OrCAD Capture™ for Windows®

User's Guide

MN-01-5090

First Edition 1 May 95

Technical support	(503) 671-9400
Bulletin board system	(503) 671-9401
Administration	(503) 671-9500
Fax	(503) 671-9501

OrCAD

9300 S.W. Nimbus Avenue
Beaverton, Oregon 97008 • USA

Contents

About this manual ix

Before you begin ix
Symbols and conventions ix
The keyboard ix
Text x

Part One — OrCAD Capture for Windows—the basics

Chapter 1 — First things first 3

Starting Capture 3
The Capture session frame 4

Chapter 2 — The Capture work environment 5

The design manager 5
The design structure pane 6
The browse pane 8
The schematic page editor 9
Viewing part instances and occurrences in the schematic page editor 9
The part editor 10
The session log 10
The toolbar 11
Displaying or hiding the toolbar 12
The tool palettes 13
Schematic page editor tool palette 13
Part editor tool palette 15
Displaying or hiding a tool palette 15
Using help and the online tutorial 15
Selecting and deselecting objects 16
Grouping objects 17
Editing objects 18
Editing properties 18
Using the spreadsheet editor to edit properties 19

Adding user-defined properties 20
Moving and resizing graphic objects 21
Undoing, redoing, and repeating an action 22

Chapter 3 **Design structure 23**
Flat designs 23
Hierarchical designs 24
Simple hierarchies 24
Complex hierarchies 24
Connecting designs 25
Hierarchical blocks 25
Hierarchical ports 26
Off-page connectors 26
An example: creating a simple hierarchy 27

Chapter 4 **Opening and saving designs 31**
Opening a design 31
Saving a design 34
Closing a design and quitting Capture 35

Part Two **Creating designs**

Chapter 5 **Setting up your design 39**
Defining your preferences 40
Defining colors 41
Controlling the grid 42
Setting pan and zoom 43
Defining selection options 44
Setting miscellaneous options 45
Setting up your design template 47
Setting up fonts for new designs 48
Defining title block information 49
Setting up the schematic page size for new designs 51
Defining the grid reference 53
Defining the default hierarchy option for new designs 54
Setting up compatibility with OrCAD's Schematic Design Tools 55
Changing properties of individual designs 56
Assigning fonts 56
Defining hierarchy 56
Setting up compatibility with OrCAD's Schematic Design Tools 56

Viewing invisible power pins without isolating them 57
Changing properties of individual schematic pages 58
Changing page size 58
Setting up new grid references 58
Viewing miscellaneous schematic page properties 59

Chapter 6 **Placing, editing, and connecting parts and electrical symbols 61**
Placing and editing parts 62
Placing parts 63
Editing parts 67
Placing and editing power and ground symbols 69
Placing power and ground symbols 69
Editing power and ground symbols 72
Placing and editing hierarchical blocks 73
Placing hierarchical blocks 73
Editing hierarchical blocks 75
Placing and editing hierarchical ports 76
Placing hierarchical ports 76
Editing hierarchical ports 79
Placing and editing off-page connectors 80
Placing off-page connectors 80
Editing off-page connectors 82
Placing and editing wires and buses 84
Placing wires 84
Editing wires 85
Placing buses 86
Editing buses 86
Placing bus entries 87
Editing bus entries 87

Chapter 7 **Adding and editing graphics and text 89**
Drawing tools 89
Drawing lines 90
Drawing rectangles and squares 91
Drawing circles and ellipses 92
Drawing arcs 93
Drawing polylines and polygons 94
Adding fill to an object 95

Mirroring an object 95
Rotating an object 95
Cutting an object 95
Copying an object 95
Pasting an object 96
Deleting an object 96
Placing bitmaps 96
Placing text 97
The text bounding box 98
Deleting text 98
Adding to text you've already typed 98
Finding text 99
Replacing text 99
Importing text 100
Exporting text 100
Character formatting 101
About screen fonts 101

Chapter 8 **Changing your view of a schematic 103**

Zooming 103
Zooming to a specified scale 104
Other viewing options 105
Jumping to a new location 106
Jumping to a specific grid reference 107
Jumping to a marked location 107
Displaying the grid and grid references 108
Finding an object 109

Chapter 9 **Printing and plotting 111**

Printing or plotting schematics or schematic pages 111
Printing or plotting a part or package 112
Previewing print output 113
Scaling a print or plot 113
Plotter pen colors 114
Special considerations for plotting 114

Part Three Libraries and parts

Chapter 10 About libraries and parts 117

Libraries 117
Parts 119
Part instances 120
    The design cache 120
Primitive and nonprimitive parts 122

Chapter 11 Creating and editing parts 123

Creating a new part 124
    Defining a part 124
    Attaching a schematic to a part 128
    Adding graphics, text, and IEEE symbols to a part 129
    Placing pins on the part 130
    About power and ground pins 135
Editing an existing part 136
    In a library 136
    On a schematic page 137
Viewing parts in a package 138
Viewing a part's convert 139

Part Four Processing your design

Chapter 12 About the design process 143

Tools overview 144
Choosing between logical and physical view 145
Working in both logical and physical view 146

Chapter 13 Preparing to create a netlist 147

Updating part references 147
Updating properties 150
Checking for design rules violations 153
Swapping gates and swapping pins 161

Chapter 14 Creating a netlist 165

Using the Create Netlist tool 165
Netlist format files 167
    Types of netlist format files 167
    Creating custom netlist format files 167

Netname resolution 168
Creating a netlist for use with OrCAD's PLD 386+ 169

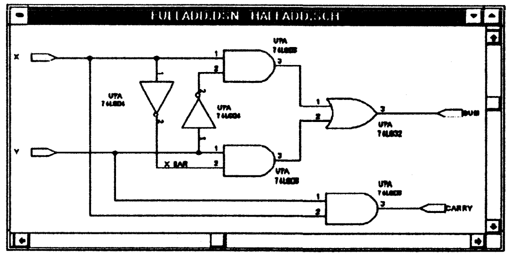
Chapter 15 **Creating reports 171**
Creating a bill of materials 171
Creating a cross reference report 174

Chapter 16 **Exporting Capture data 175**
Exporting properties to a tab-delimited file 175
Editing a property file 177
Importing properties 179
Extracting PLD source code 180

Glossary 185

Index 191

About this manual



The *OrCAD Capture for Windows User's Guide* is a comprehensive manual that contains all of the procedures you need to work with OrCAD Capture for Windows. To help you learn and use Capture efficiently, this manual is organized by tasks, beginning with the most common schematic design tasks (parts one and two), and moving on to more advanced Capture features (parts three and four). Many of the skills described in this manual are also covered in the online tutorial, *Learning Capture®*.

Before you begin

Before you can use Capture, you must install Microsoft Windows on your computer, then install Capture. For information on installing Windows, see your Windows documentation.

To install Capture, follow the printed installation instructions that accompany Capture.

Symbols and conventions

OrCAD printed documentation uses a few special symbols and conventions.

The keyboard

- The keys on your keyboard may not be labeled exactly as they are in this manual. All key names are shown using small capital letters. For example, the Control key is shown as CTRL; the Escape key is shown as ESC.
- Keys are frequently used in combinations or sequences. For example, SHIFT+F1 means to hold down the SHIFT key while pressing F1. ALT, F, A, means to press and release each of these keys in order: first ALT, then F, then A.
- *Arrow keys* is the collective name for the UP ARROW, DOWN ARROW, LEFT ARROW, and RIGHT ARROW keys.
- To choose a command from a menu, you can use the mouse or press a key combination. For example: from the File menu, choose Open (ALT, F, O).

Text

- Specific text you are to type is shown in bold. For example, if the manual says to type ***.dsn**, you type an asterisk, a period, and the lowercase letters "dsn." What you type is always shown in lowercase letters, unless it must be typed in uppercase letters to work properly.
- Placeholders for items such as filenames that you must supply are shown in italic. For example, when the manual says to type **cd** *directory_name*, you type the letters "cd" followed by a space and the name of a directory. For a directory called CIRCUITS, you would type **cd circuits**.
- Examples of syntax, netlist output, and PLD source code are displayed in monospace font—for example, `/N0001 U1(8) U2(1);`.

OrCAD Capture for Windows—the basics

Part One contains the basic information you need to get started in OrCAD Capture for Windows. It provides an overview of the structure and work environment of Capture, and tells how to open and save designs in Capture.

