

OrCAD Capture Reference Guide

**Product Version 17.4-2020
June 2020**

© 2020 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 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.

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

Contents

<u>About this Manual</u>	19
<u>Welcome</u>	19
<u>Where to Find Information in this Manual</u>	19
<u>Menu Commands</u>	19
<u>Window descriptions</u>	19
<u>Dialog Box descriptions</u>	20
<u>Where to Find Additional Information</u>	21
<u>How to Contact Technical Support</u>	21

2

<u>Project manager command reference</u>	23
<u>File menu</u>	23
<u>New command</u>	24
<u>Open command</u>	24
<u>Close Project command</u>	25
<u>Save command</u>	26
<u>Save As command</u>	26
<u>Save Project As</u>	27
<u>Archive Project command</u>	27
<u>Print Preview command</u>	28
<u>Print command</u>	28
<u>Print Setup command</u>	29
<u>1,2,3,4 command</u>	29
<u>Change Product command</u>	29
<u>Exit command</u>	30
<u>Design menu</u>	30
<u>New Schematic command</u>	31
<u>New Schematic Page command</u>	31
<u>New Part command</u>	32

OrCAD Capture Reference Guide

<u>New Part from Spreadsheet command</u>	32
<u>Rename command</u>	33
<u>Remove Occurrence Properties command</u>	33
<u>Make Root command</u>	33
<u>Replace Cache command</u>	34
<u>Update Cache command</u>	36
<u>Cleanup Cache command</u>	37
<u>Set Password Command</u>	37
<u>Remove Password</u>	37
<u>Change Password</u>	38
<u>Edit menu</u>	38
<u>Cut command</u>	39
<u>Copy command</u>	39
<u>Paste command</u>	40
<u>Lock command</u>	41
<u>UnLock command</u>	41
<u>Project command</u>	42
<u>Properties command</u>	42
<u>Object Properties command</u>	44
<u>Browse command</u>	45
<u>Find command</u>	49
<u>Rename Part Property command</u>	50
<u>Delete Part Property command</u>	51
<u>Replace command</u>	51
<u>Go To command</u>	51
<u>Clear Session Log command</u>	52
<u>View</u>	52
<u>Toolbar command</u>	53
<u>Capture Toolbar command</u>	53
<u>Draw Toolbar command</u>	53
<u>PSpice Toolbar command</u>	54
<u>Command Window command</u>	54
<u>Tools menu</u>	55
<u>Annotate command</u>	55
<u>Update Properties command</u>	55
<u>Bill of Materials command</u>	56

OrCAD Capture Reference Guide

<u>Export Properties command</u>	57
<u>Import Properties command</u>	58
<u>Generate Part command</u>	59
<u>Split Part command</u>	61
<u>Customize command</u>	61
<u>PSpice</u>	61
<u>Bias Points command</u>	62
<u>New Simulation Profile command</u>	62
<u>Edit Simulation Profile command</u>	62
<u>Run command</u>	63
<u>View Simulation Results command</u>	63
<u>View Output File command</u>	64
<u>Make Active command</u>	64
<u>Simulate Selected Profile(s) command</u>	64
<u>Create Netlist command</u>	65
<u>View Netlist command</u>	66
<u>Marker List command</u>	66
<u>Options menu</u>	66
<u>Autobackup command</u>	66
<u>Preferences command</u>	67
<u>More Preferences command</u>	67
<u>Design Template command</u>	71
<u>Design Properties command</u>	72
<u>Window menu</u>	72
<u>New Window command</u>	73
<u>Cascade command</u>	73
<u>Tile Horizontally command</u>	73
<u>Tile Vertically command</u>	73
<u>Close All Tabs of Active Project command</u>	74
<u>Close All Tabs of Active Project Except Current command</u>	74
<u>Arrange Icons command</u>	74
<u>1,2.... command</u>	74
<u>Close All Windows</u>	75
<u>Help menu</u>	75
<u>PSpice for TI Help command</u>	75
<u>Known Problems and Solutions command</u>	76

OrCAD Capture Reference Guide

<u>What's New command</u>	76
<u>About PSpice for TI command</u>	76
<u>Documentation command</u>	76
<u>Pop-Up Menu</u>	77
<u>Add File command</u>	78
<u>Annotate command</u>	78
<u>Change Project Type command</u>	78
<u>Cleanup Cache command</u>	78
<u>Copy command</u>	79
<u>Cut command</u>	79
<u>Delete command</u>	79
<u>Design Properties command</u>	79
<u>Edit Object Properties command</u>	79
<u>Edit Page command</u>	80
<u>Edit selected object properties command</u>	80
<u>Find command</u>	80
<u>Lock command</u>	80
<u>Make Root command</u>	80
<u>New Page command</u>	81
<u>New Schematic command</u>	81
<u>Open File Location command</u>	81
<u>Paste command</u>	81
<u>Properties command</u>	81
<u>Rename command</u>	81
<u>Reports – Export Placement command</u>	81
<u>Reports – Export Properties command</u>	81
<u>Save command</u>	82
<u>Save As command</u>	82
<u>Save Project As command</u>	82
<u>Schematic Page Properties command</u>	82
<u>UnLock command</u>	82

3

Schematic page editor and part editor command reference 83

<u>File menu</u>	83
------------------------	----

OrCAD Capture Reference Guide

<u>New command</u>	84
<u>Open command</u>	86
<u>Close command</u>	88
<u>Save command</u>	88
<u>Export Selection command</u>	89
<u>Import Selection command</u>	89
<u>Export ISCF command</u>	89
<u>Print Preview command</u>	90
<u>Print command</u>	90
<u>Print Setup command</u>	90
<u>Print Area command</u>	91
<u>Clear command</u>	91
<u>Import Design command</u>	91
<u>Export Design command</u>	92
<u>1,2,3,4 command</u>	92
<u>Exit command</u>	92
<u>Edit menu</u>	92
<u>Undo command</u>	93
<u>Redo command</u>	94
<u>Repeat command</u>	95
<u>Label State commands</u>	96
<u>Cut command</u>	96
<u>Copy command</u>	97
<u>Paste command</u>	97
<u>Delete command</u>	98
<u>Label command</u>	99
<u>Select All command</u>	100
<u>Part command</u>	100
<u>Properties command</u>	101
<u>Part command</u>	103
<u>Reset Location command</u>	104
<u>Mirror command</u>	105
<u>Rotate command</u>	106
<u>Lock command</u>	107
<u>UnLock command</u>	107
<u>Find command</u>	107

OrCAD Capture Reference Guide

<u>Global Replace command</u>	108
<u>Align Command</u>	109
<u>View menu</u>	114
<u>Ascend Hierarchy command</u>	115
<u>Convert command</u>	116
<u>Descend Hierarchy command</u>	116
<u>Go To command</u>	117
<u>Synchronize Up command</u>	117
<u>Synchronize Down command</u>	118
<u>Synchronize Across command</u>	118
<u>Previous Part command</u>	118
<u>Next Part command</u>	118
<u>Previous Page command</u>	119
<u>Next Page command</u>	119
<u>Package command</u>	119
<u>Zoom command</u>	120
<u>Tool Palette command</u>	122
<u>Toolbar command</u>	123
<u>Status Bar command</u>	123
<u>Command Window command</u>	123
<u>Grid command</u>	124
<u>Grid References command</u>	124
<u>Selection Filter command</u>	124
<u>Invoke UI command</u>	125
<u>Place menu</u>	125
<u>Part command</u>	127
<u>PSpice Component command</u>	127
<u>Parameterized Part command</u>	129
<u>NetGroup command</u>	129
<u>Wire command</u>	129
<u>Auto Wire Two Points command</u>	130
<u>Auto Wire Multiple Points command</u>	130
<u>Auto Wire Connect to Bus</u>	130
<u>Bus command</u>	131
<u>Junction command</u>	132
<u>Bus Entry command</u>	132

OrCAD Capture Reference Guide

<u>Net Alias command</u>	133
<u>Power command</u>	133
<u>Ground command</u>	133
<u>Off-Page Connector command</u>	134
<u>Hierarchical Block command</u>	134
<u>Hierarchical Port command</u>	135
<u>Hierarchical Pin command</u>	135
<u>No Connect command</u>	136
<u>Pin command</u>	136
<u>Pin Array command</u>	137
<u>Title Block command</u>	137
<u>Bookmark command</u>	140
<u>Text command</u>	140
<u>IEEE Symbol command</u>	140
<u>Line command</u>	141
<u>Rectangle command</u>	141
<u>Ellipse command</u>	141
<u>Arc command</u>	142
<u>Elliptical Arc command</u>	142
<u>Bezier curve command</u>	142
<u>Polyline command</u>	143
<u>Picture command</u>	143
<u>OleObject command</u>	144
<u>PSpice/Markers menu</u>	144
<u>Markers command</u>	144
<u>Marker List command</u>	145
<u>Voltage Level command</u>	145
<u>Voltage Differential command</u>	146
<u>Current Into Pin command</u>	146
<u>Power Dissipation command</u>	146
<u>Advanced command</u>	147
<u>Plot Window Templates command</u>	147
<u>Show All command</u>	147
<u>Hide All command</u>	147
<u>Delete All command</u>	148
<u>Accessories menu</u>	148

OrCAD Capture Reference Guide

<u>Accessories command</u>	148
<u>Options menu</u>	148
<u>Autobackup command</u>	149
<u>Preferences command</u>	149
<u>Design Template command</u>	150
<u>Schematic Page Properties command</u>	150
<u>Window menu</u>	150
<u>New Window command</u>	151
<u>Cascade command</u>	151
<u>Tile Horizontally command</u>	151
<u>Tile Vertically command</u>	151
<u>Arrange Icons command</u>	152
<u>1,2.... command</u>	152
<u>Help menu</u>	152
<u>PSpice for TI Help command</u>	153
<u>Known Problems and Solutions command</u>	153
<u>What's New command</u>	153
<u>About PSpice for TI command</u>	153
<u>Documentation command</u>	154
<u>Pop-up menu</u>	154
<u>Mirror Horizontally command</u>	155
<u>Mirror Vertically command</u>	155
<u>Rotate command</u>	155
<u>Edit Properties command</u>	155
<u>Edit Part command</u>	155
<u>Select Entire Net command</u>	156
<u>Edit Wire Properties command</u>	156
<u>Descend Hierarchy command</u>	156
<u>Synchronize Up command</u>	156
<u>Synchronize Down command</u>	157
<u>Synchronize Across command</u>	157
<u>Connect to Bus command</u>	157
<u>User Assigned Reference – Set command</u>	157
<u>Update Design Hierarchy command</u>	157
<u>Selection Filter</u>	157
<u>Zoom In command</u>	158

OrCAD Capture Reference Guide

<u>Zoom Out command</u>	158
<u>Go To command</u>	158
<u>Delete command</u>	158
<u>Tooltip command</u>	158
<u>Lock command</u>	158
<u>UnLock command</u>	158
<u>Add or Remove Pins on NetGroup Block command</u>	159

4

Session log command reference

<u>File menu</u>	161
<u>New command</u>	162
<u>Open command</u>	162
<u>Save command</u>	163
<u>Save As command</u>	163
<u>Print Preview command</u>	164
<u>Print command</u>	164
<u>Print Setup command</u>	165
<u>Import Design command</u>	165
<u>Exit command</u>	166
<u>1,2,3,4 command</u>	166
<u>View menu</u>	166
<u>Toolbar command</u>	167
<u>Status Bar command</u>	167
<u>Command Window command</u>	167
<u>Edit menu</u>	168
<u>Copy command</u>	168
<u>Select All command</u>	168
<u>Find command</u>	169
<u>Clear Session Log command</u>	169
<u>Options menu</u>	170
<u>Preferences command</u>	170
<u>Design Template command</u>	170
<u>Autobackup command</u>	170
<u>Window menu</u>	171

OrCAD Capture Reference Guide

<u>Cascade command</u>	171
<u>Tile Horizontally command</u>	172
<u>Tile Vertically command</u>	172
<u>Arrange Icons command</u>	172
<u>1,2.... command</u>	173
<u>Close All Windows</u>	173
<u>Help menu</u>	173
<u>PSpice for TI Help command</u>	174
<u>Known Problems and Solutions command</u>	174
<u>What's New command</u>	174
<u>About PSpice for TI command</u>	174
<u>Documentation command</u>	175

5

Command Window command reference 177

<u>Command Window pop-up menu</u>	177
<u>Font command</u>	177
<u>Background Color command</u>	177
<u>Text Color command</u>	178
<u>Save command</u>	178
<u>Clear All command</u>	178

6

Window descriptions 181

<u>Browse window</u>	181
<u>Session frame window</u>	183
<u>Session log window</u>	184
<u>Part editor window</u>	185
<u>Property Editor window</u>	185
<u>Project manager window</u>	190
<u>Schematic page editor window</u>	191
<u>Text editor window</u>	192
<u>Browse Spreadsheet editor window</u>	196
<u>Command Window</u>	197
<u>Project Manager folders</u>	198

<u>Color Reference</u>	201
<u>Introduction</u>	201
22	
<u>Dialog box descriptions</u>	205
<u>Add file to Project Folder dialog box</u>	205
<u>Add New Property dialog box</u>	205
<u>Add to Project dialog box</u>	206
<u>Annotate dialog box</u>	207
<u>Packaging tab</u>	207
<u>Archive Project dialog box</u>	212
<u>Attach Implementation dialog box</u>	214
<u>Advanced Annotation</u>	216
<u>Bill of Materials dialog box</u>	221
<u>Browse File dialog box</u>	222
<u>Color dialog box</u>	223
<u>Create Directory dialog box</u>	223
<u>Create PSpice Project dialog box</u>	223
<u>Configure Properties dialog box</u>	224
<u>Delete Part Property dialog box</u>	227
<u>Design Properties dialog box</u>	227
<u>Design Template / Design Properties dialog box</u>	229
<u>Fonts tab</u>	231
<u>Title Block tab</u>	231
<u>Grid Reference tab</u>	231
<u>Page Size tab</u>	232
<u>Hierarchy tab</u>	233
<u>SDT Compatibility tab</u>	233
<u>Display Properties dialog box</u>	234
<u>Distributions dialog box</u>	236
<u>Design XML dialog box</u>	236
<u>Design Difference dialog box</u>	237
<u>Edit Bookmark dialog box</u>	239
<u>Edit Filled Graphic dialog box</u>	239
<u>Edit Graphic dialog box</u>	239

