cadence®

OrCAD® Capture Quick Reference

Product Version 16.6
October 2012

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

Contents

1

Shortcut Keys

In addition to providing menu access keys for menu commands, Capture provides shortcut keys for miscellaneous actions, such as scrolling across an editor's window. An example of an access key is ALT, E, T for the Cut command on the Edit menu. Shortcut keys include CTRL keys (like CTRL+DELETE to delete), Single letter keys (like P to place a part), and function keys (like F4 to repeat a command).

Note: The description of each menu command includes keyboard, mouse, and other shortcuts.

Tip

Many shortcuts are available while you use another command. For example, you can use I and O to zoom in and out while you move and place objects.

Tip

You can run PSpice simulations from your Capture environment by pressing the F11 function key. Also, you can view the simulation results for the currently active profile by pressing the F12 function key.

Note: You can select a product suite from which to open Capture without having to exit the tool by choosing the File – Change Product menu command (ALT, F, C). Also, you can 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.

All Capture windows

Key	Mouse click equivalent
ALT+F4	Exit
ALT, F, X	Exit

ALT, SPACEBAR, C	Exit
F1	Help
Ctrl + F8	Switch Capture to Full Screen mode (Click Close Full Screen button or press Escape key to exit Full Screen mode)

Text editor

Key	Mouse click equivalent
CTRL+F4	If you attach a VHDL file to a hierarchical block and descend the hierarchy, you will end up within the VHDL file. Use this shortcut to close the file and return to the top level of the schematic

Schematic page editor

Key	Mouse click equivalent
CTRL+A	Select All
A	Ascend hierarchy
D	Descend hierarchy
B	Place bus
E	Place bus entry
F	Place power
G	Place ground
J	Place junction
N	Place net alias
P	Place part
T	Place text
W	Place wire

Y	Place polyline
X	Place No connect symbol
Z	Place Database Part
F7	Record macro
F8	Play macro
F9	Configure macro
CTRL+SHIFT+A	Add part(s) to group
CTRL+SHIFT+R	Remove part(s) from group
CTRL+SHIFT+W	Select entire net
CTRL+I	Opens the Selection Filter dialog box where you can specify the objects that should be selected when the mouse pointer is dragged diagonally across the schematic page.
CTRL+SHIFT+Left Click	Lock component
CTRL+U	Ungroup objects
CTRL+G	Center the view on a specific location, grid references, or bookmark.
SHIFT+B	Create test bench
SHIFT+M	Find differences and merge test benches
CTRL+F4	Close Place Part dialog **Note:** To use this shortcut, the Place Part dialog must be currently selected.

Note: Select a component on the schematic page and press SHIFT+S to place components directly from your Capture schematic design to Allegro PCB Editor.

Part editor

Key	**Mouse click equivalent**
CTRL+B	Previous part
CTRL+N	Next part

Property editor

Key	Mouse click equivalent
CTRL+B	Gives focus to the Filter by drop-down list
CTRL+D	Open the Display Properties dialog box to edit properties for a selected cell
CTRL+E	Edit properties
CTRL+F	Find a character or search string
CTRL+K	Open Propagation Delay dialog box
CTRL+L	Delete a property (cell value)
CTRL+N	Add a new column or row
CTRL+P	Apply changes
DELETE	Delete a character
CTRL+Z	Undo
CTRL+C	Copy
CTRL+V	Paste
CTRL+X	Cut
Page-Up/CTRL+<Up Arrow>	Go to the first cell in a column
Page-Down/CTRL+<Down Arrow>	Go to the last cell in a column
CTRL+<Left Arrow>	Go to the first cell in a row
CTRL+<Right Arrow>	Go to the last cell in a row
Esc	Undo edit in the selected cell
SHIFT+<Arrow key>	Select
CTRL+Home	Go to the beginning (top left corner) of the spreadsheet
CTRL+End	Go to the end (bottom right corner) of the spreadsheet
CTRL+F2	Select the contents of a cell
CTRL+F4	Close the spreadsheet

CTRL+F6	Switch to the other open windows (Works like CTRL+Tab)
CTRL+U	Opens the Propagation Delay dialog box or the Relative Propagation Delay dialog box depending on the property being edited in the Property Editor window. For example, if you select the grid corresponding to the Propagation Delay property and press CTRL+U, the Propagation Delay dialog box appears.

