Number

same net

Check if two terminals of any discrete part attached to

Checks for invalid strings in the pin number

Physically Shorted PACK-SHORT

Checks for physically shorted PACK-SHORT pins

Reports Setup tab

Electrical DRC Reports

Use this check box	To do this
Report all net names	List the names of all nets in the report file.
Report off-grid objects	List all objects that are on Fine grid in the report file.
Report hierarchical ports and off-page connectors	List all hierarchical ports and off-page connectors in the report file.
Report misleading tap connections	Checks for and reports those signals that are wrongly connected through a Bus Tap to a bus. Also checks for missing bus taps.

Physical DRC Reports

Use this check box	To do this
Report Visible unconnected power pins	List the names of all visible unconnected power pins.
Report unused part packages	List the names of any unused part packages.
Report invalid packaging	List any invalid packaging.
Report identical part references	List any identical part references.

To do this...

ERC Matrix tab

Use this control...

Matrix

Set the rules used by the Design Rules Check when testing connections between pins, hierarchical blocks, and hierarchical ports.

The pins, hierarchical ports, and off-page connectors are listed in columns and rows in the table. A test is represented by the intersection of a row and column. Either the intersection of a row and column is empty, or it contains a "W" or an "E." An empty intersection represents a valid connection, a "W" is a warning, and an "E" represents an error.

You can cycle through these three settings by pointing to an intersection and clicking the mouse button until the desired setting displays. You can also type W for warning, E for error, and N for an empty intersection. In addition to these keys, you can use the arrow keys to select other intersections.

Restore Defaults

Use this button to restore the ERC matrix to its default values.

Exception Setup tab

Use this control...

To do this...

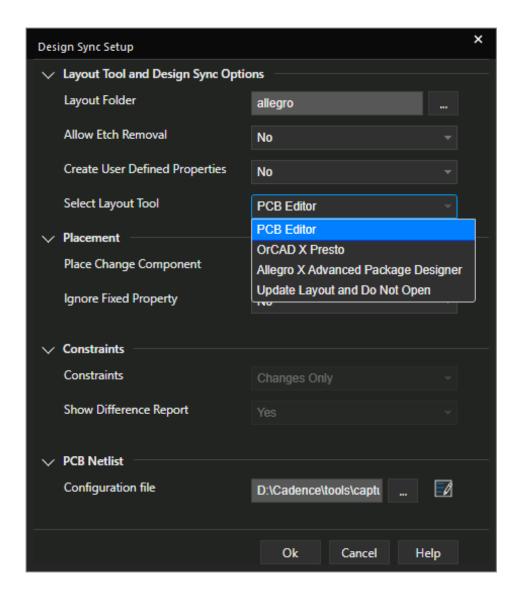
Add New

Use the *Add New* button in the Exception Setup tab to specify any DRC warnings that you do not want to be checked during the DRC check and netlisting.

For example, to ignore the ALG0051 or ALG0016 warnings during netlist, specify these in the *Exception Setup* tab of the Design Rules Check dialog box.

Design Sync Setup dialog box

To open the *Design Sync Setup* dialog box, choose *PCB – Design Sync Setup* from the main menu or click () from *Update Layout* or *Update Schematic* dialog boxes.



Use this control...

To do this...

Layout Tool and Design Sync Options

Layout Folder

Shows the default folder name allegro in your design directory, where the PCB Editor board is created or updated (corresponding to the netlist being generated).

Allow Etch Removal

Select Yes from this drop-down list, if you want PCB Editor to rip up etch of already-routed tracks for parts that need re-routing as a result of Capture changes.

The default value is No.

Selecting Yes, saves you time by automatically removing the etch up to the closest T-connection or pin, if a pin is moved due to a part placement or circuit change.

Create User-Defined Properties

Select Yes from this drop-down list to allow the creation of property definitions from the netlist.

Select Layout Tool

This option lets you choose the tool in which you want to open the output board file.

Select:

- PCB Editor to open the output board file in Allegro X PCB Editor after the netlisting is done.
- OrCAD X Presto to open the output board file in OrCAD X Presto after the netlisting is done.
- Allegro X Advanced Package Designer to open the output board file in Allegro X Advanced Package Designer after the netlisting is done.
- Update Layout and Do Not Open if you do not want to open the output board file in any of the tools but to only update the board file.

Placement

Place Changed Components

This option lets you choose what to do when a reference designator is assigned to a different component in Capture than what the reference designator is assigned to in the board layout.

Select:

■ Always

PCB Editor will replace all components in the layout with new components from the netlist according to their reference designators. PCB Editor places the new component at the same X,Y location and rotation as the old part. This is the default option unless you have netlisted previously with a different option selected.

■ If Same

Same as Always, but only if the component package symbol, value, and tolerance match the original component. If there is a difference, the old component is removed, and the new component becomes an unplaced part.

■ Never

Changes must be made by the user. All old components that had changed reference designators are removed and new ones become unplaced.

Ignore Fixed Property

Select Yes in this drop-down list, if you want PCB Editor to replace and delete symbols, rip up etch, and make other changes even if elements of your design are fixed (are assigned the FIXED property).

Constraints

Constraints

All constraints captured in schematic get transferred to PCB using the schematic to PCB create process.

You can transfer constraints using one of the following two options:

Changes Only

Use the Changes Only mode if you need to merge concurrent constraint changes done in schematic and layout.

Overwrite

Use the Overwrite mode when you need to overwrite the constraints in schematic or in board file.

Show Difference Report

Select Yes if you want to view the constraints difference report.

PCB Netlist

Configuration File

Shows the location of the last used configuration file, allegro.cfg.

Click the browse button to specify, edit, and view the information in this configuration file, which is used for netlisting and back annotating property information between Capture and PCB Editor.

Captures uses the following order of preference for accessing the allegro.cfg file is:

- 1. Last used file
- 2. File in design directory
- 3. File specified in the CDS SITE path
- 4. File configured during Capture installation

If you have run a previous netlisting or back annotation the configuration file you used is listed. If not, the first found *.CFG file in your design directory is used. If no *.CFG file is found in your design directory, then the CFG file on the path defined by the CDS_SITE environment variable will be used. Finally, if the CDS_SITE environment variable is not set, the default file will be used which is the sample allegro.cfg file that is installed with Capture.

Note: If the allegro.cfg file is not available at the location specified in this text box, Capture looks for the default file as per the order of preference.

All Rights Reserved.

Design Template or Design Properties dialog box

To open the Design Template dialog box

Select *Design Template* (see <u>Design Template command</u>) from the *Options* menu. Use this dialog box to specify default settings for new projects, designs, and schematic pages.

To open the Design Properties dialog box

© 2023

Select a design (.DSN) in the project manager then choose *Design Properties* (see <u>Design Properties</u> command) from the *Options* menu. Use this command in the project manager to

globally set design related options throughout a design.

Use this tab	To do this	
Fonts	Change the fonts for objects with text. A standard	
(available in both Design Template and Design Properties dialog box)	Windows Font dialog box appears when you click on the font display of an object.	
	These options are set once per design. Once a design is created, use the <u>Design Properties command</u> to change these options for a particular design.	
Title Block	Enter the title, organization name and address,	
(available only in Design Template dialog box)	document number, revision, and CAGE code into the title block.	
	The CAGE Code stands for Commercial and Government Entity Code. This is a number provided by	
	the federal government to its suppliers, and that can be present in the title block of a schematic page.	
	Also, enter the path and filename of the library containing the title block, and the title block name.	
	These options affect each new page. The OrCAD X -supplied title block resides in the CAPSYM.OLB library. For more information about title blocks, see <u>Setting up the default title block</u> .	
Page Size	Specify the units of measure used in the schematic pa	
(available only in Design Template dialog box)	editor. Also, change the width and height of a schematic page, as well as spacing between pins in a design. For more information, see <u>Page Size tab</u> .	
Grid Reference	Choose between alphabetic and numeric, and between	
(available only in Design Template dialog box)	ascending and descending for both horizontal and vertical grid references.	
	Also set the grid count for both horizontal and vertical grid references, set the width of the grid references, and set title block visibility. For more information, see <u>Grid</u>	

Reference tab.

Hierarchy

(available in both Design Template and Design Properties dialog box)

Miscellaneous

(available only in Design Properties dialog box)

Specify default settings of primitive or nonprimitive for hierarchical blocks and parts for future designs. These options are set once per design, and affect parts and hierarchical blocks that have the Primitive property set to Default. When parts are marked as primitive, you cannot descend into them, even if they have attached schematic folders.

Specify if you want to display the power pins in the design. This tab also displays the following information:

- Design name
- Root schematic name
- Unique ID for design
- Creation time
- Modification time
- Date format

The *Date Format* field shows the format of the modification date and/or time of each schematic page in its title block. This drop-down list also provides a list of date formatting options to select from. To change the format in which the date and time is displayed:

- **a.** Click the down arrow to expand the list.
- **b.** Select a format.
- c. Click OK.

SDT Compatibility

(available in both Design Template and Design Properties dialog box) Define the mapping to use when saving designs in SDT format. It can be changed for individual designs in the SDT Compatibility tab of the Design Properties dialog box. The Part Field to Property mapping fields are used only when you save a Capture design in an SDT format. To make SDT part fields carry over into Capture properties, you need to specify them in the SDT.CFG file.

Fonts tab

Change the fonts for objects with text. A standard Windows <u>Font dialog box</u> appears when you click on the font display of an object.

These options are set once per design. Once a design is created, use the <u>Design Properties</u> command to change these options for a particular design.

Title Block tab

Enter the title, organization name and address, document number, revision, and CAGE code into the title block.

Also enter the path and filename of the library containing the title block, and the title block name.

These options affect each new page. The OrCAD X-supplied title block resides in the CAPSYM.OLB library. For more information about title blocks, see <u>Setting up the default title block</u>.

Grid Reference tab

To open this dialog box, do

- **1.** Choose *Options Design Template*.
- 2. Select the Grid Reference tab.

(see Design Template command)

Set these options for new schematic pages. Changing these options does not affect the existing schematic pages.

Use this control	To do this
Horizontal and Vertical	
Count	Specify the number of divisions in the horizontal or vertical grid references.
Alphabetic and Numeric	Specify whether the grid references are alphabetic or numeric.
Ascending and Descending	Specify whether the grid references ascend or descend.
Width	Specify the width of the grid reference division. The width here is not the distance between grid reference division, but the amount of space taken up in the schematic page editor.

Border Visible

Displayed Specify whether the border is visible on the screen.

Printed Specify whether the border is visible on paper.

Grid Reference Visible

Displayed Specify whether the grid references are visible on the

screen.

Printed Specify whether the grid references are visible on paper.

Title Block Visible

Displayed Specify whether the title block is visible on the screen.

Printed Specify whether the title block is visible on paper.

ANSI grid references Specify if schematic pages use the ANSI Standard grid

references.

Page Size tab

To open this dialog

Select Design Template (see <u>Design Template command</u>) from the Options menu.

Set these options for future schematic pages. Changing these options won't affect schematic pages you've already created.

Use this control	To do this
Units	Specify the unit of measurement for future designs. Select either inches or millimeters.
	This only affects the schematic page editor. It doesn't affect the part editor, which is always measured in grid units.
New Page Size	Specify the size of schematic pages for future designs. The first five choices are A to E if the unit measurement is inches, or A4 to A0 if the unit measurement is millimeters.
Width	Specify the width of future schematic pages in the indicated unit measurement. The values specified here for page sizes A to E and A4 to A0 are set values in the Schematic Page Properties dialog box.

Height Specify the height of future schematic pages in the

indicated unit measurement. The values specified here for page sizes A to E and A4 to A0 are set values in the

Schematic Page Properties dialog box.

Pin-to-Pin Spacing Specify the spacing between pins in the indicated unit

measurement. Also specify grid spacing. For example, a pin-to-pin spacing of 0.1 inches means that the dots or

lines on your grid will be 0.1 inches apart.

Hierarchy tab

Specify default settings of primitive or non-primitive for hierarchical blocks and parts for future designs. These options are set once per design, and affect parts and hierarchical blocks that have the Primitive property set to Default. When parts are marked as primitive, you cannot descend into them, even if they have attached schematic folders.

SDT Compatibility tab

Define the mapping to use when saving designs in SDT format. It can be changed for individual designs in the SDT Compatibility tab of the Design Properties dialog box. The Part Field to Property mapping fields are used only when you save a Capture design in an SDT format. To make SDT part fields carry over into Capture properties, you need to specify them in the SDT.CFG file.

Differential Pair Automatic Setup dialog box

To open this dialog

Click the *Auto Setup* button in the <u>Create Differential Pair dialog box</u>.

Use this control... To do this...

All Nets/Diff Pair grid View all the flat nets or differential pairs defined in a

design.

Filter Specify the nets you want to view in the All Nets grid.

Note: To view nets of a particular type, specify the initial letters of the net in the Filter text box. All the nets of that

particular type will appear in the All Nets grid. For example, if you want to view all nets starting with the letter "A", then enter "A" in the Filter text box. All the nets starting with letter "A" will appear in the All Nets grid

Prefix Specify a string (numeric or alphabet) that you want to

precede the differential pair name.

For example, if you specify "A" in the Prefix text box, then all the differential pair names that will be created will be

preceded with "A".

+ Filter Specify the last digit of the first net's name.

For example, all net names ending with 1.

- Filter Specify the last digit of the second net's name in the -

Filter text box.

For example, all net names ending with 4.

Diff Pair/+Net/-Net grid View all the unique differential pairs that are generated

for all the nets that qualify the criteria set in the + Filter

and - Filter text boxes.

The +Net and -Net grid displays the two nets associated

with a differential pair.

Create all the differential pairs for the nets displayed in

the grid.

Remove Remove the selected differential pair from the design.

Note: If you do not want a specific differential pair to be created, select the row containing the differential pair and click the Remove button or double-click the row containing

the differential pair. The selected row disappears.

Close the Differential Pair Automatic Setup dialog box

and go back to the Create Differential Pair dialog box.

Note: An Auto Differential pair can also be created for a bus. To do so, you need to put _n_ & _p_ as prefix and the Auto command creates differential pairs for all bits in the bus.

Display Properties dialog box

To open this dialog, use one of the following methods:

In the schematic page editor:

■ Click the *Display* button in the <u>User Properties dialog box</u>.

OR

Select a part on a schematic page and choose *Properties* (see <u>Properties command</u>) from the Edit menu. In the Property Editor window, click the *Display* button.

Use this control	To do this
Name	Specifies the property's name.
Value	Specify the property's value.
Display Format	Specify the visibility for the property name and value.
Font	Displays the font name and point size.
Change	Display a standard Windows <u>Font dialog box</u> so you can change the font, font style, and font size of the property.
Use Default	Use the default value for the property. The default value is set in the <u>Design Template or Design Properties dialog</u> <u>box</u> (see <u>Design Template command</u>).

Specify the property's color.

Specify the rotation of the property.

Color

Rotation

Text Justification

Specify the text justification for displayed property text as Default, Left, Center, and Right at library level and schematic level. The default text justification is the legacy Capture behavior as performed in Capture 16.5. For more information on Text Justification, see <u>Table 8-1: Text Justification</u> table on page 370.

Note: You can override the text justification set at the library level from schematic level. Once overridden, update cache will not update the text justification set at schematic level.

Note: Text justification in a design saved using 16.6 HotFix of Capture will not be preserved if the design is opened and saved using Capture 16.6 and earlier releases.

Note: You can use the Display Properties dialog box to set the display option of an instance property and its value, but you cannot use it to display properties of an occurrence property.

Note: If you have justified a text on a library symbol using the pre-SPB 17.2 release (HotFix 009), you need to reopen the text properties dialog for the particular text and click *OK* using HotFix 009 or later. Once done, new justification settings will replace the old settings.

Table 8-1: Text Justification

Text Justification	Text after 0 degree rotation is	Text after 90 degree rotation is	Text after 180 degree rotation is	Text after 270 degree rotation is
Default	Left Justified	Right Justified	Left Justified	Right Justified
Left	Always Left Justified			
Center	Always Center Justified			
Right	Always Right Justified			

Distributions dialog box

To open this dialog

Select the Monte Carlo/Worst Case option, from the Analysis tab of the Simulate Settings dialog box, and click the Distributions button.

Use this control... To do this...

Existing distributions Display a list of existing distributions for tolerances,

defined by you. These are only used with Monte Carlo

and sensitivity/worst-case analyses.

Distribution name Specify the name of a new distribution.

Distribution curve values Specify up to 100 curve values for the distribution. Each

curve value is defined by two values in the form (x,y), where x is the deviation and y is the probability. The deviation must be between -1 and 1. The probability must

be zero, or positive.

Delete the selected distribution from the existing

distributions list.

Save the distribution defined by the distribution name

and curve values options.

Design XML dialog box

To open this dialog

Select File – Export – Design XML.

Use this control... To do this...

DSN File Specify the Capture design file path.

XML File Specify the XML file name path. The default XML file

name is design file name with .xml file extension.

Log file Select to specify the log file name path. The default log

file name is design file name with .log file extension.

View Output Select to view the output in Capture after XML

generation.

XML Schema Displays non-editable XML schema file path. Click View

to view the XML schema file.

Design Difference dialog box

To open this dialog

Select Tools - Compare Designs.

Use this control... To do this...

Design 1

Design Specify the Capture design file path

Schematic Specify the Schematic name from the selected Capture

design file

Page Specify the Schematic page name from the selected

Capture Schematic

Design 2

Design Specify the Capture design file path

Schematic Specify the Schematic name from the selected Capture

design file

Page Specify the Schematic page name from the selected

Capture Schematic

HTML Creation

HTML Type

Select one of the following options:

- Portable(Standalone) The portable html file has all the JavaScript files included in the HTML itself. The portable html file can be opened in any other machine, but is big in size compared to the lightweight html.
- Lightweight (Cadence hierarchy dependent) The lightweight html file do not contain all the JavaScript files but has links pointing to the installed Cadence hierarchy. The html file is small in size compared to the portable one, but cannot be opened in other machine.

HTML Path

Specify the output HTML file path. The default path for the HTML file is the *temp* folder.

HTML Base File Name

Specify the name of the output HTML file. The default name of the HTML is <First Design>_vs_<Second Design>.html.

Open With

Select one of the following options to open the output html file:

- Internet Explorer
- Custom Use other browsers, such as Google Chrome and Mozilla FireFox.

Note: It is strongly recommended to view the generated design difference HTML file in one of the flowing browsers, but Google Chrome is preferred:

- □ Internet Explorer (version 11 onwards)
- ☐ Google Chrome (version 45 onwards)

If browser except the recommended ones is selected in the Design Difference dialog box, ensure that it supports HTML5 Canvas.

Edit Bookmark dialog box

To open this dialog

Select a bookmark on a schematic page, then choose Properties (see <u>Properties command</u>) from the Edit menu.

Use this control... To do this...

Name Change the name of the bookmark.

Edit Filled Graphic dialog box

To open this dialog

On a schematic page

Double-click on a closed polyline, an ellipse, a rectangle.

OR

Select a closed object and choose Properties (see Properties command) from the Edit menu.

Use this control... To do this...

Fill Style Choose the fill style.

Line Style & Width Choose the line style and width.

Color Choose the color of the line. This option is not available

in the part editor.

Edit Graphic dialog box

To open this dialog

In a schematic page editor,

Double-click a line.

OR

Select a line and choose Properties (see Properties command) from the Edit menu.

Use this control... To do this...

Line Style & Width Choose the line style and width.

Color Choose the color of the line. This control is not available

in the part editor.

Edit Hierarchical Port dialog box

To open this dialog

In the schematic page editor, select a hierarchical port and choose Properties (see <u>Properties</u> <u>command</u>) from the Edit menu.

Use this control... To do this...

Name Change the name of the hierarchical port.

Type Select the hierarchical port type from the list of pin types.

Edit Net Alias dialog box

To open this dialog

Select a net alias in the schematic page editor and choose Properties (see <u>Properties</u> <u>command</u>) from the Edit menu.

Use this control... To do this...

Alias Change the net alias name.

Color Choose the color of the net alias.

Rotation Specify the rotation of the net alias or text.

Font

Change Display a Font dialog box so you can select a font.

Use Default Change the font to the default font specified in the Design

Template / Design Properties dialog box.

Edit Off-Page Connector dialog box

To open this dialog

In the schematic page editor, select an off-page connector, and choose Properties (see <u>Properties command</u>) from the Edit menu.

Use this control... To do this...

Name Change the name of the off-page connector.

Edit Part Properties dialog box

To open this dialog

While placing a part on a schematic page editor, right-click and choose Edit Properties from the shortcut menu.

Use this control... To do this...

Part Value Change the part value name.

Part Reference Specify the part reference.

Primitive

Default Use the default primitive setting. The default setting is set

in the Hierarchy tab of the Design Template dialog box

(see <u>Design Template command</u>).

Yes Indicate the part is a primitive.

No Indicate the part is nonprimitive and descends in

hierarchy.

Graphic Specify whether Capture displays the normal view or the

convert view of the part. The convert view option is only

available for parts with convert views.

Packaging

Parts per Pkg Indicates the number of parts in the package.

Part Select a part from the package list.

PCB Footprint Specify the PCB footprint name, if assigned.

Power Pins Visible Specify the visibility of the part's power pins.

User Properties Display the <u>User Properties dialog box</u> so you can modify

the part's properties.

Attach Implementation Display the <u>Attach Implementation dialog box</u> so you can

attach a schematic folder to create hierarchy. You must specify the schematic folder's name, but you only need to specify the schematic folder's library or path name if the

schematic folder is not in the current project.

Note: Be careful not to create recursion in your design. Capture cannot prevent recursion, and the Design Rules Check command does not report it.

Recursion causes Capture to process infinitely as it tries to expand the design, resulting in the loss of any changes you've made to your design since it was last saved.

Note: Except for occurrence properties, the schematics of externally-referenced libraries and designs should not be edited. You should view them as read-only designs. Trying to edit, then save, these designs from within your schematic can introduce errors such as duplicate reference designators and other problems.

When saving schematics with externally-referenced libraries or designs, occurrence properties are saved but altered instance values are not. If you want to change externally-referenced libraries or designs you should first close the referencing design. Then, open the referenced library or design, make the necessary changes, and save and close the referenced library or design. At this point, you can reopen the original design and reference the modified design.

Note: When you attach a schematic folder to a part or hierarchical block, you can specify a full path and filename in the Library text box. So, although you can specify a library that hasn't been saved, you should not try to descend into the attached schematic folder until the library that contains the schematic folder has been saved.

If you don't specify a full path and filename in the Library text box, Capture expects to find the

attached schematic folder in the same design as the part of hierarchical block to which it is attached. If the specified schematic folder doesn't exist in either the design or library, Capture creates the schematic folder when you descend hierarchy on the part or the hierarchical block.

For compatibility with future versions of Windows, Capture preserves the case of the path and filename as you specify them in the Library text box.

Edit Text dialog box

To open this dialog

Select a text object and choose Properties from the Edit menu.

Use this control... To do this...

Text Change the text to display.

Color Choose the color of the text. Text that is placed in the

part editor uses the part body color.

Rotation Specify the rotation of the text.

Font

Change Display a Font dialog box so you can select a font.

Use Default Change the font to the default font specified in the Design

Template / Design Properties dialog box.

Edit Wire(s) dialog box

To open this dialog

In a schematic page editor,

Select a wire and choose Edit Wire Properties from the shortcut menu

Use this control... To do this...

Line Style & Width Choose the line style and width.

Color Choose the color of the line. This control is not available

in the part editor.

Export ECSets from design

To open this dialog

Choose SI Analysis – Export Electrical Csets.

Use this control... To do this...

Export Directory Specify the directory to export the Electrical Csets.

Remove unused ecsets Check to remove Electrical Cset that are not assigned to

any net.

Validate Ecset Click to validate the selected Electrical Csets.

Export Click to export the selected Electrical Csets

Export Design dialog box

To open this dialog

Choose Export Design (see Export command) from the File menu.

Use this tab... To do this...

EDIF Specify a name for the EDIF file, and the scope of the export.

DXF Specify a name for the DXF file, and the scope of the export.

EDIF tab

Use this control... To do this...

Save As Specify the name of the .EDF file to save the design as.

The design is saved as a graphical EDIF file, and not an

EDIF netlist.

If you want to generate an EDIF netlist, use the EDIF tab

in the Create Netlist dialog box, accessible through the

Create Netlist command on the Tools menu.

Exporting EDIF designs saves the entire design, regardless of which window (project manager or

schematic page editor) is currently active.

Configuration File Specify the name of the configuration file (.CFG) for the

translation. Configuration files are not required for

translation.

DXF tab

Use this control... To do this...

Save As Specify the name of the .DXF file to save the schematic

page as. Capture saves .DXF files in AutoCAD's V12 file

format.

Scope

Entire Design Export the entire design. Capture appends the schematic

name and page name to the name you specified in Save

As:

<Save As><Schematic><Page>.DXF

Current Page Export the current page of the design.

Include Border Check to include the border.

Include Title Block Check to include the title block.

Note: In order to export a design or schematic to DXF, you must have a schematic page from that design open and selected as the active window. Otherwise, Capture will not allow a DXF

export.

Export Properties dialog box

To open this dialog

In the Project manager, choose Export Properties from the Tools menu.

Use this control... To do this...

Scope Process the entire design or just the selected

documents.

Contents Export part properties or part and pin properties, or flat

net properties.

Mode Export either instance properties or occurrence

properties.

Specifies to export either instance properties or

occurrence properties. Capture recommends a preferred

mode, which you can override.

Export File Specifies the name of the export output file. For more

information about property files, see Editing properties.

Browse Displays a standard Windows dialog box for selecting

files.

Export Selection dialog box

To open this dialog

In the schematic page editor, choose one or more objects, then choose Export Selection (see <u>Export Selection command</u>) from the File menu.

Use this control... To do this...

Export Selection Name Specify the export name of the selected object or

objects.

Library Specify a path, and a design or library name for the

export selection.

Browse Display a standard Windows dialog box for selecting files.

Export Variant list dialog box

To open this dialog

Use this control...

In the Part Manager, choose Export Variant List from the Tools menu.

To do this...

Output File (Variants.Ist) Path

Specify the default location for the Variants.Ist file. The default location is the allegro folder in the design directory. You can change this path also.

Config File (Variant.cfg) Path

Specify the default location for the Variants.Ist file. The default location for the Variants.Ist file. The

default path is the same as that of the Capture.exe file.

You can change this path also.

Export Variants.lst file containing information about all

the variants of the design. The properties listed are the

ones specified in the

Variant.cfg file.

Extended Preferences Setup

To open this dialog

Select Options – Extended Preferences.

Or

Select *Options – Preferences*. In the Preferences dialog box, click the *More Preferences* button.

Use this control... To do this...

Command Shell These commands are related to the TCL command win-

dow in Capture.

Journaling Select the option to enable journaling of the various

Capture commands, including TCL commands.

Use the following TCL command to enable or disable

journaling:

SetOptionBool Journaling TRUE/FALSE

Flush Commands Select the option to print the journaling commands in a

text file. By default, the text file is saved in the TEMP folder (<%TEMP%>\CAPTURELOG\<%DATE%>\

OrCaptureLogFile.captcl)

Use the following TCL command to flush the output of

the journaling commands:

SetOptionBool FlushImmediate TRUE/FALSE

Display Commands Select the option to enable display of Capture commands

in Command Window.