OrCAD Capture Reference Guide

<u>Edit Hierarchical Port dialog box</u>	240
<u>Edit Net Alias dialog box</u>	240
<u>Edit Off-Page Connector dialog box</u>	241
<u>Edit Part Properties dialog box</u>	241
<u>Edit Text dialog box</u>	243
<u>Edit Wire(s) dialog box</u>	244
<u>Export Properties dialog box</u>	244
<u>Export Selection dialog box</u>	245
<u>Extended Preferences Setup</u>	245
<u>Edit Comment Text dialog box</u>	252
<u>Font dialog box</u>	255
<u>Find dialog box</u>	255
<u>Generate Part dialog box</u>	257
<u>Go To dialog box</u>	261
<u>Location tab</u>	262
<u>Grid Reference tab</u>	262
<u>Bookmark tab</u>	262
<u>Goto Label State dialog box</u>	262
<u>Go To Line dialog box</u>	263
<u>Markers dialog box</u>	265
<u>Monte Carlo Worst-Case Output File Options dialog box</u>	266
<u>Multi-level Backup Settings dialog box</u>	267
<u>New Alias dialog box</u>	269
<u>New Page in Schematic dialog box</u>	269
<u>New Part Properties dialog box</u>	269
<u>New Project dialog box</u>	272
<u>New Property dialog box</u>	272
<u>New Schematic dialog box</u>	273
<u>New Simulation dialog box</u>	273
<u>New NetGroup / Modify NetGroup dialog box</u>	274
<u>Rename NetGroup Member dialog box</u>	274
<u>Open dialog box</u>	277
<u>Property Sheet Pane</u>	279
<u>Package Properties</u>	280
<u>Part Properties</u>	281
<u>Pin Properties</u>	282

OrCAD Capture Reference Guide

<u>Text Properties</u>	284
<u>Basic Attributes</u>	284
<u>Display Properties</u>	285
<u>Delete Current Section</u>	286
<u>Add Convert View</u>	286
<u>Delete Convert View</u>	286
<u>Edit Pins</u>	286
<u>Paste Options dialog box</u>	289
<u>Part Aliases dialog box</u>	290
<u>Part Search dialog box</u>	291
<u>Place Bookmark dialog box</u>	291
<u>Place Ground dialog box</u>	292
<u>Place NetGroup dialog box</u>	293
<u>Place Off-Page Connector dialog box</u>	294
<u>Place Hierarchical Block dialog box</u>	295
<u>Place Hierarchical Pin dialog box</u>	298
<u>Place Hierarchical Port dialog box</u>	299
<u>Place IEEE Symbol dialog box</u>	300
<u>Place Net Alias dialog box</u>	300
<u>Place Part dialog box</u>	302
<u>Place Part Pane</u>	302
<u>Search Part</u>	306
<u>Place Pin Array dialog box</u>	306
<u>Place Pin dialog box</u>	308
<u>Place Power dialog box</u>	309
<u>Place Text dialog box</u>	310
<u>Place Title Block dialog box</u>	311
<u>Preferences dialog box</u>	312
<u>Color/Print tab</u>	314
<u>Grid Display tab</u>	314
<u>Miscellaneous tab</u>	316
<u>Pan and Zoom tab</u>	320
<u>Select tab</u>	321
<u>Text Editor tab</u>	321
<u>Board Simulation tab</u>	322
<u>Print dialog box</u>	323

OrCAD Capture Reference Guide

<u>Print Preview and Print Setup dialog boxes</u>	328
<u>Print Setup dialog box</u>	331
<u>Print to File dialog box</u>	331
<u>Programmable Logic Project Wizard dialog box</u>	332
<u>Propagation Delay dialog box</u>	332
<u>Properties dialog box</u>	334
<u>General tab</u>	334
<u>Type tab</u>	335
<u>PSpice Part Search dialog box</u>	335
<u>Remove Occurrence Properties box</u>	337
<u>Rename Hierarchical Port dialog box</u>	337
<u>Rename Off-Page Connector dialog box</u>	337
<u>Rename Page dialog box</u>	338
<u>Rename Part dialog box</u>	338
<u>Rename Part Property dialog box</u>	338
<u>Rename Power Symbol dialog box</u>	339
<u>Rename Schematic dialog box</u>	339
<u>Rename Title Block dialog box</u>	340
<u>Replace Cache dialog box</u>	340
<u>Replace dialog box</u>	341
<u>Save Files in Project dialog box</u>	343
<u>Save Part As dialog box</u>	343
<u>Save Project As dialog box</u>	343
<u>Save Part Instance dialog box</u>	345
<u>Schematic Page Properties dialog box</u>	345
<u>Page Size tab</u>	346
<u>Grid Reference tab</u>	347
<u>Select Directory dialog box</u>	348
<u>Select File Type dialog box</u>	349
<u>Selection Filter dialog box</u>	350
<u>Select New Project Path dialog box</u>	350
<u>Select Occurrence dialog box</u>	351
<u>Set Label State dialog box</u>	351
<u>Setup dialog box</u>	351
<u>Simulation Settings dialog box</u>	353
<u>Analysis tab</u>	356

OrCAD Capture Reference Guide

<u>Data Collection Settings tab</u>	362
<u>General tab</u>	362
<u>Include Files tab</u>	363
<u>Libraries tab</u>	363
<u>Options tab</u>	364
<u>Probe Window tab</u>	365
<u>Stimulus tab</u>	365
<u>Specify Part Filter dialog box</u>	366
<u>Transient Output File Options dialog box</u>	367
<u>Update Properties dialog box</u>	369
<u>User Properties dialog box</u>	370
<u>Update Alias dialog box</u>	371
<u>Zoom Scale dialog box</u>	373

OrCAD Capture Reference Guide

About this Manual

Welcome

OrCAD® Capture (henceforth referred to as Capture) 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.

The OrCAD® 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 online Help.

Where to Find Information in this Manual

The OrCAD® Reference Guide contains information on both menu command 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 [Window descriptions](#) chapter provides a brief description of the different windows available in the Capture environment.

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.

Where to Find Additional Information

To access additional technical documentation from the OrCAD 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 Capture, contact the [Cadence Online Support](#)

OrCAD Capture Reference Guide

About this Manual

Project manager command reference

This chapter covers:

- [File menu](#) on page 23
- [Design menu](#) on page 30
- [Edit menu](#) on page 38
- [View](#) on page 52
- [Tools menu](#) on page 55
- [PSpice](#) on page 61
- [Options menu](#) on page 66
- [Window menu](#) on page 72
- [Help menu](#) on page 75

File menu

[New command](#) on page 24

[Open command](#) on page 24

[Close Project command](#) on page 25

[Save command](#) on page 26

[Save As command](#) on page 26

[Save Project As](#) on page 27

[Archive Project command](#) on page 27

[Print Preview command](#) on page 28

[Print command](#) on page 28

[Print Setup command](#) on page 29

[1,2,3,4 command](#) on page 29

[Change Product command](#) on page 29

[Exit command](#) on page 30

New command

Available from: File menu

Use this command to create a new [project](#), [design](#), [library](#). Choose a command from the menu that appears:

- Project
- Design
- Library

Function:

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: 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

OrCAD Capture Reference Guide

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
- Project
- VHDL File
- Verilog File

Function:

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 [New command](#) 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

Close Project 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

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

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 Save command on the File menu.

Function:

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.

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.

Archive Project command

Available from: File menu

OrCAD Capture Reference Guide

Project manager command reference

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.

Capture Toolbar: 

- Shortcuts:**
- CTRL+P
 - ALT, F, P

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 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.



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

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)

Change Product command

Available from: File menu

OrCAD Capture Reference Guide

Project manager command reference

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).

Keyboard:

Shortcuts:

- ALT+F4
- ALT, SPACEBAR, C
- ALT, F, X

Design menu

[New Schematic command](#) on page 31

[New Schematic Page command](#) on page 31

[New Part command](#) on page 32

[New Part command](#) on page 32

[New Part from Spreadsheet command](#) on page 32

[Rename command](#) on page 33

[Rename command](#) on page 33

[Remove Occurrence Properties command](#) on page 33

[Make Root command](#) on page 33

[Replace Cache command](#) on page 34

[Update Cache command](#) on page 36

[Cleanup Cache command](#) on page 37

[Set Password Command](#) on page 37

[Remove Password](#) on page 37

[Change Password](#) on page 38

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

OrCAD Capture Reference Guide

Project manager command reference

- Function:** Use this command to add a new schematic page to the selected schematic folder.
- 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 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.
-  *Tip*
- 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

Rename command

Available from: Design menu

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

Function: Use this command to remove all the unique annotations you have placed on parts

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

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:



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.



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.

OrCAD Capture Reference Guide

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 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.



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 .

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 .

Shortcuts: Keyboard: ALT, D, U

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

Function: Use this command to set a password to a Capture design. Once you click on the command, you will be asked to enter and confirm the password.

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

Function: Use this command to modify or update the already set password to a Capture design. Once you click the command, you will be asked to enter and confirm the password.

Shortcuts: -

Edit menu

[Cut command](#) on page 39

[Copy command](#) on page 39

[Paste command](#) on page 40

[Project command](#) on page 42

[Properties command](#) on page 42

[Object Properties command](#) on page 44

[Browse command](#) on page 45

[Power Pins command](#) on page 49

[Rename Part Property command](#) on page 50

[Go To command](#) on page 51

[Clear Session Log command](#) on page 52

[Delete Part Property command](#) on page 51

[Replace command](#) on page 51

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.

Note: The Cut and [Copy command](#) 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

Pop-up 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).

Capture Toolbar: 

Keyboard:

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

Pop-up menu: Copy

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.

Important

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

Pop-up menu: Paste

Lock command

Available from: *Edit* menu

Function: 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.

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

Pop-up menu: *Lock*

UnLock command

Available from: *Edit* menu

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

OrCAD Capture Reference Guide

Project manager command reference

Keyboard:

Shortcuts: ■ ALT, E, N

Pop-up menu: *UnLock*

Project command

Available from: Edit menu

Function: Use this command to add resources to your project. When you choose this command, the [Add file to Project Folder dialog box](#) 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

Properties command

Available from: Edit menu

Function: In the project manager, use this command to view properties about the selected document. Using the [Properties dialog box](#), 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:

Objects

Dialog Box

Arcs

Edit Graphic dialog box

Images (pictures)

Not applicable

OrCAD Capture Reference Guide

Project manager command reference

Objects	Dialog Box
Bookmarks	Edit Bookmark dialog box
Buses Property editor	Property editor
Bus entries	<u>User Properties dialog box</u>
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
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

OrCAD Capture Reference Guide

Project manager command reference

Objects

Dialog Box

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

Keyboard:

- CTRL+E
- ALT, E, I

Shortcuts:

Mouse: Double-click on a part

Pop-up menu: Edit Properties

Object Properties command

Available from: Edit menu

OrCAD Capture Reference Guide

Project manager command reference

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 pop-up menu.

Shortcuts:

Keyboard: ALT, E, O

Browse command

Available from: Edit menu

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

Parts 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.

Function: 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, E, B, P

Nets command

Available from: Edit menu, Browse command

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.

Shortcuts: Keyboard: ALT, E, B, N

Hierarchical Ports command

Available from: Edit menu, Browse command

OrCAD Capture Reference Guide

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

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.

Function: You can select single or multiple off-page connectors and edit them with the [Properties command](#). When you edit one off-page connector, the [Edit Off-Page Connector dialog box](#) 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

Available from: Edit menu, Browse command

Function: Use this command to list the title blocks of the selected schematic pages in the browse window. When you edit a title block, the [User Properties 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.

Function: You can select single or multiple bookmarks and edit them with the Properties command. When you edit one bookmark, the Edit Bookmark 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.

Shortcuts: Keyboard: ALT, E, B, B

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 View DRC Marker dialog box 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

OrCAD Capture Reference Guide

Project manager command reference

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

Find command

Available from: Edit menu

OrCAD Capture Reference Guide

Project manager command reference

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.

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.

Function:

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

Rename Part Property command

Available from: Edit menu

OrCAD Capture Reference Guide

Project manager command reference

- 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 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

Use this command to center the view on a specific location, grid references, or bookmark. By clicking the Go To command, the Go To Line dialog box 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 53

Capture Toolbar command on page 53

Draw Toolbar command on page 53

PSpice Toolbar command on page 54

on page 54

on page 54

Toolbar command

Available from: View menu

Function: 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

Function: 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

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.

PSpice Toolbar command

Available from: View - Toolbar menu

Function:

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.

Command Window command

Available from: View - Toolbar menu

Function:

Use this command to open the TCL Command window in the OrCAD Capture workspace.

For details see the [Command Window](#).

Tools menu

[Annotate command](#) on page 55

[Update Properties command](#) on page 55

[Bill of Materials command](#) on page 56

[Export Properties command](#) on page 57

[Import Properties command](#) on page 58

[Generate Part command](#) [on page 59](#)

[Split Part command](#) on page 61

[Customize command](#) on page 61

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: 

Shortcuts: Keyboard:

- ALT, T, O

Update Properties command

Available from: Tools menu

OrCAD Capture Reference Guide

Project manager command reference

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 ll 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.