Schematic page and part editors

Key	Mouse click equivalent
CTRL+C	Copy
CTRL+E	Edit properties
CTRL+F	Find
CTRL+G	Go to
CTRL+P	Print
CTRL+S	Save
CTRL+T	Cursor snap to grid (identical to the Preferences dialog box Grid display tab option).
CTRL+V	Paste
CTRL+X	Cut
CTRL+Y	Redo
CTRL+Z	Undo
F4	Repeat
DEL	Delete (Design and Edit menus)
DELETE	Delete (Design and Edit menus)
BACKSPACE	Delete (Design and Edit menus)
ENTER	Double-click left mouse button

ESCAPE	Deselect all and switch to selection tool (arrow pointer)
SPACE	Click left mouse button
UP ARROW	Move 1 grid up (grid on) or 0.1 grid up (grid off)
DOWN ARROW	Move 1 grid down (grid on) or 0.1 grid down (grid off)
LEFT ARROW	Move 1 grid left (grid on) or 0.1 grid left (grid off)
RIGHT ARROW	Move 1 grid right (grid on) or 0.1 grid right (grid off)
CTRL+UP ARROW	Snap pointer to nearest grid and then move 5 grids up
CTRL+DOWN ARROW	Snap pointer to nearest grid and then move 5 grids down
CTRL+LEFT ARROW	Snap pointer to nearest grid and then move 5 grids left
CTRL+RIGHT ARROW	Snap pointer to nearest grid and then move 5 grids right
PAGE UP	Pan up
PAGE DOWN	Pan down
CTRL+PAGE UP	Pan left
CTRL+PAGE DOWN	Pan right
F5	Redraw
C	Center the view at the pointer's current position
H	Mirror horizontally
I	Zoom in
O	Zoom out
R	Rotate
V	Mirror vertically
B	Begin a wire, bus, or polyline (corresponding tool active)
E	End a wire, bus, or polyline (corresponding tool active)

Session Log

Key	Mouse click equivalent
CTRL+DEL	Clear session log
CTRL+DELETE	Clear session log

Note: When an error or warning message displays in the session log, you can find the help topic for that particular error or warning by putting your cursor in the line containing the error/warning and pressing the F1 key.

Text boxes

Key	Mouse click equivalent
BACKSPACE	Delete selected text
DEL	Delete selected text
DELETE	Delete selected text
CTRL+C	Copy selected text to Clipboard
CTRL+V	Paste Clipboard contents
CTRL+X	Cut selected text to Clipboard
CTRL+Z	Undo last edit
DOUBLE CLICK	Select word and any following space
SHIFT+CLICK	Extend selection from insertion point to cursor
CTRL+RIGHT ARROW	Jump right one word
CTRL+LEFT ARROW	Jump left one word
HOME	Jump to beginning of line
END	Jump to end of line
CTRL+HOME	Jump to beginning of text box
CTRL+END	Jump to end of the text box

SHIFT+HOME	Extend selection from insertion point to beginning of multiple-line text box
SHIFT+END	Extend selection from insertion point to end of multiple-line text box

Browse spreadsheet editor

Key	Mouse click equivalent
CTRL+C	Copy value from a cell in the Browse spreadsheet editor
CTRL+V	Paste Clipboard contents onto another cell in Browse spreadsheet editor
CTRL+INSERT	Copy value from a cell in the Browse spreadsheet editor and paste it onto a cell in Microsoft Excel worksheet
SHIFT+INSERT	Paste value copied from Microsoft Excel onto a cell in the Browse spreadsheet editor

Fisheye View

Key	Mouse click equivalent
SHIFT+F11	Set Fisheye focus on selected component
SHIFT+CTRL+F11	Reset Fisheye focus
Q	Start Fisheye dynamic focus mode

2

Capture Command Line Arguments & Switches

You can invoke OrCAD Capture directly from the command line with specific arguments and switches.