Part One includes these chapters:

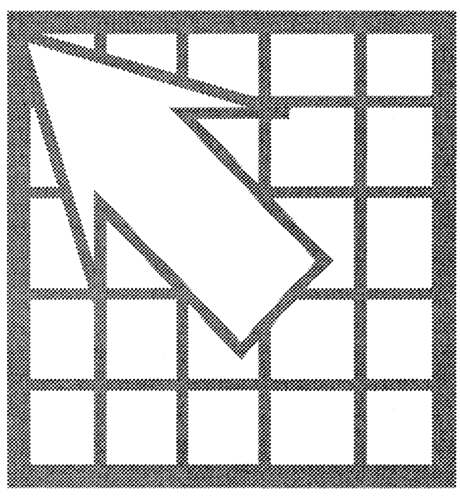
Chapter 1: First things first describes how to start Capture.

Chapter 2: The Capture work environment describes the things you'll need to know to find your way around in Capture. It shows the windows you'll see in Capture: the design manager, the schematic page editor, the part editor, and the session log. It also introduces you to the toolbar and tool palettes, and general Capture concepts such as selecting and editing objects, and undoing and repeating actions.

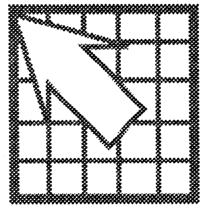
Chapter 3: Design structure describes the different types of designs that Capture supports: flat, simple hierarchical, and complex hierarchical. It introduces the electrical objects used to create these types of designs, and provides an example of how to create a simple hierarchy.

Chapter 4: Opening and saving designs shows how to open a design and navigate the schematics and schematic pages in a design. It also shows how to save a design, or a portion of a design, such as an individual schematic page.

Part One



Chapter 1



First things first

This chapter describes how to start OrCAD Capture for Windows.

Starting Capture

The Capture installation process automatically creates the OrCAD Capture program group and the Capture application icon, as well as icons for other things related to Capture (such as the online tutorial and help).

To start Capture

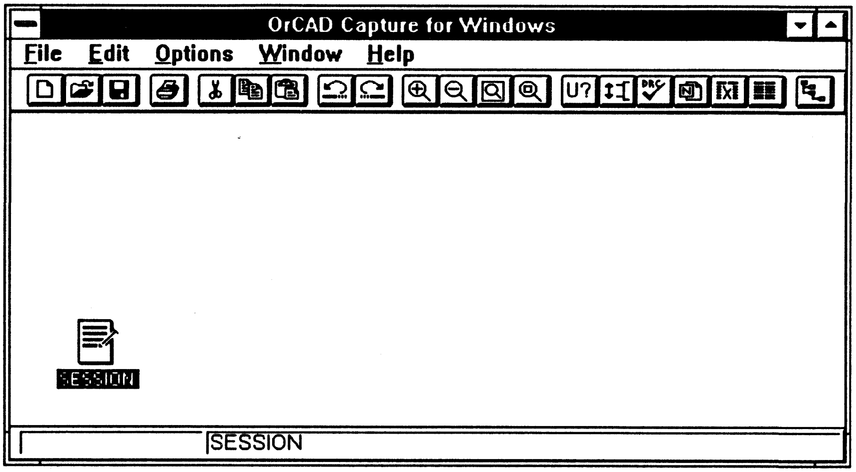
You can start Capture using either the mouse or the keyboard. If Windows is not already running, type **win** at the DOS command prompt before you perform the steps below.

Using the mouse	*Using the keyboard*
➥ Double-click the OrCAD Capture icon.	➥ If the OrCAD Capture icon is highlighted, press ENTER.

Tip You can tell if a program group is active by looking at it. If a program group is active, its title bar and an icon in the group are highlighted.

The Capture session frame

Once you start Capture, you see the Capture *session frame*. You do all your schematic design and processing from within this window.



The Session icon in the lower left portion of the window opens another window that provides information about everything you have done so far in Capture. Detailed information about this—and the other windows in Capture—is given in chapter 2.

In Capture, each *document* that you open is in a separate window. You may open as many windows as your computer's resources can handle. If you wish to work with three schematic pages, or three parts, each is available in its own window. If you need to work simultaneously with several designs, you can open them all, and each will have its own design manager window.

Depending on which type of window you have *active* (an active window is one whose title bar is highlighted), certain buttons on the toolbar and certain items on the menus may be unavailable, since Capture only lets you perform tasks and use tools based upon the type of window that is active. Also, the menus and menu choices vary slightly among the different types of windows.

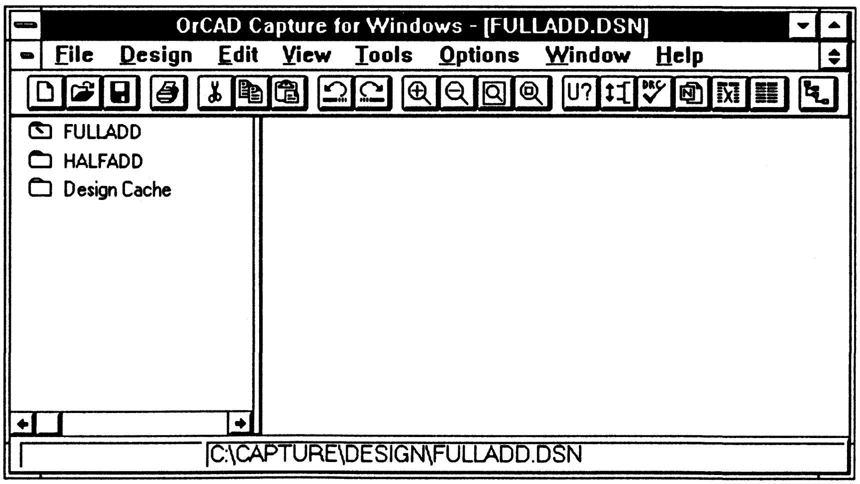
The Capture work environment

This chapter describes the things you need to know to find your way around in Capture. It shows the windows you'll see in Capture: the design manager, the schematic page editor, the part editor, and the session log. It also introduces you to the toolbar, tool palettes, and general Capture concepts such as selecting and editing objects and undoing and repeating actions.

The design manager

The design manager window provides a graphical display of the schematics and schematic pages in a design. It also provides tools—such as the tools to update part references, check for design rules violations, create netlists, and generate reports—that operate on an entire design or on a portion of a design.

Note The design manager is also used to manage libraries and the parts they contain. This is covered in detail in *Chapter 10: About libraries and parts*. The rest of this section discusses how the design manager is used to view designs.



Each open design has its own design manager window. You can move or copy schematics, schematic pages, and parts between designs and libraries by dragging them from one design manager window to another. If you close a design manager window, you close the design.

The design manager has two panes: the left pane is the *design structure pane*, and the right pane is the *browse pane*. These panes are described later in this section.

Tip—About designs and schematics A design can consist of a single schematic or a number of schematics. A schematic "contains" schematic pages in a relationship similar to the relationship between a directory and the files it contains. Files are contained by a directory; schematic pages are contained by a schematic.

A schematic page provides a graphical description of the electrical connectivity of a design. It is made up of parts, wires, and other electrical symbols such as hierarchical blocks, hierarchical ports, and off-page connectors. The schematic page may also contain borders, title blocks, text, and graphics.

The design structure pane

The left half of the design manager window is the design structure pane. It provides a graphical representation of a design's structure and a list of parts in the *design cache.* The design structure pane operates similar to the Windows File Manager, in that you can expand or collapse the structure you are viewing by double-clicking the left mouse button on an object. For example, if you double-click the left mouse button on a schematic, it opens, and icons for each schematic page in the schematic display. Then, if you double-click the left mouse button on a schematic page, that schematic page opens in the schematic page editor. Or, if the page is already open, its window comes forward and becomes active.

Tip Schematics are represented by file folder icons; schematic pages are represented by page icons. The *root schematic* for a design includes a backslash in its icon (see the illustrations in *Views—logical and physical* later in this chapter).

When a design manager window is active, any selected schematics or schematic pages limit the scope of various commands, such as the Find and Browse commands on the design manager's Edit menu, the Print command on the design manager's File menu, and the various tools on the Tools menu.

The logical view of the design has a design cache. The design cache stores all the parts in the design. It also stores other objects in the design, such as hierarchical ports and title blocks. You can update the parts in the design cache with their original library parts using the Update Cache command on the Edit menu. This command goes to the source library for each selected part and copies the part from the library. This way, you can be sure you have the current versions of any parts in your design, or you can override any changes you have made to a library part.

Schematics and schematic pages can be moved to another location by dragging and dropping—even across design manager windows. To copy rather than drag, press and hold the CTRL key while you drag the document. Note that if a schematic page is open, you cannot drag it to a different location.