Shortcuts: Keyboard: ALT, T, U

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.

Important

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

Function: 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.

For more information on property files see .

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

Import Properties command

Available from: Tools menu

Function: 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.

For more information on property files see .

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.

OrCAD Capture Reference Guide

Project manager command reference

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

Generate Part command

Available from: Tools menu

OrCAD Capture Reference Guide

Project manager command reference

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.

Shortcuts:

Keyboard: ALT, T, G

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

Customize command

Available from: Tools menu

Use this command to customize the OrCAD 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.

PSpice

[Bias Points command](#) on page 62

[New Simulation Profile command](#) on page 62

[Edit Simulation Profile command](#) on page 62

[Run command](#) on page 63

[View Simulation Results command](#) on page 63

[View Output File command](#) on page 64

[Make Active command](#) on page 64

[Simulate Selected Profile\(s\) command](#) on page 64

[Create Netlist command](#) on page 65

[View Netlist command](#) on page 66

[Marker List command](#) on page 66

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

Function: 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

Function: 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

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

OrCAD Capture Reference Guide

Project manager command reference

- Function:** 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.
- Note:** You can view the simulation results from the currently active profile by pressing the F12 function key.
- PSpice Toolbar:** 
- Shortcuts:** Keyboard:
- ALT, S, V

View Output File command

- Available from:** 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

OrCAD Capture Reference Guide

Project manager command reference

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.
- Opens PSpice using the netlist created from your design.

Function: 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.

See for procedures to simulate and view multiple profiles.

Shortcuts: Keyboard: ALT, P, A

Create Netlist command

Available from: PSpice menu

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.

Function: In OrCAD 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.
- Function:** Use this command to display or hide markers in the design. The Markers 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

Options menu

- Autobackup command on page 66
- Preferences command on page 67
- Design Template command on page 71
- Design Properties command on page 72

Autobackup command

- Available from:** Options menu

OrCAD Capture Reference Guide

Project manager 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

Preferences command

Available from: Options menu

Function: 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

OrCAD Capture Reference Guide

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

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

Function:

- **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 OrCAD Layout.

Note: By default, message-based communication is not enabled in OrCAD Capture. Enable this option if you want OrCAD Capture to communicate with the legacy tool as it impacts OrCAD 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: Unselecting the option improves user experience as Capture ignores the permissions check on the design and library files.

OrCAD Capture Reference Guide

Project manager command reference

Functions

- **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:
 - **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.

DRC

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

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)

Functions:

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.

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 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 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.

Note: For the TCL commands for these options, see

Design Template command

Available from: Options menu

OrCAD Capture Reference Guide

Project manager command reference

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 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

Design Properties command

Available from: Options menu

Function: 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 73

Cascade command on page 73

Tile Horizontally command on page 73

Tile Vertically command on page 73

Close All Tabs of Active Project command on page 74

Close All Tabs of Active Project Except Current command on page 74

Arrange Icons command on page 74

1,2.... command on page 74

Close All Windows on page 75

New Window command

- Available from:** Window menu
- Function:** Use this command to create a new window, which is a copy of the 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
- Function:** 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
- 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 from:** Window menu
- Function:** Use this command to arrange open Capture windows, one beside another, so that all are visible.
- Shortcuts:** Keyboard: ALT, W, V

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
- Function:** Use this command to close all tabs of the active project except the tab that is currently active. This option is not available for the Project Manager tab.
- Shortcuts:** Keyboard: ALT, W, C

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

- Function:** 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

[P Spice for TI Help command](#) on page 75

[Known Problems and Solutions command](#) on page 76

[What's New command](#) on page 76

[About P Spice for TI command](#) on page 76

[Documentation command](#) on page 76

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.

P Spice for TI Help command

- Available from:** Help menu
- Function:** Use this command to display the Capture help window.
- Shortcuts:** Keyboard: ALT, H, H or F1

Known Problems and Solutions command

- Available from:** Help menu
- Function:** Use this command to display a document listing the known problems in this release of OrCAD 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
- Function:** Use this command to display a document describing the new features and enhancements in this release.
- Shortcuts:** Keyboard: ALT, H, K

About PSpice for TI 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

Documentation command

- Available from:** In the project manager or schematic page editor, from the Help menu, choose Documentation.
- Function:** Use this command to launch the help landing 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

Pop-Up Menu

This section covers:

[Add File command](#) on page 78

[Annotate command](#) on page 78

[Change Project Type command](#) on page 78

[Cleanup Cache command](#) on page 78

[Copy command](#) on page 79

[Cut command](#) on page 79

[Delete command](#) on page 79

[Design Properties command](#) on page 79

[Edit Object Properties command](#) on page 79

[Edit Page command](#) on page 80

[Edit selected object properties command](#) on page 80

[Find command](#) on page 80

[Lock command](#) on page 80

[Make Root command](#) on page 80

[New Page command](#) on page 81

[New Schematic command](#) on page 81

[Open File Location command](#) on page 81

[Paste command](#) on page 81

[Paste command](#) on page 81

[Properties command](#) on page 81

[Rename command](#) on page 81

[Reports – Export Placement command](#) on page 81

[Reports – Export Properties command](#) on page 81

[Save command](#) on page 82

[Save As command](#) on page 82

[Save Project As command](#) on page 82

[Schematic Page Properties command](#) on page 82

[UnLock command](#) on page 82

Add File command

See [Project command](#) on page 42

Annotate command

See [Annotate command](#) on page 55

Change Project Type command

Available from: In the project manager, from the pop-up menu for Design Resources or the DSN file.

Use this command to change the type of a project to any of the following project types:

Function:

- Analog or Mixed A/D
- PC Board Wizard
- Programmable Logic Wizard
- Schematic.

Cleanup Cache command

See [Cleanup Cache command](#) on page 37

Copy command

See [Copy command](#) on page 39

Cut command

See [Cut command](#) on page 39

Delete command

Available from: In the project manager, from the pop-up menu

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

Function:



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.

Shortcuts

DELETE
BACKSPACE
DEL

Design Properties command

See [Design Properties command](#) on page 72

Edit Object Properties command

Available from: In the project manager, from the pop-up 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 pop-up 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 pop-up menu for schematic pages

Function: Use this command to property editor for selected components

Find command

See [Find command](#) on page 49

Lock command

See [Lock command](#) on page 41

Make Root command

See [Make Root command](#) on page 33

New Page command

See [New Schematic Page command](#) on page 31

New Schematic command

See [New Schematic command](#) on page 31

Open File Location command

Available from: In the project manager, from the pop-up menu for DSN folder

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

Paste command

See [Paste command](#) on page 40

Properties command

See [Properties command](#) on page 42

Rename command

See [Rename command](#) on page 33

Reports – Export Placement command

See [Import Properties command](#) on page 58

Reports – Export Properties command

See [Export Properties command](#) on page 57

Save command

See [Save command](#) on page 26

Save As command

See [Save As command](#) on page 26

Save Project As command

See [Save Project As](#) on page 27

Schematic Page Properties command

Available from: In the project manager, from the pop-up menu for schematic pages

Function: Use this command to open the [Schematic Page Properties dialog box](#)

UnLock command

See [UnLock command](#) on page 41

Schematic page editor and part editor command reference

This chapter covers:

- [File menu](#) on page 83
- [Edit menu](#) on page 92
- [View menu](#) on page 114
- [Place menu](#) on page 125
- [PSpice/Markers menu](#) on page 144
- [Options menu](#) on page 148
- [Window menu](#) on page 150
- [Help menu](#) on page 152
- [Pop-up menu](#) on page 154

File menu

This section covers:

[New command](#) on page 84

[Open command](#) on page 86

[Close command](#) on page 88

[Save command](#) on page 88

[Export Selection command](#) on page 89

[Import Selection command](#) on page 89

[Print Preview command](#) on page 90

[Print command](#) on page 90

[Print Setup command](#) on page 90

[Print Area command](#) on page 91

[Import Design command](#) on page 91

[Export Design command](#) on page 92

[1,2,3,4 command](#) on page 92

[Exit command](#) on page 92

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:

Function: Project
Design
Library

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.

Shortcuts: Keyboard:
ALT, F, N, P

New Design command

Available from: File menu, New command

Function: Use this command to create one schematic folder with one schematic page, which Capture opens in the schematic page editor.

Shortcuts: Keyboard:
ALT, F, N, D

New Library command

Available from: File menu, New command

Function: 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

Function: 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
- Function:** 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
- Use this command to open an existing project, design, library in a new window. Choose a command from the menu that appears:

- Function:**
- Design
 - Library
 - Project
 -

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).

- Shortcuts:** Capture Toolbar: 
Keyboard: ALT, F, O

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
- Function:** 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
- Function:** 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

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

Shortcuts: Keyboard: ALT, F, O, E

Open Text File command

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, all parts of this type in the design, discard your changes, or cancel the Close command.

Save command

Available from: File menu

Function: 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

Available from: File menu

Function: 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.

This is useful if you have portions of a schematic page that you want to use on different schematic pages.

Shortcuts: Keyboard: ALT, F, E

Import Selection command

Available from: File menu

Function: Use this command to import the contents of a file created with the [Export Selection command](#) on the File menu to the active schematic page.

Shortcuts: Keyboard: ALT, F, I

Export ISCF command

Available from: File menu, Export Command

Function: 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

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.

Capture Toolbar: 

Keyboard:

- Shortcuts:**
- CTRL+P
 - ALT, F, P

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 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.



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

Function: 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:

- Set
- Clear

Shortcuts: Keyboard: ALT, F, A

Clear command

Available from: File menu

Function: 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

Import Design command

Available from: File menu

Function: 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 Design command

Available from: File menu

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.

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)

Exit command

Available from: File menu

Function: Use this command to exit the software. If necessary, you are prompted to save your changes.

You can also exit the software by choosing the Close command on the session frame Control menu (ALT, SPACEBAR, C).

Keyboard:

- Shortcuts:**
- ALT+F4
 - ALT, SPACEBAR, C
 - ALT, F, X

Edit menu

This section covers:

[Undo command](#) on page 93

OrCAD Capture Reference Guide

Schematic page editor and part editor command reference

[Redo command](#) on page 94

[Repeat command](#) on page 95

[Label State commands](#) on page 96

[Cut command](#) on page 96

[Copy command](#) on page 97

[Paste command](#) on page 97

[Delete command](#) on page 98

[Label command](#) on page 99

[Select All command](#) on page 100

[Part command](#) on page 100

[Properties command](#) on page 101

[Properties command](#) on page 101

[Part command](#) on page 103

[Mirror command](#) on page 105

[Rotate command](#) on page 106

[Lock command](#) on page 107

[UnLock command](#) on page 107

[Find command](#) on page 107

[Global Replace command](#) on page 108

[Align Command](#) on page 109

Undo command

Available from: Edit menu

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

Function: You can perform the undo and redo operation multiple times to return your design to any particular state, as described in .

The Undo command applies to the following actions:

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

Capture Toolbar: 

Keyboard:

Shortcuts:

- CTRL+Z
- ALT, E, U

Redo command

Available from: Edit menu

Function: 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 .

The Redo command applies to the following actions:

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

- Shortcuts:**
- Capture Toolbar: 
- Keyboard:
- CTRL+Y
 - ALT, E, E

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

- Shortcuts:**
- Keyboard:
- F4
 - ALT, E, R

Label State commands

Set

Available from: Edit menu

Function: Use this command to specify a label for the current state of the active schematic page.

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

Pop-up 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.

Capture Toolbar: 

Keyboard:

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

Pop-up menu: Copy

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.

 *Important*

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

Pop-up menu: Paste

Delete command

Available from: Edit menu

Function: 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.

- Shortcuts:**
- Keyboard:
- DEL
 - DELETE
 - BACKSPACE
 - ALT, E, D
- Pop-up menu: Delete

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.

- Shortcuts:**
- Keyboard:
- Set: Ctrl + Shift + F2
 - 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.
- Keyboard:
- Shortcuts:**
- CTRL+A
 - ALT, E, L

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.
- Function:** 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

Properties command

Available from: Edit menu

In the project manager, use this command to view properties about the selected document. Using the [Properties dialog box](#), 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

- Arcs
- Images (pictures)
- Bookmarks
- Buses Property editor
- Bus entries
- DRC markers
- Ellipses
- Hierarchical blocks
- Hierarchical pins
- Hierarchical ports

Dialog box

- Edit Graphic dialog box
- Not applicable
- Edit Bookmark dialog box
- Property editor
- [User Properties dialog box](#)
- View DRC Marker dialog box
- Edit Filled Graphic dialog box

OrCAD Capture Reference Guide

Schematic page editor and part editor command reference

- IEEE symbols
- Junctions
- Lines
- Multiple objects
- Nets (wires and buses)
- Net aliases
- No connects
- Off-page connectors
- Parts
- Pictures (images)
- Part body borders
- Pins (part editor)
- Pins (schematic page editor)
- Polygons
- Polylines
- Power, ground
- Rectangles
- Text
- Title blocks
- Wires
- Place IEEE Symbol dialog box
- Not applicable
- Edit Graphic dialog box
- Property editor or Browse spreadsheet editor
- Property editor
- Property editor
- Not applicable
- Edit Off-Page Connector dialog box
- Property editor
- Not applicable
- 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:

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

Mouse: Double-click on a part

Pop-up menu: Edit Properties

Part command

Available from: Edit menu

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

Function: 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.

Shortcuts: Keyboard: ALT, E, A

Pop-up menu: Edit Part

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 Pop-up 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 Pop-up 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.

Shortcut Pop-up menu: Reset Location

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:

- Shortcuts:**
- H
 - ALT, E, M, H

Pop-up menu: Mirror Horizontally

Vertically command

Available from: Edit menu, Mirror command

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

Pop-up menu: Mirror Vertically

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

Function: Use this command to rotate selected objects counterclockwise in 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:

Shortcuts:

- R
- ALT, E, O

Pop-up menu: Rotate

Lock command

Available from:

Schematic page editor shortcut menu, when you select the schematic part.