Syntax

```
capture [-i <capture.ini location>] [dsn name] [tcl script name]
```

Switch / Argument	Description
`-i <capture.ini location>`	The location of the capture.ini to be used with the current instance of Capture.
`dsn name`	Name and location of the dsn to be opened in the current instance of Capture.
`tcl script name`	Name and location of the tcl file to be executed at startup.

Note:

- All the arguments are optional.
- The **dsn name** and **captcl name** arguments cannot be used in conjunction with each other.

3

Capture.ini Variables

When Capture starts up, it uses a pre-defined set of default values for the application settings. These default values are defined in the `Capture.ini` file. This file is placed in the `HOME` directory (`%HOME%\cdssetup\OrCAD_Capture`) in your system. However, you can specify another location for this file by using the `-i` switch when invoking Capture. See Capture Command Line Arguments & Switches section of this guide for details.

The `Capture.ini` file contains a large set of initialization variables used by Capture. This section defines a set of these variables that alter the Capture start-up behavior.

Important

The `Capture.ini` file is used by Capture during start-up. Incorrect or corrupt information in this file can cause unexpected behavior in Capture. Before making any changes to this file, you are advised to first make a backup of your installed `Capture.ini`. Also, this chapter covers only a selected list of `ini` sections and only selected variables within these sections. It is advised that you work with only these variables.

Note: The `Capture.ini` file is a text-based file. This implies that you can use any Text editor to open and edit this file.

This chapter covers initialization variables in the following sections:

- ConvertDialog on page 21

SearchToolBarSetting

This ini section includes the variables that control the selection of the Search toolbar options. You can set a Search toolbar option as selected or un-selected by settings the variable values in this section.

Use `1` to set an option as selected.

Use `0` (zero) to set an option as un-selected.

Search Toolbar Options:

- Parts
- OffPage
- BookMark
- CommentText
- Nets
- FlatNets
- HierPorts
- DRC
- TitleBlock
- PowerSymbol
- HierPins
- PartsPins
- VariantParts
- MatchCase
- Highlight

Docking

The variables in this section allow you to set the docked window status and visibility of Capture CIS windows.

session_docked

Set the value to `1` to dock the session log window.

Set the value to `0` (zero) to specify the session log as an MDI window.

session_show

Set the value to `1` to ensure the session log window is visible on start-up.

Set the value to `0` (zero) to ensure the session log window is hidden on start-up. To display the session log if it is hidden, choose the Session Log option in the Capture CIS Window menu.

Print Settings

Use the variables in this section to define the print settings in Capture CIS.

InstanceMode

Set the value to `1` ensure that the default selected Print option in the Print dialog box is set to Inst. Mode (Instance Mode).

Set the value to `0` (zero) to ensure that the default selected Print option in the Print dialog box is set to Occ. Mode (Occurrence Mode).

Preferences

The Preferences section in the ini contains variables that provide access to the settings in the Capture CIS Preference dialog box.

DockingPlacePart

Set the value to `FALSE` to ensure that the Place Part user interface available in Capture CIS versions prior to version 16.2 displays.

Set the value to `TRUE` to ensure that the Capture CIS 16.2 and later Place Part user interface displays.

SearchToolBar

Set the value to `FALSE` to ensure that the Find dialog box available in Capture CIS versions prior to version 16.2 displays.

Set the value to `TRUE` to ensure that the Capture CIS 16.2 and later Search Toolbar displays.

EnableITC

Set the value to `TRUE` to Enable Intertool Communication in Capture CIS.

Set the value to `FALSE` to disable Intertool Communication in Capture CIS.

Footprint Viewer Type

The variables in this section describe the type of footprints that will be displayed in the CIS Explorer footprint window. The section is also linked to the Layout Footprints and the Allegro Footprints sections that are then used to define the directory location for the footprint (`psm`) files.

type

This variable defines the type of footprints to display.

Set the value to `Allegro` to use Allegro files for footprints.