Use the following TCL command to enable or disable the display of capture commands in Command Window:

SetOptionBool DisplayCommands TRUE/FALSE

Enable Customer Experience Select this option to take part in the Customer

Improvement Program (CEIP) Experience Improvement Program (CEIP) in Capture.

When you participate in this program, your system will automatically send basic, anonymous information to Cadence Design Systems about how you use various Capture features. This information is combined with other CEIP data to help Cadence solve problems and improve the products and features that customers use most often. Cadence does not collect your design data or personal

information.

Design and Libraries These commands are related to Capture's designs and

libraries.

Content text instance properties Select the option to rotate the instance part properties

along with the instance part on the same axis.

TCL Command: SetOptionBool

RotateInstPropInContext TRUE/FALSE

Draw arrow on part input pins Select the option to draw arrow on part's input pins.

TCL Command: SetOptionBool DrawPinsArrows

TRUE/FALSE

Enable communication with legacy tools

Select the option to enable message-based communication with the legacy tools, such as OrCAD X

Presto.

TCL Command: SetOptionBool EnableLegacyITC

TRUE/FALSE

Note: By default, message-based communication is not

enabled in Capture. Enable this option if you want

Capture to communicate with the legacy tool as it impacts

Capture's performance.

Perform read only check on tab switch

Select the option to enable Capture to check the library and design files' permissions on every switch of tab.

TCL Command: SetOptionBool

CheckReadOnlyOnViewActivate TRUE/FALSE

Note: Unselecting the option improves user experience as Capture ignores the permissions check on the design

and library files.

Back annotate pin numbers only Select the option to ignore pin name changes during

back annotation.

TCL Command: SetOptionBool

BackAnnotatePinNumbersOnly TRUE/FALSE

Save design name as

UPPERCASE

Select the option to save the design file name (.dsn) in

uppercase letters.

TCL Command: SetOptionBool

SaveAsUpperCaseDsnName TRUE/FALSE

Enable global net ITC Select the option to start cross probing of global nets.

> TCL Command: SetOptionBool EnableGlobalNetITC 1/0

Convert images to BMP format

Select the option to convert all format images to .bmp

internally.

TCL Command: SetOptionBool SaveImageAsBitmap TRUE/FALSE

Path lookup timeout (in seconds)

Specify the time out time in seconds if the design or library files are found at the configured locations.

TCL Command: SetOptionUInt

PathLookupTimeout <valuee in seconds>

Net Naming Options (requires application restart)

The drop down menu options provides different options to select how Capture generates flat-net names for complex hierarchy designs. By default, flatname has a hierarchy block path.

TCL Command: SetOptionUInt NetNameMode
<1,2,3,4>

where

1: Always append hierarchy

2: Append hierarchy on collision

3: Always append ID

4: Append ID on collision

Following are the drop down options:

Never

Always

Only when mismatch

Append hierarchy on collision

Appends hierarchical path to the conflicting flat nets only (for global nets, numeric ID is used). The flat net that is closest to the root design and is the only flat net at that level is not renamed.

Always append hierarchy

Appends hierarchical path to all the flat nets except those in root schematic. For global nets, numeric ID is used as they do not have any hierarchical path.

Append ID on collision

Appends numeric IDs to all the conflicting flat nets only. The flat net that is closest to the root schematic and is the only flat net at that level, is not renamed.

Always append ID

Appends numeric IDs to all the flat nets except those in the root schematic.

Design Cache

Update Cache Select the Default or Forced option to update cache in

Capture. Selecting Forced will update the selected library even if it is older than the original library file in the

Design Cache.

TCL Command: SetOptionString ForceUpdateCache NONE/TRUE

DRC

Display Waived DRC Select the Display Waived DRC check box to display all

the waived DRCs on a schematic page.

TCL Command: SetOptionBool

DisplayWaivedDRC 1/0

Use Global DRC Settings Select the Use Global DRC Settings check box to use the

same DRC settings for all different designs.

CIS These commands are related to CIS operations in

Capture.

Query All Configured Tables Select this option to query all the tables that are

configured in the DBC configuration.

TCL Command: SetOptionString QueryAllTables TRUE/FALSE

Disable Regional Setting Select this option to disable regional setting.

TCL Command: SetOptionString CISRegionalSetting TRUE/FALSE

Quoted (") refdes in variant list Select this option to quote (") part references in the

variant export report.

TCL Command: SetOptionString

ExportQuotedRefDesVariant TRUE/FALSE

Use Cadence Default Library Select this option to use the default Cadence Library

located at the Installation path.

NetGroup

The following options, which are related to NetGroups, will be set only for the current active design:

- Never
- Always
- Only when mismatch (definition name mismatched instance name)

Select this option to set the character limit globally for all

Netlist

Apply Allegro Character Limits on All Projects

These command are related to Netlisting in Capture.

the projects.

TCL Command: SetOptionString AllegroCharLimitInINI TRUE/FALSE

Schematic

Schematic Descend

These commands are related to Schematic page in Capture.

The following three modes are used for schematic descend:

- Default: Opens the default page of child schematic
- First: Opens the first page of child schematic (alphabetical-wise)
- Ask: Asks user to select one of the schematic page from the list of schematic pages.

TCL Command: SetOptionString
DescendSchPage ASK/FIRST_PAGE/DEFAULT

Junction Mode

The following two modes are used for junctioning in Capture:

- Default: Select this option to place a junction at a straight wire break point
- Junction on multiple connections on wire end: Select this option to place a junction at 3 point connections.

Display underscore (_) on User Assigned Part References on Schematic Page Select this option to display underscore (_) symbol on user-assigned part references on a schematic page.

TCL Command: SetOptionString
HIGHLIGHT_USER_EDIT_REFERENCES_ON_PAGE
TRUE/FALSE

Display underscore (_) on User Assigned Part References in

Page Print

Select this option to display underscore (_) symbol on user-assigned part references in page or design print.

TCL Command: SetOptionString

HIGHLIGHT_USER_EDIT_REFERENCES_ON_PRINT

TRUE/FALSE

Distribute in a fixed area (may cause uneven distribution)

Select this option to distribute the selected objects in a fixed area.

TCL Command: SetOptionString

ObjectGridDistribution

UnevenDistributionWithSameExtents/Distrib

utionWithAutomaticExtent

Schematic Print Theme Select this option to specify the theme of the schematic

when you want to print the design or the library. The default value is Light. Irrespective of whatever is the application and canvas theme, the value selected in this drop-down is used to print the design or the library.

Enable Occurrence Copy Select this option to ensure that occurrence property is

retained on copy or cut operation along with the instance

property.

Configured Files These commands are related to customization of design

configuration files.

By default, the read-only path is displayed in the

Configuration Files tab. When you click edit, the files get stored at your system's home directory and are opened

in a text editor for any modification.

Once the changes are made, the saved files are stored at

system's home directory(%HOME%).

allegro.cfg The allegro.cfg file's path

PrefProp.txt The PrefProp.txt file's path

edif2cap.cfg The edif2cap.cfg file's path

cap2edif.cfg The cap2edif.cfg file's path

Cloud Connectors

Set one or both of the following connector options in the Extended Preferences Setup window to see the Cloud menu in Capture:

- EMA Cloud Connector
- Arrow Entrepreneur Cloud Connector

Choose options from the Cloud menu to access Capture Cloud from desktop.

Part and Symbol Editor

Snap to grid - Graphical objects

Select if graphical objects such as Line, Polyline, Rectangle, Ellipse, Arc, and Picture, which can be placed and moved on the selected grid type, Fine or Master.

This setting is saved in CAPTURE. INI and it is used whenever you start the next Capture session.

If you have selected Master:

- The snap-to-grid setting (on or off) in the toolbar is applied.
- Pressing the SHIFT key while dragging the objects toggles the grid type.

If you have selected Fine:

- The snap-to-grid setting in the toolbar is set to off, that is, the object will move on Fine grid.
- Pressing SHIFT key while dragging the objects will not change the grid type.

Snap to grid - Text objects

Specify if text objects and display property object can be placed and moved on the selected grid type, Fine or Master.

This setting is saved in CAPTURE. INI and it is used whenever you start the next Capture session.

If you have selected Master:

- Snap-to-grid setting (on or off) in the toolbar is applied.
- Pressing the SHIFT key while dragging the object toggles the grid type.

If you have selected Fine:

- The snap-to-grid setting in the toolbar is set to off, that is object will move on Fine grid.
- Pressing SHIFT key while dragging the objects will not change the grid type.

Enable the Paste Options mode Select this option if you want the open the Paste Options dialog box before pasting copied pins. This allows you to edit the pin information before pasting the pin.

> For information on how to use this dialog box, see Paste Options dialog box.

Export Design View to HTML dialog box

To open this dialog

Select File - Export - Design HTML.

Use this control	To do this	
Design		
Design	Specify the Capture design file path	
Schematic	Specify the Schematic name from the selected Capt design file	

Page

Specify the Schematic page name from the selected Capture Schematic

Output

HTML Type

Select one of the following options:

- Portable (Standalone) The portable html file has all the JavaScript files included in the HTML itself. The portable html file can be opened in any other machine, but is big in size compared to the lightweight html.
- Lightweight (Cadence hierarchy dependent) The lightweight html file do not contain all the JavaScript files but has links pointing to the installed Cadence hierarchy. The html file is small in size compared to the portable one, but cannot be opened in other machines.

Specify the output HTML file path. The default path for the HTML file is the *temp* folder.

Specify the name of the output HTML file. The default name of the HTML is <Design Name>.html.

Select one of the following options to open the output html file:

- Internet Explorer
- Custom Use other browsers, such as Google Chrome and Mozilla FireFox.

Note: It is strongly recommended to view the generated design HTML file in one of the flowing browsers, but Google Chrome is preferred:

- ☐ Internet Explorer (version 11 onwards)
- ☐ Google Chrome (version 45 onwards)

If browser except the recommended ones is selected in the dialog box, ensure that it supports HTML5 Canvas.

HTML Path

HTML Base File Name

Open With

Edit Comment Text dialog box

In the part editor, use this dialog box to add comment text on the part.

To modify the text, double-click the text. The Edit Comment Text dialog box opens. Change the text and click *OK*.

To modify the text properties:

1. Select the text.

The *Text Properties* section appears in the Property Sheet pane.

2. Modify the font, font size, font style, or the justification of the selected text.

Find and Replace dialog box

To open this dialog

In the schematic page editor, choose Global Replace (see <u>Global Replace command</u>) from the Edit menu.

Use this control... To do this...

Find what Specify the search string.

Replace with Specify the string to replace the Find what search string.

Match case Specify if the search must match the case of the string.

If this option is not selected, lowercase and uppercase

letters will be treated the same in the search.

Scope Search the Entire Design or Current Page Only.

Object Type Specify any combination of net aliases, hierarchical

ports, hierarchical pins, or off-page connectors by

selecting the check boxes.

Font dialog box

To open this dialog

Click the Change button in the <u>Edit Net Alias dialog box</u>, the <u>Place Net Alias dialog box</u>, the <u>Edit Text dialog box</u>, the <u>Place Text dialog box</u>, or the <u>Display Properties dialog box</u>.

OR

Click in a font box on the Fonts tab of the <u>Design Properties dialog box</u>, the Fonts tab of the <u>Design Template or Design Properties dialog box</u>, or the *Miscellaneous* tab of the <u>Preferences dialog box</u>.

Use this control... To do this...

Font Specifies the font for the text.

Note: Be sure to use a monospaced font (for example, Courier) as the default font for the text editor or the source editor. If you use a true-type font, the editor may distort the

appearance of the text, making it difficult to read.

Font Style Specifies the font style for the text.

Size Specifies the text size.

Sample Shows a sample of how the text will appear, based upon

the font, style and size.

Script Specifies the script used for the font.

Find dialog box

To open this dialog

Select Edit - Find, or see Find command.

Note: The Find dialog box options depend on the active window.

Use this control... To do this...

Find What Specify the search string.

Use an asterisk (*) to match any string of characters, and a question

mark (?) to match any single character.

Match Case Specify if the search must match the case of the string.

If this option is not selected, lowercase and uppercase letters will be

treated the same in the search.

Scope

Specify the properties to search for.

/Important

In the part editor, only Pins and Text is available as the options for scope.

However in project manager or schematic page editor, you can specify any of the following scopes:

- Parts
- Nets
- Title Blocks
- Off-page Connectors
- Flat Nets
- Power/GND
- Bookmarks
- Hierarchical Ports
- Text
- DRC Markers
- Parts Pin

You may specify the following scope in the design variant schematic page:

Variant Parts

For information on how to use the Variant Parts option, see the OrCAD X Capture CIS User Guide.

This option is not available in the session log or the text editor.

FPGA Export Dialog Box

To open this dialog

In the Project manager choose Export FPGA from the Tools menu for a Programmable Logic project.

Use this control	To do this
FPGA Family Vendor	Select a FPGA family from the list.
FPGA Component	Select the component you want to export. You can select All to export all the components.
Generate Wrapper File	Generate a wrapper file for the component in Verilog or VHDL
Exclude Power/GND/NC pins	Check to exclude the power, ground, and NC pins from being exported.
Output Reserve Pins	Check this option to display any reserve pin assignments made in the Reserve Pins dialog box in the output file.
	Any assignment marked as none will not appear in the output file.
Reserve Pins	Open the Reserve Pins dialog. See description of this dialog below.
Generate TCL File	Select to generate the output in the form of a TCL file for Altera and UCF for XILINX.
Net Name V/s Pin Number	Select to generate the TCL file with net names and pin numbers. Enabled for Generate TCL File option.
Pin Names V/S Pin Number	Select to generate the TCL file with pin name and pin number. Enabled for Generate TCL File option.
Generate CSV File	Select to generate the output in the form of a CSV file.
Output Directory	Specify the output file and its location.
View Output	Check to open the generated file

Reserve Pins Dialog Box

To open this dialog

In the FPGA dialog box, click the Reserve Pins button.

Use this control... To do this... Select the Input pin type from the drop-down list. Input **Bidirectional** Select the Bidirectional pin type from the drop-down list. Output Select the Output pin type from the drop-down list. Auto Assign Reserve Pin Check this option to make pin assignments as per the Assignment Global Settings for the Input, Bidirectional and Output pin types. These assignments will be made in the Pin grid. This can be overwritten for individual pins in the Pins grid. Reserve Pin Assignment Select a pin type from the grid cell drop-down for each

pin on the FPGA component.

FPGA Options Dialog Box

To open this dialog

In the Generate Part dialog box, click the FPGA Setup button.

General tab

Use this control	To do this
Create single section part	Select to create a part with only one section. This will
	result in the parts per package property to be set to 1.

Create multi-section part Select to create a part with more than one section, where

pins have separate symbols based on I/O banks

specified in pin or pad file. This is suitable for parts with

large pin counts.

Note: When you select this option Split Part Section Input Spreadsheet opens on closing the Generate part dialog box. You can modify the generated part using Split Part

Section Input Spreadsheet.

Separate symbols for power/NC

pins

Check to specify separate symbols for power or NC pins. This will create Separate symbols for Power/NC pins.

Power/NC Pin limit Specify the number of Power/NC pins to be in one

section. If number of Power/NC pins cross the specified limit, an additional power/NC symbols will be created.

Add Pin-Group to pins Check to specify auto pin-group information based either

on same I/O bank or pins with compatible I/O standard.

Based on same I/O Bank Check to add pin-group values to pins within the same

I/O Bank

Based on same I/O standard Check to add pin-group values to pins within same I/O

Bank with compatible I/O standard. Pins with missing I/O standards are grouped based on same I/O Bank and

same pin type.

Add Power/Gnd pins as invisible Check to ensure if power pins be made visible on the

symbol or made invisible.

Rename Duplicate Pins Check to ensure that pins with duplicate names are

renamed during Generate part. The new name is of the format <original name>_<pin number>. For example, for a pin named A and numbered 2, the new

name will be A 2.

Pin Defaults tab

Use this control... To do this...

Modify pin directions

Check to modify default pin direction and shape. The default pin-direction and shape settings are currently honored by Generate part during import. You can set the direction to Left, Right, Top, or Bottom and the shape to Line, Clock, Dot, Dot-Clock, Line, Short, Short Clock, Short Dot, Short Dot-Clock, Zero length.

Generate Part dialog box

To open this dialog

In the Project manager, choose Generate Part (see <u>Generate Part command</u>) from the Tools menu.

Note: To create a pin on a symbol using the Generate Part utility, the pin must have a pin to port mapping in the pin file.

Use this control... To do this...

Netlist/Source file Specifies the netlist or schematic file that Capture uses to

generate the new symbol. Typically, this is the netlist that is associated with the timing information derived from the vendor fitter tool. You can enter the path and name of the file directly, or

use the Browse button to select it.

Netlist/source file type

Specifies the format of the netlist or other source file that Capture uses to generate the symbol. If you select a netlist file name using the Browse button, Capture assigns a default vendor file type based on the netlist file extension. Otherwise, you must select a file type from the drop-down list. You can choose from the following netlist or file types:

- **Actel Pin File.** This file is typically created by the Actel Designer tool.
- Altera Pin File. This file is created by the Altera MAX+PLUS II.
- APD BGA/Die-Text File. This file is created by the Advanced Package Design tool.
- Capture Schematic/Design. A source design or library file from an external file or from the current design. This is automatically set when a schematic folder, which is in a design (.DSN) file or library (.OLB) file, is selected in the project manager at the time the Generate Part dialog box is opened.
- EDIF Netlist. Altera's MAX+PLUS II tool generates an EDIF netlist. You can also use EDIF files from any other source to generate a symbol.
- Lattice JEDEC File. This file is created by Lattice ISP products
- Lattice Pin File. This file is created by the Lattice ispExpert tool.
- Lucent ORCA Pad File. This file is created by the Lucent ORCA tool.
- PSpice Model Library. Used by the PSpice simulator. You can create your own PSpice model libraries using the PSpice Model Editor or use the model libraries that ship with PSpice and install in the Library directory.
- Verilog Netlist. Used for board simulation and for FPGA projects. You can use these netlists to generate a symbol.
- VHDL Netlist. Some vendor tools generate VHDL netlists (with embedded timing information) during place and route. You can use these netlists to generate a symbol.

Netlist/source file type (contd.)

- Xilinx M1 Pad File. This file is created by the Xilinx M1 tool.
- Xilinx Pin File. This file is created by Xilinx place and route tool set.
- XNF Netlist. XNF netlists are the results of the XACTstep place and route tool.

Primitive

Assigns No, Yes, or Default value to the Primitive property. This option is only available for the Capture Schematic/Design source file type. If the value is set to No, you can descend the hierarchy of the placed part instance to see the source schematic.

Copy schematic to library

When this check box is selected, Capture places a copy of the source schematic in the new library created in the Outputs directory of the project manager when a part symbol is generated using a schematic source file. This option is only available for the Capture Schematic/Design source file type. If the part name does not match the source schematic name, the resulting part and schematic will have the same name.

Part name

Specify the name that Capture assigns to the newly generated symbol. If you selected a netlist file name using the Browse button, Capture assigns a default symbol name to this text box that corresponds to the netlist name. Otherwise, you must enter the symbol name directly.

Destination part library

Specifies the name that Capture assigns to the symbol library that will contain the new symbol. If you selected a netlist file name using the Browse button or a schematic file in the project manager, Capture assigns a default symbol library name to this text box that corresponds to the netlist or schematic name and adds a .OLB extension. You can accept the default entry, enter the path and name of the file directly, or use the Browse button to select it.

Create new part

Specifies to create a new part using the specified netlist.

Pick symbols manually

(This check box is available only when you have a PSpice project open) Associate a PSpice model to a Capture symbol. When you click the OK button in the Generate Part dialog box, the <u>Model Import Wizard</u> appears allowing you to associate a PSpice model to an existing symbol.

(This check box is disabled if the PSpice Model Library item is not selected from the Netlist/source file type drop-down list)

Update pins on existing

Specifies to update the pins on an existing part using the specified netlist, rather than create a new part.

Sort pins

part in library

Ascending order Specify that the pins are sorted in ascending order.

Descending order Specify that the pins are sorted in descending order.

Additional pins

Specify the number of additional pins on part

Check to specify the number of pins Capture creates for the part. By default, Capture creates only the number of pins required such that each input and output specification in the netlist has a unique pin. However, if you are using a particular device, you may want to specify a number of pins that differs from the number of input and output specifications in the netlist.

Number of pins

Specify the number of pins that Capture generates for the part. This option is only available if you have activated the Specify the number of pins on part check box. Any unused pins on the symbol (pins for which there is no input or output specification in the netlist) are considered I/O pins.

Retain alpha-numeric pin-numbers. Device is pin grid array type package.

Check to retain the alphanumeric pin names for the part (for example, "P20"). This is useful for parts that model Xilinx pin grid array type packages. If the Vendor file type in this dialog box is anything other than Xilinx Pin File or Xilinx Pad File, this option is ignored.

Implementation

Implementation type

Specify the type of implementation. The implementation types available to choose from are the same as those available in the Attach Implementation dialog box.

The most common Implementation type used with the parts created from PLD vendor pin reports is either <none> or Project (which creates a hierarchy of projects for system simulation). Implementation types signify the following:

- <none> Primitive library part.
- EDIF Non-primitive library part. Contents defined by an EDIF netlist generated by a third party EDA tool.
- **Project** Primitive library part. Associated with the Simulation Resources of an OrCAD X Express project for system-level simulation.
- **Schematic View** Non-primitive library part. Contents defined by a schematic folder/page.
- VHDL Non-primitive library part. Contents defined by a VHDL model.

Implementation name

Specify the name of the attached object.

Implementation file

Specify the path and name of the library or file of the attached object.

FPGA Setup

Open the FPGA Options dialog box. Using this dialog box, you can specify settings for FPGA symbols, FPGA pins, FPGA pin swapping, and pin shape or pin direction.

Go To dialog box

To open this dialog

Select *View – Go To*, or see <u>Go To command</u>.

Note: The grid reference and bookmark options are not available in the part editor.

Location tab

Use this control... To do this...

X and Y Specify the X and Y coordinates for the jump.

Relative

Location Type: Absolute and Specify if the jump is absolute (to the indicated coordinates), or relative (using the coordinates as an

offset to the pointer's current position).

Grid Reference tab

Use this control... To do this...

Horizontal Specify a horizontal grid reference.

Vertical Specify a vertical grid reference.

Bookmark tab

Use this control... To do this...

Name Specify a jump to a bookmark. Bookmarks are made

using the Bookmark command on the Place menu.

Note: The Go To command is used to go to bookmarks

on the currently active schematic page.

Goto Label State dialog box

To open this dialog

In the schematic page editor, choose Label State - Goto from the Edit menu.

Use this control... To do this...

Enter Label Specifies the label of the state to which you want to

return the schematic.

Go To Line dialog box

To open this dialog

In the text editor, choose Go To from the Edit menu.

Use this control... To do this...

Line Number Specifies the line number to view in the text editor

window.

Hierarchical PSpice Netlist Settings dialog box

Use this	control	To do	this

design become global in scope. If the option is

cleared, the PARAM parts are local to the subcircuit in

which they occur.

Sub-circuit Patterns

Products Specifies which group of settings is active for the

netlister. Your choices are PSpice or LVS (Layout versus Schematics) netlist formats. Selecting a different group of settings changes the defaults in the Sub-circuit Patterns frame to reflect the settings of the

specified tool.

Global Net Prefix Defines the syntax of the global net of a subcircuit.

The prefix defined here is added to all global net names. For example, if the selected product is PSpice, the prefix "\$G_" would be prepended to the "GND" net name, making it "\$G_GND". If you are netlisting a PSpice design, leave the prefix as is.

Otherwise the design won't simulate.

Reference

Subcircuit Call Specifies the syntax of the subcircuit call using a

modified TEMPLATE syntax.

ParamList Element Definition Specifies the syntax of how parameters are passed

from a reference to a part definition.

Definition

Subcircuit Header Specifies the syntax of the subcircuit header using a

modified TEMPLATE syntax. If modified, you must make sure the definition header is consistent with the

call.

ParamList Element Definition Specifies the syntax of how parameters are passed

from a reference to a part definition.

Param Usage Reference Specifies the syntax used to enclose the parameters

in references.

Subcircuit Ends Specifies the syntax used for the termination of a

subcircuit.

Save as Project Default Settings Saves the current settings in the CAPTURE.INI file,

making the current settings the default settings for any

new Capture projects.

For more detailed information about the syntax for these commands, and examples of how to use them, see the *PSpice A/D Reference Guide*.

Identify DC Nets

To open this dialog

Choose SI Analysis - Identify DC Nets.

Use this control... To do this...

Global NetName Lists the DC nets.

VOLTAGE Displays the voltage values. You can add, modify, or

delete values.

Import Design dialog box

To open this dialog

Choose Import Design (see Import Design command) from the File menu.

PSpice tab

Use this control... To do this...

Open Specify the name of the .SCH design file to be translated.

Save As Specify the name of the .OPJ file for the design to be

saved as.

PSPICE.INI File Specify the path and filename of the PSPICE.INI file.

Translate Hierarchy Keep the hierarchy of the Schematics design intact,

when translating to Capture.

Consolidate all Schematic files

into one Design file

If the Translate Hierarchy option is selected, translate the

Schematics files into one Capture design file.

Create Simulation Profile for

Root Schematic Only

If the Translate Hierarchy option is selected, only create

simulation profiles for the root schematic.

EDIF tab

Use this control... To do this...

Open Specify the name of the EDIF (*.ED*) file to be

translated. The file must be a graphical EDIF design file,

and not an EDIF netlist.

Save As Specify the name of the .DSN file for the design to be

saved as.

Configuration File Specify the name of the configuration file (.CFG) for the

translation. Configuration files are not required for

translation.

PDIF tab

Use this control... To do this...

Open Specify the name of the design file to be translated.

Save As Specify the name of the .DSN file for the design to be

saved as.

Import Selection dialog box

To open this dialog

In the schematic page editor, choose Import Selection (see <u>Import Selection command</u>) from the File menu.

Use this control	To do this
Block	Specify a block to select and view. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.
Block List	Display a list of blocks in the libraries selected in the Libraries list box that match what's entered in the Block text box. When you select a block in this list, its name appears in the Block text box, and its graphic appears in the preview box.
Libraries	Select one or more libraries from the list of available libraries. The Block list displays the blocks from the selected libraries. You also select libraries from the Libraries list box to remove them from the list.
Preview box	Display the graphic of the selected block.
Add Library	Display a standard Windows Open dialog box for adding a library to the Libraries list box. You can add a library in SDT format. If you do, you can save the library in the Capture format (.OLB).
Remove Library	Remove the selected library or libraries from the libraries list box.

Intersheet References dialog box

To open this dialog

In the Project manager

Click Add Intersheet References in the Annotate dialog box, and click OK.

415

OR

Choose Intersheet References in the Tools menu.

Note: Intersheet references are derived from the page number in the title block.