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

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:

Pop-up menu: Lock

UnLock command

Available from:

Schematic page editor shortcut menu, when you select the schematic part.

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.

Shortcuts:

Keyboard: ALT, E, N

Pop-up 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

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:
ALT+E+B

Align Command



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

- If the text is justified, align command has no impact on the justified text.
- Align command is not supported for the following objects:
 - PSpice Markers
 - DRC and ERC Markers
 - Bookmarks
 - Hierarchical Pins
 - Part Pins
 - Biaspoints
 - Bus
 - Bus Entry
 - Wire
 - NetGroup
 - 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:
 - Global
 - OPC
 - Ports

- 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 *Distribute in a fixed area* check box in the Schematic Group section of the Extended Preferences Setup window is unchecked. To open this 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 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.

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.

Align Top

Available from: Edit menu

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

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

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.
3. 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 be 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 115

[Convert command](#) on page 116

[Descend Hierarchy command](#) on page 116

[Go To command](#) on page 117

[Synchronize Up command](#) on page 117

[Synchronize Down command](#) on page 118

[Synchronize Across command](#) on page 118

[Previous Part command](#) on page 118

[Next Part command](#) on page 118

[Package command](#) on page 119

[Package command](#) on page 119

[Zoom command](#) on page 120

[Tool Palette command](#) on page 122

[Tool Palette command](#) on page 122

[Toolbar command](#) on page 123

[Status Bar command](#) on page 123

[Command Window command](#) on page 123

[Grid command](#) on page 124

[Grid References command](#) on page 124

[Selection Filter command](#) on page 124

[Invoke UI command](#) on page 125

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.
- Function:** You can view and traverse the hierarchy in the project manager.
- Keyboard:
- Shortcuts:**
- SHIFT+A
 - ALT, V, A
- Pop-up: 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
- Function:** 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 active. Otherwise, it opens in a new schematic page editor window.
- You can view and traverse the hierarchy in the project manager.



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.

Shortcuts:

Keyboard:

- SHIFT+D
- ALT, V, D

Pop-up menu: Descend Hierarchy

Go To command

Available from: View menu

Function: Use this command to center the view on a specific location, grid reference, or bookmark.



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.

Shortcuts:

Keyboard:

- CTRL+G
- ALT, V, G

Pop-up menu: Go To

Synchronize Up command

Available from: View menu / Shortcut menu

Function: Updates the hierarchical blocks with all the changes made to the hierarchical ports of the underlying schematic of the selected hierarchical block.

Shortcuts:

Keyboard: Shift + U

Pop-up menu: Synchronize Up

Synchronize Down command

- Available from:** View menu / Shortcut menu
- Function:** Updates the underlying schematic with all the changes made to the hierarchical pins of the selected hierarchical block.
- Shortcuts:** Keyboard: Shift + O
Pop-up menu: Synchronize Down

Synchronize Across command

- Available from:** View menu / Shortcut menu
- Function:** 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.
- Shortcuts:** Keyboard: Shift + C
Pop-up menu: Synchronize Across

Previous Part command

- Available from:** View menu
- Function:** 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 editor. In Package view, this command selects the previous part in the package.
- Shortcuts:** Keyboard:
 - ALT, V, V
 - CTRL+B
 - SHIFT+TAB (PACKAGE VIEW ONLY)

Next Part command

- Available from:** View menu

Function: 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:

- Shortcuts:**
- ALT, V, X
 - CTRL+N
 - TAB (PACKAGE VIEW ONLY)

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

Function: Use this command to open the next page in the current schematic folder. The option is disabled for the last page in the folder.

Shortcuts: Keyboard: F10

Package command

Available from: View menu

Function: Use this command in the part editor to view all the parts in a package in a new tab. Parts cannot be edited in package view.

You can zoom in and zoom out the parts using the shortcuts \mathbb{I} and \mathbb{O} .

Shortcuts: Keyboard: ALT, V, K

Zoom command

Available from: View menu

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

- Function:**
- In
 - Out
 - Scale
 - Area
 - All
 - 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:**
- I
 - ALT, V, Z, I
- Pop-up menu: Zoom In

Out command

Available from: View menu, Zoom command

Function: Use this command to zoom out from the schematic page or part. The zoom scale is divided by the current zoom factor.

Capture Toolbar: 

Keyboard:

- Shortcuts:**
- O
 - ALT, V, Z, O
- Pop-up menu: Zoom Out

Scale command

Available from: View menu, Zoom command

Function: In the schematic page editor, use this command to zoom to a preset or user-defined scale. The new view centers on the selected objects, the pointer location, or the center of the previous view.

Shortcuts: Keyboard: ALT, V, Z, S

Area command

Available from: View menu, Zoom command

Function: 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 it.

Shortcuts: Capture Toolbar: 
Keyboard: ALT, V, Z, A

All command

Available from: View menu, Zoom command

Function: 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 objects.

Shortcuts: Capture Toolbar: 
Keyboard: ALT, V, Z, L

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

Tool Palette command

Available from: View menu

- Function:** Use this command to show or hide the tool palette. This setting is stored in your CAPTURE.INI file and thus affects the visibility of the palette in subsequent sessions.
- Shortcuts:** Keyboard: ALT, V, P

Toolbar command

- Available from:** View menu
- Function:** 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
- Function:** Use this command to show or hide the status bar. This setting is stored 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

Function: Use this command to open the TCL Command window in the OrCAD Capture workspace.
For details see the [Command Window](#).

Grid command

Available from: View menu

Function: 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

Grid References command

Available from: View menu

Function: Use this command to show or hide the grid references.
You can also set the grid references to show or hide in the Design Template dialog box and the Schematic Page Properties dialog box.

Shortcuts: Keyboard: ALT, V, R

Selection Filter command

Available from: View menu

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:

- Shortcuts:**
- CTRL+I
 - ALT, V, F

This command is also available from the schematic page editor pop-up 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.

Function: 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.

Keyboard:

- Shortcuts:**
- CTRL+U
 - ALT, E, V

Place menu

This section covers:

[Part command](#) on page 127

[PSpice Component command](#) on page 127

[Parameterized Part command](#) on page 129

[NetGroup command](#) on page 129

[Wire command](#) on page 129

OrCAD Capture Reference Guide

Schematic page editor and part editor command reference

[Bus command](#) on page 131

[Junction command](#) on page 132

[Bus Entry command](#) on page 132

[Net Alias command](#) on page 133

[Power command](#) on page 133

[Ground command](#) on page 133

[Off-Page Connector command](#) on page 134

[Hierarchical Block command](#) on page 134

[Hierarchical Port command](#) on page 135

[Hierarchical Pin command](#) on page 135

[No Connect command](#) on page 136

[Title Block command](#) on page 137

[Bookmark command](#) on page 140

[Text command](#) on page 140

[Line command](#) on page 141

[Rectangle command](#) on page 141

[Ellipse command](#) on page 141

[Arc command](#) on page 142

[Elliptical Arc command](#) on page 142

[Bezier curve command](#) on page 142

[Polyline command](#) on page 143

[Picture command](#) on page 143

[OLEObject command](#) on page 144

Part 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: 

Keyboard:

Shortcuts:

- P
- ALT, P, P

PSpice Component command

Available from: Place menu

OrCAD Capture Reference Guide

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:
 - 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:

- Shortcuts:**
- Sources: Independent Sources, PWL Sources
 - System Modules: Switch, Transformer, VCO
- Keyboard:
- ALT, P, S

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.

Shortcuts:

Draw Toolbar: 

Keyboard:

- W
- ALT, P, W

Auto Wire Two Points command

Available from: Place menu

Function: Use this command to auto-wire two points (pins or wires) on a schematic page.

Shortcuts: Draw Toolbar: 

Auto Wire Multiple Points command

Available from: Place menu

Function: Use this command to auto-wire multiple points (pins or wires) on a schematic page.

Shortcuts: Draw Toolbar: 

Auto Wire Connect to Bus

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: 

Bus command

Available from:

Place menu

Function:

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.

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.

 *Important*

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.

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

Junction command

Available from: Place menu

Function: 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:

- J
- ALT, P, J

Bus Entry command

Available from: Place menu

Function: Use this command to place a bus entry.

- Shortcuts:**
- Draw Toolbar: 
- Keyboard:
- E
 - ALT, P, E

Net Alias command

- Available from:** Place menu
- Function:** Use this command to place a net alias on the selected object. To quit placing net aliases, press ESC or choose the selection tool.
- Draw Toolbar: 
- Keyboard:
- Shortcuts:**
- N
 - ALT, P, N

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.

- Shortcuts:**
- Draw Toolbar: 
 - Keyboard:
 - G
 - ALT, P, G

Off-Page Connector command

Available from: Place menu

Function: 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.

- Shortcuts:**
- Draw Toolbar: 
 - Keyboard: ALT, P, F

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.

- Shortcuts:**
- Draw Toolbar: 
 - 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.

Shortcuts: Draw Toolbar: 
Keyboard: ALT, P, I

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.

Shortcuts: Draw Toolbar: 
Keyboard: ALT, P, R

No Connect command

Available from: Place menu

Function: 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.

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

Pin command

Available from: Place menu

Function: 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 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.

Function: You can set title block visibility in the Design Template and Schematic 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:

OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

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

Bookmark command

- Available from:** Place menu
- Function:** 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 Place IEEE Symbol dialog box appears.
- Function:** 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 pop-up menu, to change the IEEE symbol without having to quit the IEEE symbol tool.
- Draw Toolbar:** 
- Keyboard:**
- Shortcuts:**
- ALT, P, E
 - SHIFT+X

Line command

Available from: Place menu

Function: 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, 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: 

Shortcuts: Keyboard:

- ALT, P, L
- SHIFT+L

Rectangle command

Available from: Place menu

Function: Use this command to draw a rectangle. Press the left mouse button and drag the pointer to define the rectangle. To quit drawing rectangles, press ESC, or click on the selection tool.

To draw a square, hold down the SHIFT key while drawing.

Draw Toolbar: 

Shortcuts: Keyboard:

- ALT, P, R
- SHIFT+R

Ellipse command

Available from: Place menu

Function: 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 ESC, or click on the selection tool.

To draw a circle, hold down the SHIFT key while drawing.

- Shortcuts:**
- Draw Toolbar: 
 - Keyboard:
 - ALT, P, S, S
 - SHIFT+F

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.
- Shortcuts:**
- Draw Toolbar: 
 - Keyboard: SHIFT+T

Elliptical Arc command

- Available from:** Place menu
- Function:** Use this command to draw an elliptical arc. To quit drawing arcs, press ESC, or click on the selection tool.
- Shortcuts:**
- Draw Toolbar: 
 - 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.
- Function:** 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 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.

- Shortcuts:**
- Draw Toolbar: 
- Keyboard:
- ALT, P, Z
 - SHIFT+Q

Polyline command

Available from: Place menu

Function: 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 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.

- Shortcuts:**
- Draw Toolbar: 
- Keyboard:
- 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

Function: 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 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.

PSpice/Markers menu

This section covers:

[Markers command](#) on page 144

[Marker List command](#) on page 145

[Voltage Level command](#) on page 145

[Voltage Differential command](#) on page 146

[Current Into Pin command](#) on page 146

[Power Dissipation command](#) on page 146

[Advanced command](#) on page 147

[Plot Window Templates command](#) on page 147

[Show All command](#) on page 147

[Hide All command](#) on page 147

[Delete All command](#) on page 148

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 Markers 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

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.

Shortcuts: Keyboard: ALT, S, M, P

Advanced command

Available from: PSpice menu, Markers submenu

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

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

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

Function: From a schematic page, point to Markers on the PSpice menu.
Use this command to hide all the markers on the schematic

Shortcuts: Keyboard: ALT, S, M, H

Delete All command

Available from: PSpice menu, Markers submenu

Function: From a schematic page, point to Markers on the PSpice menu.
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 148

Accessories command

Available from: Accessories menu

Function: 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:

[Autobackup command](#) on page 149

[Preferences command](#) on page 149

[Design Template command](#) on page 150

[Schematic Page Properties command](#) on page 150

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

Function: 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

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 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

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 151

Cascade command on page 151

Tile Horizontally command on page 151

Tile Vertically command on page 151

Arrange Icons command on page 152

1,2.... command on page 152

New Window command

- Available from:** Window menu
- Function:** Use this command to create a new window, which is a copy of the 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
- Function:** 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
- 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 from:** Window menu
- Function:** 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 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
- Function:** 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, . . .)

Help menu

This section covers:

[PSpice for TI Help command](#) on page 153

[Known Problems and Solutions command](#) on page 153

[What's New command](#) on page 153

[About PSpice for TI command](#) on page 153

[Documentation command](#) on page 154

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.

PSpice for TI Help command

- Available from:** Help menu
- Function:** Use this command to display the Help window.
- Keyboard:
- Shortcuts:**
- F1
 - ALT, H, H

Known Problems and Solutions command

- Available from:** Help menu
- Function:** Use this command to display a document listing the known problems in this release of OrCAD 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
- Function:** Use this command to display a document describing the new features and enhancements in this release.
- Shortcuts:** Keyboard: ALT, H, K

About PSpice for TI 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

Documentation command

- Available from:** In the project manager or schematic page editor, from the Help menu, choose Documentation.
- Function:** Use this command to launch the help 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

Pop-up menu

This section covers:

[Mirror Horizontally command](#) on page 155

[Mirror Vertically command](#) on page 155

[Rotate command](#) on page 155

[Edit Properties command](#) on page 155

[Edit Part command](#) on page 155

[Select Entire Net command](#) on page 156

[Descend Hierarchy command](#) on page 156

[Synchronize Up command](#) on page 156

[Synchronize Down command](#) on page 157

[Synchronize Across command](#) on page 157

[Connect to Bus command](#) on page 157

[User Assigned Reference – Set command](#) on page 157

[Update Design Hierarchy command](#) on page 157

[Selection Filter](#) on page 157

[Zoom In command](#) on page 158

[Zoom Out command](#) on page 158

[Go To command](#) on page 158

[Delete command](#) on page 158

[Tooltip command](#) on page 158

[Lock command](#) on page 158

[UnLock command](#) on page 158

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 [Horizontally command](#) on page 105

Mirror Vertically command

See [Vertically command](#) on page 106

Rotate command

See [Rotate command](#) on page 106

Edit Properties command

See [Properties command](#) on page 101

Edit Part command

See [Part command](#) on page 103

Select Entire Net command

Available from: Popup menu

Function: 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.