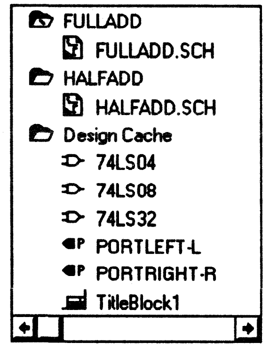
Views—logical and physical

Designs can be displayed in one of two views—logical or physical—corresponding to the Logical or Physical commands on the design manager's View menu.

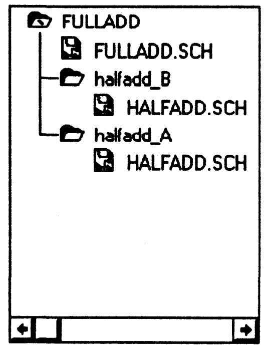
In logical view, the design displays as a series of independent schematics. Any operations performed in logical view affect the part *instances* in the design, and thus carry forward to the *occurrences* of part instances in physical view.

In physical view, the design displays as an "unfolded" view of the design's hierarchy. You see each occurrence of each schematic page in this mode. In the examples at right, you see only one instance of the HALFADD.SCH schematic page in logical view. When you change to physical view, however, you see each occurrence of HALFADD.SCH.

In physical view, you can only edit part properties. You cannot place or edit parts, graphics, or text. Any changes you make to part properties apply to the specific part occurrences—they are not carried back to the part instances in logical view.



Logical view.



Physical view.

The browse pane

The browse pane occupies the right side of the design manager window. It displays the items selected using the Browse command on the Edit menu, or the results of any searches done using the Find command on the Edit menu.

Both the Browse and Find commands apply to all the schematics and schematic pages you select in the design structure pane. If you double-click on one of the items listed in the browse pane, the schematic page that contains that object displays in the schematic page editor window, with the object selected.

FULLADD.DSN

Reference	Value	Source Part	Source Library	Page	Schematic
U?	74LS04	74LS04	\CAPTURE\LI...	HALFADD.S...	HALFADD
U?	74LS04	74LS04	\CAPTURE\LI...	HALFADD.S...	HALFADD
U?	74LS08	74LS08	\CAPTURE\LI...	HALFADD.S...	HALFADD
U?	74LS08	74LS08	\CAPTURE\LI...	HALFADD.S...	HALFADD
U?	74LS08	74LS08	\CAPTURE\LI...	HALFADD.S...	HALFADD
U?	74LS32	74LS32	\CAPTURE\LI...	FULLADD.SCH	FULLADD
U?	74LS32	74LS32	\CAPTURE\LI...	HALFADD.S...	HALFADD
halfadd_A	HALFADD.S...			FULLADD.SCH	FULLADD
halfadd_B	HALFADD.S...			FULLADD.SCH	FULLADD

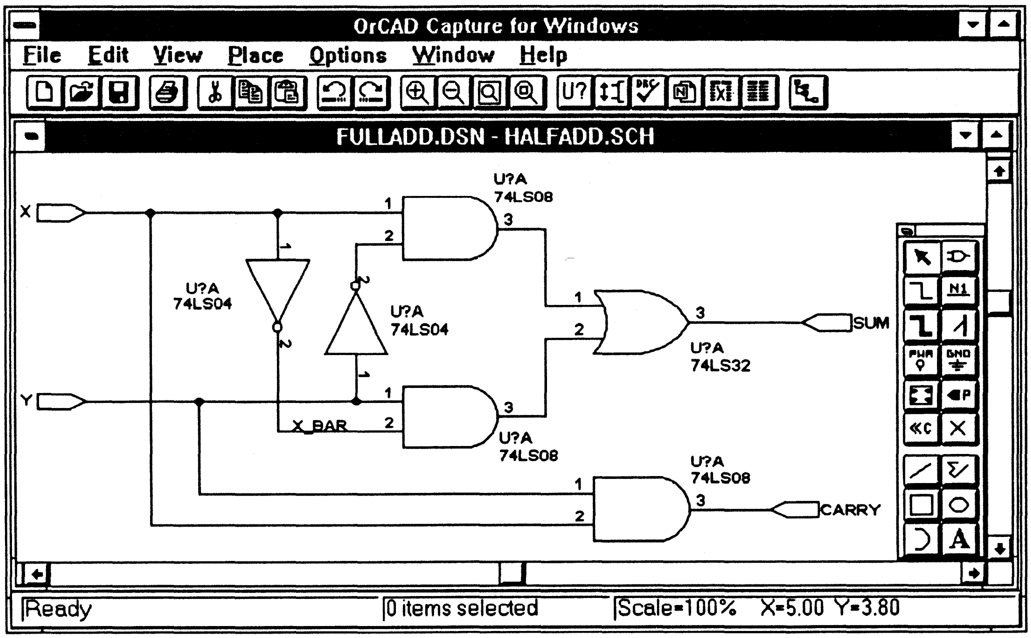
Browse pane in the design manager.

When you browse a design or library, you can sort the results using the buttons at the top of the browse pane, as shown in the figure above. When you choose one of these buttons, Capture sorts the selections alphabetically according to the value of the corresponding property. Each type of object offers a different set of buttons. For example, the sort fields for parts are part reference, part value, source part, source library, schematic page, and schematic, but the sort fields for nets are net alias name, net name, schematic page, and schematic.

The schematic page editor

The schematic page editor window is used to display and edit schematic pages. You can place parts, wires, buses, and even draw graphics. The schematic page editor has a tool palette that contains tools to draw and place everything you need to create a schematic. You can print a schematic page from within the schematic page editor.

See also See chapters 5, 6, 7, and 8 for complete information about using the schematic page editor.



Schematic page editor window.

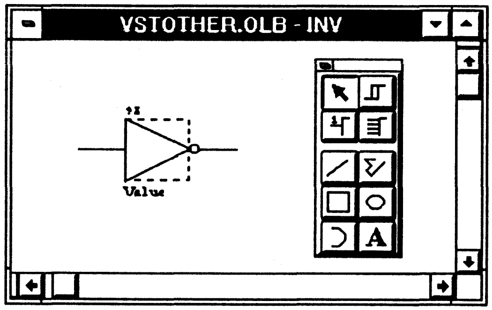
Viewing part instances and occurrences in the schematic page editor

As described in the *Design manager* section, designs have two views: logical and physical. When you look at a schematic page in logical view, you see part instances. All parts and other objects can be edited, moved, or deleted.

When you look at a schematic page in physical view, you see occurrences of part instances. All parts and objects are dimmed to indicate that you can't edit, move, or delete them. You can, however, edit the properties associated with an occurrence of a part or object. Any changes that you make to the properties of a part occurrence stay with the occurrence. This means that you won't be able to see the properties when you change the view to logical. But if you change back to physical view, you'll again be able to see the properties that apply to specific part occurrences.

See also For more information about logical and physical view, see *Views—logical and physical* in *Design structure pane* earlier in this chapter.

The part editor



The part editor window is used to create and edit parts. From the View menu of the part editor you can choose either Part or Package. In Part view you can:

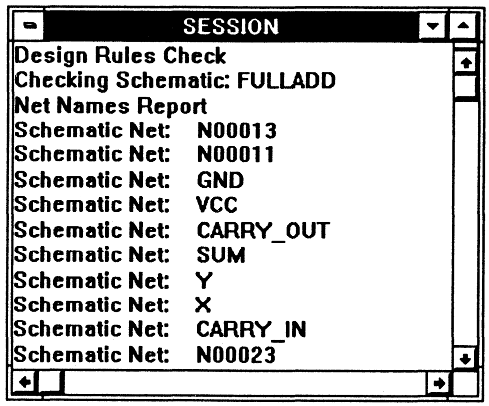
- Create and edit parts and symbols, and store them in new or existing libraries.
- Create and edit power and ground symbols, off-page connector symbols, hierarchical port symbols, and title blocks.

Part view has a tool palette that provides graphical tools to draw parts and symbols, as well as tools to place pins on parts.

Package view shows you the entire *package*. You cannot edit parts in this view, but you can select parts to edit.

See also See *Chapter 10: About libraries and parts* for complete definitions of parts and packages. See *Chapter 11: Creating and editing parts* for a complete description of the part editor.

The session log



The session log displays a listing of all events that have occurred during the current Capture session. This includes messages from tools.

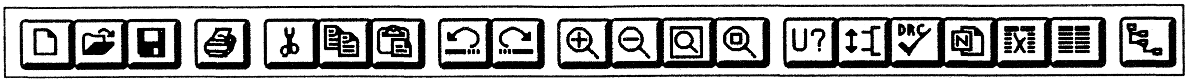
You can search for information in the session log using the Find command on the Edit menu. You can also save the contents of the session log to a file. The default filename is SESSION.TXT.