Use this control	To do this
Place On Off Page Connectors	Specify if intersheet references are placed on off-page connectors. If this option is not selected, intersheet references won't be placed on off-page connectors.
Position	
Offset Relative to Port	Specify that the positions of intersheet references are relative to their respective ports.
Offset Relative to Port Name	Specify that the positions of intersheet references are relative to their respective port names.
Reset Positions	Specify that existing intersheet references will be reset if their port or port name has moved. If this option is not specified, then existing intersheet references are not moved.
X Offset	Specify the distance between the intersheet reference and either its relative port or port name
Format	
Standard (1, 2, 3)	Specify that all intersheet references listed for a given port.
	For example, if a signal exists on pages 1,2,3 and 5, then on page 1 the intersheet reference is defined as 2,3,5.
Abbreviated (13)	Specify an abbreviated list of intersheet references listed for a given port.
	For example, if a signal exists on pages 1,2,3,4 and 5, then on page 1 the intersheet reference is defined as 25. This indicates all pages from 2 through 5.
Grid(1A5[Zone][Num])	Specify a list of intersheet references by schematic page zone for a given port.
	For example, if a signal exists on pages 1, 2 and 3. On page 2 the connecting signal exists in the schematic page zone 3D. On page 3 the connecting signal exists in the zone 4C. In this case, the intersheet reference on page 1 is defined as 3D2, 4C3.

Prefix Specify a prefix that is appended to the front of all

intersheet references.

Suffix Specify a suffix that is appended to the back of all

intersheet references.

Port Type Match Matrix Specify which pin-to-pin pairings that Capture will

generate intersheet references for.

For example if you select the Input/Output option, then intersheet references will be placed in all cases when an input port on one schematic page connects to an output

port on another schematic page.

SelectAll Selects all the check boxes options in the matrix

Deselect All Deselects all the check boxes options in the matrix

Restore Defaults Option Restores the application default selections in the Port

Type Match matrix.

Report File Specify (or browse for) the name and location of a new

CSV file that Capture will use to generate a report of all

the intersheet references on the design.

View Output Specify to view the Report file as soon as the intersheet

reference generation is complete.

ISCF Export dialog box

To open this dialog box

Use this control... To do this...

Input Design Specify the input design(.dsn) path.

Output File Specify the output file (.iscf) path.

Log File Specify the log file (.log) path.

Properties When the dialog box is launched, some default part

properties and pin properties are present.

To add new part properties and pin properties, write the property name in the input text field and add them using

the addition symbol (+).

Ground Net When the dialog box is launched, some default ground

nets are present.

Add new ground nets by writing next to the default

ground nets.

XML To DSN dialog box

To open this dialog

Choose File – Import – Design XML.

Use this control	To do this		
XML File	Specify the input XML(.xml) path.		
DSN File	Specify the output design file (.DSN) path.		
Overwrite Mode	Select the following options from the drop-down list:		
	■ Don't overwrite; generate new design name		
	■ Replace existing design		
	■ Add/Replace page in existing design		
Log File	Select to specify the log file (.log) path.		
Generate Tcl	Select to specify the TCL file (.tcl) path.		
XML Schema	Displays the non-editable path of the XML schema used to generate the Capture design file from the XML file.		

XML To OLB dialog box

To open this dialog

Choose File - Import - Library XML.

Use this	control	То	do	this
OSC tills		10	uU	

XML File Specify the input XML(.xml) path.

OLB File Specify the output library file (.olb) path.

Overwrite Mode Select the following options from the drop-down list:

Don't overwrite; generate new library name

■ Replace existing library

Add/Replace page in existing library

Log File Select to specify the log file (.log) path.

Generate Tcl Select to specify the TCL file (.tcl) path.

XML Schema Displays the non-editable path of the XML schema used

to generate the Capture library file from the XML file.

Library Management dialog box

To open this dialog

Click Launch Library Management in the Library Setup (SI Analysis) dialog box (SI Analysis – SI Library Setup).

Use this control...

Working Library

Select to specify the library as the working library.

Ignore Library

Select to ignore the library.

Click to launch Model Integrity.

Library Setup (SI Analysis) dialog box

To open this dialog

In the Project manager, choose SI Analysis – SI Library Setup.

Use this control	To do this
	Click to add a new library.
×	Click to remove the selected library from the list.
LM	Click to open the Library Management dialog box.
+	Click to move up the selected library in the list.
4	Click to move down the selected library in the list.
Working	Displays the name of the working directory.

Library XML dialog box

To open this dialog

Choose File - Export - Library XML.

Use this control... To do this...

OLB File Specify the path of the Capture library file (.olb).

XML File Specify the path of the XML file (.xml).

Log File Select to specify the log file (.log) path.

View Output Select to view the XML output file in Capture

XML Schema Displays the non-editable path of the XML schema

used to generate the XML file from the Capture

Library file.

Markers dialog box

To open this dialog

In the Project manager, choose Marker List from the PSpice menu.

OR

In a PSpice schematic page, point to Markers on the PSpice menu and choose List.

Use this control	To do this
Markers List	Display or hide markers on your design. Select the check box next to the listed marker to display markers on the schematic.
	If the check box next to a marker is not selected, it is hidden and will not display in Capture. However, the marker still exists in the profile. This feature is useful for printing a design for documentation.
Go To	If only one marker is selected, click this button to open the schematic page that contains the marker.
	Note: Removing any or all markers from the list removes them from the schematic. You cannot undo the Remove or Remove All operation.
Remove All	Remove all markers from the list and from the design.
Remove	Remove the selected markers from the list and from the design.

Monte Carlo Worst-Case Output File Options dialog box

To open this dialog

Select the Monte Carlo/Worst Case option, from the Analysis tab of the Simulation Settings dialog box, and click the More Settings button.

Find

Find the indicated function on the values of the output variable and reduce these to a single value. The value is the basis for the comparisons between the nominal and subsequent runs. The following functions are available:

- YMAX. Find the absolute value of the greatest difference in each waveform from the nominal run.
- MAX. Find the maximum value of each waveform.
- MIN. Find the minimum value of each waveform.
- RISE_EDGE. Find the first occurrence of the waveform crossing above the threshold value. The waveform must have one or more points at or below the threshold value, followed by one above. The output value listed is the first point that the waveform increases above the threshold value.
- **FALL_EDGE.** Find the first occurrence of the waveform crossing below the threshold value. The waveform must have one or more points above the threshold value, followed by one below. The output value listed is the first point that the waveform decreases below the threshold value.

Threshold value

Specify the value used in the RISE_EDGE and FALL_EDGE functions.

Evaluate only when the sweep variable is in the range

Specify a beginning and ending range to evaluate the sweep variable in.

List model parameter values in the output file for each turn

List the model parameter values in the output file for each run of the Monte Carlo/Worst Case analysis.

Multi-level Backup Settings dialog box

To back up your design, enter the values to determine the duration, number of backups and storage location.

To open this dialog

Choose Autobackup option from the Options menu.

Use this control... To do this...

Backup time (in minutes)

Enables you to determine the time after which Capture

will perform automatic backup.

No of backups to keep Enables you to determine the total number of backups

that will be stored.

Directory for backup Enables you to determine the storage location for the

backup.

Model Import Wizard

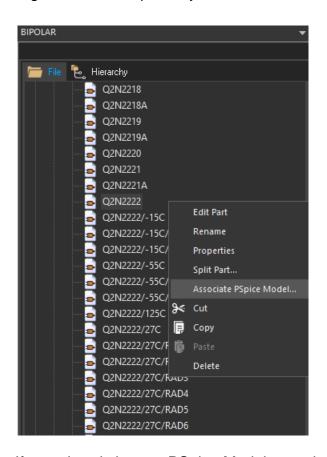
Associating PSpice Model to Capture Symbols

The procedure covered in this section can be used to attach PSpice models only to homogeneous part symbols.

To associate a PSpice model to a Capture Symbol, do the following:

- 1. Open the library (.OLB) containing the symbol for which you want to associate the PSpice model.
- **2.** Select the required Capture symbol and choose *Tools Associate PSpice Model* (see Associate PSpice Model command).

OR



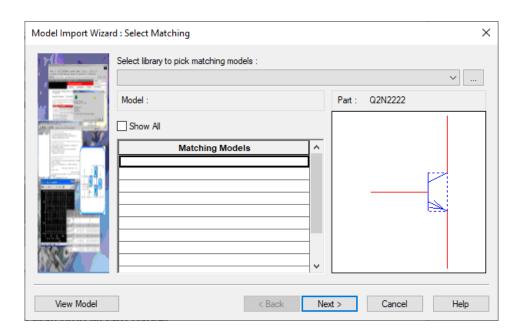
Right-click the Capture symbol and choose Associate PSpice Model.

3. If you already have a PSpice Model associated with the selected symbol, you get a warning stating that the implementation property is already defined. Ignore the warning and click *Yes*.

Note:

- If the symbol is open in Capture, the *Associate PSpice Model* menu option will appear disabled.
- If the selected symbol in the . OLB file has a Convert view, it will be ignored by the Model Import Wizard. The PSpice model gets attached to the symbol in the normal view.

The Select Matching page of the Model Import Wizard appears.



Select Matching wizard page

Use this control...

To do this...

Select library to pick matching models

Specify the path to the library that contains the PSpice model to be associated with the selected Capture symbol.

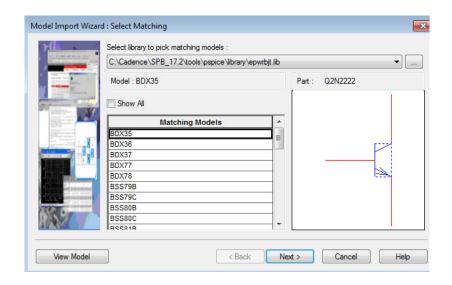
Matching Models

You can either select the library from the model library drop-down list or can browse to the library location.

Displays a list of PSpice models in the selected model library that can be associated with the Capture symbol.

Once you have selected the .lib file, the Model Import Wizard lists the matching models, and displays the currently selected model in the list.

In the Select Matching page of the wizard, select a model from the Matching Models list box and click Next.



View Model

Displays the model definition for the PSpice model currently selected from the *Matching Models* list.

Symbol pane

Displays the name and the graphic for the Capture symbol to which PSpice model is to be attached.

Next

Move to the next step.

Cancel

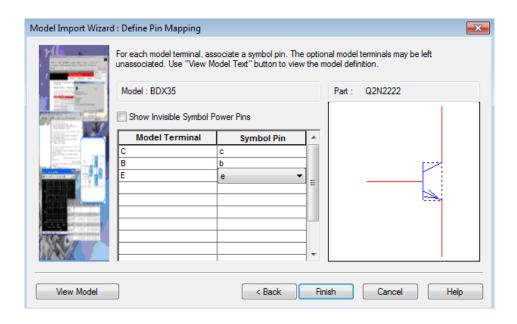
Cancel the process of associating an existing symbol to a simulation model.

Define Pin Mapping wizard page

In the *Define Pin Mapping* page of the wizard, map each of the model terminal to a symbol pin. The pin names on the selected symbol appear in the *Symbol Pin* drop-down list.

While you complete the pin-port mapping, you can view the symbol shape in the Symbol pane on the right of the wizard, and the use the *View Model* button to view the model definition.

All the Capture symbol pins must be mapped to a model terminal. After you have mapped each symbol pin to a unique model terminal, if there are any optional model terminals left, you may leave them unmapped.



Use this control...

Model Terminal
Symbol Pin

To do this...

List the port names from the model definition

List the symbol pin names.

From the drop-down list, select the pin name that is to be associated with the listed model terminal.

Optional Model Terminals List the optional ports from the model definition.

Depending on the availability of symbol pins, you may or

may not map these.

Symbol pane Displays the name and the graphic for the Capture

symbol to which PSpice model is to be attached.

Show Invisible Symbol Power

Pins

Select this check box to view and map any invisible

power pins in the symbol.

Back Move to the previous step, where you selected a

matching symbol.

Cancel Cancel the process of associating an existing symbol to a

simulation model.

Save Symbol Complete the process of associating a symbol to the

selected model and to go back to the Associate/Replace

Symbol page.

Note: If you are using Model Import Wizard to associate a PSpice model to a Capture symbol, the *Finish* button is visible instead of the *Save Symbol* button. Click the *Finish* button to complete the process of associating the selected PSpice model to the Capture symbol and close

the wizard.

View Model Display the model text for the selected model in a new

window.

For mapping you may want to view the model definition.

For this use the *View Model* button.

Finish To complete attaching a PSpice model to the selected

symbol and to close the Model Import wizard, click

Finish.

A message appears indicating that the PSpice model is now attached to the selected symbol.

When you use Model Import Wizard to associate a model to a symbol, the following symbol properties are updated:

- Value of the Implementation Type property
- Value of the Implementation property

Note: The value of the Implementation property attached to the part symbol, should match the name of the PSpice model as it appears in the .MODEL or .SUBCKT definition.

Value of the property (not required for template-based models)

Associating Parts to a PSpice Model

Associate/Replace Symbol wizard page

Use this control... To do this...

Models with symbol List the PSpice models, for which the Model Import

Wizard can find matching symbols, along with the

corresponding symbol names.

Models without symbol List the PSpice models for which matching symbols

could not be found.

Associate Symbol or Replace

Symbol

Select this toggle button when you want to attach an existing symbol to the selected simulation model.

The Associate Symbol button is appears when you select a model from the Models without symbols list. Use the Associate Symbol button to associate an

existing symbol to a model without symbol.

The *Replace Symbol* button is appears when the selected model already has a symbol associated to it. Use the *Replace Symbol* button to replace a symbol associated to a model, by an existing symbol of your

choice.

Finish Stop the process of creating symbols.

In case you have models that do not have any symbols associated to them, a message appears, asking you whether rectangular symbols should be attached to

these models or not.

View Model Display the model text for the selected model in a new

window.

Select Matching wizard page

Use this control... To do this...

Select a library to pick Specify the library that contains the symbol you want to

matching symbols associate the selected model.

You can either use the Browse button to navigate to the desired .OLB file, or select the file from the drop-down list. The drop-down menu lists a maximum of 10 most

recently used libraries.

Matching Symbols List all the symbols that can be associated with the

model selected in Associate/Replace Symbol page.

The matching symbols list is generated based on model

definition of the selected model by the user.

Symbol pane Display the graphical shape of the symbol currently

selected in the *Matching Symbols* list.

Next Move to the next step.

Cancel the process of associating an existing symbol to a

simulation model.

View Model Display the model text for the selected model in a new

window.

Define Pin Mapping wizard page

Use this page for pin to port mapping between the selected symbol shape and the model definition. While you complete the pin-port mapping, you can view the symbol shape in the symbol pane on the right of the wizard, and the use the *View Model* button to view the model definition.

All the symbol pins must be mapped to a model terminal. After you have mapped each symbol pin to a unique model terminal, if there are any optional model terminals left, you may leave them unmapped.

Use this control... To do this...

Model Terminal List the port names from the model definition.

Symbol Pin List the symbol pin names.

From the drop-down list, select the pin name that is to be

associated with the listed model terminal.

Optional Model Terminals List the optional ports in the model definition.

Depending on the availability of symbol pins, you may or

may not map these.

Symbol pane Display the graphical shape of the symbol currently

selected in the *Matching Symbols* list.

Back Move to the previous step, where you selected a

matching symbol.

Cancel the process of associating an existing symbol to a

simulation model.

Save Symbol Complete the process of associating a symbol to the

selected model and to go back to the Associate/Replace

Symbol page.

Note: If you are using Model Import Wizard to associate a PSpice model to a Capture symbol, the *Finish* button is visible instead of the *Save Symbol* button. Click the *Finish* button to complete the process of associating the selected PSpice model to the Capture symbol and close

the wizard.

View Model Display the model text for the selected model in a new

window.

Push Occ. properties to instance dialog box

To open this dialog

In the Project manager, choose Transfer Occ. Prop. to Instance - Push Occ, Prop into Instance (see <u>Push Occ. Prop into Instance command</u>) from the Accessories menu.

If you copy a circuit or part of a circuit from design A and pasted it in design B, you might see occurrence and instance level properties with different values on the pasted parts/nets in design B. For example, the reference designators of the occurrences and instances may be different. To avoid confusion in the future you have to ensure that each part has only one reference designator by replacing the instance value of the Part Reference property with the occurrence value of the Part Reference property for each part. You can do this automatically using this dialog box.

To transfer occurrence property values of the Part Reference and PCB footprint properties as instance level property values, select the first radio button and click OK.

To remove all occurrence properties from the design and change the preferred mode of design to instance check the occurrence level properties check box.

To transfer occurrence property values of the flat nets to schematic nets, select the Transfer flat net properties to schematic net properties radio button and click OK.

NC Verilog Simulation dialog box

To open this dialog

Choose Board Simulation (see Board simulation command) from the Tools menu.

OR

If you have specified Verilog as the simulation language, use <u>Board Simulation tab</u> of the <u>Preferences dialog box</u>.

Use this option	To do this
Use Interactive	Specify that Capture invoke NC Verilog in interactive mode.
Specify Batch File	Specify that Capture invoke NC Verilog in batch mode, using the batch file specified in the associated text box.
Setup	Specify the parameters for the NC Verilog session through the NC Verilog Simulation Setup dialog box
Run	Start the simulation.

NC Verilog Simulation Setup dialog box

To open this dialog

Click on the Setup button in the NC Verilog Simulation dialog box. This dialog box is organized into a series of tabs that provide a method for setting up various aspects of the simulation session.

Use this tab	To do this
Simulation tab (NC Verilog)	Set options for the simulation run.
Testbench tab (NC Verilog)	Specify or create a testbench that provides stimulus for the design.
Model compilation tab (NC Verilog)	Specify simulation files to compile, and options for the compilation.

Simulation tab (NC Verilog)

The Simulation tab of the NC Verilog dialog box provides a method for specifying the simulation options you will use for your simulation session.

Use this control	To do this
HDL.VAR file	Specify the path to the HDL.VAR file used by NC Verilog during the simulation. This file is a text file that specifies simulation configuration variables, variables that specify options for compiling, elaborating, and simulating your design, and variables that specify the locations for various support files and invocation scripts.
Log Directory	Specify the directory to which NC Verilog stores the simulation log file.
Compile	Specify that NC Verilog compile the Verilog netlist (and any associated testbench) before simulation. During compilation, NC Verilog checks the netlist for syntax and semantics, and notifies you of any errors.
	You can specify NC Verilog command line options for the

compile in the Cmd Options text box. Enter options of the exactly as you would from the command line. For example, to use the *Append_log* command line option, in the Cmd Options text box, type:

-Append_log

Note: You must compile your design at some point before simulation can occur. This option allows you to skip the compile step in cases where you have compiled your design earlier and do not wish to recompile. If you have not compiled your design, however, and do not select this option, NC Verilog will be unable to simulate your design.

Elaborate

Specify that NC Verilog elaborate the Verilog netlist (and any associated testbench) before simulation. During elaboration, NC Verilog constructs a design hierarchy based on the instantiation and configuration information in the design, establishes signal connectivity, and computes initial values for all objects in the design.

You can specify NC Verilog command line options for the elaboration in the Cmd Options text box. Enter options exactly as you would from the command line. For example, to use the *Append_log* command line option, in the Cmd Options text box, type:

-Append log

Note: You must elaborate your design at some point before simulation can occur. This option allows you to skip the elaboration step in cases where you have elaborated your design earlier. If you have not elaborated your design, however, and do not select this option, NC Verilog will be unable to simulate your design.

Note: Note, also, that you must compile your design (using the Compile option) before you can elaborate it.

Start the simulation session.

You can specify NC Verilog command line options for the simulation in the Cmd Options text box. Enter options exactly as you would from the command line. For example, to use the *message* command line option, in the Cmd Options text box, type:

-message

Note: You can choose not to start the simulation session by deselecting this option. This is useful when you want to compile or elaborate your design without performing the actual simulation.

Simulate

Start SimVision

Start the NC Verilog SimVision application (which includes waveform displays).

Note: If you do not select this option, you can still compile, elaborate, or simulate your design (by selecting the corresponding options on the dialog box, but the SimVision application will not be displayed. In this case, when simulation occurs without the waveform display, and the results are reported to a file that you specify as one of the NC Verilog command options.

Enter Interactive Mode

Sets the simulation time for the SimVision session to zero and pauses the simulation to await your input. This option is only available if you choose the Start SimVision option. If, after selecting the SimVision option, you do not select the Enter Interactive Mode option, NC Verilog runs the simulation according to the testbench associated with the design (as specified in the Testbench tab (NC VHDL).)

Note: For specific information on the various NC Verilog commands, as well as the HDL.VAR file, please refer to your NC Verilog documentation.

Testbench tab (NC Verilog)

The Testbench tab of the NC Verilog Simulation dialog box provides a method to specify (or create) a Verilog testbench for your design.

Use this control...

To do this...

Generate Testbench

Create a Verilog testbench template from the top level of your design. The inputs and outputs of the template correspond to the ports and the top level of the design.

Note: When you generate a testbench in this manner, you must still edit the testbench in order to apply stimuli to the design inputs. In order to edit the testbench, choose the Edit button, which causes Capture to open the testbench in a text editor. You can also open the testbench from the project manager, in which case, the testbench is opened using Capture's Verilog editor.

Include Testbench Add an existing testbench to your design in order to

supply stimuli for the simulation. When you select this option, you must specify the path to the testbench in the

Testbench Path field.

None Specify that there is no existing testbench for the design.

Use this option if you want to compile and elaborate your

design without simulating.

Note: For specific information on the various NC Verilog commands, please refer to your NC Verilog documentation.

Model compilation tab (NC Verilog)

The NC Verilog Model Compilation tab provides a method for you to specify and compile the simulation models you will use for your board level simulation. This tab is only available for board level simulation. You will not see this tab as an option for FPGA simulation (functional or timing).

Use this control	To do this
Library extensions	Specify the file extensions of the simulation models to be compiled. Typically, Verilog models have .v or .verilog extensions. You can specify more than one extension by using the "+" symbol. For example:
	+.v +.verilog
Path	Specify the directory in which the simulation models reside.
Verilog Files/Directories	Navigate to the directory containing the simulation models by using the controls in the upper right corner.
Compile location	Specify the directory into which the compiled models will be stored.
Compile at Source	Specify that the compiled models be stored in the same directory as the source files.
Compile	Start the compilation.

New Property dialog box

To open this dialog

Click the New button in the <u>Display Properties dialog box</u>.

Use this control... To do this...

Name Add the name of the new property column in the Part

Editor.

Value Specify an appropriate value for the new property added.

NC Verilog Simulation Setup dialog box

To open this dialog

Click the Setup button on the NC Verilog Simulation dialog box.

This dialog box is organized into a series of tabs that provide a method for setting up various aspects of the simulation session.

Use this tab...To do this...Simulation tab (NC Verilog)Set options for the simulation run.Testbench tab (NC Verilog)Specify or create a testbench that provides stimulus for the design.Model compilation tab (NC Verilog)Specify simulation files to compile, and options for the compilation.

Simulation tab (NC Verilog)

The Simulation tab of the NC Verilog dialog box provides a method for specifying the simulation options you will use for your simulation session.

Use this control... To do this...

HDL.VAR file

Specify the path to the HDL.VAR file used by NC Verilog during the simulation. This file is a text file that specifies simulation configuration variables, variables that specify options for compiling, elaborating, and simulating your design, and variables that specify the locations for various support files and invocation scripts.

Log Directory

Specify the directory to which NC Verilog stores the simulation log file.

Compile

Specify that NC Verilog compile the Verilog netlist (and any associated testbench) before simulation. During compilation, NC Verilog checks the netlist for syntax and semantics, and notifies you of any errors.

You can specify NC Verilog command line options for the compile in the Cmd Options text box. Enter options exactly as you would from the command line. For example, to use the *Append_log* command line option, in the Cmd Options text box, type:

-Append log

Note: You must compile your design at some point before simulation can occur. This option allows you to skip the compile step in cases where you have compiled your design earlier and do not wish to recompile. If you have not compiled your design, however, and do not select this option, NC Verilog will be unable to simulate your design.

Elaborate

Specify that NC Verilog elaborate the Verilog netlist (and any associated testbench) before simulation. During elaboration, NC Verilog constructs a design hierarchy based on the instantiation and configuration information in the design, establishes signal connectivity, and computes initial values for all objects in the design.

You can specify NC Verilog command line options for the elaboration in the Cmd Options text box. Enter options exactly as you would from the command line. For example, to use the *Append_log* command line option, in the Cmd Options text box, type:

-Append log

Note: You must elaborate your design at some point before simulation can occur. This option allows you to skip the elaboration step in cases where you have elaborated your design earlier. If you have not elaborated your design, however, and do not select this option, NC Verilog will be unable to simulate your design.

Note: Note, also, that you must compile your design (using the Compile option) before you can elaborate it.

Start the simulation session.

You can specify NC Verilog command line options for the simulation in the Cmd Options text box. Enter options exactly as you would from the command line. For example, to use the *message* command line option, in the Cmd Options text box, type:

-message

Note: You can choose not to start the simulation session by deselecting this option. This is useful when you want to compile or elaborate your design without performing the actual simulation.

Simulate

Start SimVision

Start the NC Verilog SimVision application (which includes waveform displays).

Note: If you do not select this option, you can still compile, elaborate, or simulate your design (by selecting the corresponding options on the dialog box, but the SimVision application will not be displayed. In this case, when simulation occurs without the waveform display, and the results are reported to a file that you specify as one of the NC Verilog command options.

Enter Interactive Mode

Sets the simulation time for the SimVision session to zero and pauses the simulation to await your input. This option is only available if you choose the Start SimVision option. If, after selecting the SimVision option, you do not select the Enter Interactive Mode option, NC Verilog runs the simulation according to the testbench associated with the design (as specified in the <u>Testbench tab (NC VHDL)</u>.)

Note: For specific information on the various NC Verilog commands, as well as the HDL.VAR file, please refer to your NC Verilog documentation.

Testbench tab (NC Verilog)

The Testbench tab of the <u>NC Verilog Simulation dialog box</u> provides a method to specify (or create) a Verilog testbench for your design.

Use this control...

To do this...

Generate Testbench

Create a Verilog testbench template from the top level of your design. The inputs and outputs of the template correspond to the ports and the top level of the design.

Note: When you generate a testbench in this manner, you must still edit the testbench in order to apply stimuli to the design inputs. In order to edit the testbench, choose the Edit button, which causes Capture to open the testbench in a text editor. You can also open the testbench from the project manager, in which case, the testbench is opened using Capture's Verilog editor.

Include Testbench Add an existing testbench to your design in order to

supply stimuli for the simulation. When you select this option, you must specify the path to the testbench in the

Testbench Path field.

None Specify that there is no existing testbench for the design.

Use this option if you want to compile and elaborate your

design without simulating.

Note: For specific information on the various NC Verilog commands, please refer to your NC Verilog documentation.

Model compilation tab (NC Verilog)

The NC Verilog Model Compilation tab provides a method for you to specify and compile the simulation models you will use for your board level simulation. This tab is only available for board level simulation. You will not see this tab as an option for FPGA simulation (functional or timing).