Set the value to `Layout` to use Layout files for footprints.

Note: Use the **dir0** variable in the **Layout Footprints** section to define the directory location containing the footprint files to use if the footprint type is defined as **Layout**. Or use the **dir0** variable in the **Allegro Footprints** section to define the directory location containing the footprint files to use if the footprint type is defined as **Allegro**.

Allegro Footprints

The variable (or variables) in this section define the directory location (or locations) for the `psm` and `pad` files used by the PCB Editor 3D Footprint Viewer to display a footprint. A `psm` file is used by the PCB Editor 3D Footprint Viewer to display the 3D footprint. A `pad` file is used by the Viewer to display the pin information in the viewer.

You can define any number of paths that contain the `psm` and `pad` files. If you define multiple paths, Capture CIS will search the directories, for the footprint files, in the order in which they are defined in this section.

Note: If the footprint information (corresponding `psm` file) is not found in any of the paths defined by the variables in this, Capture CIS will display an error message in the Session log window.

Example:

```
[Allegro Footprints]

Dir1=D:\cadence163\share\pcb\pcb_lib\symbols

Dir2=D:\SPB162\tools\fsp\examples\libraries\footprints\allegro
```

Part Management

The variables in this section define paths to the database configuration file used in the Capture CIS flow. The configuration file identifies the ODBC data source to use, Part properties that are transferred to your design and the relationship across the tables in your preferred parts database.

Note: If this section is missing or the configuration file paths are incorrect, you will experience errors in any CIS database operations.

Configuration File

Use this variable to define the path for the database configuration file if you are using the Licensed version of Capture CIS.

Example:

```
Configuration
File=D:\Cadence_10Nov\SPB_16.3\tools\Capture\Samples\BENCHACC.DBC
```

DemoConfiguration File

Use this variable to define the path for the database configuration file if you are using the Lite version of Capture CIS.

Example:
`D:\Cadence_Demo\SPB_16.3\tools\Capture\Samples\DemoBENCHACC.DBC`

Frame view options in CIS Explorer window

The following sections allow you to toggle the visibility of the corresponding frames in Local Part Database and ICA tabs of the Capture CIS Explorer window.

- Visibility Frame

  This section defines the variables of the visibility frame in CIS Explorer.

- Schematic Part Frame

  This section defines the variables of the schematic part frame in CIS Explorer.

- FootPrint Frame

  This section defines the variables of the footprint frame in CIS Explorer.

- Relational Table Frame

  This section defines the variables of the relational table frame in CIS Explorer.

Each of the above sections has two variables that define the frame visibility in the Local Part Database and ICA tabs in the Capture CIS Explorer window.

Visible

Set the value to `1` to ensure the corresponding frame is visible in the Local Part Database tab.

Set the value to `0` (zero) to ensure the corresponding frame is hidden in the Local Part Database tab.

Visible In ICA Mode

Set the value to `1` to ensure the corresponding frame is visible in the ICA tab.

Set the value to `0` (zero) to ensure the corresponding frame is hidden in the ICA tab.

ConvertDialog

The variables in this section define the default behavior of the Save Converted Libraries and Designs dialog box. This dialog box appears when you exit Capture if you have you included a configured library or a referenced design from a version of Capture prior to 16.3 in design that is based on a 16.3 or later version of Capture.

Choice

Set this variable to `Save All` to ensure that all displayed libraries or designs are saved to the upgraded version.

Set this variable to `No All` to ensure that none of the displays libraries or designs are saved to the upgraded version.

Do Not Ask

Set this variable to `1` to ensure that the dialog box will not be displayed every time you close Capture in the upgrade library and design mode. Capture will use the option that is set for the Choice in this section.

Note: This option applies to referred libraries only. These are libraries that are included in a project through the Add Library command on the Place Part dialog. In addition, when you close an older version library directly opened in Capture or when you exit Capture after opening an older version library, the application always prompts you to upgrade the library.

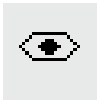
Set this variable to 0 to ensure that the dialog box will appear every time you close Capture in the upgrade library and design mode.