The information in the session log is useful when working with OrCAD's technical support staff to solve technical problems.

Tip To delete the contents of the session log, press CTRL+DEL.

The toolbar

The toolbar gives you easy access to the most frequently used Capture commands. By clicking a button, you can quickly perform a task. If a tool button is dimmed, you can't perform that task in the current situation. For example, you can only use the Update Part References, Design Rules Check, and Create Netlist tools in the design manager, so those tool buttons are dimmed in the schematic page editor.

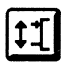
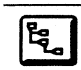


Some tools work on whatever you have selected, while others give you a choice of either operating on what is selected or expanding the scope to the entire design. For example, if you select one schematic page from a design, then choose Update Part References, a dialog box appears with options to Update selection (which only updates the part references on the selected schematic page) or Update entire design (which updates all part references in the design, regardless of which individual page you have selected).

The following table briefly summarizes the tools on the toolbar. The tasks that these tools perform are described throughout this manual.

Tool	*Name*	*Description*
	New	Create a new document of the same type as the active document. Similar to the New command on the File menu.
	Open	Open an existing design or library. Similar to the Open command on the File menu.
	Save	Save the active schematic page or part. Equivalent to the Save command on the File menu.
	Print	Print the active schematic page or part. Equivalent to the Print command on the File menu.
	Cut	Remove the selected objects from your document and place them on the Clipboard. Equivalent to the Cut command on the Edit menu.
	Copy	Copy the selected objects to the Clipboard. Equivalent to the Copy command on the Edit menu.
	Paste	Paste the contents of the Clipboard into your document at the location of the pointer. Equivalent to the Paste command on the Edit menu.
	Undo	Undo the last command performed, if possible. Equivalent to the Undo command on the Edit menu.
	Redo	Redo the last command performed, if possible. Equivalent to the Redo command on the Edit menu.

Tools on the toolbar (page 1 of 2).

Tool	*Name*	*Description*
	Zoom In	Zoom in to see a closer, enlarged view. Equivalent to choosing Zoom and In from the View menu.
	Zoom Out	Zoom out to see more of your document. Equivalent to choosing Zoom and Out from the View menu.
	Zoom Area	Specify an area of the schematic page or part to enlarge to fill the entire window. Equivalent to choosing Zoom and Area from the View menu.
	Zoom All	View the entire document. Equivalent to choosing Zoom and All from the View menu.
	Update Part References	Assign part references to parts in the selected schematic pages. Equivalent to the Update Part References command on the Tools menu.
	Gate and Pin Swap	Back annotate the selected schematic pages. Equivalent to the Gate and Pin Swap command on the Tools menu.
	Design Rules Check	Check for design rule violations in the selected schematic pages. Equivalent to the Design Rules Check command on the Tools menu.
	Create Netlist	Create a netlist from the selected schematic pages. Equivalent to the Create Netlist command on the Tools menu.
	Cross Reference	Create a cross reference report of the selected schematic pages. Equivalent to the Cross Reference command on the Tools menu.
	Bill of Materials	Create a bill of materials from the selected schematic pages. Equivalent to the Bill of Materials command on the Tools menu.
	Design Manager	Display a design manager window for the active document, providing an overview of design contents. Equivalent to choosing a design manager window by number from the Window menu.

Tools on the toolbar (page 2 of 2).

Displaying or hiding the toolbar

If you need more room on the screen to view a schematic page or a part, you can hide the toolbar and then display it again later when you need it.

To display or hide the toolbar

➥ From the View menu, choose Toolbar (ALT, V, T).

The tool palettes

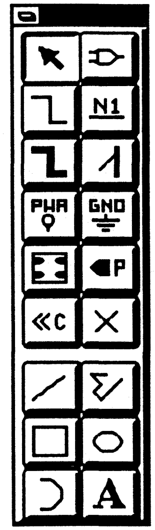
There are two tool palettes: one for the schematic page editor window, and one for the part editor window. While many of the tools on these two tool palettes are the same, each tool palette has some unique tools.

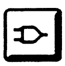
After you choose a tool (and, in the case of some tools, after you also respond to the accompanying dialog box necessary to activate the tool), you can press the right mouse button to display a short pop-up menu. This pop-up menu has options related to the tool you are using. For example, if you are placing text, the menu has options to zoom in, zoom out, and go to a different location.

Schematic page editor tool palette

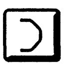
The schematic page editor tool palette has two groups of tools. The top tools are electrical. The bottom tools are drawing tools, and are used to create things that don't have electrical connectivity, or more specifically, things that don't show up in a netlist.

The following table briefly summarizes the tools on the schematic page editor tool palette. For detailed descriptions of each of these tools, see *Chapter 6: Placing, editing, and connecting parts and electrical symbols* and *Chapter 7: Adding and editing graphics and text.*



Tool	*Name*	*Description*
	Selection tool	Selects objects. This is the normal mode.
	Part selector	Selects parts from a library for placement on a schematic page. Equivalent to the Part command on the Place menu.
	Wire	Draws wires. Press SHIFT to draw wires that are non-orthogonal (not a multiple of 90°). Equivalent to the Wire command on the Place menu.
	Bus	Draws buses. Press SHIFT to draw buses that are non-orthogonal (not a multiple of 90°). Equivalent to the Bus command on the Place menu.

Tools on the schematic page editor tool palette (page 1 of 2).

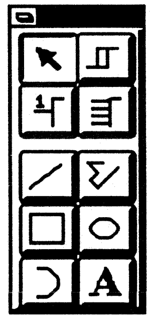
Tool	*Name*	*Description*
	Net Alias	Places aliases on wires and buses. Equivalent to the Net Alias command on the Place menu.
	Bus Entry	Places a bus entry. Equivalent to the Bus Entry command on the Place menu.
	Power	Places a power symbol. Equivalent to the Power command on the Place menu.
	Ground	Places a ground symbol. Equivalent to the Ground command on the Place menu.
	Hierarchical Block	Draws hierarchical blocks. Equivalent to the Hierarchical Block command on the Place menu.
	Hierarchical Port	Places hierarchical ports. Equivalent to the Hierarchical Port command on the Place menu.
	Off-Page Connector	Places off-page connectors. Equivalent to the Off-Page Connector command on the Place menu.
	No Connect	Places no-connect symbols on pins. Equivalent to the No Connect command on the Place menu.
	Line tool	Draws lines. SHIFT constrains lines to multiples of 90°. Equivalent to the Line command on the Place menu.
	Polyline tool	Draws polylines (lines with multiple segments). SHIFT constrains lines to multiples of 90°. Equivalent to the Polyline command on the Place menu.
	Rectangle tool	Draws rectangles and squares. SHIFT constrains the shape to a square. Equivalent to the Rectangle command on the Place menu.
	Ellipse tool	Draws ellipses and circles. SHIFT constrains the shape to a circle. Equivalent to the Ellipse command on the Place menu.
	Arc tool	Draws arcs. Equivalent to the Arc command on the Place menu. After the arc is drawn, using the pointer tool with SHIFT constrains the arc to the same radius.
	Text tool	Places text. Equivalent to the Text command on the Place menu.

Tools on the schematic page editor tool palette (page 2 of 2).

Part editor tool palette

The part editor tool palette has the basic drawing tools that you can use to create part body graphics and to add pins.

The tools that are unique to the part editor are described in the table below. For detailed descriptions of these tools, see *Chapter 11: Creating and editing parts.*



Tool	*Name*	*Description*
	Pin tool	Places pins on a part. Equivalent to the Pin command on the Place menu.
	Pin array tool	Places multiple pins on a part. Equivalent to the Pin Array command on the Place menu.
	IEEE symbol tool	Places IEEE symbols on a part. Equivalent to the IEEE Symbol command on the Place menu.

Tools on the part editor tool palette.

Displaying or hiding a tool palette

Like the toolbar, you can hide a tool palette and then display it again later when you need it.

To display or hide the tool palette

➥ From the View menu, choose Tool Palette (ALT, V, P).

Using help and the online tutorial

Capture's online help includes information to help OrCAD's Schematic Design Tools users become familiar with Capture. Topics covered include:

- Simple steps for converting SDT 386+ and Release IV designs and libraries to Capture.
- Mapping SDT objects, terms, and commands to their Capture equivalents.