Shortcuts: Keyboard: CTRL+SHIFT+W
Pop-up menu: Select Entire Net

Edit Wire Properties command

Available from: Popup menu

Function: 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 [Descend Hierarchy command](#) on page 116

Synchronize Up command

See [Synchronize Up command](#) on page 117

Synchronize Down command

See [Synchronize Down command](#) on page 118

Synchronize Across command

See [Synchronize Across command](#) on page 118

Connect to Bus command

See [Auto Wire Connect to Bus](#) on page 130

User Assigned Reference – Set command

Available from: Pop-up with part selected in schematic page or Reference property in the Property Editor, User Assigned Reference – Set

Use this command to mark references as user assigned

OrCAD Capture marks a reference as user assigned if:

- Function:**
- Reference is changed in the Property Editor
 - Reference is changed in the Schematic canvas
 - Reference is changed in board (Backannotation)

Shortcuts: Pop-up menu: User Assigned Reference – Set

Update Design Hierarchy command

Available from: Pop-up menu from the Hierarchy tab in the Project manager

Function: Use this command to update the design hierarchy for the current project.

Shortcuts: Pop-up menu: Update Design Hierarchy

Selection Filter

[Selection Filter command](#) on page 124

Zoom In command

See [In command](#) on page 120

Zoom Out command

See [Out command](#) on page 121

Go To command

See [Go To command](#) on page 117

Delete command

See [Delete command](#) on page 98

Tooltip command

Schematic page editor shortcut menu, when you select the schematic part.

Available from:

Note: The Tooltip menu item will not appear if you select a component on the schematic page and then invoke the pop-up menu.

Function:

Use this command to toggle the display on tooltips for pins, parts and nets on the schematic page.

Shortcuts:

Pop-up menu: ToolTip

Lock command

See [Lock command](#) on page 107

UnLock command

See [UnLock command](#) on page 107

Add or Remove Pins on NetGroup Block command

Available from: Schematic page when you select a NetGroup Block.

Function: Use this command to open the Add/Remove the Pins on NetGroup Block dialog box, where you can add or remove the pins on a NetGroup Block.

OrCAD Capture Reference Guide
Schematic page editor and part editor command reference

Session log command reference

This chapter covers:

- [File menu](#) on page 161
- [View menu](#) on page 166
- [Edit menu](#) on page 168
- [Options menu](#) on page 170
- [Window menu](#) on page 171
- [Help menu](#) on page 173

File menu

[New command](#) on page 162

[Open command](#) on page 162

[Save command](#) on page 163

[Save As command](#) on page 163

[Print Preview command](#) on page 164

[Print command](#) on page 164

[Print Setup command](#) on page 165

[Import Design command](#) on page 165

[Exit command](#) on page 166

[1,2,3,4 command](#) on page 166

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

Function: ■ Library

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, in a new window. Choose a command from the menu that appears:

Design

Function: Library

Project

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).

Shortcuts: Toolbar: 
Keyboard:
■ ALT, F, O

Save command

Available from: File menu

Function: 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.

Shortcuts: Toolbar: 
Keyboard:
■ CTRL+S
■ ALT, F, S

Save As command

Available from: File menu

Function: 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.

OrCAD Capture Reference Guide

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 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.

Toolbar: 

Keyboard:

Shortcuts:

- CTRL+P
- ALT, F, P

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 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.



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: File menu

Function: 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 from:** File menu
- Function:** Use this command to exit the software. If necessary, you are prompted to save your changes.
You can also exit the software by choosing the Close command on the session frame Control menu (ALT, SPACEBAR, C).
- Shortcuts:** Keyboard:
 - ALT+F4
 - 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 167

[Status Bar command](#) on page 167

[Command Window command](#) on page 167

Toolbar command

- Available from:** View menu
- Function:** 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 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
- Function:** Use this command to show or hide the status bar. This setting is stored 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
- Function:** Use this command to open the TCL Command window in the OrCAD Capture workspace.
- For details see the [Command Window](#).

Edit menu

This section covers:

[Copy command](#) on page 168

[Select All command](#) on page 168

[Find command](#) on page 169

[Clear Session Log command](#) on page 169

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).

Toolbar: 

Shortcuts:

- CTRL+C
- ALT, E, C

Select All command

Available from: Edit menu

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

Shortcuts: Keyboard: Ctrl+A

Find command

Available from: Edit menu

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

Function: 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.

Keyboard:

Shortcuts:

- CTRL+F
- ALT, E, F

Clear Session Log command

Available from: Edit menu

Function: Use this command to clear the session log.

Shortcuts: Keyboard: ALT, E, S or CTRL+DEL

Options menu

[Preferences command](#) on page 170

[Design Template command](#) on page 170

[Autobackup command](#) on page 170

Preferences command

Available from: Options menu

Function: 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

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 [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

OrCAD Capture Reference Guide

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 171

Tile Horizontally command on page 172

Tile Vertically command on page 172

Arrange Icons command on page 172

1,2.... command on page 173

Close All Windows on page 173

Cascade command

Available from:

Window menu

OrCAD Capture Reference Guide

Session log command reference

Function: 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

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 from: Window menu

Function: 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 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
- Function:** 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

[PSpice for TI Help command](#) on page 174

[Known Problems and Solutions command](#) on page 174

[What's New command](#) on page 174

[About PSpice for TI command](#) on page 174

[Documentation command](#) on page 175

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.

PSpice for TI Help command

- Available from:** Help menu
- Function:** Use this command to display the Capture Help window.
- Shortcuts:**
- Toolbar:
- Keyboard:
- F1
 - ALT, H, H

Known Problems and Solutions command

- Available from:** Help menu
- Function:** Use this command to display a document listing the known problems in this release of OrCAD 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
- Function:** Use this command to display a document describing the new features and enhancements in this release.
- Shortcuts:** Keyboard: ALT, H, K

About PSpice for TI command

- Available from:** Help menu

OrCAD Capture Reference Guide

Session log command reference

Function: Use this command to get the software version number, copyright information, registration number, and license information.

Shortcuts: Keyboard: ALT, H, A

Documentation command

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

Function: Use this command to launch the help 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

OrCAD Capture Reference Guide

Session log command reference

Command Window command reference

This chapter covers:

- [Command Window pop-up menu](#) on page 177

Command Window pop-up menu

[Font command](#) on page 177

[Background Color command](#) on page 177

[Text Color command](#) on page 178

[Save command](#) on page 178

[Clear All command](#) on page 178

Font command

Available from: Pop-up menu

Function: Use this command to change the font of the Command window

Shortcuts: None

Background Color command

Available from: Pop-up menu

Function: Use this command to change the background color of the Command window

Shortcuts: None

Text Color command

Available from: Pop-up menu

Function: Use this command to change the text color of the Command window

Shortcuts: None

Save command

Available from: Pop-up menu

Function: Use this command to save the commands in the Command window to a TCL file.

Shortcuts: None

Clear All command

Available from: Pop-up menu

Function: Use this command to clear all the commands from the Command window

Shortcuts: None

OrCAD Capture Reference Guide
Command Window command reference

OrCAD Capture Reference Guide
Command Window command reference

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.

OrCAD Capture Reference Guide

Window descriptions

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.
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.

OrCAD Capture Reference Guide

Window descriptions

Schematic Order by the schematic folder the off-page connector is in.

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 session frame 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.

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.

Note: The following Capture utilities are found on the Tools menu:

- 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)

OrCAD Capture Reference Guide

Window descriptions

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.

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 pop-up 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.

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.

OrCAD Capture Reference Guide

Window descriptions

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-Allegro 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.

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.

OrCAD Capture Reference Guide

Window descriptions

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.

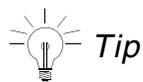
Each column is a placeholder that you can use to add properties. The cells in the property editor show the property values for each instance or occurrence. A cell with hash marks indicates that the property does not exist on the object that the cell represents. You can add a value by clicking inside 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.

OrCAD Capture Reference Guide

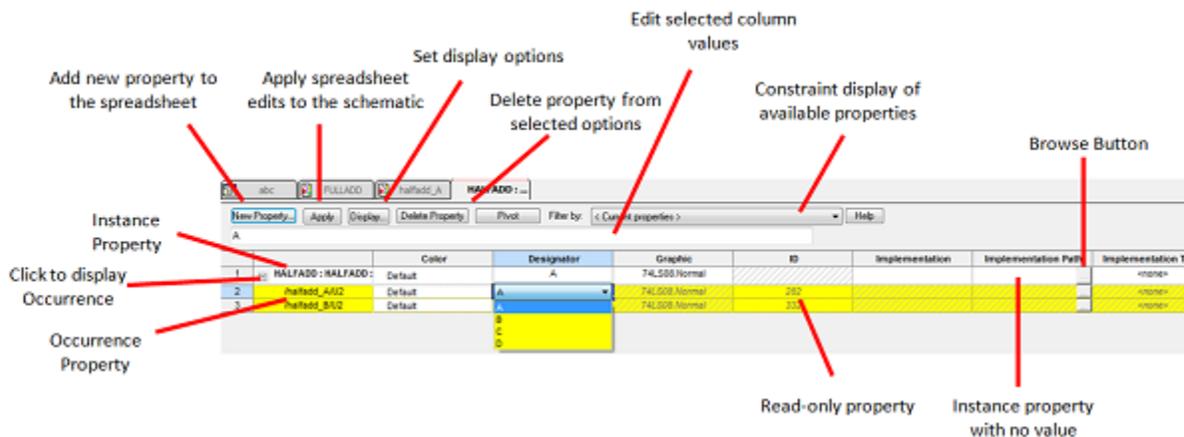
Window descriptions



Tip

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.



Short-cut keys

The following short-cut keys apply to the Property Editor:

Operation/command	Short-cut key
Undo	CTRL+Z
Copy	CTRL+C
Paste	CTRL+V
Cut	CTRL+X
Find	CTRL+F
Move to first cell in column	PageUp/CTRL+<Up-Arrow>
Move to last cell in column	PageDown/CTRL+<Down-Arrow>
Move to first cell in row	CTRL+<Left-Arrow>
Move to last cell in row	CTRL+<Right-Arrow>

OrCAD Capture Reference Guide

Window descriptions

Operation/command	Short-cut key
Undo Edit (within a cell)	Esc
Select	SHIFT+<Arrow key>
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 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

OrCAD Capture Reference Guide

Window descriptions

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.

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.



Tip

Roll the mouse wheel up and down to scroll through vertically.



Tip

Hold down the CTRL key and roll the mouse wheel to zoom in and zoom out.



Tip

Hold down the SHIFT key and roll the mouse wheel up and down to scroll through horizontally.



Tip

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 Text Editor tab 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 pop-up 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:

OrCAD Capture Reference Guide

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
Copy	CTRL+C
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

OrCAD Capture Reference Guide

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

OrCAD Capture Reference Guide

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

Pop-up menu

A pop-up menu is available in the text editor window. Click the right mouse button to bring up the pop-up menu. The following commands are available in this menu:

- Cut command
- Copy command
- Paste command
- Delete command
- Select All command
- Undo command
- Find command

Browse Spreadsheet editor window

You 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 display the Browse spreadsheet editor from the project manager, schematic page editor, or the part editor.

From the project manager - Select the schematic design and select the Browse command from the Edit menu. You can select a component from the resulting popup menu. To display the Browse spreadsheet editor, 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

From schematic page editor - Select the schematic page in the project manager and select the Browse command from the Edit menu. You can select a component from the resulting popup menu. To display the Browse spreadsheet editor, 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)

OrCAD Capture Reference Guide

Window descriptions

- Nets (including constituent nets within a bus) occurrences
- Hierarchical ports
- Off-page connectors
- Title blocks
- Bookmarks
- DRC markers

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.

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 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

OrCAD Capture Reference Guide

Window descriptions

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 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.
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.

OrCAD Capture Reference Guide

Window descriptions

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.

OrCAD Capture Reference Guide

Window descriptions

Color Reference

Introduction

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	

OrCAD Capture Reference Guide

Color Reference

Capture Color ID	RGB Value	Color Sample
10	128,255,0	
11	0, 255, 64	
12	0, 255, 255	
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	

OrCAD Capture Reference Guide

Color Reference

Capture Color ID	RGB Value	Color Sample
28	0, 0, 255	
29	0, 0, 160	
30	128, 0, 128	
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	

OrCAD Capture Reference Guide

Color Reference

Capture Color ID	RGB Value	Color Sample
46	64, 0, 64	
47	255, 255, 255	
48	Windows Default Color	

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...

Look in

File name

Files of type

To do this...

Browse the hierarchical drive and directory structure for your system.

Select or type the name of the project or file(s) that you want to add to the Project Folder.

Filter files by extension.

Add New Property dialog box

Use this dialog to add a new property to the selected object (or objects).

OrCAD Capture Reference Guide

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 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.

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...

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.

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.

Packaging tab

Use this control...

To do this...

OrCAD Capture Reference Guide

Dialog box descriptions

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.