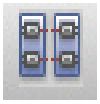
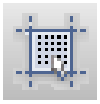
4

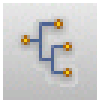
Capture Toolbar

The Capture toolbar provides shortcuts for many of the most frequently used commands. The following table describes the icons on the toolbar.

Icon	Name	Description
	New document	Creates a new document based on the active document. Equivalent to the New command on the File menu.
	Open document	Opens an existing document based on the active document. Equivalent to the Open command on the File menu.
	Save document	Saves the active schematic page or part. Equivalent to the Save command on the File menu.
	Print	Prints the active schematic page or part. Equivalent to the Print command on the File menu.
	Cut to clipboard	Removes the selected object and places it on the Clipboard. Equivalent to the Cut command on the Edit menu.

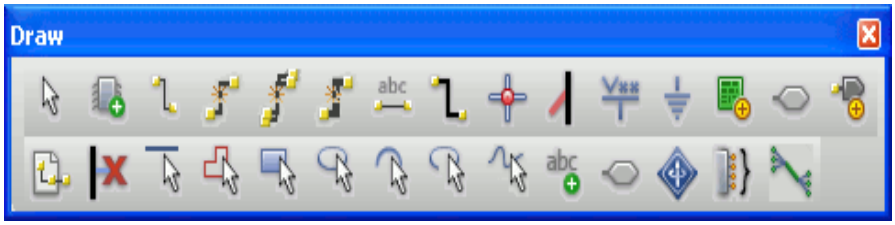
	Copy to clipboard	Copies the selected object to the Clipboard. Equivalent to the Copy command on the Edit menu.
	Paste from clipboard	Pastes the contents of the Clipboard at the cursor. Equivalent to the Paste command on the Edit menu.
	Undo	Undoes the last command performed. Equivalent to the Undo command on the Edit menu.
	Redo	Redoes the last command performed. Equivalent to the Redo command on the Edit menu.
	Most recently used	Displays the most recently placed part name in the drop-down list. Capture automatically adds part names as you select them from the Place Part dialog box. Select from the list to place parts again later.
	Zoom in	Zooms in to present a closer, enlarged view. Equivalent to the In command on the Zoom menu (on the View menu).
	Zoom out	Zooms out to present more of your document. Equivalent to the Out command on the Zoom menu (on the View menu).
	Zoom to region	Specifies an area of the schematic page or part to enlarge to fill the entire window. Equivalent to the Area command on the Zoom menu (on the View menu).
	Fisheye	Toggles the Fisheye mode on and off.
	Zoom to all	Views the entire document. Equivalent to the All command on the Zoom menu (on the View menu).
	Annotate	Assigns part references to parts on the selected schematic pages. Equivalent to the Annotate command on the Tools menu.

	Back annotate	Back annotates the selected schematic pages. Equivalent to the Back Annotate command on the Tools menu.
	Design rules check	Checks for design rules violations in the selected schematic pages. Equivalent to the Design Rules Check command on the Tools menu.
	Create netlist	Creates a netlist from the selected design. Equivalent to the Create Netlist command on the Tools menu.
	Cross reference	Creates a cross reference report of the selected schematic pages. Equivalent to the Cross Reference command on the Tools menu.
	Bill of materials	Creates a bill of materials report from the selected schematic pages. Equivalent to the Bill of Materials command on the Tools menu.
	Snap to grid	When Snap to Grid is turned on  , then Capture automatically aligns all objects as you place or move them on the page, by snapping them to the page grid. When Snap to Grid is turned off  , then Capture allows you to place or move objects anywhere on the page.

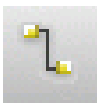
	Area Select - Fully Enclosed Vs Intersecting	If the button is in the Intersecting mode state  , the objects are selected when the selection area border intersects them. If the button is in the Fully Enclosed state  , the objects are selected only when they are completely enclosed in the selection area. This is equivalent to selecting the Area Select options under the Select tab in the Preferences dialog box. **Note:** If the button is in the Enclosed mode state, ensure that the object along with its name and number are enclosed in the selection area. Otherwise, the object does not get selected.
	Drag connected object	If the button is in the  state then Capture allows to drag and place objects on the schematic, even if the connectivity changes. If the button is in the  state then the selected object attaches to the cursor and does not get placed on the schematic, if it changes the connectivity.
	Project manager	Displays a project manager window for the active document, providing an overview of project contents.
	Help	Displays the online help. Equivalent to the Help Topics command on the Help menu.