Selecting and deselecting objects

Once you select an object, you can perform many operations on it, including moving, copying, cutting, mirroring, rotating, resizing, or editing. You can also select multiple objects and edit them, or group them into a single object. Grouping objects is a convenient way to maintain the relationship among several objects while you move them to another location.

This section describes the different ways to select individual objects and groups of objects. These selection methods work both in the schematic page editor and in the part editor.

Tip To select a part, you click within the part itself. To select a graphical object, you click on an outside edge of the object (it is easier to do this if you zoom in).

To select an object

➥ Position the pointer on the object and click the left mouse button. The object displays in the *selection color.*

Tip To change the selection color: from the Options menu, choose Preferences, then choose the Colors tab. Click the left mouse button over the Selection color. Select a new color from the color palette window, choose the OK button to dismiss the color palette, then choose the OK button to dismiss the dialog box.

To select multiple objects

➥ For each object to select, position the pointer on the object and press CTRL while you click the left mouse button. Every object in the selection set displays in the selection color.

Tip You can quickly and easily edit the properties of a group of homogeneous objects using the spreadsheet editor. See *Editing objects* in this chapter.

To select all objects in an area

1 From the tool palette, choose the selection tool. 

2 Click on an area where there are no objects or parts. Move the pointer to one corner of the area to select. Press and hold the left mouse button while you drag the mouse to the opposite corner, then release the left mouse button. Every object in the selection set displays in the selection color.

Tip You can control whether the selection set includes all objects intersected by your selection rectangle or only those objects entirely inside the selection rectangle. From the Options menu, choose Preferences, then click on the Select tab. Select one of the Area Select options, then choose the OK button.

To select all objects on a schematic page or part

➥ From the Edit menu, choose Select All. All objects display in the selection color.

Note The selection set behaves as if it is one object. This means that you can perform operations such as moving, copying, cutting, deleting, mirroring, or rotating the entire selection set.

To deselect objects

➥ Click on an area where there are no objects or parts. All selected objects become unselected. Note that a part occupies a rectangular area encompassing all its graphics. This means that a part may occupy a larger area than is initially apparent.

To remove one object from a selection set

➥ Place the pointer over the object and press CTRL while you click the left mouse button.

Note To select from objects stacked one atop another, position the pointer over the objects and press the TAB key while you click the left mouse button. This cycles through the objects in the stack.

Grouping objects

Use the Group command on the Edit menu to group multiple objects into one selectable object. This is a convenient way to maintain the relationship among several objects while moving them to another location. You can *nest* groups, meaning a group can contain other groups as well as objects. The Group command is only available when multiple objects are selected. Objects remain grouped until you ungroup them or close all windows on the schematic page or part that contains them.

To group multiple selected objects

1. Select the objects you want to group, as described in the previous section.
2. From the Edit menu, choose Group. You can move the objects as a group.
3. When you are through manipulating the objects as a group, you can ungroup them. From the Edit menu, choose Ungroup.

Editing objects

Each object has a set of properties, and you can edit the value associated with each property. For some objects (parts, packages, part instances, hierarchical blocks, pins, nets, and buses), you can add your own *user-defined properties*. Note that you cannot add user-defined properties to graphic objects, bookmarks, IEEE symbols, no-connect objects, net aliases, power and ground symbols, off-page connectors, or bus entries.

Properties can be used to store information, such as a part's value or reference. They can also be used to define the appearance of an object. For example, properties are used to define the color, line weight, and fill of graphic objects.

For some objects—such as wires, buses, lines, ellipses, rectangles, and so on—you can edit the object's size and shape by clicking on it and dragging its resize handles. To quickly change a rectangle to a square, or an ellipse to a circle, you can hold down the SHIFT key and drag the object's resize handles.

Editing properties

Capture uses properties to describe objects. For example, a capacitor has characteristics that define its type, its height in millimeters, and its color. In Capture, type, height, and color are property names, while ceramic, 6, and brown are property values.

To edit an object's properties

Using the mouse	*Using the keyboard*
➥ Double-click on the object.	1 Click once on the object to select it.
	2 From the Edit menu, choose Properties.

A dialog box containing properties for the object displays. Edit the properties in the dialog box. When you are done, choose the OK button.

See also Each type of object has its own set of properties. For information about the specific properties of each object, see Capture's online help.

Using the spreadsheet editor to edit properties

The properties of a group of homogeneous objects can be edited using the spreadsheet editor. In the spreadsheet editor, you can edit:

- Multiple part instances
- Pins on part instances
- Hierarchical ports
- Wires
- Buses
- Nets
- Off-page connectors
- DRC markers
- Bookmarks

To edit an object's properties using the spreadsheet editor

1. Select the group of objects, as described earlier in this chapter.
2. From the Edit menu, choose Properties. Note that if the objects in the selection set are not homogeneous, the Properties command is unavailable.

   The spreadsheet editor displays. You can do these operations in the spreadsheet editor:

   - Click the left mouse button to select a cell for copying or pasting. Double-click to select the cell for editing.
   - Click on a row or column heading to select the entire row or column.
   - With one or more cells selected for copying or pasting, press and hold the SHIFT key while you click on an adjacent cell to extend the selection set.
   - Choose the New button to display the New Property dialog box. Enter the property name. If you want all members of the current selection set to have a particular value, enter the value also.
3. Choose the OK button to close the spreadsheet editor.

Tip To assign a cell's value to all of the cells within the same column in the spreadsheet editor, select a cell's value, click on the Copy command, select the entire column, and click on the Paste command.

Adding user-defined properties

You can add user-defined properties to electrical objects. For example, if you want your schematic to include the supplier and price-per-hundred for all parts, you can create user-defined properties to contain this information. You can add as many user-defined properties to electrical objects as you like, and you can remove them when you find that you don't need them. Graphic objects do not allow user-defined properties, but you can add user-defined properties to part instances and part occurrences. For example, you can add a user-defined property for price quotes to a part instance, and you can define a property for power expenditure in a particular part occurrence.

Because you can add user-defined properties to electrical objects, the properties of a library part are not necessarily identical to those of a part instance or occurrence. You can, however, add a user-defined property, such as a stock number, to a library part. If you know that a property will have the same value for every instance or occurrence, you can add the value to the library part: the property's value will then be present on all instances and occurrences of that part, unless you change or remove it.

You can add user-defined properties to these objects:

- Parts in libraries
- Part instances and occurrences
- Hierarchical blocks
- Pins
- Nets (not wires—the property is on the net)
- Buses

You cannot add user-defined properties to graphic objects, bookmarks, IEEE symbols, no-connect symbols, net aliases, power and ground symbols, off-page connectors, or bus entries.

To add a user-defined property

1. Select the object.
2. From the Edit menu, choose Properties.
3. Choose the User Properties button.
4. In the User Properties dialog box that displays, choose the New button. The New Property dialog box displays.
5. Enter a name for the new property.
6. Enter a value for the new property.
7. Choose the OK button to close the New Property dialog box.
8. Choose the OK button twice.

Moving and resizing graphic objects

Before you can move or resize a graphic object, you must first select it. A selected object has resize handles that you use to change the size of the object. To quickly change a rectangle to a square, or an ellipse to a circle, you can hold down the SHIFT key and drag the object's resize handles.

To resize and move objects

1. Select the object to resize or move.
2. To resize the object, press the left mouse button on a resize handle, and drag the resize handle until the object is the size you would like it. Release the mouse button.
   or
   To move the object, press the left mouse button anywhere on the object—except on a resize handle—and drag the object until it is where you want it. Release the mouse button.

Tip If you move electrically connected items, they will "rubberband" or stretch the electrical connections to maintain connectivity. To prevent this, either group the items before you move them, or press and hold ALT while you move them.

3. To deselect the object, click in an area where there are no parts or objects.

See also For descriptions of other ways to manipulate objects, see *Chapter 7: Adding and editing graphics and text.*

Undoing, redoing, and repeating an action

If you make a mistake, you can use the Undo command to undo your action. To repeat an edit action, you can use the Repeat command. For example, you might move a selected object five grid units, then realize you also need to move a different object the same distance. Select the second object, then from the Edit menu, choose the Repeat command.

You can use the Undo, Redo, and Repeat commands with the following actions:

- Placing objects
- Deleting objects (except for the Repeat command)
- Copying objects
- Moving objects
- Resizing objects
- Rotating objects
- Mirroring objects

To undo an action

➥ From the Edit menu, choose Undo.

To undo an Undo command

➥ From the Edit menu, choose Redo.

To repeat a command

1 Perform the command once.

2 From the Edit menu, choose Repeat.

Tip You can use the Repeat command to align objects or to quickly create repetitive structures such as buses.