Use this control	To do this
Library extensions	Specify the file extensions of the simulation models to be compiled. Typically, Verilog models have .v or .verilog extensions. You can specify more than one extension by using the "+" symbol. For example:
	+.v +.verilog
Path	Specify the directory in which the simulation models reside.
Verilog Files/Directories	Navigate to the directory containing the simulation models by using the controls in the upper right corner.
Compile location	Specify the directory into which the compiled models will be stored.
Compile at Source	Specify that the compiled models be stored in the same directory as the source files.
Compile	Start the compilation.

NC VHDL Postroute Simulation dialog box

To open this dialog

Choose Postroute as the simulation configuration in the <u>Select Simulation Configuration</u> <u>dialog box</u> dialog box and click OK.

Use Interactive

Specify that Capture invoke NC VHDL in interactive mode.

Specify Batch File

Specify that Capture invoke NC VHDL in batch mode, using the batch file specified in the associated text box.

Setup

Display the NC VHDL Postroute Simulation Setup dialog box, from which you can specify the parameters of your

simulation

simulation.

Run Start the simulation.

NC VHDL Postroute Simulation Setup dialog box

To open this dialog

Click on the Setup button in the NC VHDL Preroute Simulation dialog box.

This dialog box is organized into a series of tabs that provide a method for setting up various aspects of the simulation session.

Use this tab	To do this
Simulation tab (NC VHDL)	Set options for the simulation run.
Testbench tab (NC VHDL)	Specify or create a testbench that provides stimulus for the design.

Simulation tab (NC VHDL)

The Simulation tab of the NC VHDL Preroute or Postroute Simulation Setup dialog box provides a method for specifying the simulation options you will use for your simulation session.

Use this control	To do this
HDL.VAR file	Specify the path to the HDL.VAR file used by NC VHDL during the simulation. This file is a text file that specifies simulation configuration variables, variables that specify options for compiling, elaborating, and simulating your design, and variables that specify the locations for various support files and invocation scripts.
Log Directory	Specify the directory to which NC VHDL stores the simulation log file.
Compile	Specify that NC VHDL compile the VHDL netlist (and any associated testbench) before simulation. During compilation, NC VHDL checks the netlist for syntax and semantics, and notifies you of any errors.

You can specify NC VHDL command line options for the compile in the Cmd Options text box. Enter options exactly as you would from the command line. For example, to use the *Append_log* command line option, in the Cmd Options text box, type:

-Append_log

Note: You must compile your design at some point before simulation can occur. This option allows you to skip the compile step in cases where you have compiled your design earlier and do not wish to recompile. If you have not compiled your design, however, and do not select this option, NC VHDL will be unable to simulate your design.

Elaborate

Specify that NC VHDL elaborate the VHDL netlist (and any associated testbench) before simulation. During elaboration, NC VHDL constructs a design hierarchy based on the instantiation and configuration information in the design, establishes signal connectivity, and computes initial values for all objects in the design.

You can specify NC VHDL command line options for the elaboration in the Cmd Options text box. Enter options exactly as you would from the command line. For example, to use the *Append_log* command line option, in the Cmd Options text box, type:

-Append log

Note: You must elaborate your design at some point before simulation can occur. This option allows you to skip the elaboration step in cases where you have elaborated your design earlier. If you have not elaborated your design, however, and do not select this option, NC VHDL will be unable to simulate your design.

Note: Note, also, that you must compile your design (using the Compile option) before you can elaborate it.

Specify that NC VHDL start the simulation after the design has been compiled and elaborated.

You can specify NC VHDL command line options for the simulation in the Cmd Options text box. Enter options exactly as you would from the command line. For example, to use the *message* command line option, in the Cmd Options text box, type:

-message

Note: You can choose not to start the simulation session by deselecting this option. This is useful when you want to compile or elaborate your design without performing the actual simulation.

Simulate

Start SimVision Start the NC VHDL SimVision application (which

includes waveform displays).

Note: If you do not select this option, you can still compile, elaborate, or simulate your design (by selecting the corresponding options on the dialog box, but the SimVision application will not be displayed. In this case, when simulation occurs without the waveform display, and the results are reported to a file that you specify as one of

the NC VHDL command options.

Enter Interactive Mode Sets the simulation time for the SimVision session to

zero and pauses the simulation to await your input. This option is only available if you choose the Start SimVision option. If, after selecting the SimVision option, you do not select the Enter Interactive Mode option, NC VHDL runs the simulation according to the testbench associated with

the design (as specified in the Testbench tab (NC

VHDL).)

Note: For specific information on the various NC VHDL commands, as well as the HDL.VAR file, please refer to your NC VHDL documentation.

Testbench tab (NC VHDL)

The Testbench tab of either the NCVHDL Preroute or Postroute Simulation dialog box provides a method to specify (or create) a VHDL testbench for your design.

Use this control... To do this...

Generate Testbench Create a VHDL testbench template from the top level of your design. The inputs and outputs of the template

correspond to the ports and the top level of the design.

Note: When you generate a testbench in this manner, you must still edit the testbench in order to apply stimuli to the design inputs. In order to edit the testbench, choose the Edit button, which causes Capture to open the testbench in the text editor. You can also open the testbench from the project manager, in which case, the testbench is opened using Capture's Verilog editor.

Include Testbench Add an existing testbench to your design in order to

supply stimuli for the simulation. When you select this option, you must specify the path to the testbench in the

Testbench Path field.

None Specify that there is no existing testbench for the design.

Use this option if you want to compile and elaborate your

design without simulating.

Note: For specific information on the various NC VHDL commands, please refer to your NC VHDL documentation.

Model compilation tab (NC VHDL)

The NC VHDL Model Compilation tab provides a method for you to specify and compile the simulation models you will use for your board level simulation. This tab is only available for board level simulation. You will not see this tab as an option for FPGA simulation (functional or timing).

To do this
Specify the file extensions of the simulation models to be compiled. Typically, VHDL models have .vhd or .vhdl extensions. You can specify more than one extension by using the "+" symbol. For example:
+.vhd +.vhdl
Specify the directory in which the simulation models reside.
Navigate to the directory containing the simulation models by using the controls in the upper right corner.
Specify the directory into which the compiled models will be stored.
Specify that the compiled models be stored in the same directory as the source files.
Start the compilation.

NC VHDL Preroute Simulation dialog box

To open this dialog

Choose Preroute on the <u>Select Simulation Configuration dialog box</u>.

Use this option...

Use Interactive

Specify that Capture invoke NC VHDL in interactive mode.

Specify Batch File

Specify that Capture invoke NC VHDL in batch mode, using the batch file specified in the associated text box.

dailing the bator hie apecined in the associated text b

Setup Specify the parameters for the NC VHDL session

through the NC VHDL Simulation Setup dialog box.

Run Start the simulation.

NC VHDL Preroute Simulation Setup dialog box

To open this dialog

Click the Setup button on the NC VHDL Simulation dialog box.

This dialog box is organized into a series of tabs that provide a method for setting up various aspects of the simulation session.

Use this tab	To do this
Simulation tab (NC VHDL)	Set options for the simulation run.
Testbench tab (NC VHDL)	Specify or create a testbench that provides stimulus for the design.
Model compilation tab (NC VHDL)	Specify simulation files to compile, and options for the compilation.

NC VHDL Simulation dialog box

To open this dialog

Choose Board Simulation (see Board simulation command) from the Tools menu.

OR

If you specified VHDL as the simulation language, use <u>Board Simulation tab</u> of the <u>Preferences dialog box</u>.

Use this option	To do this
Use Interactive	Specify that Capture invoke NC VHDL in interactive mode.
Specify Batch File	Specify that Capture invoke NC VHDL in batch mode, using the batch file specified in the associated text box.
Setup	Specify the parameters for the NC VHDL session through the NC VHDL Simulation Setup dialog box.
Run	Start the simulation.

NC VHDL Simulation Setup dialog box

To open this dialog

Click the Setup button on the NC VHDL Simulation dialog box.

This dialog box is organized into a series of tabs that provide a method for setting up various aspects of the simulation session.

Use this tab	To do this
Simulation tab (NC VHDL)	Set options for the simulation run.
Testbench tab (NC VHDL)	Specify or create a testbench that provides stimulus for the design.
Model compilation tab (NC VHDL)	Specify simulation files to compile, and options for the compilation.

NC VHDL Library Compilation dialog box

This dialog box provides a method for compiling the simulation libraries for the target vendor of your FPGA design.

To open this dialog

In the Project manager, choose Compile Vendor Libraries from the Tools menu.

Use this control... To do this...

Simulation library Select the VHDL simulation libraries that correspond to

the target technology for your FPGA design.

Run display in session log Specify that Capture display the commands and results

of the compilation in the session log.

CMD path Specify the path to, and name of, the command file that

contains the compilation command settings for the target

library.

New Alias dialog box

To open this dialog

Click the New button in the Part Aliases dialog box.

Use this control... To do this...

Name Specify the name of the new alias.

New Layout dialog box

To open this dialog box, select *PCB – New Layout*.

Using this dialog box, the Capture design is transferred and creates the PCB Editor board.

Use this control...

To do this...

PCB Layout Folder

This folder provides the location where the .BRD files and PST*.DAT files are to be saved. The default location is the PCB layout folder of the board on which an operation was done the last time.

The default location is the directory named the last time this dialog box was invoked for the current design.

- If this is the first time the design is being netlisted, the default location will be an allegro subfolder in your design directory.
- If the netlist files have been generated previously for the project, then the default is last directory used with this dialog box for a design.

Input Board File

This board is a base (or template file) on top of which the logical schematic data is placed to create a new Output Board File. If there is a board which the netlist was previously imported to, this is the default board file.

If you have not previously created or specified any input board file, then this field is empty and the logic is imported into an empty board.

Output Board File

This is the location for a new board file. The output board uses the input board as a template and the output board receives the logical data from the design.

- If this is the first time the netlist has been run to create or update a board, the name will be DESIGN_FILENAME.BRD found in an allegro subfolder where the design is found. If the path and board name are invalid, Capture displays an error message.
- If you have previously netlisted, the board file listed is the one on which you had performed any operation the last time.

New Page in Schematic dialog box

To open this dialog

Select a schematic folder in the Project manager and choose New Schematic Page from the Design menu.

Use this control... To do this...

Name Specify the schematic folder or schematic page name.

New Part Creation Spreadsheet

You can use the New Part Creation Spreadsheet to create new parts (multi-section/single section). The New Part Creation Spreadsheet has a spreadsheet-like interface that allows you to paste contents copied from a part data sheet to the spreadsheet.

Each row in the New Part Creation Spreadsheet corresponds to a pin while each column corresponds to properties associated with the pins. The property names are listed as the column header.



The multi-section parts created using the New Part Creation Spreadsheet cannot be split. You can only split either a single-section part or parts already split using the New Part Creation Spreadsheet.



To sort on any property, double-click its name in the column header.



You can hide or show a property column in the New Part Creation Spreadsheet. To do this, right-click the property column header you want to hide and select Hide from the shortcut menu. The selected property column will not appear now. To show a property column, right-click the property column header next, on the right-hand side of the hidden property column and select *Unhide*. The hidden property column appears in the New Part Creation Spreadsheet. Alternatively, you can show a property column by:

- □ Double-clicking the column handle (♣) of the property column header.
- Dragging the column handle of the property column header.

(only the last two methods can be used to show a property column, which is the last column in the New Part Creation Spreadsheet).

Note: You can change the order in which the property columns appear in the New Part Creation Spreadsheet. To do this, select the property column header you want to move and drag and drop it to the location where you want it in the New Part Creation Spreadsheet.

Note: The spreadsheet window is resizable. You can resize the window using the resize cursor you see when you move the mouse pointer to any of the edges of the dialog. You can also use the standard Maximize button on the top right corner of the window.

Use this control... To do this...

Part Name Specify a name for the new part.

Part Ref Prefix Specify a part reference for the new part.

Part Numbering

Specify a numbering format (alphabetic or numeric) that should be added as suffix to the current part reference for the new part.

If you select Alphabetic, an alphabet (between A to Z) will be added as a suffix to the current part reference for each of the new part.

If you select Numeric, a number (between 1 and 1024) will be added as a suffix to the current part reference for each of the new part.

Note: The Section property column changes based on your selection in the Part Numbering group. For example, if Alphabetic is selected, the Section property column displays "A".

No. of Sections

If you want to create a multi-section part, specify the number of sections you want to have in your new part in the No. of Sections text box. The New Part Creation Spreadsheet creates single-section parts, by default.

Note: If you select alphabetic numbering, then you can create up to a maximum of 26 sections only. If you select numeric numbering, then you can create up to a maximum of 1024 sections.

Number Specify the pin number.

Name Specify the name of the pin.

Type Specify the type of pin. To change a pin type, select the Type cell, and select *Input, Output, Passive, Open Emitter, Open collector, 3 State, Bidirectional,* or *Power.*

September 2023

© 2023

458



You can select the Type cells for multiple pins simultaneously using the SHIFT+Down Arrow keys and then enter the pin type. The selected Type cells get populated with the pin type of your choice. Alternatively, you can:

- Select the Type cells for multiple pins simultaneously using the SHIFT+Left mouse button click, then press the CTRL key, and then select a pin type of your choice from the list box. The selected Type cells get populated with the pin type of your choice.
- Click the first cell of the range, and then drag to the last cell, and then enter the pin type of your choice. The selected Type cells get populated with the pin type of your choice.

(You can use these methods to make selection in the Shape, Position, and Section property column list boxes also).

Shape

Specify a shape for the pin. To change a pin shape, select the Shape cell, and select Clock, Dot, Dot-Clock, Line, Short, Short Clock, Short Dot, Short Dot-Clock, Zero length

PinGroup

Specify a value for each swappable (input) pin of the part.

Position

Specify the pin position as left, right, top or bottom. To change position for a pin, select the Position cell, and select *Left*, *Right*, *Top*, or *Bottom*.

Section

Specify a section number.

To change a section for a pin, select the Section cell, and select the required section number from the list.



You can select Section cells for multiple pins simultaneously using the SHIFT+Down arrow keys and enter the section number. Alternatively, you can:

- Select the Section cells for multiple pins simultaneously using the SHIFT+Left mouse button click, then press the CTRL key, and then select a section number of your choice from the list box. The selected Section cells get populated with the section number of your choice.
- Click the first cell of the range, and then drag to the last cell, and then enter the section number of your choice. The selected Section cells get populated with the section number of your choice.



You can select alternate Section cells for multiple pins simultaneously using the CTRL+Left mouse button click and enter the section number.

Add Pins

Add new pins at the end of the current row set in the New Part Creation Spreadsheet.

Delete Pins

Delete selected row containing the pin information from the New Part Creation Spreadsheet.



Once you delete a pin from the New Part Creation Spreadsheet, you cannot retrieve it later.

Save

Save the new part. If any warnings are generated during the save operation, a message box appears asking you whether you want to view the warnings. If you want to view the warnings, click the View Warnings button. The New Part Creation Spreadsheet expands and displays a grid showing warnings messages. If you select the Continue button, the part is saved as is.

Hide Warnings Hide the warnings messages.

Show Warnings Show the warnings messages again.

New Part Properties dialog box

To open this dialog

Select a library in the project manager and choose New Part (see <u>New Part command</u>) from the Design menu.

Use this control	To do this
Name	Specify the name of the part. This is used as the default part value when the part is placed on a schematic page. Part names can be up to 31 characters long.
	Note that the "!" character cannot be used in the part name.
Part Reference Prefix	Specify the part reference prefix, such as "C" for capacitor or "R" for resistor. For example:
	C?1(capacitor)
	R?1 (resistor)
PCB Footprint	Specify the PCB footprint name to be included for this part in the netlist. Contains a value for a device consisting of zero or more pads, other objects, and a name.
Create Convert View	Specify whether the part has a convert. You might use the convert to define a <u>DeMorgan equivalent</u> . A part with this option specified will have two views (a normal and a convert) you can switch between once the part is placed.
Multiple-Part Package	
Parts per Pkg	If there are multiple parts in the package, specify the number of parts in the package.

Package Type If the part is a package, specify whether all the parts in

the package have the same graphical representation (homogeneous) or different graphical representations

(heterogeneous).

Note: The package type can only be set at creation time. These options are not available when you edit the part

later.

You should not cut and paste parts between homogeneous and heterogeneous packages.

Part Numbering If the part is a multiple-part package, specify whether

parts in the package are identified by letter or number.

For example:

U?A (alphabetic)

■ U?1 (numeric)

Part Aliases Display the Part Aliases dialog box to add or remove

aliases. Part aliases show up in a library represented by the part symbol with a horizontal line through the center.

Attach Implementation Display the Attach Implementation dialog box so you can

attach a schematic folder to create hierarchy. You must specify the schematic folder's name, but you only need to specify the schematic folder's library or path name if the

schematic folder is not in the current project.

Pin Number Visible Specify whether the pin number (s) for the part should be

displayed when you open the part in the Part editor

window or view the part in the package view.

Note: Be careful not to create recursion in your design. Capture cannot prevent recursion, and the Design Rules Check command does not report it.

Recursion causes Capture to process infinitely as it tries to expand the design, resulting in the loss of any changes you've made to your design since it was last saved.

Note: If you attach external schematic folders or other files to hierarchical blocks in a design or parts in a library, be sure to include the attachments when you pass the design or library to a board fabrication house or to another engineer. Attached schematic folder and other files are not carried along automatically when you copy or move a part, schematic folder, or schematic page to another library, design, or schematic folder. Only the "pointers" to the attached schematic folder and files—that is, their names and the names of the designs or

libraries that contain them—are carried along.

Attached files work much like their counterparts in email—they do not provide an alternative definition of the part (as do attached schematic folders).

Note: When you attach a schematic folder to a part or hierarchical block, you can specify a full path and filename in the Library text box. So, although you can specify a library that hasn't been saved, you should not try to descend into the attached schematic folder until the library that contains the schematic folder has been saved.

If you don't specify a full path and filename in the Library text box, Capture expects to find the attached schematic folder in the same design as the part of hierarchical block to which it is attached. If the specified schematic folder doesn't exist in either the design or library, Capture creates the schematic folder when you descend hierarchy on the part or the hierarchical block.

For compatibility with future versions of Windows, Capture preserves the case of the path and filename as you specify them in the Library text box.

Note: You can access this dialog box after you place a new part. To change the part parameters, change to package view and choose the Package Properties command from the Options menu.

Note: Once you have attached a file and associated a text editor with it, you can use the Descend Hierarchy command to open that file. If you have an attached schematic folder as well as an attached file, Descend Hierarchy opens the schematic folder and not the file.

New Project dialog box

To open this dialog

Choose New - Project (see Project command) from the File menu.

Use this control... To do this...

Name Specify the name of the new project.

Location Specify the path where you want the new project files to

be saved, or use the Browse button to locate the

directory.

Enable PSpice Simulation Select this check box if you intend to have simulation

capabilities in your PCB design.

If you select this check box, the Create PSpice Project

dialog box opens.

New Property dialog box

To open this dialog

Click the New button on the <u>User Properties dialog box</u>.

OR

Click the New button on the Edit Part Properties dialog box.

Note: Property names and values can have up to 256 characters each.

Use this control... To do this...

Name Specify the new property's name.

Value Specify the new property's value.

New Schematic dialog box

To open this dialog

Select a design (.DSN) in the Project manager and choose New Schematic command from the Design menu.

Use this control... To do this...

Name Specify the schematic folder or schematic page name.

New Simulation dialog box

To open this dialog

For a PSpice project in the Project manager, choose New Simulation Profile (see New Simulation Profile command) from the PSpice menu.

Use this control... To do this...

Name Specify the name for the new profile. This is not a file

name.

Inherit From Specify a simulation model to inherit properties from. The

list box displays all of the simulation models in designs

open in Capture.

Root Schematic Shows selected schematic to act as the root for the

simulation profile.

New Symbol Properties dialog box

To open this dialog

Select a library in the Project manager and choose New (see New Symbol command) from the Design menu.

Use this control... To do this...

Name Specify the name for the new symbol.

Note: A Symbol name cannot be more than 31

characters long.

Symbol Type Select the symbol type. Symbols may be one of the

following:

Power

■ Off-Page Connector

Hierarchical Port

■ Title Block

■ Pin Shape

New NetGroup / Modify NetGroup dialog box

The New NetGroup or Modify NetGroup dialog boxes displays when you click the Add NetGroup or Modify NetGroup buttons on the NetGroup dialog box.

Use this control... To do this...

NetGroup Name Enter the name of the NetGroup.

Note: This text box is disabled in the Modify NetGroup

dialog.

Apply Use this button to save the name of the NetGroup.

Note: You cannot add members to a NetGroup, until you

have applied the NetGroup name.

Add a member (NetGroup, bus, or scalar) to the

NetGroup.

Delete a NetGroup member.

Rename a NetGroup member.

UP Move the NetGroup member position up.

Down Move the NetGroup member position down.

Rename NetGroup Member dialog box

The Rename NetGroup Member dialog displays when you click the Rename NetGroup button on the New NetGroup or Modify NetGroup dialog boxes.

Use this control... To do this...

Name Enter the name of the NetGroup member to rename.

Note: When you rename a NetGroup member that is a NetGroup, you need to choose the name of a NetGroup that already exists in this design. Also, the NetGroup to which you rename this member to, must not already exist

in the current NetGroup.

Open dialog box

Use this command to open projects or files.

This is a standard Windows Open dialog box in which you can locate and select the project or file of your choice.

To open this dialog

Choose Open from the File menu.

Use this control	To do this
Look in	Browse the hierarchical drive and directory structure for your system.
File name	Select or type the name of the project or file.
Files of type	Filter files by extension.

Property Sheet Pane

This pane contains the following sections related to part and pins:

- Package Properties
- Part Properties
- Pin Properties
- Text Properties
- Basic Attributes

Note: In an existing part, depending upon the object selected, the required section in the property pane opens.

This pane has the following functions:

- Display Properties
- Delete Current Section
- Add Convert View
- Delete Convert View
- Edit Pins
- Associate PSpice Model

Package Properties

To open this section in the Property Sheet pane

When you select a part in the part editor, and related package properties appear in the *Package Properties* section of the *Property Sheet* pane.

Use this control	To do this
Part Numbering	Displays the numbering format (Alphabetic or Numeric) that is added as suffix to the current part reference for the new or selected part.
	If Alphabetic is selected, you can create a maximum of 26 sections.
	If Numeric is selected, you can create a maximum of 1024 sections.
	This field has a drop-down list using which you can change the part numbering format after part creation or during part editing.
Package Type	Displays whether all the parts in the package have the same graphical representation (homogeneous) or different graphical representations (heterogeneous).
	The package type can only be set at creation time. This option is not available when you edit the part later.
	You should not cut and paste parts between homogeneous and heterogeneous packages.
PCB Footprint	Specify the PCB Footprint name to be included for this part in the netlist. Contains a value for a device consisting of zero or more pads, other objects, and a name.
	PCB Footprint is a reserved property name. If you want to make its value visible on the schematic page, you must do so in the property editor.
Part Reference Prefix	Specify the part reference prefix, such as $\mathbb C$ for capacitor or $\mathbb R$ for resistor.

Section Count	Specify the number of parts in the package. The Section Count field shows the count specified in the Parts per Pkg or No. of Sections field during part creation.
	If Alphabetic is selected in <i>Part Numbering</i> (while part creation), section names will start with A.
	Note: If you create a single-section part with Alphabetic numbering, section name is not displayed and it is automatically converted into a part with Numeric numbering.
	If Numeric is selected in <i>Part Numbering</i> (while part creation), section names will start with 1.
	You can modify the number of sections after part creation.
Part Aliases	Click the <i>Update</i> button to open the Update Alias dialog box, which is used to add and edit part aliases.

Part Properties

To open this section in the Property Sheet pane

Open or select a part in the part editor, and the related part properties appear in the *Part Properties* section of the *Property Sheet* pane.

Use this control	To do this
Name	Displays both the name and normal or convert view of the part. The part name appears to the left of the period, and the view appears to the right. This property is read-only.
suffix	Is used to indicate if the view is Normal or Convert for the current section of the part. This is a read-only property.
Implementation Path	Specify the filename and directory to the child schematic.
	Click (a) icon to browse the location of the child schematic.

Implementation Specify the name of the child schematic for the part.

Implementation Type Specify the implementation type. For information about

various implementation types, see Attach

Implementation dialog box.

Value Specify the part value. If this is not specified when you

place the part in a schematic folder, Capture uses the

part name.

Pin Name Visible Select this option to specify if the pin names are visible in

the schematic page editor and part editor.

Pin Number Visible Select this option to specify if the pin numbers are visible

in the schematic page editor and part editor.

Pin Name Rotate Select this option to specify if the pin names and pin

numbers rotate with top and bottom pins.

Click this icon to add user-defined properties.

Add New Property You can add any user-defined properties in this section.

Some of the commonly used properties are available in

the drop-down list.

You can also delete an existing user-defined property. Click the Delete icon () next to any user-defined part

property to delete it.

Any user property added to the current section of the

part is applied to all sections.

Pin Properties

To open this section in the Property Sheet pane

Select a pin in the part editor, and the properties related to the pin appear in the *Pin Properties* section of the *Property Sheet* pane.

Use this control... To do this...

Name Specify the pin name. You can create a pin name with an

overbar by adding a backslash (\) after every letter in the

pin name.

Number Specify the pin number. The pin number does not need to be a number; it can be alphabetic. If it ends in a number, it is incremented by one after each pin is placed. The pin number has a limit of 32 characters. Characters that exceeds this limit will be truncated. Shape Select the pin shape from the list of pin shapes. Type Select the pin type from the list of pin types. Pin Visible Specify the pin visibility on the schematic page. Only power pins can be set to not visible. Order Specify the order in which pin is placed in a part. This is a read-only property. **User Properties** Click the add icon, : in this section to add a user-defined property. You can also delete an existing user-defined property. Click the Delete icon () next to any user-defined pin property to delete it. Any pin property added to the current section of a homogeneous part is applied to all sections in the same view (Normal or Convert). For a heterogeneous part, pin properties are not copied to other sections. **Note:** Bus pins cannot be used directly as netlisting pins. Their main purpose is to make it possible to use nonprimitive parts more easily by connecting large numbers of signals to a child schematic folder. **Note:** You can place one pin on a part that represents all pins for a bus. Such a pin is called a bus pin. Bus pins use the same naming convention as buses. You can use bus pins in most cases where you can use scalar pins. For example: Off-page connectors Hierarchical ports Hierarchical pins of nonprimitive parts and hierarchical blocks Do not use pins in the following situations: Hierarchical pins of primitive parts and hierarchical blocks

Any design that you intend to use with your board layout tool

Text Properties

To open this section in the Property Sheet pane

Select a comment text in the part editor, and related text properties appear in the *Text Properties* section of the *Property Sheet* pane.

Use this control	To do this
------------------	------------

Text Displays the text.

Font Displays the current font style and also allows you to

change it from the drop-down list.

Font Size Displays the current font size and also allows you to

change it from the drop-down list.

Bold Select this check box to modify the text format.

Italic Select this check box to modify the text format.

Justification Specify the text justification for the comment text as

Default, Left, Center, or Right.