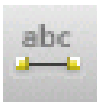
5

Draw Toolbar



The Draw Toolbar provides shortcuts for commands to place components, pins, wires, bus, and drawing objects, such as arcs, polyline, ellipse, and text. The following table describes the icons on the toolbar.

Tool	Name	Description
	Select	Selects objects. This is the normal mode.
	Place part	Selects parts from a library for placement. Equivalent to the Part command on the Place menu.
	Place wire	Draws wires. SHIFT allows any angle drawing. Equivalent to the Wire command on the Place menu.
	Auto Connect two points	Switches the Schematic page editor into the Auto connect mode and allows you to connect two points on the page.
	Auto Connect multi points	Switches the Schematic page editor into the Auto connect mode and allows you to connect multiple points on the page.

	Auto Connect multi points	Switches the Schematic page editor into the Auto connect mode and allows you to connect points to a bus.
	Place net alias	Places aliases on wires and buses. Equivalent to the Net Alias command on the Place menu.
	Place bus	Draws buses. SHIFT allows any angle drawing. Equivalent to the Bus command on the Place menu.
	Place junction	Places or deletes junctions. Equivalent to the Junction command on the Place menu.
	Place bus entry	Draws bus entries. Equivalent to the Bus Entry command on the Place menu.
	Place power	Places power symbols. Equivalent to the Power command on the Place menu.
	Place ground	Places ground symbols. Equivalent to the Ground command on the Place menu.
	Place hierarchical block	Places hierarchical blocks. Equivalent to the Hierarchical Block command on the Place menu.
	Place port	Places hierarchical ports on schematic pages. Equivalent to the Hierarchical Port command on the Place menu.
	Place pin	Places hierarchical pins in the selected hierarchical block. Equivalent to the Hierarchical Pin command on the Place menu.
	Place off-page connector	Places off-page connectors. Equivalent to the Off-Page Connector command on the Place menu.
	Place no connect	Places no-connect symbols on pins. Equivalent to the No Connect command on the Place menu.

	Place line	Draws lines. Equivalent to the Line command on the Place menu.
	Place polyline	Draws polylines. SHIFT allows any angle drawing. Equivalent to the Polyline command on the Place menu.
	Place rectangle	Draws rectangles. SHIFT constrains the shape to a square. Equivalent to the Rectangle command on the Place menu.
	Place ellipse	Draws ellipses. SHIFT constrains the shape to a circle. Equivalent to the Ellipse command on the Place menu.
	Place arc	Draws arcs. Equivalent to the Arc command on the Place menu.
	Place elliptical arc	Draws elliptical arcs. Equivalent to the Elliptical Arc command on the Place menu.
	Place bezier curve	Draws bezier curves. Equivalent to the Bezier Curve command on the Place menu.
	Place text	Places text. Equivalent to the Text command on the Place menu.
	Place IEEE Symbol	Places IEEE symbols. Equivalent to the IEEE Symbol command on the Place menu. This command is available in the Part Editor.
	Place Pin Array	Places pin arrays. Equivalent to the Pin Array command on the Place menu. This command is available in the Part Editor.
	Place Bundle	Opens the Bundle dialog box that allows you to create, modify, delete bus bundles. You can also import Xml bundle definitions into the current design.

6

PSpice Toolbar

The PSpice toolbar provides shortcuts for many of the most frequently used PSpice commands. This toolbar appears only if you have PSpice license and open a project that uses PSpice. The following table describes the icons on the toolbar.