To repeat a Place operation

1 Place an object on a schematic page.

2 Press CTRL and drag the object to a new location. This creates a copy of the object. Leave the object selected.

3 From the Edit menu, choose Repeat. The pointer repeats the relative move in step 2 and an additional object is placed.

Chapter 3

Design structure

Many designs can fit on one schematic page. Some designs, however, are too large for even the biggest page, and even if a very complex design could fit on one sheet, there are good reasons for dividing it:

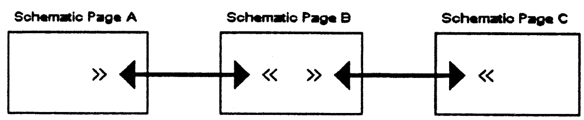
- To fit at full scale on your printer's page.
- To partition a design so that several people can work on it at once.
- To develop the design using a top-down approach. That is, you may want to begin with a block diagram in which each block represents a major function and then construct more detailed diagrams for each of the blocks.
- To organize your design by functional parts.

Capture offers two ways of handling multiple-page designs: flat designs and hierarchical designs.

Flat designs

Flat designs are practical for small designs with few schematic pages. A flat design is a schematic structure in which the output lines of one schematic page connect laterally to the input lines of another page in the same schematic through objects called *off-page connectors*. A flat design has no hierarchy (no hierarchical blocks or ports; no parts with attached schematics).

All schematic pages in a flat design are contained within a single schematic, and are on a single level, as shown at right.



Flat design.

Since you must manage all of the interconnections between the pages of a flat design by the names assigned to the off-page connectors, it is best to keep a flat design relatively small.

Hierarchical designs

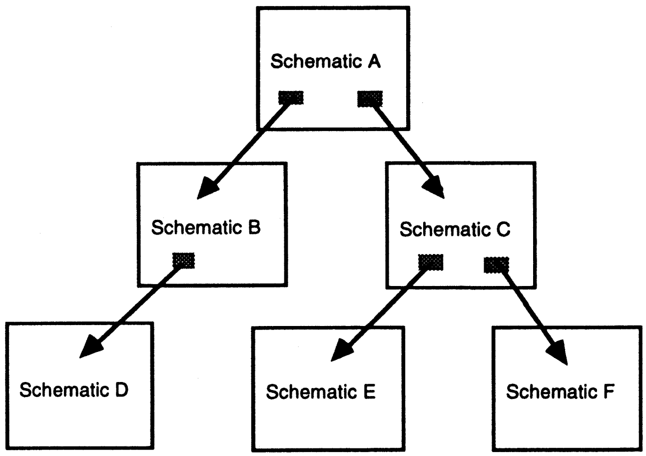
As an alternative to a flat design, you can create schematics that contain symbols representing other schematics. These symbols are called *hierarchical blocks*. The layered arrangement created by placing schematics inside other schematics is called a *hierarchy*.

Any schematic can contain hierarchical blocks (or parts with attached schematics) that refer to other schematics, and this nesting structure can be made many levels deep. The schematic at the top of a hierarchy, which directly or indirectly refers to all other schematics in the design, is called the *root* schematic page. In the design manager, the root schematic page has a backslash in its icon.

Simple hierarchies

A one-to-one correspondence between hierarchical blocks (or parts with attached schematics) and the schematics they reference is called a *simple hierarchy*. The picture at right is an example of a simple hierarchy.

In a simple hierarchy, each hierarchical block or part with an attached schematic (represented by gray boxes in the picture at right) represents a unique schematic.

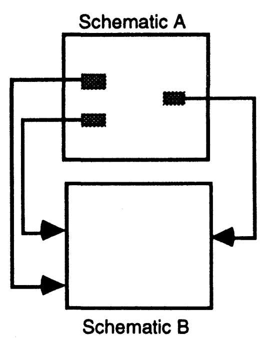


Simple hierarchy.

Complex hierarchies

A many-to-one correspondence between hierarchical blocks (or parts with attached schematics) and the schematics they reference is called a *complex hierarchy*. In the picture at right, schematic A references schematic B three different times. These references can be via hierarchical blocks or parts with attached schematics.

In the design manager, you can view a complex hierarchy two ways: in logical view, you see one schematic that represents all references to that schematic; in physical view, you see a separate schematic for each reference to that schematic. To change views, choose Physical or Logical from the design manager's View menu.



Complex hierarchy.

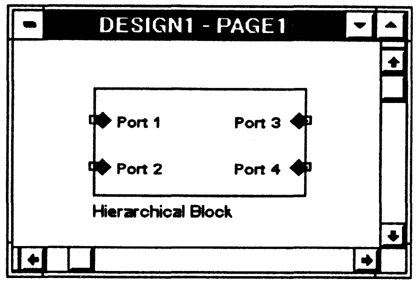
Connecting designs

In Capture, you connect schematics and schematic pages by extending nets between them, using hierarchical blocks, hierarchical ports, and off-page connectors. Hierarchical blocks and hierarchical ports carry nets between schematics and schematic pages, while off-page connectors carry nets between schematic pages within a single schematic.

See also For information about placing hierarchical blocks, hierarchical ports, and off-page connectors, see *Chapter 6: Placing, editing, and connecting parts and electrical symbols.*

Hierarchical blocks

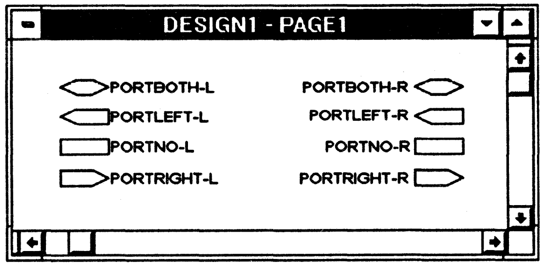
A hierarchical block is a representation of a schematic, which is attached to the hierarchical block. It provides vertical (downward-pointing) connection only. The hierarchical ports in a hierarchical block act as points of attachment for electrical connections between the hierarchical block and other electrical objects in the attached schematic. A hierarchical block functions just like a part with an attached schematic.



A part with an attached schematic functions the same as a hierarchical block, and pins on such a part function the same as hierarchical ports within a hierarchical block. You can use the same attached schematic for either method of defining a hierarchy. The only difference between the two methods is that a part with an attached schematic is easier to reuse.

Caution If you attach a schematic to a hierarchical block, be sure to include the attachment when you pass the design to a board fabrication house or to another engineer. Attached schematics external to the design *are not* carried along automatically when you copy or move a schematic or schematic page to another design, library, or schematic. Only the *pointer* to the attached schematic—that is, its name and the name of the design or library that contains it—is carried along. For this reason, you should copy all attached schematics into your design when you want your design to be "portable."

Hierarchical ports

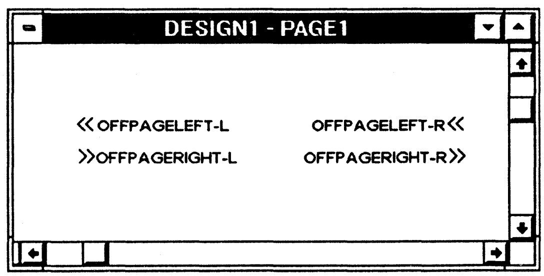


Hierarchical ports provide connection between schematics and between schematic pages. Inside a hierarchical block, a hierarchical port provides vertical (downward-pointing) connection only. It is connected by name to hierarchical ports on schematic pages within the attached schematic. You can think of its function as bringing a net "up" from the attached schematic into the hierarchical block, but not out onto the schematic page.

A free-standing hierarchical port (that is, a hierarchical port outside a hierarchical block) provides vertical (upward-pointing) and *lateral connection*. It's connected vertically to the like-named hierarchical port inside any hierarchical block to which it is attached. It's connected laterally to like-named nets, hierarchical ports, and off-page connectors within the same schematic. You can think of its function as carrying a net out of the schematic.

Note Before you create or resize a hierarchical block, make sure the Snap to grid option is turned on (from the schematic page editor's Options menu, choose Preferences). If the hierarchical block is off grid, then hierarchical ports inside it are also off grid—even if you change the Snap to grid setting before you place them—and it may be difficult to connect to these off-grid hierarchical ports.

Off-page connectors



Off-page connectors provide connection between schematic pages within the same schematic. An off-page connector is connected by name to other off-page connectors within the same schematic. Like-named off-page connectors in different schematics are not connected. Capture's symbol library CAPSYM.OLB contains two types of off-page connectors: one points right, while the other points left. These are shown in the picture at right.