Basic Attributes

To open this section in the Property Sheet pane

Select any object in the part editor. All properties related to the object (text, graphical, or position-related) appear in this section. Some of the properties are listed in the following table:

Use this control	ГО	do	this
------------------	----	----	------

<Object Name> Displays the value of the selected object.

Rotation Displays the rotation of the selected object.

Location Displays the X and Y coordinates of the selected object

on the canvas.

Font Displays the current font style of the selected object. You

can also change the font style from the drop down list.

Color Displays the color of the selected object and also allows

you to change it from the drop-down list.

Font Size Displays the current font size. You can also change the

font size from the drop down list.

Bold Select this check box to make the object value bold.

Italic Select this check box to italicize the object value.

Justification Specify the justification for the object value as Default,

Left, Center, or Right.

Display Properties

Click the Display Property icon () next to any part property to specify the visibility for the property name and value. It allows you to set the display option of the selected property and its value, such as Do Not Display, Value Only, and Name Only.

Delete Current Section

Click the *Delete Current Section* button to delete the current section of the part.

/Important

This button is disabled for a part with a single section and for a heterogeneous part with only two sections.

To select any specific section of a heterogeneous part, click the drop-down at the left bottom corner of the canvas.

Add Convert View

Click the *Add Convert View* button to specify the convert view for a part.

Delete Convert View

Click the Delete Convert View button to delete the convert view for a part.

Edit Pins

To open this dialog box, do any one of the following:

- Click the Edit Pins button in the Property Sheet pane.
- Select the desired pins, right-click and select Edit Pins.
- Select the desired pins, and choose *Edit Edit Pins*.
- Press SHIFT+H.

This dialog box enables you to view pin information according to pin selection. You can view pin information for:

- All the pins of all the sections
- All the pins of the current section
- Selected pins only

Use this control... To do this...

Pin Number Select this check box to show the <Section Name>:

Pin Number column.

Pin Group Select this check box to show the *Pin Group* column.

Pin Ignore Select this check box to show the <Section Name>:

Pin Ignore column.

Order Select this check box to show the *Order* column.

Pin Type Select this check box to show the <View>: Pin Type

column.

Pin Shape Select this check box to show the <*View*>: Pin Shape

column.

<View>: Pin Name
Displays the name of the pin.

Section>: Pin Number Displays the pin number in this section.

 Section>: Pin Ignore Select to ignore the pin in this section.

Order Shows the order in which the pins are placed in

a section.

Pin Group Specify a value for each swappable (input) pin of the

part.

Valid input can be integers from 0 to 126.

<View>: Pin Shape
Select the pin shape from a list of pin shapes.

<View>: Pin Type
Select the pin type from a list of pin types.

<View>: Pin Visible
Specify the pin visibility when the part is placed on the

schematic page.

Only power pins can be set to not be visible.

Operations in the Edit Pins dialog box

This table lists the various operations you can perform in the Edit Pins dialog box.

Operation	Function
Press F2	Edit the current cell.
Press Tab	Shift to the next cell across the row.

Operation	Function
Press SHIFT+Tab	Shift to the next cell across the row in the reverse direction.
Press Enter	Apply changes to the current cell and shift to the next row.
Press CTRL+C and CTRL+V	Copy and paste data in the same table.
	Or
	Copy and paste data from Microsoft Excel.
	In the edit pins table:
	Pasting data from one cell into multiple cells copies the data to all the target cells.
	Pasting data from many cells to multiple cells populates data only from the copied cells.
Drag the plus sign that appears at the bottom-right corner of a cell to copy its contents into multiple cells.	Use the range extender to populate other cells of a column with the same value.
Select a check box to display a column.	Select the required column header check boxes above the edit pins table to show or hide the required columns.
Clear the check box to hide a column.	
	Selection of the check boxes is maintained within a session of the part editor.
Double-click a column header.	Sort column data in ascending or descending order.

Operations between Part Editor Canvas and Edit Pins Dialog Box

The following table lists the various operations you can perform between the canvas and the Edit Pins dialog box.

Operation	Description
Select the required pins on canvas and open the Edit Pins dialog box.	Only the selected pins and their corresponding information is displayed in the table.
	The size of the Edit Pins dialog box gets adjusted according to the number of pins selected.
Select all the pins on the canvas and open the Edit Pins dialog box.	Pin information of all the pins of the current section is displayed.
If you do not select any pin and open the Edit Pins dialog box.	All the pins for all the sections appear in the table.

Associate PSpice Model

You can associate a PSpice model at the time of part creation itself. Click the *Associate PSpice Model* button. The <u>Model Import Wizard</u> opens.

Paste Options dialog box

To open this dialog box

- **1.** Ensure that the setting *Enable the Paste Options mode* is selected in the Extended Preferences Setup window.
- 2. Select any number of pins.

or

Select any graphical object with any number of pins selected along with it.

- 3. Copy the selection.
- **4.** Select Edit Paste, press CTRL+V, or right-click and select Paste.

If you copy only pins, the Paste Options dialog box opens immediately.

If you copy any graphical object along with the pins, the Paste Options dialog box opens after you place the copied graphical object(s).

The pin location of the copied part is retained only if the destination part has no pins else the copied pins are pasted as a pin array.

- **5.** Modify the pin information from the required section.
- **6.** Click the *Paste* button.
- 7. Click to place the pin at the required location.

Part Aliases dialog box

To open this dialog box

Click the *Part Aliases* button in the <u>New Part Properties dialog box</u>.

Use this control... To do this...

Alias Names Select an alias name from the displayed list.

New Open the New Alias dialog box to add new aliases.

Delete the selected alias from the list.

Part Search dialog box

To open this dialog

Click Part Search in the Place Part dialog box.

Use this control... To do this...

Part Name Specify the part name to search for. Use an asterisk (*) to

match any string of characters, and a question mark (?)

to match any single character.

Libraries Displays the location of all libraries for the part specified

in Part Name. After Capture has searched for the part and located it, select the part from one of the libraries and click OK. If the library is configured in the Place Part dialog box, then Capture closes this dialog box, and

selects the part in the Place Part dialog box.

Library Path Specify the path containing libraries for Capture to

search through. Set this to your library directory.

Begin Search Begin searching for the part specified by Part Name, and

in the directory specified by Library Path.

Browse Display a standard Windows dialog box for selecting files.

PCB Project Wizard dialog box

PCB Project Wizard creates your project as a PCB design. The PCB Wizard helps you configure libraries for your project.

To create a PC Board project:

- 1. Select the Enable project simulation check box if you intend to have simulation capabilities in your PCB design. If you selected Enable project simulation, go to step 2. Otherwise, go to step 3.
- **2.** Select the type of simulation resources you want to include:
 - Analog or mixed signal
 - □ VHDL-based
 - Verilog-based
- 3. Click Next.
- **4.** Continue with the steps below that are appropriate for the simulation resource you chose.

Analog or mixed signal simulation:

- 1. Select a PSpice symbol library you want to include in your project and click the Add button (or double-click the library name). Continue this step until you have chosen all the libraries you want.
- 2. Click the Finish button.

VHDL-based simulation:

- 1. Select the PCB part symbol libraries you want to include in your project, and click the Add button.
- 2. Click the Finish button.

Note: Typically, referenced projects are FPGA projects that you want to include in your PCB project. This is useful for board simulation that includes the appropriate timing and functionality information for an FPGA that is included in your printed circuit board.

Note: Some symbol libraries do not have corresponding simulation models.

Place and Route Settings dialog box

To open this dialog

For a programmable logic project, in the <u>Select Simulation Configuration dialog box</u> choose Postroute and click OK.

Use this control... To do this...

Build Netlist Path Specify the path to the netlist that resulted from the

place-and-route of your project using your vendor

place-and-route tool.

SDF File Path Specify the path to the standard delay file that resulted

from the place-and-route of your project using your

vendor place-and-route tool.

Place Bookmark dialog box

To open this dialog

In the schematic page editor, choose Bookmark (see <u>Bookmark command</u>) from the Place menu.

Use this control... To do this...

Name Specifies the bookmark's name.

Place Ground dialog box

To open this dialog

In the schematic page editor

Choose Ground (see **Ground command**) from the Place menu.

OR

Click the Ground button on the Draw Electrical toolbar.

Use this control	To do this
Symbol	Specify a symbol to select and view. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.
Symbol List	Displays a list of power symbols and ground symbols in the libraries selected in the Libraries list box that match what's entered in the Symbol text box. When you select an object in this list, its name appears in the Symbol text box, and its graphic appears in the preview box. Select an object from the list of symbols available in the selected libraries.
Libraries	Select one or more libraries from the list of available libraries. The symbol list displays the objects from the selected libraries. You also select libraries from the Libraries list box to remove them from the list.
	Note: The CAPSYM.OLB, which is the default library in Capture now includes the PSpice ground (0) symbol. Use the '0' symbol to place a PSpice ground 0 symbol in your design. If your design does not have a PSpice ground (0) symbol, then the PSpice analog simulation may not run. To place a PSpice ground '0' symbol in your design, see Placing PSpice ground 0 symbols for PSpice simulations.
Preview box	Displays the graphic of the selected object.

Name

Specify the object's name.

Add Library Display a standard Windows Open dialog box for adding

a library to the Libraries list box. You can add a library in SDT format. If you do, you can save the library in the

Capture format (.OLB).

Remove Library Remove the selected library or libraries from the libraries

list box.

Place NetGroup dialog box

The NetGroup dialog displays when you choose NetGroup from the Place menu.

Use this control... To do this...

Add NetGroup Click this button to open the New NetGroup dialog to

create a NetGroup.

Modify NetGroup Click this button to open the Modify NetGroup dialog to

modify a NetGroup.

Note: You need to click the check mark next to a

NetGroup name to open the associated NetGroup

definition in this dialog.

Delete NetGroup Click this button to delete selected NetGroups.

Import NetGroups Click this button to import NetGroups definition Xml files.

Instance Name Enter the instance name of a named NetGroup to place

on the schematic page.

Place NetGroup Block Check this box to place the NetGroup as a block on the

schematic page.

Place Unnamed NetGroup Check this box to place the NetGroup as an unnamed

NetGroup block on the schematic page.

Place Off-Page Connector dialog box

To open this dialog

In the schematic page editor

Choose Off-Page Connector (see Off-Page Connector command) from the Place menu.

OR

Click the Off-Page Connector button on the Draw toolbar.

Use this control	To do this
Symbol	Specify a symbol to select and view. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.
Symbol List	Displays a list of off-page connectors in the libraries selected in the Libraries list box that match what's entered in the Symbol text box. When you select an object in this list, its name appears in the Symbol text box, and its graphic appears in the preview box. Select an object from the list of symbols available in the selected libraries.
Libraries	Select one or more libraries from the list of available libraries. The symbol list displays the objects from the selected libraries. You also select libraries from the Libraries list box to remove them from the list.
Preview box	Displays the graphic of the selected object.
Name	Specify the object's name.
Add Library	Display a standard Windows Open dialog box for adding a library to the Libraries list box. You can add a library in SDT format. If you do, you can save the library in the Capture format (.OLB).
Remove Library	Remove the selected library or libraries from the libraries list box.
NetGroup OffPage	Place the off-page connector as a named NetGroup off-page connector.
	Note: Select the NetGroup, to be used, from the drop-down list of NetGroups.
Show UnNamed NetGroup	Place the off-page connector as an unnamed NetGroup off-page connector.

Place Hierarchical Block dialog box

To open this dialog

In the schematic page editor

Choose Hierarchical Block (see Hierarchical Block command) from the Place menu.

OR

Click the Hierarchical Block button on the Draw toolbar.

Use this control... To do this...

Reference Specify the hierarchical block's name.

Primitive

Default Use the default primitive setting, which for hierarchical

blocks is nonprimitive. The default setting for hierarchical blocks is set in the Hierarchy tab of the Design Template

dialog box (see Design Template command).

Yes Indicate the hierarchical block is a primitive.

No Indicate the hierarchical block is nonprimitive and

descends in hierarchy.

User Properties Display the <u>User Properties dialog box</u> so you can modify

the hierarchical block's user defined properties.

Implementation

Implementation Type

Schematic View Indicate that the attached implementation is a schematic folder. Capture automatically generates the appropriate hierarchical pins for the block based on the hierarchical ports.

VHDL Indicate that the attached implementation is a VHDL entity. Capture automatically generates the appropriate hierarchical pins for the hierarchical block based on the port declarations in the VHDL entity.

EDIF Indicate that the attached implementation is an EDIF netlist. If your design includes EDIF implementations for hierarchical blocks, you must specify the hierarchical pins for the block; Capture will not generate them from the EDIF netlist. Also, if your design includes EDIF implementations, you can simulate them, but you cannot compile or build them.

Project Indicate that the attached implementation is a Capture programmable logic project. You must specify the hierarchical pins for the hierarchical block; Capture will not generate them.

PSpice Model Indicate that the attached implementation is a PSpice model file. You must specify the hierarchical pins for the block; Capture will not generate them.

PSpice Stimulus Indicate that the attached implementation is PSpice stimulus file. You must place the hierarchical pins on the block. Capture will not generate them.

Verilog Indicate that the attached implementation is a Verilog model. Capture automatically generates the appropriate hierarchical pins for the hierarchical block based the Verilog model.

Attaching an implementation does not automatically add that file, project, or schematic folder to the project. You must specifically add the implementation to the project with the Project command (Edit menu).

Implementation name

Specify the name of the attached schematic folder, VHDL entity, netlist, or project for the hierarchical block.

Path and filename

Specify the path and filename for the library of the attached object. Use the Browse button to locate the file, or supply both the path and filename. If the attached object is in the same design as the hierarchical block you are placing, leave this option unspecified.

Note: If you attach external schematic folders or other files to hierarchical blocks in a design or parts in a library, be sure to include the attachments when you pass the design or library to a board fabrication house or to another engineer. Attached schematic folder and other files are not carried along automatically when you copy or move a part, schematic folder, or schematic page to another library, design, or schematic folder. Only the "pointers" to the attached schematic folder and files—that is, their names and the names of the designs or libraries that contain them—are carried along.

Attached files work much like their counterparts in email—they do not provide an alternative definition of the part (as do attached schematic folders).

Note: Be careful not to create recursion in your design. Capture cannot prevent recursion, and the Design Rules Check command does not report it.

Recursion causes Capture to process infinitely as it tries to expand the design, resulting in the loss of any changes you've made to your design since it was last saved.

Note: When you attach a schematic folder to a part or hierarchical block, you can specify a full path and filename in the Library text box. So, although you can specify a library that hasn't been saved, you should not try to descend into the attached schematic folder until the library that contains the schematic folder has been saved.

If you don't specify a full path and filename in the Library text box, Capture expects to find the attached schematic folder in the same design as the part of hierarchical block to which it is attached. If the specified schematic folder doesn't exist in either the design or library, Capture creates the schematic folder when you descend hierarchy on the part or the hierarchical block.

For compatibility with future versions of Windows, Capture preserves the case of the path and filename as you specify them in the Library text box.

Note: Once you have attached a file and associated a text editor with it, you can use the <u>Descend Hierarchy command</u> to open that file. If you have an attached schematic folder as well as an attached file, Descend Hierarchy opens the schematic folder and not the file.

Place Hierarchical Pin dialog box

To open this dialog

In the schematic page editor

Choose Hierarchical Pin (see <u>Hierarchical Pin command</u>) from the Place menu.

OR

Click the Hierarchical Pin button on the Draw toolbar.

Use this control	To do this
Name	Specify the hierarchical pin's name.
Туре	Select the pin type from the list of pin types.
Width	Specify if the pin connects to a bus or a wire. If bus is specified, the hierarchical pin must connect to a bus; otherwise, it must connect to a wire.
User Properties	Display the <u>User Properties dialog box</u> so you can edit the pin's properties.

Note: Bus pins cannot be used directly as netlisting pins. Their main purpose is to make it possible to use nonprimitive parts more easily by connecting large numbers of signals to a child schematic folder.

Note: You can place one pin on a part that represents all pins for a bus. Such a pin is called a bus pin. Bus pins use the same naming convention as buses.

Note: You can use bus pins in most cases where you can use scalar pins. For example:

- Off-page connectors
- Hierarchical ports
- Hierarchical pins of nonprimitive parts and hierarchical blocks
- Do not use pins in the following situations:
- Hierarchical pins of primitive parts and hierarchical blocks
- Any design that you intend to use with your board layout tool

Place Hierarchical Port dialog box

To open this dialog

In the schematic page editor

Choose Hierarchical Port (see Hierarchical Port command) from the Place menu.

OR

Click the Hierarchical Port button on the Draw toolbar.

Use this control	To do this
Symbol	Specify a symbol to select and view. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.
Symbol List	Displays a list of hierarchical ports in the libraries selected in the Libraries list box that match what's entered in the Symbol text box. When you select an object in this list, its name appears in the Symbol text box, and its graphic appears in the preview box. Select an object from the list of symbols available in the selected libraries.
Libraries	Select one or more libraries from the list of available libraries. The symbol list displays the objects from the selected libraries. You also select libraries from the Libraries list box to remove them from the list.
Preview box	Displays the graphic of the selected object.
Name	Specify the object's name.
Add Library	Display a standard Windows Open dialog box for adding a library to the Libraries list box. You can add a library in SDT format. If you do, you can save the library in the Capture format (.OLB).
Remove Library	Remove the selected library or libraries from the libraries list box.

NetGroup Port Place the hierarchical port as a named NetGroup port.

Note: Select the NetGroup, to be used, from the

drop-down list of NetGroups.

Show UnNamed NetGroup Place the hierarchical port as an unnamed NetGroup

port.

Place IEEE Symbol dialog box

To open this dialog

In the part editor, do one of the following:

■ Select Place – IEEE Symbol, see <u>IEEE Symbol command</u>.

Click the IEEE Symbol button on the Draw toolbar.

Use this control... To do this...

IEEE Symbols Select a symbol from the list of available symbols.

Preview box Displays the graphic of the selected symbol.

Place Net Alias dialog box

To open this dialog

In the schematic page editor

Choose Net Alias (see Net Alias command) from the Place menu.

OR

Choose the Place net alias button on the Draw toolbar.

OR

Press the N key.

Use this control... To do this...

Alias Enter the net alias name in the text box.

Color Specify the color of the net alias.

Rotation Specify the rotation of the net alias or text.

Font

Change Display the Font dialog box so you can select a font.

Use Default Change the font to the default font specified in the Design

Template dialog box.

NetGroup

Use this control

NetGroup Aware Aliases Select to assign a NetGroup to a bus. Select an existing

NetGroup from the list or edit to add a new NetGroup.

The width of the alias is the same as the specified

NetGroup.

To do this

Click the Off-Page Connector button on the Draw toolbar.

Ose this control	To do tins
Symbol	Specify a symbol to select and view. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.
Symbol List	Displays a list of off-page connectors in the libraries selected in the Libraries list box that match what's entered in the Symbol text box. When you select an object in this list, its name appears in the Symbol text box, and its graphic appears in the preview box. Select an object from the list of symbols available in the selected libraries.

Libraries Select one or more libraries from the list of available

libraries. The symbol list displays the objects from the selected libraries. You also select libraries from the

Libraries list box to remove them from the list.

Preview box Displays the graphic of the selected object.

Name Specify the object's name.

Add Library Display a standard Windows Open dialog box for adding

a library to the Libraries list box. You can add a library in SDT format. If you do, you can save the library in the

Capture format (.OLB).

Remove Library Remove the selected library or libraries from the libraries

list box.

NetGroup Port Place the off-page connector as a named NetGroup

off-page connector.

Note: Select the NetGroup, to be used, from the

drop-down list of NetGroups.

Show UnNamed NetGroup Place the off-page connector as an unnamed NetGroup

off-page connector.

Place Part dialog box

To open this dialog

In the schematic page editor

Choose Part (see Part command) from the Place menu.

OR

Click the Part button on the Draw toolbar.

Place Part Pane

Use this control... To do this...

Part Specify the name of the part. Use an asterisk (*) to match

any string of characters, and a question mark (?) to

match any single character.

Part List Displays a list of parts in the libraries selected in the

Libraries list box that match what's entered in the Part text box. When you select a part in this list, its name appears in the Part text box, and its graphic appears in the preview box. Select a part from the list of parts

available in the selected libraries.

Note: You can filter the part list, using the Filter button

described below.

Filter Display the Specify Part Filter dialog box that allows you

to restrict part searches in part libraries based on specific

criteria.

For example, if you are a PSpice user, you can restrict

your part library search such that only parts with

associated PSpice simulation models will be listed in the

Part List.

 ∇

Libraries Select one or more libraries from the list of available

libraries. The part list displays the parts from the selected libraries. You also select libraries from the Libraries list

box to remove them from the list.

Graphic Select either Normal or Convert view. All parts have a

normal view. Some parts have a convert view that can be

used for things such as a <u>DeMorgan equivalent</u> part.

Add Library Display a standard Windows Open dialog box for adding

a library to the Libraries list box. You can add a library in SDT format. If you do, you can save the library in the

Capture format (.OLB).



Remove Library

Remove the selected library or libraries from the libraries list box.



Reload Library Parts

Reloads the Capture library parts in the selected library.



Note: As you reload the library parts, remember the following points:

- To see the updated part, reselect library part in Place Part dialog. To update library with newly added or deleted parts, reload the library in Place Part dialog.
- To enable the library parts update on selection, select the Refresh part on selection option in Preferences – Miscellaneous tab.

Packaging

Parts per Pkg Displays the number of parts in the package.

Part Select the part in the package to place on the schematic

page.

Type A package may be either homogeneous or

heterogeneous.

Preview box Displays the graphic of the selected part.

Application indicator

This feature uses icons to indicate whether the selected part has certain properties. The icon displayed for each property is described below.

Property: PCB Footprint



Property: PSpice Template



The following icon is not associated with any property. It indicates that the part is associated with a template-based parameterized PSpice model



Search Part

The Search for Part section of the Place Part dialog box is viewed if you click the Expand button.

Search For Enter the part name (you can include wildcard

characters) to search through the libraries for a specific

part or parts.

Path Enter the path of the library to search for the part. You

can use the Browse button to select the library.

Libraries Displays a list of libraries in which you searched for parts.

Add libraries from the search library list to the Place Part

library list.

Place Pin Array dialog box

To open this dialog

In the part editor, do one of the following:

- Select Place Pin Array or see Pin Array command.
- Click the Pin Array button on the Draw toolbar.

Use this control	To do this
Starting Name	Specify the name for the pin array. If the name ends with a digit (09), each pin in the array is incremented by the value specified in the <i>Pin# Increment for Next Pin</i> field.
	You can create a pin name with an overbar for a negated pin by adding a backslash (\) after every letter in the pin name.
	When you open this dialog box to add more pin arrays, you can see that <i>Starting Name</i> is automatically incremented from the last usage of the pin array.
Starting Number	Specify the starting number for the pin array. Pin numbers can be alphanumeric. If a pin number ends in a number, it is incremented by the value specified in the <i>Pin# Increment for Next Pin</i> field.
	When you open this dialog box to add more pin arrays, you can see that <i>Starting Number</i> is automatically incremented from the last usage of the pin array.
Number of Pins	Specify the number of pins in the pin array.
Pin Spacing	Specify the spacing between pins for the pin array.
Shape	Select the pin shape from the list of pin shapes.
	When you open this dialog box to add more pin arrays, you can see that last selection of pin shape is retained.
Туре	Select the pin type from the list of pin types.
	When you open this dialog box to add more pin arrays, you can see that last selection of pin type is retained.

Pin Visible Specify the pin visibility when the part is placed on the

schematic page. Only power pins can be set to not

visible.

Additional Options

Pin# Increment for Next Pin Specify the increment for the next pin number in the pin

array.

Pin# Increment for Next Section Specify the increment between pin numbers for the next

section. This is valid only for homogeneous parts.

Place Pin dialog box

To open this dialog

In the part editor, do one of the following:

■ Select Place – Pin, or see Pin command.

Click the Pin button on the Draw toolbar.

Use this control	To do this
Name	Specify the pin name. You can create a pin name with an overbar by adding a backslash (\) after every letter in the pin name.
	If the name ends with a digit (09), each pin name is automatically incremented, by the value specified in the <i>Pin# Increment for Next Pin</i> field. You can see that the pin name value is automatically incremented from the last usage of the pin.
Number	Specify the pin number. Pin numbers can be alphanumeric.
	If it ends in a number, it is incremented by the value specified in the <i>Pin# Increment for Next Pin</i> field. You can see that the pin number value is automatically incremented from the last usage of the pin.

Shape Select the pin shape from the list of pin shapes.

When you open this dialog box to add more pins, you can

see that last selection of pin shape is retained.

Type Select the pin type from the list of pin types.

When you open this dialog box to add more pins, you can

see that last selection of pin type is retained.

Width Specify whether the pin connects to a bus or a wire. If

bus is specified, the pin must connect to a bus;

otherwise, it must connect to a wire.

Pin Visible Specify the pin visibility when the part is placed on the

schematic page. Only power pins can be set to not

visible.

User Properties Click this button to open the <u>User Properties dialog box</u>,

so you can add or edit any user-defined property.

Additional Options

Pin# Increment for Next Pin Specify the increment for the next pin.

Pin# Increment for Next Section Specify the increment between pin numbers for the next

section. This is valid only for homogeneous parts.

Note: Bus pins cannot be used directly as netlisting pins. Their main purpose is to make it possible to use nonprimitive parts more easily by connecting large numbers of signals to a child schematic folder.

Note: You can place one pin on a part that represents all pins for a bus. Such a pin is called a bus pin. Bus pins use the same naming convention as buses.

You can use bus pins in most cases where you can use scalar pins. For example:

- Off-page connectors
- Hierarchical ports
- Hierarchical pins of nonprimitive parts and hierarchical blocks

Do not use pins in the following situations:

- Hierarchical pins of primitive parts and hierarchical blocks
- Any design that you intend to use with your board layout tool

Place Power dialog box

To open this dialog

In the schematic page editor

Choose Power (see Power command) from the Place menu.

OR

Click the Power button on the Draw toolbar.

OR

Press the F key.

Use this control	To do this
Symbol	Specify a symbol to select and view. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.
Symbol List	Displays a list of power symbols and ground symbols in the libraries selected in the Libraries list box that match what's entered in the Symbol text box. When you select an object in this list, its name appears in the Symbol text box, and its graphic appears in the preview box. Select an object from the list of symbols available in the selected libraries.
Libraries	Select one or more libraries from the list of available libraries. The symbol list displays the objects from the selected libraries. You also select libraries from the Libraries list box to remove them from the list.
Preview box	Displays the graphic of the selected object.
Name	Specify the object's name.
Add Library	Display a standard Windows Open dialog box for adding a library to the Libraries list box. You can add a library in SDT format. If you do, you can save the library in the Capture format (.OLB).

Remove Library Remove the selected library or libraries from the libraries

list box.

Place Text dialog box

To open this dialog

In the schematic page editor, do one of the following:

■ Select *Place - Text*, or see <u>Text command</u>.

Click the Text button on the Draw toolbar.

Press the T key.

Use this control	To do this
------------------	------------

Text Enter the text in the text box.