This functionality works independently from the existing annotation behavior of Capture.



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 [Customizing part references in a design](#).

Scope

- Update entire design
- Update selection
- Schematic Pages
- Hierarchical Blocks

Specify whether to list 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)

Note: This grid appears only when you select the Refdes control required check box.

- Pages or H-Blocks column

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.

OrCAD Capture Reference Guide

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.



Tip

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. For more information, see [Working with Parts](#).

OrCAD Capture Reference Guide

Dialog box descriptions

Annotation Type	<p>Specifies the sequence in which the components on the design are annotated.</p> <p>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.</p>
Mode	<p>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.</p>
Physical Packaging	<p>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.</p> <p>The value in Additionally From INI is updated using the value specified by you in the Capture.ini file <code>AnnotateAdditionalPropertyString</code> property. For example, to specify <code>PartGroup</code> as the value, set it in Capture.ini as</p> <pre>AnnotateAdditionalPropertyString={PartGroup}</pre> <p>You can also specify more than one value such as <code>{Value}{Source Package}</code>. If you change this value in the field, Capture.ini will be updated accordingly.</p> <p>Note: Do not use <code>{GROUP}</code> as a property string in combined property strings text box. This may cause problems while annotating your design for a PCB Editor tool, like Allegro PCB Editor. The <code>GROUP</code> property is used in PCB Editor for a specific purpose.</p>

OrCAD Capture Reference Guide

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=<property 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.

OrCAD Capture Reference Guide

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 pop-up menu for the Reference property in the Property Editor or from the pop-up 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.

Archive Project dialog box

To open this dialog

In the Project manager, choose Archive Project (see [Archive Project command](#)) from the File menu.

Use this control...

To do this...

OrCAD Capture Reference Guide

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>\<design_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\bipolar.lib`.
- ❑ In case of global-level model libraries:
 - a copy of model library is created under the existing `<design_name>.lib`, if it exists
 - 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

Include any testbench which is part of the current project.

Output files

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.

OrCAD Capture Reference Guide

Dialog box descriptions

Archive directory	<p>Specify the drive and directory for the project to be archived in.</p> <p>Use the ... button to display the <u>Select Directory dialog box</u> where you can locate and select a new drive, directory, or both.</p>
Create single archive file	<p>Activate the File name text box for specifying the name of the compressed archive file.</p>
File name	<p>Specify a name for the compressed archive file. The default name is <projectname-current date>.</p>
Add more files	<p>Add more files and folders to be archived.</p>
Browse for	<p>Specify whether you want to add more files or directories to your archive.</p> <p>Select the Directories option to add a directory or the Files option to add more files to your archive.</p>
Additional Files/Directories	<p>Specify files and directories you want to be archived.</p> <p>Use the ... button to select the files and directories you want to archive.</p>

Attach Implementation dialog box

To open this dialog

Choose Attach Implementation

- From the New Part Properties dialog box.

OR

- From the Edit Part Properties dialog box

Use this control...

To do this...

OrCAD Capture Reference Guide

Dialog box descriptions

Implementation Type	<p>Specify the type of implementation from one of the following:</p> <p>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.</p> <p>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.</p> <p>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.</p> <p>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.</p> <p>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.</p> <p>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>.</p>
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.

Advanced Annotation

To open this dialog

Select the Capture design file in the Project Manager, choose *Tools* – *Annotate*, and click *Advanced Annotation*.

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-down 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

OrCAD Capture Reference Guide

Dialog box descriptions

Inherited Ranges

Specifies the inherited ranges from the top-level, such as Capture design

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?"

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

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.

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.

OrCAD Capture Reference Guide

Dialog box descriptions

Combined Property String	<p>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.</p> <p>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 tool, like Allegro PCB Editor. The GROUP property is used in PCB Editor for a specific purpose.</p> <p>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.</p> <p>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.</p>
Additional from INI	
Include Non-Primitive Parts	<p>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.</p>
Preserve Designator	<p>Specifies that the designator information of a homogeneous part during unconditional or reset annotation is to be preserved.</p>

OrCAD Capture Reference Guide

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 pop-up menu for the Reference property in the Property Editor or from the pop-up 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.

OrCAD Capture Reference Guide

Dialog box descriptions

Bill of Materials dialog box

To open this dialog

Choose Bill of Materials (see [Bill of Materials command](#)) 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.

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.

OrCAD Capture Reference Guide

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 .
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.
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...

Look in
File name
Files of type
Open as read-only

To do this...

Specifies the drive and directory to locate the file in.
Specifies the name of the file to look for.
Specifies the type of files to look for.
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 [Preferences dialog box](#).

Use this control...

Basic colors

To do this...

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.

Create Directory dialog box

To open this dialog

Click the Create Dir button in the [Select Directory dialog box](#).

Use this control...

Current Directory

To do this...

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 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.

OrCAD Capture Reference Guide

You need to select one of two options in this dialog box.

Use this control...

Create based upon an existing project

To do this...

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.

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...

Left Arrow

Right Arrow

To do this...

Remove the property from the Find window

Add the property to the Find window

OrCAD Capture Reference Guide

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

OrCAD Capture Reference Guide

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...

Property Name

To do this...

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 pop-up menu.

Alternatively:

1. Select a design in project manager.
2. Select *Options – Design Properties* (see [Design Properties command](#)).

Use this tab...

Fonts

To do this...

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.

OrCAD Capture Reference Guide

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 `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.

Miscellaneous

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*.

Design Template / Design Properties dialog box

To open the Design Template dialog box

Select *Design Template* (see [Design Template command](#)) from the *Options* menu.

To open the Design Properties dialog box

Select a design (.DSN) in the project manager then choose *Design Properties* (see [Design Properties command](#)) from the *Options* menu.

Use this tab...

Fonts

(available in both Design Template and Design Properties dialog box)

Title Block

(available only in Design Template dialog box)

Page Size

(available only in Design Template dialog box)

To do this...

Change the fonts for objects with text. A standard 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 [Design Properties command](#) to change these options for a particular design.

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-supplied title block resides in the CAPSYM.OLB library. For more information about title blocks, see .

Specify the units of measure used in the schematic page editor. Also, change the width and height of a schematic page, as well as spacing between pins in a design. For more information, see [Page Size tab](#).

OrCAD Capture Reference Guide

Grid Reference

(available only in Design Template dialog box)

Choose between alphabetic and numeric, and between 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 [Grid Reference tab](#).

Hierarchy

(available in both Design Template and 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.

Miscellaneous

(available only in Design Properties 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*.

OrCAD Capture Reference Guide

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 [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 [Design Properties](#) 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-supplied title block resides in the CAPSYM.OLB library. For more information about title blocks, see [Setting up the default title block](#).

Grid Reference tab

To open this dialog

Select Design Template (see [Design Template](#) command) 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...

OrCAD Capture Reference Guide

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 [Design Template command](#)) 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...

OrCAD Capture Reference Guide

Units	<p>Specify the unit of measurement for future designs. Select either inches or millimeters.</p> <p>This only affects the schematic page editor. It doesn't affect the part editor, which is always measured in grid units.</p>
New Page Size	<p>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.</p>
Width	<p>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.</p>
Height	<p>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.</p>
Pin-to-Pin Spacing	<p>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.</p>

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.

Display Properties dialog box

To open this dialog

In the schematic page editor:

- Click the *Display* button in the User Properties dialog box.
- OR
- Select a part on a schematic page and choose *Properties* (see Properties command) from the Edit menu. In the Property Editor window, click the *Display* button.

Use this control...

Name

Value

Display Format

Font

Change

Use Default

Color

Rotation

To do this...

Specifies the property's name.

Specify the property's value.

Specify the visibility for the property name and value.

Displays the font name and point size.

Display a standard Windows Font dialog box so you can change the font, font style, and font size of the property.

Use the default value for the property. The default value is set in the Design Template / Design Properties dialog box (see Design Template command).

Specify the property's color.

Specify the rotation of the property.

OrCAD Capture Reference Guide

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 [Table 8-1: Text Justification](#) table on page 235.

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	Delete the selected distribution from the existing distributions list.
Save	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.

OrCAD Capture Reference Guide

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	

OrCAD Capture Reference Guide

HTML Type	<p>Select one of the following options:</p> <ul style="list-style-type: none">■ 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	<p>Specify the output HTML file path. The default path for the HTML file is the <i>temp</i> folder.</p>
HTML Base File Name	<p>Specify the name of the output HTML file. The default name of the HTML is <First Design>_vs_<Second Design>.html.</p>
Open With	<p>Select one of the following options to open the output html file:</p> <ul style="list-style-type: none">■ Internet Explorer■ Custom - Use other browsers, such as Google Chrome and Mozilla FireFox. <p>Note: It is strongly recommended to view the generated design difference HTML file in one of the following browsers, but Google Chrome is preferred:</p> <ul style="list-style-type: none"><input type="checkbox"/> Internet Explorer (version 11 onwards)<input type="checkbox"/> Google Chrome (version 45 onwards) <p>If browser except the recommended ones is selected in the Design Difference dialog box, ensure that it supports HTML5 Canvas.</p>

Edit Bookmark dialog box

To open this dialog

Select a bookmark on a schematic page, then choose Properties (see [Properties command](#)) from the Edit menu.

Use this control...

Name

To do this...

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...

Fill Style

Line Style & Width

Color

To do this...

Choose the fill style.

Choose the line style and width.

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...

Line Style & Width

Color

To do this...

Choose the line style and width.

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 [Properties command](#)) from the Edit menu.

Use this control...

Name

Type

To do this...

Change the name of the hierarchical port.

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 [Properties command](#)) from the Edit menu.

Use this control...

Alias

Color

To do this...

Change the net alias name.

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 [Properties command](#)) 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 pop-up 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 Design Template command).
Yes	Indicate the part is a primitive.

OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

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...

Text

Color

Rotation

Font

Change

Use Default

To do this...

Change the text to display.

Choose the color of the text. Text that is placed in the part editor uses the part body color.

Specify the rotation of the text.

Display a Font dialog box so you can select a font.

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 pop-up menu

Use this control...

Line Style & Width

Color

To do this...

Choose the line style and width.

Choose the color of the line. This control is not available in the part editor.

Export Properties dialog box

To open this dialog

In the Project manager, choose Export Properties from the Tools menu.

Use this control...

Scope

Contents

Mode

Export File

Browse

To do this...

Process the entire design or just the selected documents.

Export part properties or part and pin properties, or flat net properties.

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.

Specifies the name of the export output file. For more information about property files, see .

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 [Export Selection command](#)) 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.

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 window in Capture.

OrCAD Capture Reference Guide

Journaling	<p>Select the option to enable journaling of the various Capture commands, including TCL commands.</p> <p>Use the following TCL command to enable or disable journaling:</p> <pre>SetOptionBool Journaling TRUE/FALSE</pre>
Flush Commands	<p>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)</p> <p>Use the following TCL command to flush the output of the journaling commands:</p> <pre>SetOptionBool FlushImmediate TRUE/FALSE</pre>
Display Commands	<p>Select the option to enable display of Capture commands in Command Window.</p> <p>Use the following TCL command to enable or disable the display of capture commands in Command Window:</p> <pre>SetOptionBool DisplayCommands TRUE/FALSE</pre>
Enable Customer Experience Improvement Program (CEIP)	<p>Select this option to take part in the Customer 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.</p>
Design and Libraries	<p>These commands are related to Capture's designs and libraries.</p>
Content text instance properties	<p>Select the option to rotate the instance part properties along with the instance part on the same axis.</p> <p>TCL Command: <pre>SetOptionBool RotateInstPropInContext TRUE/FALSE</pre></p>

OrCAD Capture Reference Guide

Draw arrow on part input pins	<p>Select the option to draw arrow on part's input pins.</p> <p>TCL Command: <code>SetOptionBool DrawPinsArrows TRUE/FALSE</code></p>
Enable communication with legacy tools	<p>Select the option to enable message-based communication with the legacy tools, such as OrCAD Layout.</p> <p>TCL Command: <code>SetOptionBool EnableLegacyITC TRUE/FALSE</code></p> <p>Note: By default, message-based communication is not enabled in OrCAD Capture. Enable this option if you want OrCAD Capture to communicate with the legacy tool as it impacts OrCAD Capture's performance.</p>
Perform read only check on tab switch	<p>Select the option to enable Capture to check the library and design files' permissions on every switch of tab.</p> <p>TCL Command: <code>SetOptionBool CheckReadOnlyOnViewActivate TRUE/FALSE</code></p> <p>Note: Unselecting the option improves user experience as Capture ignores the permissions check on the design and library files.</p>
Save design name as UPPERCASE	<p>Select the option to save the design file name (.dsn) in uppercase letters.</p> <p>TCL Command: <code>SetOptionBool SaveAsUpperCaseDsnName TRUE/FALSE</code></p>
Enable global net ITC	<p>Select the option to start cross probing of global nets.</p> <p>TCL Command: <code>SetOptionBool EnableGlobalNetITC 1/0</code></p>
Convert images to BMP format	<p>Select the option to convert all format images to .bmp internally.</p> <p>TCL Command: <code>SetOptionBool SaveImageAsBitmap TRUE/FALSE</code></p>
Path lookup timeout (in seconds)	<p>Specify the time out time in seconds if the design or library files are found at the configured locations.</p> <p>TCL Command: <code>SetOptionUInt PathLookupTimeout <valuse in seconds></code></p>

OrCAD Capture Reference Guide

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

OrCAD Capture Reference Guide

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.

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

These command are 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.

TCL Command: `SetOptionString
AllegroCharLimitInINI TRUE/FALSE`

Schematic

These commands are related to Schematic page in Capture.

OrCAD Capture Reference Guide

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.

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/DistributionWithAutomaticExtent`

OrCAD Capture Reference Guide

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

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.

OrCAD Capture Reference Guide

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 to 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](#).