An example: creating a simple hierarchy

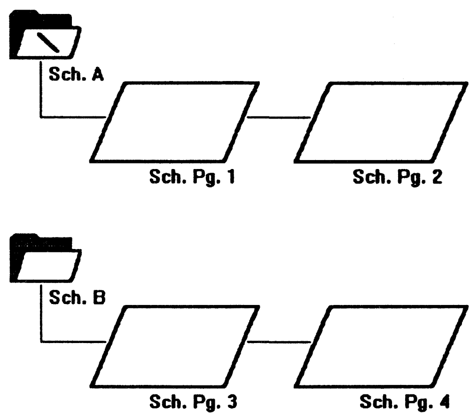
As described earlier in this chapter, you connect designs, schematics, and schematic pages by extending nets between them using off-page connectors, hierarchical ports, and hierarchical blocks.

Off-page connectors carry nets between schematic pages within a single schematic. Hierarchical blocks and hierarchical ports carry nets between schematics, which need not be in the same design.

The rest of this section contains an example of how to use off-page connectors, hierarchical ports, and hierarchical blocks to create a simple hierarchy.

See also See *Chapter 6: Placing, editing, and connecting parts and electrical symbols* for complete information about placing off-page connectors, hierarchical ports, and hierarchical blocks.

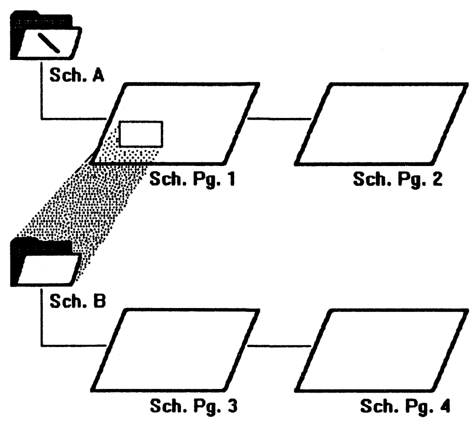
This figure shows two schematics, each with two schematic pages. The schematic marked with a backslash (\) is called the *root schematic*.



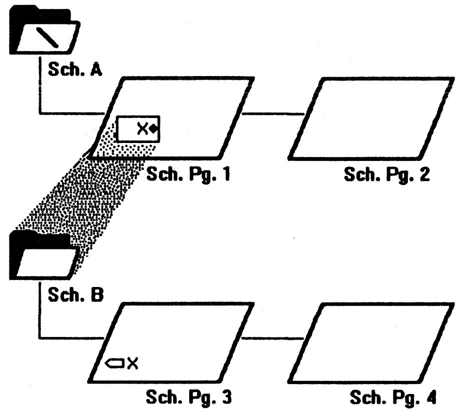
To establish the hierarchy with schematic A “above” schematic B:

1 Place a hierarchical block on schematic page 1.

2 In the Place Hierarchical Block dialog box, use the Attach Schematic button to attach schematic B.



To carry a net between schematic A and schematic B:



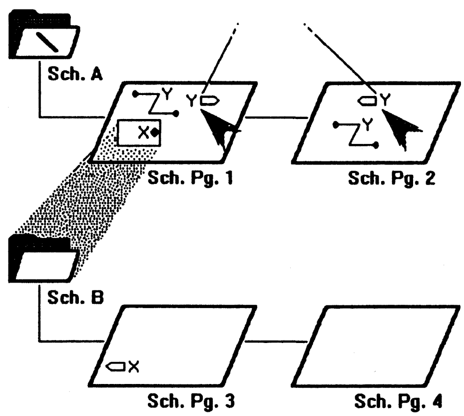
1 Select the hierarchical block on schematic page 1 and place a hierarchical port named "X" inside it.

This hierarchical port is like a pin—it is a point of attachment for electrical connections between the hierarchical block and other objects on schematic page 1.

2 Place another hierarchical port named X on schematic page 3.

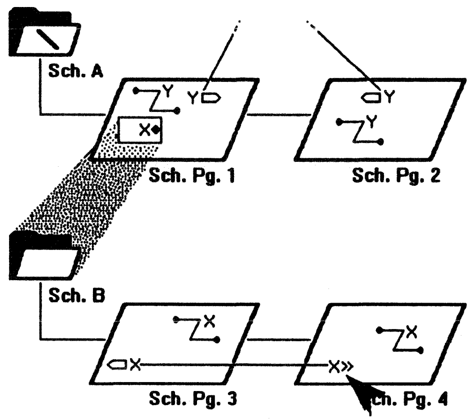
This hierarchical port is a point of attachment for electrical connections between schematic page 3 and other schematic pages. It is connected by name to the hierarchical port inside the hierarchical block on schematic page 1.

Free-standing hierarchical ports generally carry a net "up" through a hierarchy. In the root schematic, they usually represent external signals such as physical connectors on a PC board.

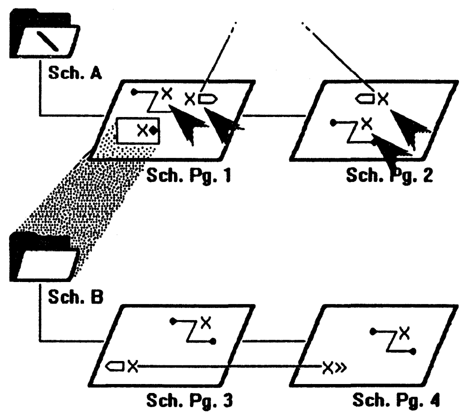


Note that these free-standing hierarchical ports in schematic A are electrically connected by name, so any like-named electrical objects on schematic pages 1 and 2 are part of a single net named Y. You could make one (but not both) of these hierarchical ports an off-page connector without affecting the electrical connections.

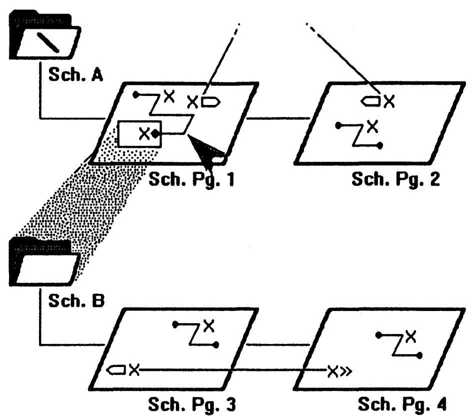
To connect the schematic pages in schematic B, place an off-page connector named X on schematic page 4. Any like-named electrical objects on schematic pages 1 and 2 are part of a single net named X.



To connect the X and Y nets, it is not enough simply to rename one set of objects, as shown here. Again, the hierarchical port inside the hierarchical block on schematic page 1 is like a pin—it doesn't bring schematic B's net X out of schematic A's hierarchical block and onto schematic page 1.



When you physically connect any part of schematic page 1's net X to the hierarchical port inside the hierarchical block, the nets are joined.



Chapter 4

Opening and saving designs

All the schematic drawings and library parts for one project are stored in a single file called a *design*. A design contains a design cache plus one or more schematics. The design cache is like an embedded library; it contains a copy of all the parts used in the design. A schematic is a container for one or more schematic pages. The schematic page is analogous to your drafting board.

For example, you might have a simple widget project composed of input, processing, and output circuits. It's possible that the output circuit will not fit on your plotter paper, but is conveniently divided into display, gain, and power circuits. In this case, the design is Widget, and the file is WIDGET.DSN. Widget contains three schematics: Input, Processing, and Output. Input and Processing contain one schematic page each, while Output contains three schematic pages: Display, Gain, and Power. The design cache for Widget contains all the parts that are on the five schematic pages.

Note As schematics and schematic pages reside inside a design, parts reside in a library. Symbols and title blocks also reside in libraries. Each design can access any number of libraries, and any library can serve any number of designs.

The information in this chapter describes opening and saving designs. The steps also work for opening libraries—instead of choosing Design, choose Library.

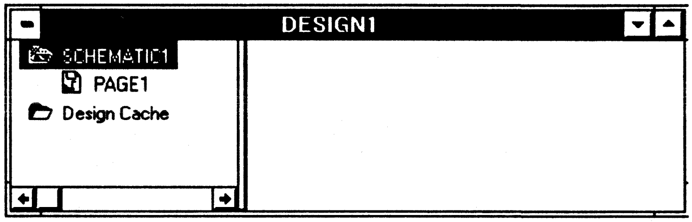
Opening a design

You can open a new design or an existing design. You can also open schematics created with OrCAD's Schematic Design Tools Release IV or 386+ software.

See also For specific information about opening SDT Release IV and SDT 386+ files, see the *Help for SDT Users* portion of Capture's online help.