Color Specify the color of the text. Text that is placed in the part

editor uses the part body color.

Rotation Specify the rotation of the text.

Font

Change Display a Font dialog box so you can select a font.

Use Default Change the font to the default font specified in the Design

Template dialog box.

Text Justification Specify the text justification for the comment text as

Default, Left, Center, and Right. The default text

justification is the legacy Capture behavior as performed

in Capture 16.5. For more information on Text

Justification, see Table 8-1: Text Justification table on

page 370.

Note: Text justification in a design saved using 16.6 HotFix of Capture will not be preserved if the design is opened and saved using Capture 16.6 and earlier

releases.

Place Title Block dialog box

To open this dialog

In the schematic page editor

Choose Title Block (see <u>Title Block command</u>) from the Place menu.

OR

Click the Title Block button on the Draw toolbar.

Use this control	To do this
Symbol	Specify a symbol to select and view. Use an asterisk (*) to match any string of characters, and a question mark (?) to match any single character.
Symbol List	Displays a list of title blocks in the libraries selected in the Libraries list box that match what's entered in the Symbol text box. When you select an object in this list, its name appears in the Symbol text box, and its graphic appears in the preview box. Select an object from the list of symbols available in the selected libraries.
Libraries	Select one or more libraries from the list of available libraries. The symbol list displays the objects from the selected libraries. You also select libraries from the Libraries list box to remove them from the list.
Preview box	Displays the graphic of the selected object.
Name	Specify the object's name.
Add Library	Display a standard Windows Open dialog box for adding a library to the Libraries list box. You can add a library in SDT format. If you do, you can save the library in the Capture format (.OLB).
Remove Library	Remove the selected library or libraries from the libraries list box.

Preferences dialog box

To open this dialog

In the schematic page editor or part editor, choose Preferences (see <u>Preferences command</u>) from the Options menu.

Use this tab	To do this
Application Theme	Click the drop-down list to select the theme of the application.
Schematic Theme	Click the drop-down list to select the theme of the schematic page.
Colors/Print	Define the default color of objects such as aliases, wires, design variants, part not present, and pins. A standard Windows default Color dialog box appears when you click on the color of an item.
	The check boxes next to the objects control whether the objects will be printed or plotted. If an object's box is selected, the object can be printed or plotted. Objects always appear on your screen, regardless of the setting of their check boxes.
	The Use Defaults button resets colors to the default colors shipped with Capture.
	Note: The border and grid references of schematic pages use the color specified for title blocks.
	Important
	Color specified in part editor for Pin Name, Pin

Grid Display

Control the behavior and appearance of the grid display for both the schematic page editor and the part editor. See <u>Grid Display tab</u> for more information.

Number, Part Properties, Part Reference Prefix, Part Value, and any user-defined property overrides the color selected in the Preferences

dialog box.

Pan And Zoom Set auto scrolling options and zoom factor for both the

schematic page editor and the part editor. See Pan and

Zoom tab for more information.

Select Specify selection options, change the maximum number

of objects you can drag, and set tool palette visibility for both the schematic page editor and the part editor. For

more information, see Select tab

Miscellaneous Specify the fill style, line style, and line width for both the

schematic page editor and the part editor. Also specify the line color for the schematic page editor. You can also define the session log font, set text rendering, set auto recovery intervals, and enable intertool communication.

For more information, see Miscellaneous tab.

Text Editor Specify font and color information for the text editor. Also

specify tab setting in terms of character spacing, and highlighting options. For more information, see <u>Text</u>

Editor tab.

Board Simulation Specify the hardware development language to be used

during board level simulation. For more information, see

Board Simulation tab.

Color/Print tab

Use these options to specify the color settings for the different object types on a schematic page.

Use this control... To do this...

Basic colors Shows the color of the object selected in the Colors tab.

To change the color, click the left mouse button on a

different color and then click OK.

Custom colors This feature is disabled in Capture.

Define Custom Colors This feature is disabled in Capture.

Grid Display tab

Use these options to specify the appearance of the grid for your working area. You use this dialog box to set grid properties for both the schematic page grid and the part and symbol grid.

Use this control	To do this
Visible	Specify whether the schematic page or part's grid is visible or hidden on the screen.
Grid Style	Specify whether the grid appears as grid dots or lines in the editor.
Grid spacing	Specify the grid spacing on the schematic page as a fraction of pin-to-pin spacing.
	For example, a setting of 1/2 specifies that the grid spacing on the schematic page is set to exactly half the specified pin-to-pin spacing.
Pointer snap to grid	Specify whether the pointer snaps to the grid in the editor. This check box is selected by default.
Fine	Specify whether connectivity (part, bus) and drawing
(only for schematic page editor)	objects, like Line, Polyline, Text, Rectangle, Ellipse, Arc, and Picture can be placed and moved on the fine grid. For description of various scenarios for this option, see Customizing placement and movement of objects on the schematic.
	Note: This setting is saved in the CAPTURE.INI file and it is used whenever you start the next Capture session.
Coarse	Specify whether connectivity and drawing objects can be
(only for schematic page editor)	placed and moved on the coarse grid. For description of various scenarios for this option, see <u>Customizing</u> placement and movement of objects on the schematic.
	Note: This setting is saved in the CAPTURE.INI file and it is used whenever you start the next Capture session.

Master

(only for schematic page editor)

Specify whether connectivity and drawing objects can be placed and moved on the master grid, if *Pointer snap to grid* check box is selected. This setting is equivalent to the Snap To grid toolbar state ...

Specify whether connectivity and drawing objects can be placed and moved on the fine grid, if *Pointer snap to grid* check box is not selected. This setting is equivalent to the Snap To grid toolbar state ...

For description of various scenarios for this option, see Customizing placement and movement of objects on the schematic.

Note: This setting is saved in the CAPTURE.INI file and it is used whenever you start the next Capture session.

Note: Ensure that the *Pointer snap to grid* check box is selected and the Connectivity Elements is set to Coarse while placing connectivity objects. Otherwise, your part pins may be placed on the fine grid, making it difficult to connect them properly.

Note: When you place a part on fine grid, it remains on fine grid through any cut-and-paste and drag-and-drop operations.

Miscellaneous tab

Use these options to specify miscellaneous options in Capture.

Use this control	To do this
Fill Style	Specify a fill pattern for rectangles, ellipses, and polygons.
Line Style	Specify line style for lines, polylines, rectangles, ellipses, and arcs.
Line Width	Specify line width for lines, polylines, rectangles, ellipses, and arcs.

Color Specify the color of lines, rectangles, and ellipses.

Polylines and arcs use the default color of objects set in

the Colors tab. This option only applies to lines, rectangles, and ellipses in the schematic page editor.

This color is not the default color, but can be set to use the default color. Objects placed while this option is not set to the default color, won't use the default color. Changing this option won't change the color of objects

already placed in the schematic page editor.

Note: You can change the fill style, line and width style, and color on individual objects using the Properties command on the Edit menu.

Junction Dot Size Specify the dot size as Small, Medium, Large, and Very

Large. This will increase or decrease the size of the dots

created when a wire is connected to another wire.

Project Manager and Session

Log

Specify the font for the session log. If you click on this box, a standard Windows Font dialog box for font selection appears. This option is neither a schematic

page nor a part editor option.

Text Rendering

Render TrueType fonts with

strokes

Specify that text appears as a series of lines, connected to resemble the outlines of the corresponding TrueType

letters or numbers they represent.

Fill text Specify that text outlines be filled in.

Auto Recovery

Enable Autorecover Enable or disable auto recovery. If the option is checked,

then auto recovery is enabled.

Update every N minutes Specify the time interval in minutes (where N is the

number of minutes) after which Capture performs an

auto-save.

Note: Auto recovery is not an automatic saving feature. If you intentionally exit Capture without first saving your changes, they will be lost. Autorecovered files are automatically deleted when you exit Capture normally.

Auto Reference

Automatically reference placed parts

(Default: **ON**)

Enable automatic part referencing. When a part is placed on the schematic page, the next available reference designator will automatically be assigned. Disabled, parts placed on the schematic will be assigned the reference designator found in the library. For example: U?A or JP?. This is the default selection.

You can choose to perform design level annotation by selecting the *Design Level (Only PCB Designs)* option. This option is not selected by default. This option should be used for PCB Designs only. For more information on Design Level Annotation, see the following table:

Table 8-2: Design Level Annotation

Project Created Using	Simulation Enabled	Will Design Level Annotation Work?
Schematic	-	Yes
PCB Wizard Board	No	Yes
PCB Wizard Board	Yes	No
PCB Wizard Board	Yes, using Analog or Mixed Signal Simulation	No
PCB Wizard Board	Yes, using Verilog-based digital simulation	No
Analog or Mixed Signal Simulation	Yes, using VHDL-based digital simulation	No
Programmable Logic Wizard	-	No

Note: The Auto Reference feature should be used to provide unique part references for simulation, and is not intended to replace the packaging process for a PCB design. For packaging you should Annotate your design.

Preserve reference on copy

Enable part references to be preserved while pasting a part to a schematic page. When you copy a part and paste it on a schematic page, the part will retain the same reference designator as that of the copied part. But, if you place a new part on a schematic page, Capture will assign the reference designator found in the library. For example: U?A or J?P.

Note: This option is not supported for complex hierarchical designs.

/Important

You can select only one option at a time.

Depending on the check box selection state, the following scenarios are possible:

- If both the check boxes are disabled:
 - the new part that you place on a schematic page will not be annotated.
 - the part references of the copied part will not be preserved while pasting that part on a schematic page.
- If the Automatically reference placed parts check box is enabled:
 - the new parts that you place on a schematic page will be annotated.
 - the part references of the copied part will not be preserved while pasting that part on a schematic page, rather they will be incremented.
- If the Preserve reference on copy check box is enabled:
 - □ the new part that you place on a schematic page will not be annotated.
 - the part references of the copied part will preserved while pasting that part on a schematic page.

Intertool Communication

Enable intertool communication

(Default: OFF)

Enable intertool communication with other OrCAD X products, such as PSpice or OrCAD X Presto. For more information about intertool communication, see Intertool communication. This option is not specific to either the schematic page editor or the part editor.

Wire Drag

Allow component move with connectivity changes

(Default: ON)

If this check box is selected or the toolbar button is in the Drag Connected Object state, then Capture allows you to drag and place the selected part or wire on the schematic, even if it results in connectivity changes. Also, Capture flags a warning with a changed cursor and will show the temporary markers.

If the check box is not selected or the toolbar button is in the Drag Connected Object state, then the selected part or wire attaches to the cursor and does not get placed on the schematic, if it results in connectivity changes. Also, Capture flags only a warning with a changed cursor and does not show the temporary markers.

Docking

Docking Place Part

(Default: **ON**)

If this option is checked, the dockable Place Part dialog is invoked when you select the Place Part command. Unchecking this option will ensure that the modal Place Part dialog displays.

Note: You need to restart Capture to make changes to this option effective.

IREF Display Property

Global Visibility

(Default: ON)

Checking this option ensures that the intersheet references are visible. Unchecking this option will hide the intersheet references.

Note: If you deselect this option (to hide the intersheet references in a design), and then run the Intersheet References command, the intersheet references will be displayed on the relevant pages of the design. If you then open the Miscellaneous tab, notice the Global Visibility option is checked.

Place Part

Refresh part on selection

(Default: OFF)

If this option is checked, then the library parts are updated in the Place Part dialog. Unchecking this option ensures that the library parts are not updated in the Place Part dialog.

Pan and Zoom tab

Use these options to specify panning and zooming factors. Set them separately for the schematic page editor, and the part and symbol editor.

Use this control	To do this
Zoom Factor	Specifies the zoom factor for the editor.
	Note: The zoom factor can be any number between 1 and 10.
Auto Scroll Percent	Specifies how much of the schematic page or part scrolls across the screen when the pointer drags a selected object into the border area of the editor.

Select tab

Use these options to specify selection factors.

Use this control	To do this
Area Select	Specify whether objects are selected when the selection area border intersects them, or only when they are completely enclosed in the selection area.
	Note: If the Fully Enclosed option is selected and you select an object on a schematic page, make sure that you select the object along with its name and number. Otherwise, the object does not get selected.
Maximum number of objects to display at high resolution while dragging	Specify the maximum number of objects that are visible at high resolution while performing a drag-and-drop operation. When you drag a number of objects greater than this value, a rectangular placeholder appears in lieu of the selected objects.
Show Palette	Specifies whether the tool palette is visible or hidden.

Text Editor tab

Use these options to specify the text settings like syntax highlight colors.

Use this control... To do this...

Syntax Highlighting Displays the current colors for VHDL and Verilog

keywords, comments, and quoted strings. Select one of the color buttons to display the standard Windows Color dialog box. In the Color dialog box, you can change the

color of the selected language element.

Current Font Setting Displays the current settings of the font item selected in

the Set Font For option. The settings displayed are the

font, font size, style, effects, and color.

Set Display the standard Windows Font dialog box for setting

font options. The settings apply to the selected font item

only.

Note: Be sure to use a monospaced font (for example, Courier) as the default font for the text editor or the source editor. If you use a true-type font, the editor may distort the appearance of the text, making it difficult to read.

Tab Setting Specify the spacing between tabs in terms of character

spaces for the text editor. The range is limited to 1-100.

This tab setting is stored in the CAPTURE.INI file. Capture will ignore existing CAPTURE.INI files that specify the tab setting in inches and will reset spacing to

a default of four character spaces.

Highlight Keywords, Comments,

and Quoted Strings

Specify that Capture highlight all VHDL or Verilog

keywords, comments, and quoted strings in the active

file.

Show line numbers Specify that the text editor display line numbers in text

files

Save text files on deactivate Specify that any open text files are saved in their current

state whenever the text editor window loses focus

Auto reload text files Specify that the text editor reload text files automatically

when there is a more recent version of the file as a result

of a Capture operation (such as netlist generation).

Save text files before running Specify that the text editor save any open text files before

tools running any Capture tools (such as design rules check or

netlist generation).

Reset all the tab options to Capture's default values.

Board Simulation tab

Use these options to specify the hardware development language used for board simulation.

Use this control... To do this...

VHDL Specify that Capture use VHDL for the board simulation

netlist and testbench.

Verilog Specify that Capture use Verilog for the board simulation

netlist and testbench.

Print dialog box

To open this dialog

Choose Print (see <u>Print command</u>) from the File menu.

Note: Click the Setup button to go to the <u>Print Setup dialog box</u> and check settings before you print. All settings you choose (except the number of copies) are saved to CAPTURE.INI and will become the default when you restart Capture.

Use this control... To do this...

Printer Displays the active printer and printer connection.

Scale

Specify the scaling factor to print by, or let Capture automatically scale to the specification.

- Scaling to paper size Forces Capture to scale the printing job to the page size specified in the Page Size tab in the Schematic Page Properties dialog box. Use the Page Size tab in the Design Template dialog box on the Options menu to reset the Custom page size.
- Scale to page size Forces Capture to scale the printing job to the page size specified by the Page size option in the Print dialog box.
- Scale Specifies the scaling factor to print by, or lets Capture automatically scale to the specified scale.

Page size

If Scale to page size is selected, select the page size to which the image will be scaled. Choose one of the standard page sizes, or Custom. These sizes are defined in the Schematic Page Properties dialog box, in the Page Size tab.

Print offsets

Specifies horizontal and vertical printing offsets in inches or millimeters (mm) depending on the setting you've chose in the Design Template dialog box. Capture centers the page horizontally and vertically when both check boxes are selected.

Your entire schematic page will be output to the printing or plotting device, regardless of the use of offsets. The following rules define the number of output pages that will be printed or plotted:

- The device and its driver determine the dimensions of the printed page area.
- The number of pages is calculated from the physical dimensions of the schematic page and the driver-provided area dimensions.
- A positive offset shifts the entire schematic page to the right in the X direction, and down in the Y direction. Additional pages are output as necessary so the entire schematic is printed. There is no truncation.
- A negative offset shifts the schematic page left, and up. The effect of a negative offset is to start the drawing on a "previous" page. Previous pages are "prepended" so drawing can start at the starting portion of the schematic. There is no truncation.
- Only the number of pages required to print or plot the schematic page will be printed. Extra, blank pages are omitted.

Specify whether you want to print instances or occurrences of a page.

- Inst. Mode (Instance mode) Enables you to print only the instances of a page in a schematic. This option is selected by default.
- Occ. Mode (Occurrence mode) Enables you to print multiple occurrences of a page in a schematic.

Specify the resolution of the print. Choose a setting from the drop-down list.

Specify the number of copies to print.

Print option

Print quality

Copies

Print to file Print the object to a file. If you select this option, the Print

to File dialog box appears after you click OK.

Print all colors in black

Causes difficult-to-read colors to print in black.

Collate copies Print copies organized in order of page numbers

Print area When a specific print area is set for the schematic page

and this option is selected, the print output is the print area of the schematic page. Clear this check box to print

the entire schematic page.

When Print area is selected, all print options except Print quality, Copies, Print to file, and Print all colors in black are unavailable. Output of selected print area is zoomed

in and centered.

Include pages outside hierarchy Specifies to also print pages in the design that are not

included in the root hierarchy. If you do not select this option, only those files included in the root hierarchy will

print.

Include referenced pages in

other libraries or designs

Specifies to also print pages outside of the design that the root hierarchy references. If you do not select this option, only the files in the design's root hierarchy will

print.

Print statistics

Printed pages per document page - Specifies the number of horizontal and vertical pages needed to print a document. If a schematic page (document page) takes more than one page to print out, Capture reports the total number of pages under the Total column heading. Horizontal and Vertical refers to the pages that will make up the printed schematic page.

For example, a schematic page may take four pages to print, so the Total is 4. Depending on physical shape of the design, the Horizontal may be 2 and the Vertical may be 2 (2 x 2), or the Horizontal may be 4 and the Vertical may be 1 (4 x 1).

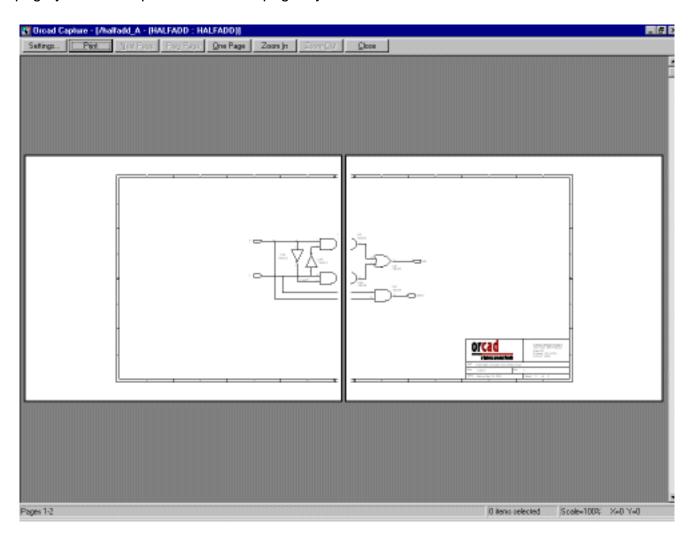
- Maximum page size for selected printer These dimensions identify the printable page area of the printer and are dependent on the printer and paper size.
- Size from schematic page properties Specifies the dimensions of the page size selected on the Page Size tab of the Schematic Page Properties dialog box.
- Size of actual printout Specifies the scaled document size as it changes with different settings in the Scale options or when you change the page orientation in the Print Setup dialog box.

The Maximum size for the page layout, Size from schematic page properties, and Size of actual printout dimensions are shown in inches or millimeters, depending on the setting you've chose in the Design Template dialog box.

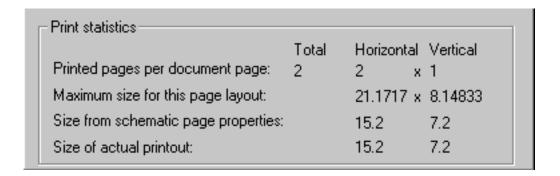
Setup

Display a standard Windows dialog box to set printer or plotter configuration.

The Print statistics section of the Print and Print Preview dialog boxes reports the number of horizontal and vertical pages it takes to print out the selected document. If the schematic page you want to print takes a two-page layout as shown here:



You would see this in the Printed pages per document page statistics:



Note: When printing a multi-page schematic, make sure that the pages do not have multiple Title Blocks with different page numbers. Otherwise, the pages will not be printed in the correct order. If you change the page numbers in the Title Blocks manually, then make sure that the Do not change the page number check box is checked in the <u>Annotate dialog box</u>.

Print Preview and Print Setup dialog boxes

To open this dialog

Choose Print Preview (see <u>Print Preview command</u>) from the File menu.

Note: Click the Setup button to go to the Print Setup dialog box before you print. All settings you choose (except the number of copies) are saved to CAPTURE.INI and will become the default when you restart Capture.

Use this control... To do this...

Printer

Displays the active printer and printer connection.

Scale

Specify the scaling factor to print by, or let Capture automatically scale to the specification.

- Scaling to paper size Forces Capture to scale the printing job to the page size specified in the Page Size tab in the Schematic Page Properties dialog box. Use the Page Size tab in the Design Template dialog box on the Options menu to reset the Custom page size.
- Scale to page size Forces Capture to scale the printing job to the page size specified by the Page size option in the Print dialog box.
- Scale Specifies the scaling factor to print by, or lets Capture automatically scale to the specified scale.

Page size

If Scale to page size is selected, select the page size to which the image will be scaled. Choose one of the standard page sizes, or Custom. These sizes are defined in the Schematic Page Properties dialog box, in the Page Size tab.

Print offsets

Specifies horizontal and vertical printing offsets in inches or millimeters (mm) depending on the setting you choose in the Design Template dialog box. Capture centers the page horizontally and vertically when both check boxes are selected.

Your entire schematic page will be output to the printing or plotting device, regardless of the use of offsets. The following rules define the number of output pages that will be printed or plotted:

- The device and its driver determine the dimensions of the printed page area.
- The number of pages is calculated from the physical dimensions of the schematic page and the driver-provided area dimensions.
- A positive offset shifts the entire schematic page to the right in the X direction, and down in the Y direction. Additional pages are output as necessary so the entire schematic is printed. There is no truncation.
- A negative offset shifts the schematic page left, and up. The effect of a negative offset is to start the drawing on a "previous" page. Previous pages are "prepended" so drawing can start at the starting portion of the schematic. There is no truncation.
- Only the number of pages required to print or plot the schematic page will be printed. Extra, blank pages are omitted.

Print option

Specify whether you want to print instances or occurrences of a page.

- Inst. Mode (Instance mode) Enables you to print only the instances of a page in a schematic. This is the default option.
- Occ. Mode (Occurrence mode) Enables you to print multiple occurrences of a page in a schematic.

Print quality

Specify the resolution of the print. Choose a setting from the drop-down list.

Copies

Specify the number of copies to print.

Print to file

This option is not available in Print Preview. If you want to print to file, use the Print dialog box.

Print all colors in black Causes difficult-to-read colors to print in black.

Collate copies

Print copies organized in order of page numbers

Print area

When a specific print area is set for the schematic page and this option is selected, the print output is the print area of the schematic page. Clear this check box to print the entire schematic page.

When Print area is selected, all print options except Print quality, Copies, Print to file, and Print all colors in black are unavailable. Output o selected print area is zoomed in and centered.

hierarchy

Include pages outside Specifies to also print pages in the design that are not included in the root hierarchy. If you do not select this option, only those files included in the root hierarchy will print.

Include referenced pages in other libraries or designs Specifies to also print pages outside of the design that the root hierarchy references. If you do not select this option, only the files in the design's root hierarchy will print.

Print statistics

Printed pages per document page - Specifies the number of horizontal and vertical pages needed to print a document. If a schematic page (document page) takes more than one page to print out, Capture reports the total number of pages under the Total column heading. Horizontal and Vertical refers to the pages that will make up the printed schematic page.

For example, a schematic page may take four pages to print, so the Total is 4. Depending on physical shape of the design, the Horizontal may be 2 and the Vertical may be 2 (2 x 2), or the Horizontal may be 4 and the Vertical may be 1 (4 x 1).

- Maximum page size for selected printer These dimensions identify the printable page area of the printer and are dependent on the printer and paper size.
- Size from schematic page properties Specifies the dimensions of the page size selected on the Page Size tab of the Schematic Page Properties dialog box.
- Size of actual printout Specifies the scaled document size as it changes with different settings in the Scale options or when you change the page orientation in the Print Setup dialog box.

The Maximum size for the page layout, Size from schematic page properties, and Size of actual printout dimensions are shown in inches or millimeters, depending on the setting you've chose in the Design Template dialog box.

Setup Display a standard Windows dialog box to set printer or plotter

configuration.

Print Setup dialog box

To open this dialog

choose the Print Setup (see Print Setup command) from the File menu.

OR

Click the Setup button on the Print dialog box or Print Preview and Print Setup dialog boxes.

The Print Setup dialog box is a standard windows dialog box for configuring your printer or plotter. Use it to choose a printer, paper source, and orientation before printing. For more information on setting up printers and plotters, refer to the documentation for your configured printer driver.

Many times, the options for your printer are not available in the standard setup dialog box. If you do not find the options you need, try the printer setup in the Windows Control Panel.

Print to File dialog box

To open this dialog

In the Print dialog box, select the Print to File option and then click OK.

Use this control... To do this...

Output file name Specify the name of the output file.

Programmable Logic Project Wizard dialog box

Programmable logic project for designs may include Verilog or VHDL models as part of the structure. Projects of this nature will often use simulation and synthesis tools as part of the design flow. When you create a programmable logic project, certain folders are added to the project. These are discussed in Working with the project manager.

524

To create a programmable logic project:

- **1.** Select the logic vendor and target family for the project (for example, Actel, Altera, and Xilinx family).
- 2. Click the Finish button.

Propagation Delay dialog box

Use this dialog box to specify a pin-pair and select valid minimum and maximum value for the PROPAGATION DELAY property.

The PROPAGATION DELAY property has the following syntax:

<Pin pair>:<min value>:<max value>

To open this dialog

While editing the PROPAGATION_DELAY property in the Property Editor, choose *Invoke UI* from the *Edit* menu press CTRL+U.

OR

Right-click the grid corresponding to the PROPAGATION_DELAY property and select *Invoke UI* from the popup menu.

Use this control...

To do this...

Pin Pair

Applies Min and/or Max delay constraint to various pin-pairs. This value may be set to one of the following:

- Longest/Shortest pin-pair—Minimum delay is applied to the shortest pin-pair and maximum delay is applied to the longest pin-pair.
- Longest/Shortest Driver/Receiver—Minimum is applied to the shortest Driver/Receiver pin-pair and maximum is applied to the longest Driver/Receiver pin-pair.
- All Drivers/All Receivers—Min/Max constraints apply to all Driver/Receiver pin-pairs.

Min

Specifies the minimum allowable propagation delay/length for the pin-pairs.

Min Rule Specifies whether the minimum allowable propagation

delay is measured as DELAY in ns, %MANHATAN, or LENGTH in mills (mils), micron (um), millimeter (mm),

centimeter (cm), and inches (in).

Max Specifies the maximum allowable propagation

delay/length for the pin-pairs.