Tool	Name	Description
	New simulation profile	Creates a new simulation profile. Equivalent to the New Simulation Profile command on the Pspice menu.
	Edit simulation profile	Opens simulation profile for editing. Equivalent to the Edit Simulation Profile command on the Pspice menu.
	Run PSpice	Runs PSpice simulation for active profile. Equivalent to the Run Pspice command on the Pspice menu.
	View simulation results	Shows simulation results for the active profile. Equivalent to the View Simulation Results command on the Pspice menu.
	Place voltage marker	Places voltage/digital level markers. Equivalent to the Voltage Level command on the Markers sub-menu of the Pspice menu.

	Place differential marker	Places voltage differential marker. Equivalent to the Voltage Differential command on the Markers sub-menu of the Pspice menu.
	Place current marker	Places marker showing current into a pin. Equivalent to the Current Into Pin command on the Markers sub-menu of the Pspice menu.
	Place power marker	Places marker showing power dissipation of a device. Equivalent to the Power Dissipation command on the Markers sub-menu of the Pspice menu.
	Enable bias voltage display	Enables display of bias voltages. Equivalent to the Enable Bias Voltage Display command on the Bias Points sub-menu of the Pspice menu.
	Toggle bias voltage display	Toggles display of voltage bias point value for selection. Equivalent to the Toggle Selected Bias Voltage Display command on the Bias Points sub-menu of the Pspice menu.
	Enable bias current display	Enables display of bias currents. Equivalent to the Enable bias current display command on the Bias Points sub-menu of the Pspice menu.
	Toggle bias current	Toggles display of current bias point value for selection. Equivalent to the Toggle Selected Bias Current Display command on the Bias Points sub-menu of the Pspice menu.
	Enable bias power display	Enables display of bias point quiescent power. Equivalent to the Enable bias power display command on the Bias Points sub-menu of the Pspice menu.
	Toggle bias power	Toggles display of bias point quiescent power for selection. Equivalent to the Toggle bias power command on the Bias Points sub-menu of the Pspice menu.

7

FPGA Toolbar

The FPGA toolbar offers a quick and easy way to simulate, synthesize, and compile vendor library projects. This toolbar is active only when you open a project of type Programmable Logic Design. The following table describes the icons on the toolbar.

Tool	Name	Description
	Simulate	Initiates setup for simulation of FPGA design.
	Synthesize	Initiates setup for synthesis of FPGA design.
	P & R	Starts the place-and-route tool specific to the target vendor of the programmable logic design.
	Compile vendor libraries	Initiate compilation of the vendor simulation models.

8

Part Manager Toolbar

The Part Manager toolbar offers a quick and easy way to perform common tasks. This toolbar is active only when you open OrCAD Capture CIS. The following table describes the icons on the toolbar.

Tool	Name	Description
	Link database part	Links a part from the database to a schematic part. Equivalent to the Link Database command on the Tools menu.
	Update All Part Status	Checks all parts against the parts database. Equivalent to the Update All Part Status command on the Tools menu.
	Bill of materials	Creates a bill of materials. Equivalent to the Standard command on the CIS Bill of Materials sub-menu of the Reports menu.
	Crystal Reports bill of materials	Generates bill of materials using Crystal Reports interface. Equivalent to the Crystal Reports command on the CIS Bill of Materials sub-menu of the Reports menu.
	Variant report	Generates a variant report. Equivalent to the Variant command on the Reports menu.

	Expand/ collapse tree item	Expands or collapses the tree view. Equivalent to the Expand/ collapse tree item command on the View menu.
	Show/Hide tree view	Shows or hides the tree view. Equivalent to the Show/Hide tree view command on the View menu.
	Resolve ambiguity	Resolves ambiguity for selected item. Equivalent to the Resolve ambiguity command on the Tools menu.

9

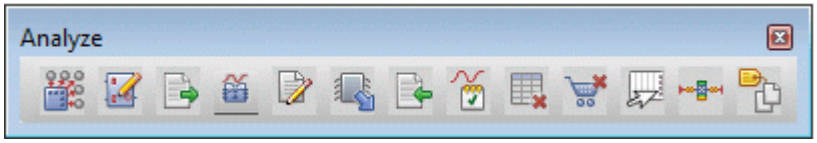
Search Toolbar

The Search toolbar offers a quick and easy way to perform common search tasks in Capture. This toolbar is active only when you open OrCAD Capture CIS. The following table describes the icons on the toolbar.