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.

OrCAD Capture Reference Guide

OrCAD Capture Reference Guide

Font dialog box

To open this dialog

Click the Change button in the Edit Net Alias dialog box, the Place Net Alias dialog box, the Edit Text dialog box, the Place Text dialog box, or the Display Properties dialog box.

OR

Click in a font box on the Fonts tab of the Design Properties dialog box, the Fonts tab of the Design Template / Design Properties dialog box, or the *Miscellaneous* tab of the Preferences dialog box.

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...

OrCAD Capture Reference Guide

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

Generate Part dialog box

To open this dialog

In the Project manager, choose Generate Part (see [Generate Part command](#)) 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...

Netlist/Source file

To do this...

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.

OrCAD Capture Reference Guide

- 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.

OrCAD Capture Reference Guide

Netlist/source file type (contd.)	<ul style="list-style-type: none">■ 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.

OrCAD Capture Reference Guide

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 Model Import Wizard 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 part in library
Specifies to update the pins on an existing part using the specified netlist, rather than create a new part.

Sort pins

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

OrCAD Capture Reference Guide

Implementation type	<p>Specify the type of implementation. The implementation types available to choose from are the same as those available in the Attach Implementation dialog box.</p> <p>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:</p> <ul style="list-style-type: none">■ <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 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 [Go To command](#).

Note: The grid reference and bookmark options are not available in the part editor.

Location tab

Use this control...

X and Y

Location Type: *Absolute* and *Relative*

To do this...

Specify the X and Y coordinates for the jump.

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...

Horizontal

Vertical

To do this...

Specify a horizontal grid reference.

Specify a vertical grid reference.

Bookmark tab

Use this control...

Name

To do this...

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...

Enter Label

To do this...

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...

Line Number

To do this...

Specifies the line number to view in the text editor window.

OrCAD Capture Reference Guide

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...

Markers List

To do this...

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.

Use this control...

Find

To do this...

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...

Backup time (in minutes)

No of backups to keep

Directory for backup

To do this...

Enables you to determine the time after which Capture will perform automatic backup.

Enables you to determine the total number of backups that will be stored.

Enables you to determine the storage location for the backup.

OrCAD Capture Reference Guide

New Alias dialog box

To open this dialog

Click the New button in the [Part Aliases dialog box](#).

Use this control...

Name

To do this...

Specify the name of the new alias.

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...

Name

To do this...

Specify the schematic folder or schematic page name.

New Part Properties dialog box

To open this dialog

Select a library in the project manager and choose New Part (see [New Part command](#)) from the Design menu.

Use this control...

Name

To do this...

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.

OrCAD Capture Reference Guide

Part Reference Prefix	<p>Specify the part reference prefix, such as "C" for capacitor or "R" for resistor. For example:</p> <p>C?1(capacitor)</p> <p>R?1 (resistor)</p>
PCB Footprint	<p>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.</p>
Create Convert View	<p>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.</p>
Multiple-Part Package	
Parts per Pkg	<p>If there are multiple parts in the package, specify the number of parts in the package.</p>
Package Type	<p>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).</p> <p>Note: The package type can only be set at creation time. These options are not available when you edit the part later.</p> <p>You should not cut and paste parts between homogeneous and heterogeneous packages.</p>
Part Numbering	<p>If the part is a multiple-part package, specify whether parts in the package are identified by letter or number. For example:</p> <ul style="list-style-type: none">■ U?A (alphabetic)■ U?1 (numeric)
Part Aliases	<p>Display the <u>Part Aliases dialog box</u> to add or remove aliases. Part aliases show up in a library represented by the part symbol with a horizontal line through the center.</p>

OrCAD Capture Reference Guide

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 <u>Part editor window</u> or view the part in the <u>package</u> 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...

Name

Location

To do this...

Specify the name of the new project.

Specify the path where you want the new project files to be saved, or use the *Browse* button to locate the directory.

New Property dialog box

To open this dialog

Click the New button on the [User Properties dialog box](#).

OR

Click the New button on the [Edit Part Properties dialog box](#).

OrCAD Capture Reference Guide

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 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	Add a member (NetGroup, bus, or scalar) to the NetGroup.
Delete	Delete a NetGroup member.
Rename	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.

OrCAD Capture Reference Guide

OrCAD Capture Reference Guide

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...

Look in

File name

Files of type

To do this...

Browse the hierarchical drive and directory structure for your system.

Select or type the name of the project or file.

Filter files by extension.

OrCAD Capture Reference Guide

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

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 `C` for capacitor or `R` for resistor.

OrCAD Capture Reference Guide

Section Count	<p>Specify the number of parts in the package. The <i>Section Count</i> field shows the count specified in the <i>Parts per Pkg</i> or <i>No. of Sections</i> field during part creation.</p> <p>If <i>Alphabetic</i> is selected in <i>Part Numbering</i> (while part creation), section names will start with <i>A</i>.</p> <p>Note: If you create a single-section part with <i>Alphabetic</i> numbering, section name is not displayed and it is automatically converted into a part with <i>Numeric</i> numbering.</p> <p>If <i>Numeric</i> is selected in <i>Part Numbering</i> (while part creation), section names will start with <i>1</i>.</p> <p>You can modify the number of sections after part creation.</p>
Part Aliases	<p>Click the <i>Update</i> button to open the Update Alias dialog box, which is used to add and edit part aliases.</p>

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 <i>Normal</i> or <i>Convert</i> for the current section of the part. This is a read-only property.
Implementation Path	<p>Specify the filename and directory to the child schematic.</p> <p>Click  icon to browse the location of the child schematic.</p>

OrCAD Capture Reference Guide

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.
	<p>Click this icon to add user-defined properties.</p> <p>You can add any user-defined properties in this section. Some of the commonly used properties are available in the drop-down list.</p> <p>You can also delete an existing user-defined property. Click the Delete icon () next to any user-defined part property to delete it.</p> <p>Any user property added to the current section of the part is applied to all sections.</p>

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...

Name

To do this...

Specify the pin name. You can create a pin name with an overbar by adding a backslash (\) after every letter in the pin name.

OrCAD Capture Reference Guide

Number	<p>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.</p> <p>The pin number has a limit of 32 characters. Characters that exceeds this limit will be truncated.</p>
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	<p>Click the add icon,  in this section to add a user-defined property.</p> <p>You can also delete an existing user-defined property. Click the Delete icon () next to any user-defined pin property to delete it.</p> <p>Any pin property added to the current section of a homogeneous part is applied to all sections in the same view (Normal or Convert).</p> <p>For a heterogeneous part, pin properties are not copied to other sections.</p>

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...

Text

Font

Font Size

Bold

Italic

Justification

To do this...

Displays the text.

Displays the current font style and also allows you to change it from the drop-down list.

Displays the current font size and also allows you to change it from the drop-down list.

Select this check box to modify the text format.

Select this check box to modify the text format.

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...

<Object Name>

To do this...

Displays the value of the selected object.

OrCAD Capture Reference Guide

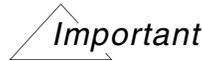
Rotation	Displays the rotation of the selected object.
Location	Displays the X and Y coordinates of the selected object on the canvas.
Font	Displays the current font style of the selected object. You can also change the font style from the drop down list.
Color	Displays the color of the selected object 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.



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* from the pop-up menu.
- 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

OrCAD Capture Reference Guide

■ 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.

OrCAD Capture Reference Guide

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.
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	<ul style="list-style-type: none">■ Copy and paste data in the same table.Or■ Copy and paste data from Microsoft Excel. In the edit pins table: <ul style="list-style-type: none">■ 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.

OrCAD Capture Reference Guide

Operation	Function
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.	 Important 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.

Paste Options dialog box

To open this dialog box

OrCAD Capture Reference Guide

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 New Part Properties dialog box.

Use this control...

Alias Names

New

Delete

To do this...

Select an alias name from the displayed list.

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.

Place Bookmark dialog box

To open this dialog

In the schematic page editor, choose Bookmark (see [Bookmark command](#)) 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.

OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

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 User Properties dialog box so you can modify the hierarchical block's user defined properties.

Implementation

OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

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 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.

Place Hierarchical Pin dialog box

To open this dialog

In the schematic page editor

Choose Hierarchical Pin (see Hierarchical Pin command) from the Place menu.

OR

Click the Hierarchical Pin button on the Draw toolbar.

Use this control...

Name

Type

Width

User Properties

To do this...

Specify the hierarchical pin's name.

Select the pin type from the list of pin types.

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.

Display the User Properties dialog box 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...

Symbol

Symbol List

Libraries

Preview box

To do this...

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.

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.

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.

Displays the graphic of the selected object.

OrCAD Capture Reference Guide

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 [IEEE Symbol command](#).
- Click the IEEE Symbol button on the Draw toolbar.

Use this control...

IEEE Symbols

Preview box

To do this...

Select a symbol from the list of available symbols.

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.

OrCAD Capture Reference Guide

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

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.

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.

OrCAD Capture Reference Guide

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...

OrCAD Capture Reference Guide

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.

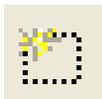
For example, if you are a PSpice user, you can restrict your part library search such that only parts with associated PSpice simulation models will be listed in the Part List.



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 DeMorgan equivalent 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).



OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

Application indicator

This feature uses icons to indicate whether the selected part has certain properties. The icon displayed for each property is described below. Note that, in the case of the Implementation Type property, only Implementation Type properties with values of “Schematic View” or “VHDL” are indicated with an icon.



Property: PCB Footprint. **Value:** Various values related to PCB layout.



Property: Implementation Type. **Value:** “Schematic View”



Property: Implementation Type. **Value:** “VHDL”



Property: Implementation Type. **Value:** “Verilog”



Property: PSpice Template. **Value:** PSpice syntax for part netlist entry



This 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	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...

OrCAD Capture Reference Guide

Starting Name Specify the name for the pin array. If the name ends with a digit (0--9), each pin in the array is incremented by the value specified in the *Pin# Increment for Next Pin* 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 *Starting Name* 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 *Pin# Increment for Next Pin* field.

When you open this dialog box to add more pin arrays, you can see that *Starting Number* 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.

Type 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 (0--9), each pin name is automatically incremented, by the value specified in the *Pin# Increment for Next Pin* 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 *Pin# Increment for Next Pin* 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.

OrCAD Capture Reference Guide

User Properties Click this button to open the [User Properties dialog box](#), 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.

OrCAD Capture Reference Guide

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 [Text command](#).
- Click the Text button on the Draw toolbar.

- Press the T key.

Use this control...

Text

Color

Rotation

Font

Change

Use Default

Text Justification

To do this...

Enter the text in the text box.

Specify the color of the text. Text that is placed in the part editor uses the part body color.

Specify the rotation of the text.

Display a Font dialog box so you can select a font.

Change the font to the default font specified in the Design Template dialog box.

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 235.

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 [Title Block command](#)) from the Place menu.

OR

Click the Title Block button on the Draw toolbar.

Use this control...

To do this...

OrCAD Capture Reference Guide

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 [Preferences command](#)) from the Options menu.

Use this tab...

Application Theme

Schematic Theme

To do this...

Click the drop-down list to select the theme of the application.

Click the drop-down list to select the theme of the schematic page.

OrCAD Capture Reference Guide

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 Number, Part Properties, Part Reference Prefix, Part Value, and any user-defined property overrides the color selected in the Preferences dialog box.

Grid Display

Control the behavior and appearance of the grid display for both the schematic page editor and the part editor. See [Grid Display tab](#) for more information.

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](#).

OrCAD Capture Reference Guide

<i>Text Editor</i>	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 Text Editor tab .
<i>Board Simulation</i>	Specify the hardware development language to be used during board level simulation.

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.

OrCAD Capture Reference Guide

Grid spacing	<p>Specify the grid spacing on the schematic page as a fraction of pin-to-pin spacing.</p> <p>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.</p>
Pointer snap to grid	<p>Specify whether the pointer snaps to the grid in the editor. This check box is selected by default.</p>
Fine <i>(only for schematic page editor)</i>	<p>Specify whether connectivity (part, bus) and drawing 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.</p> <p>Note: This setting is saved in the CAPTURE.INI file and it is used whenever you start the next Capture session.</p>
Coarse <i>(only for schematic page editor)</i>	<p>Specify whether connectivity and drawing objects can be placed and moved on the coarse grid. For description of various scenarios for this option, see Customizing placement and movement of objects on the schematic.</p> <p>Note: This setting is saved in the CAPTURE.INI file and it is used whenever you start the next Capture session.</p>
Master <i>(only for schematic page editor)</i>	<p>Specify whether connectivity and drawing objects can be placed and moved on the master grid, if <i>Pointer snap to grid</i> check box is selected. This setting is equivalent to the Snap To grid toolbar state .</p> <p>Specify whether connectivity and drawing objects can be placed and moved on the fine grid, if <i>Pointer snap to grid</i> check box is not selected. This setting is equivalent to the Snap To grid toolbar state .</p> <p>For description of various scenarios for this option, see Customizing placement and movement of objects on the schematic.</p> <p>Note: This setting is saved in the CAPTURE.INI file and it is used whenever you start the next Capture session.</p>

Note: Ensure that the *Pointer snap to grid* check box is selected and the Connectivity Elements is set to Coarse while placing

OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

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 [Table 8-2: Design Level Annotation](#) table on page 318.

OrCAD Capture Reference Guide

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.

Note: You can select only one option at a time.

OrCAD Capture Reference Guide

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 be preserved while pasting that part on a schematic page.

Intertool Communication

Enable intertool communication (Default: OFF)	Enable intertool communication with other OrCAD products such as Simulate or Layout. 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

OrCAD Capture Reference Guide

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 un-check 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...

Zoom Factor

To do this...

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.

OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

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 tools	Specify that the text editor save any open text files before running any Capture tools (such as design rules check or netlist generation).
Reset	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 [Print command](#)) from the File menu.

Note: Click the Setup button to go to the [Print Setup dialog box](#) 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...

Printer

Scale

To do this...

Displays the active printer and printer connection.

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.

OrCAD Capture Reference Guide

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 "prepending" 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 option is selected by default.
- 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.

OrCAD Capture Reference Guide

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	<p>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.</p> <p>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.</p>
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.

OrCAD Capture Reference Guide

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.

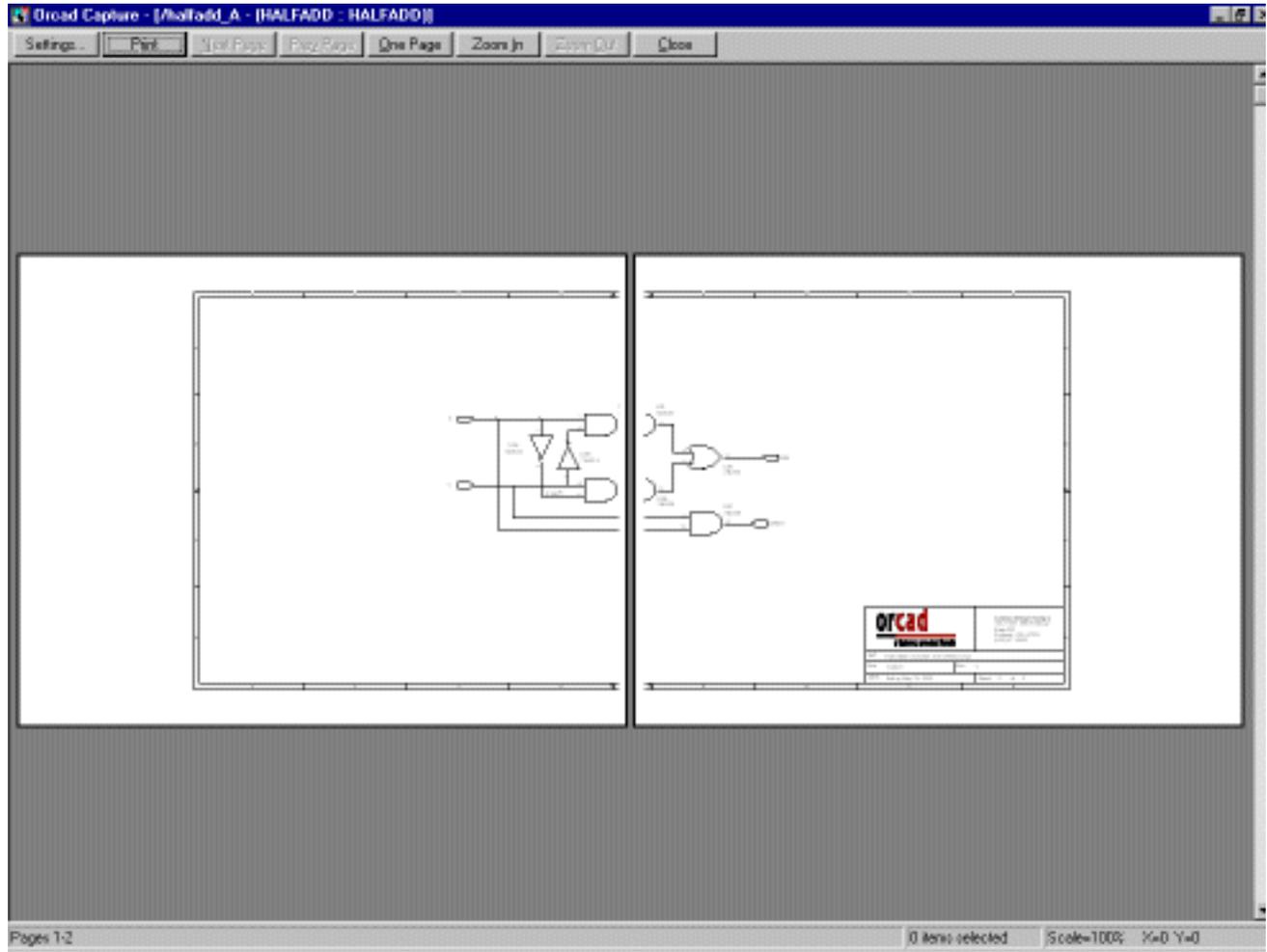
Setup

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

OrCAD Capture Reference Guide

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:

Print statistics			
	Total	Horizontal	Vertical
Printed pages per document page:	2	2	x 1
Maximum size for this page layout:		21.1717	x 8.14833
Size from schematic page properties:		15.2	7.2
Size of actual printout:		15.2	7.2

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 [Annotate dialog box](#).

Print Preview and Print Setup dialog boxes

To open this dialog

Choose Print Preview (see [Print Preview command](#)) 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.

OrCAD Capture Reference Guide

Print offsets	<p>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.</p> <p>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:</p> <ul style="list-style-type: none">■ 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	<p>Specify whether you want to print instances or occurrences of a page.</p> <ul style="list-style-type: none">■ 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	<p>Specify the resolution of the print. Choose a setting from the drop-down list.</p>
Copies	<p>Specify the number of copies to print.</p>
Print to file	<p>This option is not available in Print Preview. If you want to print to file, use the Print dialog box.</p>
Print all colors in black	<p>Causes difficult-to-read colors to print in black.</p>
Collate copies	<p>Print copies organized in order of page numbers</p>

OrCAD Capture Reference Guide

Print area	<p>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.</p> <p>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.</p>
Include pages outside hierarchy	<p>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.</p>
Include referenced pages in other libraries or designs	<p>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.</p>
Print statistics	<ul style="list-style-type: none">■ 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. <p>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).</p> <ul style="list-style-type: none">■ 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. <p>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.</p>

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...

Output file name

To do this...

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*.

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...

OrCAD Capture Reference Guide

Pin Pair	<p>Applies Min and/or Max delay constraint to various pin-pairs. This value may be set to one of the following:</p> <ul style="list-style-type: none">■ <i>Longest/Shortest pin-pair</i>—Minimum delay is applied to the shortest pin-pair and maximum delay is applied to the longest pin-pair.■ <i>Longest/Shortest Driver/Receiver</i>—Minimum is applied to the shortest Driver/Receiver pin-pair and maximum is applied to the longest Driver/Receiver pin-pair.■ <i>All Drivers/All Receivers</i>—Min/Max constraints apply to all Driver/Receiver pin-pairs.
Min	<p>Specifies the minimum allowable propagation delay/length for the pin-pairs.</p>
Min Rule	<p>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).</p>
Max	<p>Specifies the maximum allowable propagation delay/length for the pin-pairs.</p>
Max Rule	<p>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).</p>
Add Pin Pair	<p>Displays the Create Pin Pairs dialog box. Use this dialog box to define a pin-pair.</p> <p><i>Keyboard shortcut: ALT, A</i></p>
Delete Pin Pair	<p>Deletes the pin-pair corresponding to the selected row.</p> <p><i>Keyboard shortcut: ALT, D</i></p>
OK	<p>Performs syntax checking and if the syntax is correct assigns the PROPAGATION_DELAY property on the selected net.</p>

Note: You can use the [User Properties dialog box](#) 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 pop-up menu.

Use this tab...

General tab

Type tab

Project tab

To do this...

Provide information about the file name, size, and the date it was last modified. For more information, see [General tab](#)

Specify the type of the file, such as schematic or simulation model. For more information, see [Type tab](#).

Specify project options. These options include project type, vendor, and family. For more information, see [Type tab](#).

General tab

Use this control...

File name

To do this...

Displays the name of the file currently selected in the project manager window.

OrCAD Capture Reference Guide

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 type drop-down list	The drop-down list provides a list of file descriptions. The 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.

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 Categories View	Toggle Hide Categories View or Show Categories View to hide or unhide the Category View.
Search Online	Opens new window in Capture for searching PSpice models in OrCAD 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

OrCAD Capture Reference Guide

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

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...

Name

To do this...

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 [Rename command](#)) 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 and 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...

Name

To do this...

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 [Rename command](#)) from the Design menu.

Use this control...

Name

To do this...

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 [Rename command](#)) from the Design menu.

Use this control...

To do this...

Name

Specify the name of the title block.

Replace Cache dialog box

To open this dialog

Select a part in the design cache folder and choose Replace Cache (see [Replace Cache command](#)) 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 *Part name* list box, select a library in the *Part Library* 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 *Part Name* 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.

OrCAD Capture Reference Guide

Preserve Refdes

Preserve the reference designator of parts and/or symbols 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.

OrCAD Capture Reference Guide

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	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.

OrCAD Capture Reference Guide

To open this dialog

Perform one of the following:

- Choose *File – Save Project As*.
- Select DSN in the project manager and from the pop-up menu choose *Save Project As*.
- Select *Design Resources* in the project manager and choose *Save As* from the pop-up menu

Use this tab...

Destination Directory

Project Name

Settings

To do this...

Specify the location to which the project should be saved.

Shows the current project name. You can edit to change it to a different name.

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 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 Project

Select to rename the design file to match the project name. Selected by default.

Copy All Referred Files Present Within Project Folder

Select to copy all referred files that are in the current 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 Present Out of Project Folder Select to copy all referred files that are outside the 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.

||

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 [Page Size tab](#).

OrCAD Capture Reference Guide

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 [Schematic Page Properties command](#) from the Options menu, when you are in the schematic page editor.

The Design Template dialog box appears when you choose the [Design Template command](#) 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.

OrCAD Capture Reference Guide

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 Design Template command 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	<p>Select a file type for the file you are adding to the project. Choose the appropriate type from the provided list:</p> <ul style="list-style-type: none">■ EDIF Netlist■ List■ OrCAD 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...

Check box under the Schematic Page Drag Selection Filter group

Select All

Clear All

To do this...

Specify the objects that should be selected when the mouse pointer is dragged diagonally across the schematic page.

Select all the check boxes in the Schematic Page Drag Selection Filter group.

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...

Occurrence list

To do this...

Specify which occurrence of the schematic page you want to open. This option lists all occurrences available for the selected schematic page.

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...

Enter Label

To do this...

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 [Updating the PCB Editor configuration file](#).

OrCAD Capture Reference Guide

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

Specifies 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.

OrCAD Capture Reference Guide

Ignore Electrical constraints	<p>Specifies that the following electrical constraints will be ignored during netlist:</p> <ul style="list-style-type: none">■ 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 <p>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.</p>
Output Warnings	<p>Specifies if netlist warnings (ALG*) are to be logged during netlisting.</p>
Suppress Warnings	<p>Specifies the netlist warnings (ALG*) to be ignored during netlist.</p>

Simulation Settings dialog box

To open this dialog

Choose Edit Simulation Profile (see [Edit Simulation Profile command](#)) from the PSpice menu.

OrCAD Capture Reference Guide

This dialog box is similar to the Simulation Settings dialog box in PSpice. Both dialog boxes set simulation profile properties.

Use this tab...

General

Analysis

Include Files

To do this...

Specify file information and simulation notes. For more information, see [General tab](#)

Specify analysis options. This includes time domain, DC sweep, AC sweep and noise, and bias point options. For more information, see [Analysis tab](#)

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

Create include files using any text editor, such as Notepad. Include file names typically have an .INC extension.

For more information, see the [Include Files tab](#) help topic.

OrCAD Capture Reference Guide

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 [Libraries tab](#) help topic.

Stimulus

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 [Stimulus tab](#) help topic.

Options

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 [Options tab](#) help topic.

OrCAD Capture Reference Guide

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](#)

OrCAD Capture Reference Guide

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 [Transient Output File Options 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.

OrCAD Capture Reference Guide

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...

Monte Carlo

Worst-case/Sensitivity

To do this...

Perform a Monte Carlo (statistical) analysis of the circuit.

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.

OrCAD Capture Reference Guide

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. <ul style="list-style-type: none">■ <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 Distributions dialog box .
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...
---------------------	---------------

OrCAD Capture Reference Guide

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...

Run the simulation at temperature

To do this...

Specifies a temperature at which the analysis is done. The value is specified in degrees Centigrade.

OrCAD Capture Reference Guide

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...

To do this...

OrCAD Capture Reference Guide

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 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 states	Collect data for voltages, currents, and digital states.
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 (.CSD)	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.

OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

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

OrCAD Capture Reference Guide

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.

OrCAD Capture Reference Guide

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...
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 associated VHDL or Verilog models.
Contains Packaging Information	Specify that you want to restrict the part search to only parts that have packaging information.

Transient Output File Options dialog box

The Transient Output File Options dialog box appears when you select the General Settings option from the Analysis tab 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 Harmonics

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.

OrCAD Capture Reference Guide

Update Properties dialog box

To open this dialog

In the project manager, choose Update Properties (see [Update Properties command](#)) 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.
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.

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...

Name

Value

Attributes

New

Remove

Display

To do this...

List the item's properties.

Display the value of each property.

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.

Display the New Property dialog box so you can create a property for the item.

Remove the selected property from the item's property list.

Display the 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 PROPAGATION DELAY and RELATIVE PROPAGATION DELAY properties to all the bits of a bus at the same time. Make sure that the syntax is correct. For more information, see [Assigning signal flow properties](#).

Update Alias dialog box

Use this control...

Part Aliases



Apply

To do this...

In this section, click the add row icon () to specify a new part alias.

Click the *Add Alias* button () to add the new alias.

Click *Apply* to save the alias. You can see the new alias added to the library in the project manager tab.

OrCAD Capture Reference Guide

Zoom Scale dialog box

To open this dialog

In the schematic page editor, choose *Zoom – Scale* from the View menu.

Use this control...

X%

Custom X%

To do this...

Choose a predefined zoom scale ranging from 25% to 400%.

Specify a custom zoom scale.