Max Rule Specifies whether the maximum allowable propagation

delay is measured as DELAY in ns, MANHATAN, or LENGTH in mills (mils), micron (um), millimeter (mm),

centimeter (cm), and inches (in).

Add Pin Pair Displays the <u>Create Pin Pairs dialog box</u> dialog box. Use

this dialog box to define a pin-pair.

Keyboard shortcut: ALT, A

Delete Pin Pair Deletes the pin-pair corresponding to the selected row.

Keyboard shortcut: ALT, D

OK Performs syntax checking and if the syntax is correct

assigns the PROPAGATION DELAY property on the

selected net.

Note: You can use the <u>User Properties dialog box</u> to assign the PROPAGATION_DELAY property to all the bits of a bus at the same time. Make sure that you use the correct syntax for specifying a value for the PROPAGATION DELAY property. The syntax is:

<Pin pair>:<min value>:<max value>

The pin-pairs can only be:

- L:S
- D:R
- AD:AR

Properties dialog box

Use the dialog box to access information about the current project, or the file currently selected in the project manager window. You can also change the description of a file.

To open this dialog

Choose Properties (see Properties command) from the Edit menu or from the shortcut menu.

Use this tab	To do this
General tab	Provide information about the file name, size, and the date it was last modified. For more information, see General tab
Type tab	Specify the type of the file, such as schematic or simulation model. For more information, see <u>Type tab</u> .
Project tab	Specify project options.

General tab

Use this control... To do this...

File name Displays the name of the file currently selected in the

project manager window.

Last modified date Displays the date that the file was last modified.

Size Displays the size (in bytes) of the file currently selected in

the project manager window.

Type tab

Use this control... To do this...

file description of the file that is currently selected in the project manager window appears in the window. Select another file description from the drop list to change the file type of the file currently selected in the project

manager window.

Project tab

Use this control... To do this...

Project Type Displays the type of project, PSpice and PCB.

PSpice Part Search dialog box

Use this dialog box to place PSpice Parts on the schematic page by searching them in the libraries using PSpice Part Search.

To open this dialog

Select Place - PSpice Component - Search.

Use this control... To do this...

Category View

Hide Categories View/Show Toggle Hide Categories View or Show Categories View

Categories View to hide or unhide the Category View.

Search Online Opens new window in Capture for searching PSpice

models in OrCAD X Capture Marketplace

Categories tab Displays PSpice parts in categories defined by PSpice

parts function

Library tab Displays PSpice parts in the libraries they are associated

to

Search Window View

Search text box Enter the search text to search for PSpice parts in the

PSpice Part Search database

Search drop-down list Provides option to search in selected category or all

categories

Symbol Viewer Displays symbol of the selected part

Part Table View

Part Name Displays part name of the PSpice part

Description Displays brief description of the PSpice part

Symbol Library Displays library path of the selected PSpice part

PDF Export

Use this dialog box to create PDF of the Schematic Design.

To open this dialog

Choose File - Export - PDF.

Use this control... To do this...

Use this Control... To do this...

Output Directory Specify the output directory folder.

Output PDF File Specify the output PDF file name.

Printing Mode Select either Instance mode or Occurrence mode of the

design to be printed

Orientation Select either Portrait or Landscape orientation mode

Create Properties PDF File Select to generate the properties PDF file. The PDF file

is saved in the Output Directory as prop<Output PDF

File>.pdf.

You can also access the Properties PDF file by clicking

on an object and selecting the CrossProbe option.

Create Net & Part Bookmarks Select this option to create Net and Part Bookmarks in

the PDF file

Exclude Properties Select this option to exclude the part's properties from

getting exported to the PDF

Output Paper Size Select the output paper size

Driver Specify the driver that prints the PDF file. By default,

Capture install the OrcadPSPrinter driver.

Converter Select the PDF Converter type from the drop-down list.

You can select one of the following converters:

Adobe Distiller

■ GhostScript/equivalent

■ GhostScript 64/equivalent

Custom

Converter Path Specify the path where the converter executable file

(.exe) is located

Converter Arguments

Specify the converter arguments. Following are the default converter arguments:

/N /q /o \$::capPdfUtil::mPdfFilePath
 \$::capPdfUtil::mPSFilePath

Relative Propagation Delay dialog box

Use the Relative Propagation Delay dialog box to specify a pin-pair and select valid match_group, scope, pin-pair, delta and tolerance for the RELATIVE_PROPAGATION_DELAY property.

The Relative Propagation Delay dialog box allows you to define an error-free RELATIVE PROPAGATION DELAY property.

The RELATIVE_PROPAGATION_DELAY property can have one of the following two syntax:

1. For the target pin-pair

```
<match_group>:<scope>:<pin-pair>::
where <pin-pair> has the following syntax:
<pin1>:<pin2>
```

2. For non-target pin-pairs

<match group>:<scope>:<pin-pair>:<delta>:<tolerance>

To open this dialog

While editing the RELATIVE_PROPAGATION_DELAY property in the Property Editor, choose Invoke UI from the Edit menu press CTRL+U.

OR

Right-click the grid corresponding to the RELATIVE_PROPAGATION_DELAY property and select Invoke UI from the popup menu.

Use this control...

To do this...

Matched Group

Displays the current match group. To change the match group:

- Select a group from the list box.
- Type a new match group name.

Do not press Enter after typing match group name as it will close the dialog box.

Based on the match group selected, all nets contained in it will display in the *Nets Attached* box.

Applies Min and/or Max delay constraint to various pin-pairs. This value may be set to one of the following:

Longest/Shortest pin-pair—Minimum delay is applied to the shortest pin-pair and maximum delay is applied to the longest pin-pair.

A constraint when set on the longest pin-pair of a net is most stringent. If the constraint is met by the longest pin-pair, it is ensured that the constraint will be met by all other pin-pairs of the net also.

- Longest/Shortest Driver/Receiver—Minimum is applied to the shortest Driver/Receiver pin-pair and maximum is applied to the longest Driver/Receiver pin-pair.
- All Drivers/All Receivers—Min/Max constraints apply to all Driver/Receiver pin-pairs.

Specifies whether the scope of the pin-pair is Global or Local. By setting the scope as global, you can define the RELATIVE_PROPAGATION_DELAY property on different nets of same match group. When scope is set as Local, you can define the RELATIVE_PROPAGATION_DELAY property on different pin-pairs of same net.

Specifies the relative value from the target net that all nets in the group should match.

Note: If a delta value is not defined, all members of the group will be matched within the specified tolerance.

Note: The delta value may be negative, in which case the delta is subtracted from the routed length of the Target. If it is positive (or unsigned), the value is added to the routed length of the Target.

Specifies whether the measurement unit for delta is DELAY (ns) or LENGTH mills (mils), micron (um), millimeter (mm), centimeter (cm), and inches (in).

Specifies the maximum allowable propagation delay/length for the pin-pairs.

Scope

Delta

Units

Tolerance

Tol. Units Specifies the unit for Tolerance. You can select one of the

following options in Tol. Units field:

■ %

■ DELAY (ns)

■ LENGTH (mils, um, mm, cm, and in)

Add Pin Pair Displays the Create Pin Pairs dialog box dialog box. Use

this dialog box to define a pin-pair.

Keyboard shortcut: ALT, A

Delete Pin Pair Deletes the pin-pair corresponding to the selected row.

Keyboard shortcut: ALT, D

Set Target Specifies the selected pin-pair as the target pin-pair. The

minimum and maximum propagation delay values for

other nets will be set relative to the target net.

Note: When you select a target net pair, the *Target Net Name*. *Target Pin Pair*, and *Scope* fields above the grid

get populated.

Keyboard shortcut: ALT, S

Delete Target Removes the target status from the specified net. You

can now select a new pin-pair and assign it as target.

Keyboard shortcut: ALT, T

Nets Attached This view-only section displays all the nets that are

attached to the Match group.

OK Performs syntax checking and if the syntax is correct

assigns the RELATIVE PROPAGATION DELAY property

on the selected net.

Note: You can use the <u>User Properties dialog box</u> to assign the

RELATIVE_PROPAGATION_DELAY property to all the bits of a bus at the same time. Make sure that you use the correct syntax for specifying a value for the RELATIVE PROPAGATION DELAY property. The syntax is:

For the target pin-pair:

<match_group>:<scope>:<pin-pair>::

where <pin-pair> has the following syntax:

<pin1>:<pin2>

For non-target pin-pairs:

<match_group>:<scope>:<pin-pair>:<delta>:<tolerance>

The pin-pairs can only be:

- AD:AR
- L:S
- D:R

Remove ECSets from Design

To open this dialog

Choose SI Analysis – Remove Electrical Csets Assignments.

Use this control... To do this...

Remove Click to remove the selected Electrical Csets

assignments.

Note: When you remove an Electrical Cset from an object, the Cset is removed only from the object and not from the design. As a result, when you extract Electrical Cset from the object, again you might get a message

stating that the Cset already exists.

Remove Occurrence Properties box

To open this dialog

Select a design (.DSN) in the Project manager and choose Remove Occurrence Properties (see Remove Occurrence Properties command) from the Design menu.

Click Yes, if you want to remove all backannotation and occurrence properties from the design. Otherwise, click No.



If you chose to remove all backannotation and occurrence properties from your design, these properties are permanently removed from the design. You cannot undo this action.

Rename Hierarchical Port dialog box

To open this dialog

Select a hierarchical port in a library in the Project manager and choose Rename (see Rename command) from the Design menu.

Use this control... To do this...

Name Specify the name of the hierarchical port.

Rename Off-Page Connector dialog box

To open this dialog

Select an off-page connector in a library in the Project manager and choose Rename (see Rename command) from the Design menu.

Use this control... To do this...

Name Specify the name of the off-page connector.

Rename Page dialog box

To open this dialog

Select a schematic page in the Project manager and choose Rename (see <u>Rename command</u>) from the Design menu.

Use this control... To do this...

Name Specify the name of the schematic page.

Rename Part dialog box

To open this dialog

Select a schematic part in a library in the Project manager an choose Rename (see Rename command) from the Design menu.

Use this control... To do this...

Name Specify the name of the part.

Rename Part Property dialog box

To open this dialog

Select a design (.DSN) or a schematic page in the Project manager and choose Rename Part Property (see Rename Part Property command) from the Edit menu.

Use this control... To do this...

Find User Property Type the name of the part property you want to change.

Replace with User Property Type the name of the part property you want.

Rename Power Symbol dialog box

To open this dialog

Select a power or ground symbol in a library in the Project manager and choose Rename (see Rename command) from the Design menu.

OR

Select a power or ground symbol in the schematic page editor and choose Properties from the Edit menu.

Note: If you rename a power or ground symbol using this dialog box, the name is limited to 31 characters.

Use this control... To do this...

Name Specify the name of the power or ground symbol.

Rename Schematic dialog box

To open this dialog

Select a schematic folder in the project manager and choose Rename (see <u>Rename command</u>) from the Design menu.

Use this control... To do this...

Name Specifies the name of the schematic folder.

Rename Title Block dialog box

To open this dialog

Select a title block in a library in the Project manager and choose Rename (see <u>Rename command</u>) from the Design menu.

Use this control... To do this...

Name Specify the name of the title block.

Reorder UnNamed NetGroup Pins dialog box

To open this dialog

Right-click an UnNamed NetGroup and choose Reorder pins for UnNamed NetGroup...

Use this control... To do this...

NetGroup Select pin to change order.

Up Click to move pin up.

Down Click to move pin down.

Replace Cache dialog box

To open this dialog

Select a part in the design cache folder and choose Replace Cache (see <u>Replace Cache command</u>) from the Design menu.

Use this control	To do this
New Part Name	Specify the part's name. The current name appears in the text box.
	Note: If the list of parts is not available in the <i>Part name</i> list box, select a library in the <i>Part Library</i> field.
Part Library	Specify the path and library containing the replacement part. The current path and library appear in the text box.
	Note: When you select a library, the <i>Part Name</i> field displays a sorted list of all parts that you can select.
Browse	Display a standard Windows dialog box for selecting files.
Actions	
Preserve schematic part properties	Retain all instance and occurrence properties of the schematic part in the design, bringing in the graphics, pins, and package properties from the library.
Replace schematic part properties	Bring in graphics, pins, and package properties from the library, totally replacing the schematic part in the design.

Preserve Refdes Preserve the reference designator of parts and/or sym-

bols that you want to change.

Note: This option is not available for symbols that do not require preserving of the reference designator. For example, if you have selected a title block, off-page connector, h-port, or power ground symbols, the *Preserve Refdes* check box will be unavailable for

selection.

Replace dialog box

To open this dialog

In a text editor window, choose Replace from the Edit menu.

Use this control	To do this
Find what	Specify the string to be found and replaced.
Replace with	Specify the string to replace the one specified in the Find What option.
Match whole word only	Specify that the search cannot match the Find What string within another word.
Match case	Specify the search must match the case of the string specified in the Find What option.
Find Next	Find the next occurrence of the specified text, without replacing the currently selected string.
Replace	Replace the currently selected string with the one specified in the Replace With option.
Replace All	Replace all occurrences of the string specified in Find what with the string specified in Replace with. The search and replace takes place in the specified section of the file.

Save Files in Project dialog box

The Save Files in Project dialog box appears when you save a file in a project without first saving the project. Capture may also display this dialog box if you perform an action that requires the project be saved first, and it has not been saved.

Use this control... To do this...

Yes Save the design file (*.DSN) and the project file (*.OPJ).

Yes All Save the design file (*.DSN) and the project file (*.OPJ).

Also save any other open files that are part of the project.

No Don't save the design file (*.DSN).

No All Don't save any file in the project (*.OPJ).

Cancel the action.

Save Part As dialog box

To open this dialog

In the Part editor, choose Save As from the File menu.

Use this control... To do this...

Name Specify the name of the part to be saved.

Library Specify the path and filename of the library that the part

is saved in.

Save Project As dialog box

Save Project As saves the project to a new location along with the associated files present inside or outside the project directory maintaining their internal and external links. Associated files include referred projects, designs, libraries, simulation profiles, output files and so on.

To open this dialog

Perform one of the following:

- Choose File Save Project As.
- Select DSN in the project manager and from the shortcut menu choose Save Project As.
- Select *Design Resources* in the project manager and choose *Save As* from the shortcut menu.

Use this tab	To do this
Destination Directory	Specify the location to which the project should be saved.
Project Name	Shows the current project name. You can edit to change it to a different name.
Settings	Specify whether or not to copy the design file and referred files.
	Referred files include Projects, Libraries, Output Files, Simulation Files and so on referred from the current project.
	Important
	Irrespective of the option selected to copy the referred files, the links to the referred files are

referred files, the links to the referred files are always updated for the new saved project. In case of PSpice projects, PSpice files are always copied to the new location, irrespective of the options selected in the settings tab.

Copy DSN to Project Folder Select to copy the design file to the destination project folder. Selected by default.

Rename DSN to match Select to rename the design file to match the project name. Selected by default.

Copy All Referred Files Select to copy all referred files that are in the current Present Within Project Folder project folder into the destination folder. Selected by default.

Note: The hierarchical structure of these referred files is preserved in the target project directory as well.

Copy All Referred Files

Select to copy all referred files that are outside the Present Out of Project Folder current project folder into the destination folder. The files are copied to a new subfolder MovedFiles inside the target project directory

> **Note:** The hierarchical structure of the referred files is not maintained while copying them.

Save Part Instance dialog box

The Save Part Instance dialog box appears when you finish editing a part instance in the part editor window and close its part editor window.

Use this control	To do this
Update Current	Apply the changes only to the selected part instance.
II	Apply the changes to all instances of the selected part. If you have an identical part from another library, the instances of the part from the second library won't be affected.
Discard	Return to the schematic page editor without applying any changes to the selected part instance.
Cancel	Return to the part editor and continue editing the part.

Schematic Page Properties dialog box

To open this dialog

In the schematic page editor, choose Schematic Page Properties (see Schematic Page Properties command) from the Options menu.

Use this tab	To do this
Page Size	Specify the measuring scale used, the page width and height, and the spacing between pins on a schematic page. For more information, see <u>Page Size tab</u> .

Grid Reference Choose between alphabetic and numeric, and between

ascending and descending for both horizontal and vertical grid references. Also use it to set the grid count for both horizontal and vertical grid references, and set the width of grid references. For more information, see

Grid Reference tab.

Miscellaneous Displays the schematic folder's creation time, last

modification time, and the number of the schematic page

being viewed in the schematic page editor.

Page Size tab

The Schematic Page Properties dialog box appears when you choose the <u>Schematic Page Properties command</u> from the Options menu, when you are in the schematic page editor.

The Design Template dialog box appears when you choose the <u>Design Template command</u> from the Options menu.

Set these options for future schematic pages. Changing these options won't affect schematic pages you've already created.

Use this control	To do this
Units	Specify the unit of measurement future designs are measured in. Select either inches or millimeters. This only affects the schematic page editor. It doesn't affect the part editor, which is always measured in grid units.
New Page Size	Specify the size of new schematic pages in the current design. The first five choices are A to E if the unit measurement is inches, or A4 to A0 if the unit measurement is millimeters.
Width	Displays the width of new schematic pages in the indicated unit measurement. You may specify the width of the custom schematic page. All other schematic page widths are permanently set for the current design.
Height	Displays the height of new schematic pages in the indicated unit measurement. You may specify the height of the custom schematic page. All other schematic page heights are permanently set for the current design.

Pin-to-Pin Spacing Displays the pin-to-pin spacing in the indicated unit

measurement.

Grid Reference tab

The Schematic Page Properties dialog box appears when you choose Schematic Page Properties from the Options menu, when you are in the schematic page editor.

The Design Template dialog box appears when you choose the <u>Design Template command</u> from the Options menu.

Set these options for existing schematic pages. Changing these options won't affect future schematic pages.

Use this control	To do this
Horizontal and Vertical	
Count	Specify the number of divisions in the horizontal or vertical grid references.
Alphabetic and Numeric	Specify whether the grid references are alphabetic or numeric.
Ascending and Descending	Specify whether the grid references ascend or descend.
Width	Specify the width of the grid reference division.
Border Visible	
Displayed	Specify whether the border is visible on the screen.
Printed	Specify if the border is visible on paper.
Grid Reference Visible	
Displayed	Specify whether the grid references are visible on the screen.
Printed	Specify whether the grid references are visible on paper.
Title Block Visible	
Displayed	Specify whether the title block is visible on the screen.
Printed	Specify whether the title block is visible on paper.
ANSI grid references	Specify if schematic pages use the ANSI Standard grid references.

Select Directory dialog box

To open this dialog

Choose the ... (browse Directories) button in the Archive Project dialog box.

Use this control... To do this...

Directories Displays the current selected drive and directory.

Drives Specify the drive to create the new project on.

Create Dir Display the <u>Create Directory dialog box</u> to create a new

directory on the current drive. The new directory is

created below the current selected directory.

Network Display a standard Windows Map Network Drive dialog

box to select a different drive.

Select File Type dialog box

The Select File Type dialog box appears after you add a file to a project.

Use this control... To do this...

File Types list

Select a file type for the file you are adding to the project.
Choose the appropriate type from the provided list:

- EDIF Netlist
- List
- OrCAD X Project
- Report
- Schematic Design
- Schematic Library
- Simulate Stimulus
- Standard Delay File
- Unknown
- VHDL Netlist
- VHDL SimModel
- VHDL Source
- VHDL Synthesis Macro Library
- VHDL Synthesis Target Library
- VHDL Testbench
- Waveform
- XNF Netlist

Selection Filter dialog box

The Selection Filter dialog box allows you to control the selection of objects in a schematic page during a block-select operation. It provides check box options that allow you to include or exclude objects from a list. For example, if you select the Parts, Nets, and Power/GND check boxes, only these objects will be selected when you perform the block-select operation.

To open this dialog

In the schematic page editor

Right-click the schematic page and choose Selection Filter from the popup menu.

OR

Choose Selection Filter from the View menu.

OR

Use the keyboard shortcut, Ctrl + I.

Use this control	To do this
Check box under the Schematic Page Drag Selection Filter group	Specify the objects that should be selected when the mouse pointer is dragged diagonally across the schematic page.
Select All	Select all the check boxes in the Schematic Page Drag Selection Filter group.
Clear All	Clear all the selected check boxes in the Schematic Page Drag Selection Filter group.

Select New Project Path dialog box

The Select New Project Path dialog box allows you to specify a different location for the project in the new format.

New Project Path

Specify the location where you want the project in the new format to be created. The project in the old format will be retained in its original location.

Select Occurrence dialog box

This dialog box displays when you open a schematic page that has multiple occurrences.

Use this control... To do this...

Occurrence list Specify which occurrence of the schematic page you

want to open. This option lists all occurrences available

for the selected schematic page.

Select Simulation Configuration dialog box

To open this dialog

Choose Simulate from the PICFlow menu.

Use this control... To do this...

Simulation configuration Generate a VHDL netlist of your design and store it in the

project manager's In Design folder (if you choose Preroute) or Timed folder (if you choose Postroute) for

simulation.

Note: Before generating the netlist, Capture checks the .DSN file to see if it is current (that is, if you have saved it). If the .DSN file is not current, Capture prompts you to save

it.

Legacy programmable logic projects (that is, projects created with a version of Capture previous to version 14.2) that include schematic components from the Xilinx 4000E library, must use the OrCAD X C4KE VHDL libraries for simulation. However, Xilinx 4000E schematics created with Capture version 14.2 (or any later versions), or schematics that have been modified to include the new Xilinx 4000E part symbols, must use the UNISIM VHDL libraries for simulation.

Set Label State dialog box

To open this dialog

In the schematic page editor, choose Label State, Set from the Edit menu.

Use this control... To do this...

Enter Label Specify a label for the current state of the schematic

page.

Setup dialog box

The Setup dialog box is used to set up, edit and view information about the configuration file used for netlisting and back annotating property information between Capture and PCB Editor. For more information about the configuration file, see <u>Updating the PCB Editor configuration file</u>.

To open this dialog

Click the Setup button in the PCB tab of the Create Netlist or Back Annotate dialog boxes.

Use this control... To do this...

Configuration File Mapping file used to pass properties back and forth

between Capture and PCB Editor.

Order of Preference:

Last Used > Design Directory > CDS_SITE path >

Capture Install Directory

If you have run a previous netlisting or back annotation the configuration file you used is listed. If not, the first found *.CFG file in your design directory is used. If no *.CFG file is found in your design directory, then the CFG file on the path defined by the CDS_SITE environment

variable will be used. Finally, if the CDS_SITE

environment variable is not set, the default will be to use the sample ALLEGRO. CFG file that was installed with

Capture.

Edit Click this button to edit the configuration file listed in the

Configuration File field. The file opens in your default text editor. You can edit the configuration in any text editor. Only changes that are saved to the file get used when you run the netlist. Just having the file open with changes

won't be enough.

Backup Versions Species the number of backup versions of the PST*.DAT

files you want to maintain in your design directory. The

highest value is the latest (newest). For example,

PSTXNET.DAT,2 is the third version of the

PSTXNET.DAT netlist file saved, and PSTXNET.DAT is

the most recent version saved.

Device/Net/Pin name char limit Specifies the maximum permissible character limit or

length of a component, net, or pin name. The default is 31. The maximum value permitted by Capture is 255.

Ignore Electrical constraints

Specifies that the following electrical constraints will be ignored during netlist:

- PROPAGATION_DELAY
- RATSNEST_SCHEDULE
- RELATIVE_PROPAGATION_DELAY
- DIFFERENTIAL_PAIR
- NET_SPACING_TYPE
- NET_PHYSICAL_TYPE
- ELECTRICAL_CONSTRAINT_SET
- RATSNEST_SCHEDULE
- VOLTAGE
- MIN_LINE_WIDTH
- MIN_NECK_WIDTH
- MATCHED_DELAY

Note: These electrical warnings are ignored when the design is forward annotated to layout. However, if any of the above constraints is defined on the board, these constraints will again be ignored in the backannotation process.

Output Warnings

Specifies if netlist warnings (ALG*) are to be logged during netlisting.

Suppress Warnings

Specifies the netlist warnings (ALG*) to be ignored during netlist.

.

Simulation Settings dialog box

To open this dialog

Choose Edit Simulation Profile (see Edit Simulation Profile command) from the PSpice menu.

This dialog box is similar to the Simulation Settings dialog box in PSpice. Both dialog boxes set simulation profile properties.

Use this tab	To do this
General	Specify file information and simulation notes. For more information, see <u>General tab</u>
Analysis	Specify analysis options. This includes time domain, DC sweep, AC sweep and noise, and bias point options. For more information, see <u>Analysis tab</u>
Include Files	Add include files for the current design only, or globally for all designs.
	PSpice searches model libraries, stimulus files, and include files for any information it needs to complete the definition of a part or to run a simulation. Include files are user-defined files that contain:
	■ PSpice commands
	or
	Supplemental text comments that you want to appear in the PSpice output file

extension.

Create include files using any text editor, such as Notepad. Include file names typically have an .INC

Libraries

- Add new model libraries that were created outside of Capture or the model editor
- Remove libraries from the configuration list.

 Removing a library using this dialog box means that you are removing it from the configured list. The library still exists on your computer and you can add it back in if needed.
- Establish whether a model library is for the current design only, or global for all designs
- Change the order in which PSpice searches the model libraries
- Change or add directory search paths

PSpice searches model libraries, stimulus files, and include files for any information it needs to complete the definition of a part or to run a simulation.

For more information, see the <u>Libraries tab</u> help topic.

Add a stimulus file which contains time-based definitions for analog and digital input waveforms.

PSpice searches model libraries, stimulus files, and include files for any information it needs to complete the definition of a part or to run a simulation.

You can create a stimulus file by:

- Manually using the Model Text View of the model editor (or a standard text editor) to create the definition. Stimulus files typically have an .STM extension.
- Using the stimulus editor, which automatically generates a file with an .STL extension

For more information, see the <u>Stimulus tab</u> help topic.

Specify values, limits, and conditions for analog and gate-level simulation, and the information to include in the output file. For more information, see the <u>Options tab</u> help topic.

Stimulus

Options

Data Collection Specify which data is collected, and what file format is

used to write the information. For more information, see

the Data Collection Settings tab help topic.

Probe Window Determine if the Probe window appears when the profile

is opened, and either during simulation or after

simulation has completed. You can also choose to show all markers on open schematics or show the last plot. For more information, see the Probe Window tab help topic.

Analysis tab

The Analysis tab specifies analysis options for the simulation.

Use this control... To do this...

Analysis Type Specify the analysis type. The type determines the

options that are available in the Options box. Analysis

types include:

Time Domain (Transient)

DC Sweep

AC Sweep/Noise

Bias Point

Options Depending on the analysis type selected, a sub-set of

the following options are available.

General settings options

Primary and secondary sweep options

Monte Carlo/Worst Case options

Parametric Sweep options

Temperature Sweep options

Save Bias Point options

Load Bias Point options

General settings options

Use this control... To do this...

Run to time Specifies a finishing time, or stopping time, for the

simulation. You can specify periods shorter than

seconds, by including the time increment (such as "ns")

immediately after the time period.