Tool	Name	Description
	Find	Searches for items based on the selection in the Project Manager and the options selected from the Search Options list. You can search at any level of the project hierarchy from design down to page. You can use the * (star) and ! (exclamation mark) wildcard characters.
	Search Options	Narrows the search filter down on various parameters such as match case and schematic component types.
	Previous	Searches the previous instance of the last search hit.
	Next	Searches the next instance of the last search hit.

10

SI Analysis Toolbar



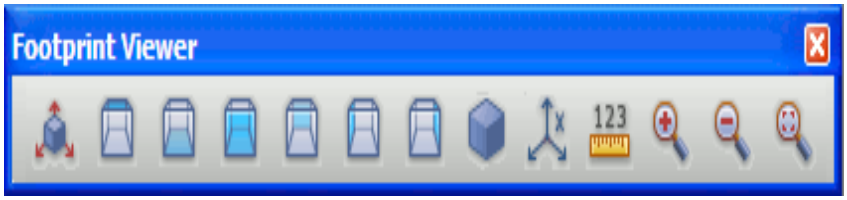
The SI Analysis toolbar for SI Analysis tasks provides shortcuts for many of the most frequently used SI Analysis commands. The following table describes the icons on the toolbar.

Tool	Name	Description
	Auto Assign Discrete SI Models	Assigns discrete SI models to the design parts.
	Validate SI Model Assignments	Validates the SI model assignments.
	Export SI Models Used	Exports the SI models used in the design into a DML library file.
	SI Library Setup	Opens the Library Setup (SI Analysis) window to set the DML libraries.
	SI Model Integrity	Opens Model Integrity with the DML file for the devices.

	Export Electrical Constraint sets	Opens the Export Electrical Csets from design window. You can export a topology file that can then be updated using SigXPlorer offline in a distributed design environment.
	Import Electrical Csets	Opens the Select Directory window where you can specify the directory from where you want to import an Electrical Cset. This allows you to import topology files that have been updated offline in a distributed design environment.
	Validate Electrical Cset Assignments	Opens the Validate ECSets in design window where you can selectively validate Electrical Csets in the design.
	Remove SI Model Assignments	Removes SI model assignments.
	Remove Electrical Cset Assignments	Opens the Remove ECSets from design window where you can selectively remove existing Electrical Csets from the design.
	Assign SI Model	Opens the SI Model Assignment window where you can assign models from the listed libraries. You can also autogenerate models
	Explore Signal	Opens SigXplorer to explore the signal.
	Associate Electrical Cset	Associates an Electrical Csets using a topology file.

11

Footprint Viewer Toolbar



The Footprint Viewer toolbar offers a quick and easy ways to view the footprint on the canvas in various different angles. It also provides access to the measure tool and the different zoom options in the footprint viewer. The following table describes the icons on the toolbar.

Tool	Name	Description
	Show Footprint Viewer	Toggle (show or hide) the 3-D Footprint Viewer display
	Show Top	Show top view of the PCB Footprint in the Viewer.
	Show Bottom	Show bottom view of the PCB Footprint in the Viewer.
	Show Front	Show front view of the PCB Footprint in the Viewer.
	Show Back	Show back view of the PCB Footprint in the Viewer.

	Show Left	Show left view of the PCB Footprint in the Viewer.
	Show Right	Show left view of the PCB Footprint in the Viewer.
	Show Isometric View	Show isometric view of the PCB Footprint in the Viewer.
	Show Axis	Toggle (show or hide) the display of the axis in the Footprint Viewer.
	Show Measure Tool	Toggle (show or hide) the display of the Footprint Viewer Measure Tool
	Zoom In	Zoom in to the view on the canvas
	Zoom Out	Zoom out of the view on the canvas
	Zoom Fit	Zoom to fit the PCB footprint on the canvas