Start saving data after Specifies the time to wait before saving data. This is

useful for saving only specific portions of the analysis data and reducing the size of the data file. You can specify periods shorter than seconds, by including the time increment (such as "ns") immediately after the time

period.

Transient options

Maximum step size Specifies a smaller internal time step value than the

default value.

Skip the initial transient bias

point calculation

Specifies to skip the calculation of the bias point.

Output File Options This button displays the <u>Transient Output File Options</u>

dialog box

Primary and secondary sweep options

Use this control... To do this...

Sweep variable

Voltage source Specifies the source's voltage is used to set the sweep.

Current source Specifies the source's current is used to set the sweep.

Global parameter Specifies that during the sweep, the global parameter's

value is set to the sweep value and all expressions are

reevaluated.

Model parameter Specifies that the parameter in the model is set to the

sweep value.

Temperature Specifies to set the temperature to the sweep value. For

each value in the sweep, all the circuit components have their model parameters updated to that temperature.

Name Specifies the name of the source.

Model type Specifies the model type.

Model name Specifies the model name.

Parameter name Specifies the name of the global or model parameter.

Sweep Variable

Linear Specifies that the sweep variable is swept linearly from

the starting to the ending value.

Logarithmic Specifies that the sweep variable is swept logarithmically

by octaves or decades.

Value list Specifies that the sweep uses a list of values.

Start value Specifies the starting value for the sweep.

End value Specifies the ending value for the sweep.

Increment Specifies the step size for the sweep.

Monte Carlo/Worst Case options

The Monte Carlo/Worst Case options in the Simulation Settings dialog box's Analysis tab specifies analysis options for the simulation.

Monte Carlo/Worst Case analyses vary the lot or device tolerances of devices, among multiple runs of an analysis (DC sweep, AC sweep, or transient).

You can run either a Monte Carlo or a worst-case analysis, but not both at the same time. Before running either analysis, you must set up the device and lot tolerances of the model parameters to be investigated.

Use this control... To do this...

Monte Carlo Perform a Monte Carlo (statistical) analysis of the circuit.

Worst-case/Sensitivity Perform a sensitivity and worst-case analysis of the

circuit.

Monte Carlo options

Number of runs Specifies the total number of runs to be performed.

Use distribution Specifies whether to use the Uniform or Gaussian

distribution curve. Uniform is the default distribution. You can also create your own distribution curves by clicking

the Distributions button.

Random number seed Specify the seed for the random number generator within

the Monte Carlo analysis. This value must be an odd integer ranging between 1 and 32,767. If this is not

specified, the default value of 17,533 is used.

Save data from Produce output from subsequent runs, after the nominal

(first) run.

<none> Only the nominal run produces output.

All Forces all output to be generated.

 First Generates output only during the specified number of runs.

 Every Generates output every specified number of runs.

Runs (list) Does analysis for the listed runs.

Distributions Display the <u>Distributions dialog box</u>.

Worst-case/Sensitivity options

Vary devices that have Specify which devices are included in the analysis by the

model parameter indicating use of DEV or LOT

tolerance.

Limit devices to type(s) Specify the types of devices to include in the analysis.

The list is a string containing the initial letters of PSpice

primitives.

Save data from each sensitivity

run

Save the worst case data for every run of the DC, AC, or

Time Domain analysis.

More Settings Display the Monte Carlo Worst-Case Output File Options

dialog box.

Parametric Sweep options

Use this control... To do this...

Sweep options

Voltage source Specifies the source's voltage is used to set the sweep.

Current source Specifies the source's current is used to set the sweep.

Global parameter Specifies that during the sweep, the global parameter's

value is set to the sweep value and all expressions are

reevaluated.

Model parameter Specifies that the parameter in the model is set to the

sweep value.

Temperature Specifies to set the temperature to the sweep value. For

each value in the sweep, all the circuit components have their model parameters updated to that temperature.

Name Specifies the name of the source.

Model type Specifies the model type.

Model name Specifies the model name.

Parameter name Specifies the name of the global or model parameter.

Sweep Type

Linear Specifies that the sweep variable is swept linearly from

the starting to the ending value.

Logarithmic Specifies that the sweep variable is swept logarithmically

by octaves or decades.

Value list Specifies that the sweep uses a list of values.

Start value Specifies the starting value for the sweep.

End value Specifies the ending value for the sweep.

Increment Specifies the step size for the sweep.

Temperature Sweep options

Use this control... To do this...

Run the simulation at temperature

Specifies a temperature at which the analysis is done. The value is specified in degrees Centigrade.

Repeat the simulation for each of the temperatures

Specifies that the analysis must be performed for each of the temperatures listed. The values are specified in degrees Centigrade, and must be separated by spaces.

Save Bias Point options

Use this control	To do this
Save bias information in filename	Specifies the path and filename to save the bias point node voltages and inductor currents in.
Options	
Save bias information	Specifies to save the bias either at a specific time, or at each time interval. All times are specified in seconds. If a time interval is specified, then only the latest bias is saved.
When Primary Sweep value is	Specifies the first DC sweep value at which the bias point is to be saved. If there are two sweep variables, Primary Sweep value specifies the first value.
When Secondary Sweep value is	Specifies the second value, if there are two DC sweep variables. If there is only one variable, type the value in the Primary Sweep value text box.
When Parametric Sweep value is	Specifies the transient analysis time at which the bias point is to be saved.
When Monte Carlo run number is	Specifies the number of Monte Carlo or worst-case analysis run for which the bias point is to be saved.
When Temperature and Sweep temperature is	Specifies the temperature at which the bias point is to be saved.
Do not save subcircuit voltages and currents	When selected, specifies that node voltages and inductor currents for subcircuits are not saved.

Load Bias Point options

Use this control...

Load bias information from filename	Specifies the name of the file to load bias points. It is used in setting initial bias conditions for subsequent
	simulations. However, loading a bias point file does not

To do this...

guarantee convergence.

Data Collection Settings tab

The Data Collection Settings tab specifies data collection options.

Use this control... To do this...

Schematic/Circuit Data

All voltages, currents, and digital Collect data for voltages, currents, and digital states.

states

(.CSD)

All but internal subcircuit data

Exclude subcircuit data.

At Markers only Collect data for the marked node only.

None Do not collect data.

Save data in the CSDF format Specify that PSpice will write simulation results to the

data file in ASCII format following the CSDF convention.

General tab

The General tab specifies file properties, and simulation profile notes.

Use this control... To do this...

Simulation Profile Specifies the name of the current simulation profile.

Input

Project Name (.OPJ) Specifies the path and file name of the project file

containing the schematic for simulation.

Schematic filename (.DSN) If Schematic is selected, this specifies the path and file

name of the design file containing the schematic for

simulation.

Schematic name

Specifies the name of the root schematic for the

simulation profile.

Output

Output filename Specifies the file name for simulation output.

Probe data filename Specifies the name for Probe data output.

Notes Provides a text area for you to record notes on the active

simulation profile.

Include Files tab

Include files contain PSpice circuit file commands. PSpice reads these before reading the netlist or simulation profile. Include files are useful for defining mathematical functions used in expressions.

Use this control	To do this
Filename	Display an include file name to be added to the current design only, or globally for all designs.
Include Files	Lists the include files to be loaded for the simulation.
Add as Global	Add the include file named in the Filename box to the Include Files list box, as global to all designs.
Add to Design	Add the include file named in the Filename box to the Include Files list box, for the current design only.
Edit File	Open a file you select from the Include Files list for editing.

Libraries tab

Use this tab to configure global and design only model libraries.

Use this control	To do this
Filename	Display a selected model library file name to be added to the current design only, or globally for all designs.
Include Files	Lists the model library files to be loaded for the simulation.
Add as Global	Add the include file named in the Filename box to the Include Files list box, as global to all designs.
Add to Design	Add the model library file named in the Filename box to the Library Files list box, for the current design only.
Edit File	Open a model library you select from the Library Files list for editing.

Options tab

Use this control... To do this...

Analog Simulation Use the Analog Simulation settings to fine-tune analog

simulation accuracy, set iteration limits, set operating

temperature, and specify MOSFET parameters.

General Enter values for speed level, tolerances, and minimum

conductance.

Autoconverge Enter relaxed limits for various options that PSpice can

modify during a simulation to achieve convergence.

MOSFET options Enter values for the default drain area, default source

area, default length, and default width.

Analog Advanced options Enter values for the total transient iteration limit, relative

magnitude for matrix pivot, and absolute magnitude for

matrix pivot.

General Enter values for solver, ITL5, and minimum conductance.

Bias Point Enter values for various options to enable GMIN

stepping, ITL6, PSEUDOTRAN

Transient Enter values related to transient analysis, such as

METHOD, CSHUNT, TRANCONV.

Gate-level Simulation Use the Gate-Level Simulation settings to set timing, I/O

levels for A/D interfaces, drive strength, and error

message limits.

General Enter various options for default delay selector, flip-flop

and latches' initial state, default digital level, suppressing

simulation error messages in the .dat file

Advance options Enter values for the minimum output drive resistance,

maximum output drive resistance, overdrive ratio, default

delay calculation, and error message limits.

Output file

General Use the Output File settings to select the types of

information PSpice A/D saves to the simulation output

file.

Note: The option names shown correspond to the option names used in the PSpice OPTIONS command. For more information about this command, refer to the PSpice

Reference document.

Probe Window tab

Use this control... To do this...

Display Probe window when

profile is opened

Display the Probe windows that were displayed the last

time the profile was opened.

Display Probe window

during simulation. Display the Probe windows when the simulation is

running, and update the waveforms as the simulation

progresses.

after simulation has completed. Display the Probe windows when the simulation is

finished.

Show

All markers on open schematics Show the traces for all the markers that are placed on

currently open designs in Capture.

Last plot Show the traces that were used the last time the profile

was opened.

Nothing Show the traces that were used the last time the profile

was opened.

Stimulus tab

Use this tab to configure global and design stimulus files.

Use this control... To do this...

Filename Display a selected stimulus file name to be added to the

current design only, or globally for all designs.

Include Files Lists the stimulus files to be loaded for the simulation.

Add as Global Add the stimulus file named in the Filename box to the

Stimulus Files list box, as global to all designs.

Add to Design Add the stimulus file named in the Filename box to the

Stimulus Files list box, for the current design only.

Edit File Open a stimulus file you select from the Stimulus Files

list for editing.

Specify Part Filter dialog box

To open this dialog

In the Place Part dialog box click the Filter button.

Use this control	To do this
USE HIIS COILLIOL	10 40 1113

Contains Simulation Model Specify that you want to restrict the part search to

only parts that have associated PSpice or HDL

simulation models.

PSpice Model Specify that you want to restrict the part search to

only parts that have associated PSpice models. You can also restrict the search to only parts that have associated parameterized or non-parameterized

PSpice models.

Smoke information Specify that you want to restrict the part search to

only parts that have associated PSpice models that

contain smoke information.

HDL Model Specify that you want to restrict the part search to

only parts that have associated HDL models. You can also restrict the search to only parts that have

Specify that you want to restrict the part search to

associated VHDL or Verilog models.

Contains Packaging

Information only parts that have packaging information.

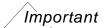
Split Part Section Input Spreadsheet

The Split Part Section Input Spreadsheet allows you to divide a part into multiple sections.

Each row in the Split Part Section Input Spreadsheet corresponds to a pin while each column corresponds to properties associated with the pins. The property names are listed as the column header. You can sort a property by double-clicking its column header.

■ Part Name—Displays the name of the part. This is a non-editable field.

■ Part Ref Prefix—Displays the part reference. This is a non-editable field.



You need to select a single-sectioned part from a library. You can split a multi-sectioned part only when it has already been split using the Split Part Section Input Spreadsheet.



To sort on any property, double-click its name in the column header.



You can hide or show a property column in the Split Part Section Input Spreadsheet. To do this, right-click the property column header you want to hide and select *Hide*. The selected property column will not appear now. To show a property column, right-click the property column header next, on the right-hand side of the hidden property column and select *Unhide*. The hidden property column appears in the Split Part Section Input Spreadsheet. Alternatively, you can show a property column by:

- □ Double-clicking the column handle (♣) of the property column header.
- Dragging the column handle of the property column header.

(only the last two methods can be used to show a property column, which is the last column in the Split Part Section Input Spreadsheet).

You can change the order in which the property columns appear in the Split Part Section Input Spreadsheet. To do this, select the property column header you want to move and drag and drop it to the location where you want it in the Split Part Section Input Spreadsheet.

You can use standard copy and paste feature to copy all data from the Split Part Section Input Spreadsheet to MS Excel. You can later use the MS Excel file for archiving or documentation. It is recommended that you avoid using MS Excel to paste information into the Split Part Section Input Spreadsheet.

The spreadsheet window is resizable. You can resize the window using the resize cursor you see when you move the mouse pointer to any of the edges of the dialog. You can also use the standard Maximize button on the top right corner of the window.

Use this control... To do this...

Part Numbering Specify a numbering format (alphabetic or numeric) that should be

added as suffix to the current part reference for the split part.

If you select Alphabetic, an alphabet (between A to Z) will be added as a suffix to the current part reference for each of the split parts.

If you select Numeric, a number (between 1 and 1024) will be added as a suffix to the current part reference for each of the split parts.

Note: The Section property column changes based on your selection in the Part Numbering group. For example, if Alphabetic is selected, the Section property column displays "A".

No. of Sections Specify the number of sections you want to have in the split part.

Note: If you select alphabetic numbering, then you can create up to a maximum of 26 sections only. If you select numbering,

then you can create up to a maximum of 1024 sections.

Number Specify the pin number.

Name Specify the name of the pin.

Type

Specify the type of pin. To change a pin type, select the Type cell, and select *Input, Output, Passive, Open Emitter, Open collector, 3 State, Bidirectional,* or *Power.*



You can select the Type cells for multiple pins simultaneously using the SHIFT+Down Arrow keys and then enter the pin type. The selected Type cells get populated with the pin type of your choice. Alternatively, you can:

- Select the Type cells for multiple pins simultaneously using the SHIFT+Left mouse button click, then press the CTRL key, and then select a pin type of your choice from the list box. The selected Type cells get populated with the pin type of your choice.
- Click the first cell of the range, and then drag to the last cell, and then enter the pin type of your choice. The selected Type cells get populated with the pin type of your choice.

(You can use these methods to make selection in the Shape, Position, and Section property column list boxes also).

Shape

Specify a shape for the pin. To change a pin shape, select the Shape cell, and select Clock, Dot, Dot-Clock, Line, Short, Short Clock, Short Dot, Short Dot-Clock, Zero length.

PinGroup

Specify a value for each swappable (input) pin of the part.

Position

Specify the pin position as left, right, top or bottom. To change position for a pin, select the Position cell, and select *Left*, *Right*, *Top*, or *Bottom*.

Section

Specify a section number. By default, all pins are assigned the sections 1 or A depending on the selection made in the Part Numbering group.

To change a section for a pin, select the Section cell, and select the required section number from the list.



You can select Section cells for multiple pins simultaneously using the SHIFT+Down arrow keys and enter the section number. Alternatively, you can:

- Select the Section cells for multiple pins simultaneously using the SHIFT+Left mouse button click, then press the CTRL key, and then select a section number of your choice from the list box. The selected Section cells get populated with the section number of your choice.
- Click the first cell of the range, and then drag to the last cell, and then enter the section number of your choice. The selected Section cells get populated with the section number of your choice.



You can select alternate Section cells for multiple pins simultaneously using the CTRL+Left mouse button click and enter the section number.

Add Pins

Add new pins at the end of the current pin set in the Split Part Section Input Spreadsheet.

Delete Pins

Delete selected row containing the pin information from the Split Part Section Input Spreadsheet.



Once you delete a pin from the Split Part Section Input Spreadsheet, you cannot retrieve it later.

Save Splits the part into multiple sections as specified in the Section

property column and saves the current part.

If any warnings are generated during the save operation, a message box appears asking you whether you want to view the warnings. If you want to view the warnings, click the View Warnings button. The Split Part Section Input Spreadsheet expands and displays a grid showing warnings messages. If you select the Continue button, the

split part is saved as is.

Hide Warnings Hide the warning messages.

Show Warnings Show the warning messages again.

Save As Retain the original part and save the changed part as a new part in

the same library.

Synthesis Option dialog box

This dialog box provides a method to set certain options for Synplify (Synplicity's synthesis tool).

Use this control To	do this
---------------------	---------

Interactive Mode Specify that Synplify is started in interactive mode,

allowing you to set any particular synthesis or

optimization options, "on the fly."

Batch Mode Specify a batch file that runs Synplify "in the

background," according to the parameters specified in

the file.

Create Synplify Project Specify that Capture create a new Synplify project for the

current synthesis run. If you have already created a Synplify project, perhaps from a previous synthesis run, you can specify a path to that project in the appropriate

text box.

Note: For specific information on the various Synplify commands, please refer to your Synplify documentation.

Transient Output File Options dialog box

The Transient Output File Options dialog box appears when you select the General Settings option from the <u>Analysis tab</u> of the Simulate Settings dialog box, and click the Output File Options button.

Use this control	To do this
Print values in the output file every:	Specify the interval for printing transient values to the output file.
Perform Fourier Analysis	Perform a Fourier analysis, which decomposes the results of transient analysis to Fourier components.
Center Frequency	Specify the fundamental frequency. Not all of the transient results are used; only the interval from the end, back to 1/frequency before the end is used. This means that the transient analysis must be at least 1/frequency seconds long.
Number of Harmonies	Specify the number of harmonics of the selected voltages and currents to be calculated in the transient analysis.
Output Variables	Specify a list of output variables. The output is split up depending upon the width of the data columns and the output width.
Include detailed bias point information for nonlinear controlled sources and semiconductors (/OP)	Include detailed information about the bias point. The bias point is calculated regardless of whether this option is selected or not. However, if this option is not selected, then the only information about the bias point is a list of the node voltages, voltage source currents, and total power dissipation. Selecting this option can cause the small-signal (linearized) parameters of all the nonlinear controlled sources and all the semiconductor devices to be placed in the output file. This option controls the output for the regular bias point only.

Update Properties dialog box

To open this dialog

In the project manager, choose *Update Properties* (see <u>Update Properties command</u>) from the *Tools* menu.

Use this control	To do this
Scope	Specify whether to process all the properties in the design or just in the selected schematic page or pages.
Mode	Update either instances or occurrences. Capture automatically sets this option based on the project type. FPGA and PSpice projects default to use instances, while PCB and Schematic projects default to occurrences.
Action	
Update parts and Update nets	Specify whether to update the properties of parts or nets.
Use case insensitive compares	Match the combined property string with update properties without regards to case sensitivity.
Convert the update property to uppercase	Convert the case of characters in the update property to uppercase. The update file itself remains unchanged.
Unconditionally update the property (normally only updated if empty)	Unconditionally change the specified property. By default, a property is only updated if it is empty. That is, properties with values already in them are not updated.
Do not change updated properties visibility	Specify that the visibility of the updated properties is not changed
Make the updated property visible	Specify that the updated property is to be made visible.
Make the updated property invisible	Specify that the updated property is to be made invisible.
Create a report file	Specify whether Capture creates a report file.
Report file	Specify a report filename and path.
Property update file	Specify an update file. The update file must be in ASCII format.

Browse

Displays a standard Windows dialog box for selecting files.

Update Old Project wizard

The Update Old Project wizard allows you to convert your project to the Capture 10.0 format. Converting your analog project to the Capture 10.0 format has the following benefits:

- Capture 10.0 introduces a new directory structure for analog projects that makes it easier to manage the files for the project.
- You can use the new simulation profile features in PSpice if your project is in the Capture 10.0 format. For more information on the new simulation profile features in PSpice, see the PSpice online help.

Important

Once you convert your project to the new format, you cannot open the project in the new format in Capture version 9.2.3 or older versions. You can create the project in the new format in a different location so that you have a backup of the project created in Capture 9.2.3 or older versions.

Important

Before you convert a project to the new format, ensure that the schematic names do not have the / (forward slash) or the \ (backward slash) character. If you do not do this, the conversion will fail. To rename a schematic that has the / or \ character in its name, select the schematic name in the Capture Project Manager and choose Rename from the Design menu.

Convert the Project

Select this option if you want to convert the project to the new format.

Retain Old Project

Select this check box if you want to retain the project in its original location and create the project in the new format in a different location.

If you do not select this check box, the project in the new format will overwrite the project in the old format.

Note: Cadence recommends that you select this check box so that you have a backup of the project in the old format.

None

Select this option if you do not convert the project to the new format.

Do not ask me this question again

Select this check box if you want Capture to remember the settings in the Update Old Project wizard.

■ If the Convert the Project option is selected, any analog project created using version Capture 9.2.3 or older versions that you open in Capture 10.0 will be automatically converted to the new format.

Later on, if you want to disable automatic conversion of old analog projects to the new project format, delete the entry given below that exists in the [PSPICE] section of the PSPICE.INI file.

□ CONVERTDESIGN=CONVERT

Or

- CONVERTDESIGN=CONVERT_AND_RETAIN
- If the None option is selected, Capture will not prompt you to convert any analog project created using Capture version 9.2.3 or older versions to the new format.

Later on, if you want to enable conversion of old analog projects to the new project format, delete the following entry that exists in the <code>[PSPICE]</code> section of the PSPICE.INI file located in the <code>/tools/pspice</code> directory under your installation directory:

CONVERTDESIGN=NOCONVERT

User Properties dialog box

To open this dialog

- In the part editor:
 - Click the *User Properties* button in Place Pin dialog box. It contains name and value of the user-defined property.

- In the *Property Sheet* pane, click the add row icon () to add user-defined properties. Some of the commonly used properties are available in the drop-down list.
- In the schematic page editor:

Click the *User Properties* button in Edit Part Properties dialog box.

Use this control... To do this...

Name List the item's properties.

Value Display the value of each property.

Attributes Display the attributes of each property. An "R" indicates

the property is read-only, and a "V" indicates the property is visible to the user. You cannot remove or change the values of read-only properties, but you can set their

visibility in the Display Properties dialog box.

New Display the New Property dialog box so you can create a

property for the item.

Remove Remove the selected property from the item's property

list.

Display Properties dialog box so you can

change the appearance of the selected property.

Note: You can use the User Properties dialog box to assign <u>PROPAGATION DELAY</u> and <u>RELATIVE_PROPAGATION_DELAY</u> properties to all the bits of a bus at the same time. Make sure that the syntax is correct. For more information, see <u>Assigning signal flow properties</u>.

Update Alias dialog box

Use this control... To do this...

Part Aliases In this section, click the add row icon

(🕎) to specify a new part alias.

Click the Add Alias button () to

add the new alias.

Use this control... To do this...

Apply Click *Apply* to save the alias. You can

see the new alias added to the library in

the project manager tab.

Update Layout dialog box

To open the Update Layout dialog box, do one of the following:

■ Select *Update Layout* from the *PCB* menu.

Right-click the board file under the *Layout* folder of project manager and select *Update Layout* from the shortcut menu.

Click the icon, in the PCB toolbar.

This dialog box allows you to view the differences with respect to the selected board, and synchronize the layout from the schematic. It shows the type of change, addition, modification or removal of a design object.

Use this control	To do this
Schematic	Displays the location of the project file.
Layout	Displays the location of the currently active layout file.
	If there are multiple layout files, click the drop-down list to select the required file.

Preview section

This section displays the list of differences based on the direction of the arrow:

Change Type

Type can be any of the following:

- □ Gate Swap
- □ Ref Des
- □ Pin Swap
- Component Property
- □ Net Property
- Pin Property
- Component
- □ Net
- Connection
- Object
- Action
- New Value
- Old Value

If you click *Sync*, forward annotation of changes takes place from Capture to PCB Editor.

Allows you to view changes by:

- Sorting using column header
- Filtering using the selected values in the drop-down list below the column header

Sync button

Filter icon

Update Schematic dialog box

To open the Update Schematic dialog box, do one of the following:

- Select *Update Schematic* from the *PCB* menu.
- Right-click the board file under the *Layout* folder of project manager and select *Update Schematic*.
- Click the icon, in the PCB toolbar.

This dialog box allows you to view the connectivity differences with respect to the selected board, and synchronize the schematic from the layout. It shows the type of change, addition, modification or removal of a design object.

Use this control	To do this
Schematic	Displays the location of the project file.
Layout	Displays the location of the currently active layout file.
	If there are multiple layout files, click the drop-down list to select the required file.

Preview section

This section displays the list of differences based on the direction of the arrow:

Change Type

Type can be any of the following:

- □ Gate Swap
- □ Ref Des
- □ Pin Swap
- Component Property
- Net Property
- Pin Property
- Component
- □ Net
- Connection
- Object
- Action
- New Value
- Old Value

Sync button

When you click *Sync*, changes in the PCB Editor board are back annotated to the Capture schematic to ensure the physical board design is synchronized with the logical schematic design.

Filter icon

Allows you to view changes by:

- Sorting using column header
- Filtering using the selected values in the drop-down list below the column header

Validate ECSets in design

To open this dialog

Choose SI Analysis – Validate Electrical Csets Assignments.

Use this control... To do this...

Remove association on fail Check to remove Electrical Cset association if validation

fails

Validate Click to validate the selected Electrical Csets.

VHDL Samples dialog box

To open this dialog

Choose Samples from the Edit menu.

When you select a sample in the upper box, the associated sample lines appear in the lower box. Double-click on the sample type in the upper box or select it and click OK to copy the sample into the text editor.

The VHDL Samples dialog box contains an alphabetical reference of VHDL language keywords and samples.

In addition to the samples, there are overviews of four important VHDL packages compatible with Capture:

- Numeric_Std (IEEE 1076.3)
- Standard (IEEE 1076)
- Std_Logic (IEEE 1076-1164)
- Textio (IEEE 1076)

Upper box

Displays a list of VHDL sample types. When you select a sample type, the associated sample lines appear in the lower box.

Use this control	To do this
Component Instance	This sample provides a template for component instantiation in a VHDL model.
ENTITY/ARCHITECTURE	This sample provides a template for the ENTITY and ARCHITECTURE statements required in all VHDL models.
PROCESS STATEMENT	This sample provides a template for sequential PROCESS statements in your VHDL model.
Testbench clock	This sample provides a template for a VHDL clock definition.
Testbench Self-Check (ASSERT)	This sample provides a template for defining report statements that appear when a particular condition is met during simulation.
Testbench Table of Vectors	This sample provides a template for defining a table of test bench vectors.
Testbench Wire to Bus	This sample merges scalar ports into a vector signal.

Note: You can add your own samples to the list by editing the STANDARD.VHX file in the /Capture directory.

Lower box

Displays a list of VHDL sample file lines. The sample lines displayed are associated with the selected sample type in the upper box.

OK

Copies the contents of the lower box to the active VHDL file and to the Clipboard.

View DRC Marker dialog box

To open this dialog

Select a DRC marker and choose Properties from the Edit menu.

Use this control... To do this...

DRC Marker text box Displays the DRC marker number and message. This is

the same message that appears in the session log and

the DRC report.

Zoom Scale dialog box

To open this dialog

In the schematic page editor, choose *Zoom – Scale* from the View menu.

Use this control... To do this...

X% Choose a predefined zoom scale ranging from 25% to

400%.

Custom X% Specify a custom zoom scale.