To open a new design

➥ From the File menu in any Capture window (the design manager, schematic page editor, part editor, or session log), choose New, then choose Design (ALT, F, N, D).



The new design opens in a new design manager window. The design is given the default name DESIGN*n* where *n* is a number given to distinguish the new design from other new designs created in the current session. A new design contains a design cache, and one schematic—named SCHEMATIC1—that contains one schematic page named PAGE1.

To open an existing design

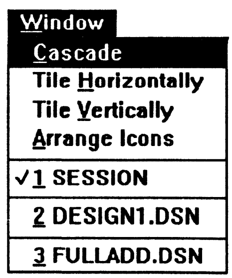
1. From the File menu in any Capture window, choose Open, then choose Design (ALT, F, O, D).
2. If the design you wish to open is not listed in the File Name text box, do one or more of the following:
   - In the Drives box, select a new drive.
   - In the Directories box, select a new directory.
   - In the List Files of Type box, select the type of file to open.
   - In the File Name text box, enter the extension of the file to open.
3. Select the design or type the name in the File Name text box, and choose the OK button.

The design opens in a new design manager window, as shown in the Tip on the next page.

Tip Capture lists the four designs or libraries that you last had open as the last items in the File menu. To open one of these most recently used files, choose it from the File menu.

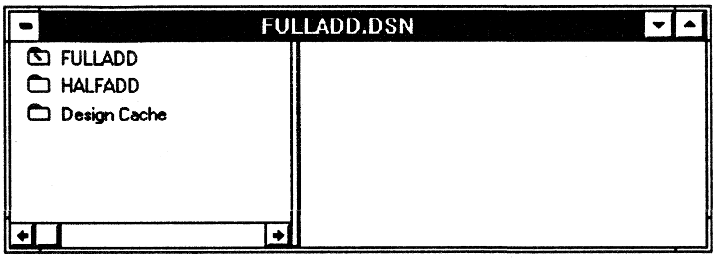
Working with several designs at once

You can have several designs open at once, each in its own design manager window. To make a different design the active window, just click on its window. If you can't see the window, choose the design from the Window menu.

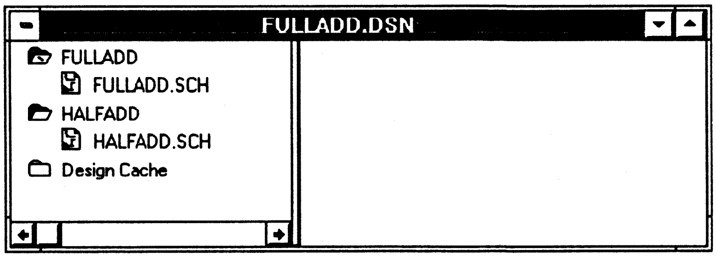


Tip When you open an existing design, its schematics (represented by file folder icons) are closed. To open a schematic and see what schematic pages (represented by page icons) it contains, double-click on it, as shown in this example.

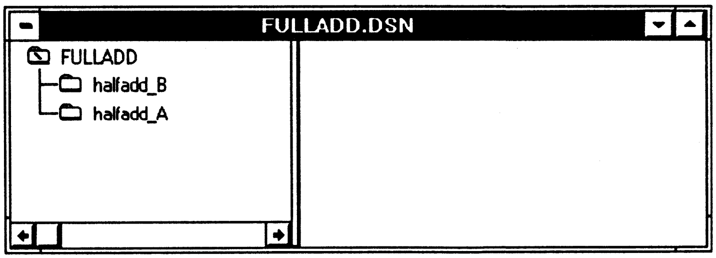
The picture at right shows the design FULLADD, and its two schematics: FULLADD and HALFADD.



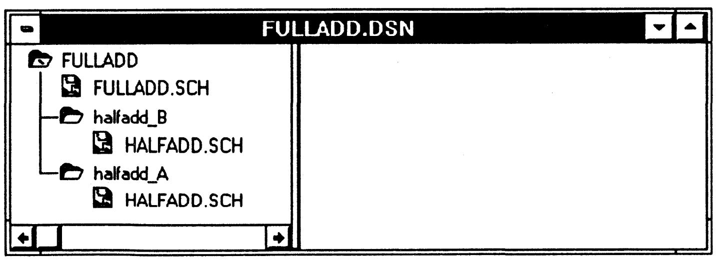
If you open the schematics FULLADD and HALFADD (by double-clicking on them), you see the schematic pages contained in each schematic: FULLADD.SCH and HALFADD.SCH.



If you change to physical view (from the View menu, choose Physical), you will see that FULLADD contains two occurrences of HALFADD. They are named halfadd_B and halfadd_A.



If you open the schematics (again, by double-clicking on them), you see that the FULLADD design contains one occurrence of the schematic page FULLADD.SCH and two occurrences of the schematic page HALFADD.SCH.



Saving a design

You can save a new or an existing design or library. You can also save a design or library in either Schematic Design Tools Release IV or 386+ format.

Note When Capture saves a design or library, it also automatically saves a backup version of the design or library, and gives it a .D~N extension if it's a design, and an .O~B extension if it's a library.

See also For specific information about saving in SDT Release IV or SDT 386+ format, see the *Help for SDT Users* portion of Capture's online help.

To save a new design

1 From the File menu in the design manager, choose Save. Since the design is new and hasn't yet been saved, Capture displays a standard Save As dialog box.

2 Enter the name you would like the design to have in the File Name text box. To control where the file is saved, do one of the following:

- In the Saved File Type box, select the type of file to save.
- In the Directories box, select a new directory.
- In the Drives box, select a new drive.

3 Choose the OK button.

The design is saved, and remains open in the design manager.

Note—Guidelines for naming documents By default, Capture designs have a .DSN extension and Capture libraries have an .OLB extension. Designs saved for use in SDT Release IV or SDT 386+ should have a .SCH extension. Libraries saved for use in SDT Release IV or SDT 386+ should have a .LIB extension.

To save an existing design

➥ From the File menu in the design manager, choose Save.

The design is saved, and remains open in the design manager.

Note If you choose Save when a schematic page window is active, only that page is saved, instead of the entire design.

To save a copy of a design

1. From the File menu in the design manager, choose Save As.

   The Save As dialog box displays.

2. Enter the name you would like the design to have in the File Name text box. To control where the file is saved, do one of the following:

   - In the Saved File Type box, select the type of file to save.
   - In the Directories box, select a new directory.
   - In the Drives box, select a new drive.

3. Choose the OK button.

The design is saved, and remains open in the design manager.

To save all open designs

➥ From the File menu in the design manager, choose Save All.

When you choose the Save All command, Capture asks if you want to save the changes in just the open designs or libraries that have been modified. For example, if you have one design and two libraries open, then modify the design and one of the libraries, but don't modify the other library, Capture will ask if you want to save the modified design and the modified library, but will not ask you about the unmodified library. The Save All command is only available if there is at least one open design or library that has actually been modified.

Closing a design and quitting Capture

Closing a design or library

➥ From the File menu in the design manager, choose Close (ALT, F, C).

When you close a design or library, Capture asks if you want to save your changes.

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

Quitting Capture

➥ From the File menu in the design manager, choose Exit (ALT, F, X).

When you choose the Exit command, Capture asks if you want to save your changes.

Creating designs

Part Two contains information relating to schematic design tasks. It provides instructions for setting up your design environment, adding parts and graphics to your design, navigating around your design using the Zoom command, and printing or plotting your design.

Part Two includes these chapters:

Chapter 5: Setting up your design describes how to customize the working environment specific to your system, how to create default settings for new designs, and how to override default settings in individual designs.

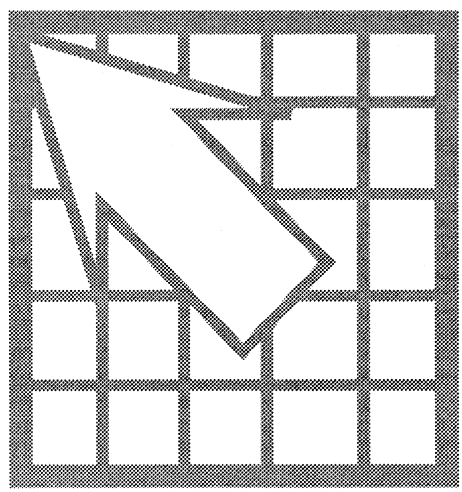
Chapter 6: Placing, editing, and connecting parts and electrical symbols describes how to place and edit parts and symbols. It also describes how to connect the elements of your design using hierarchical blocks, hierarchical ports, off-page connectors, wires, and buses.

Chapter 7: Adding and editing graphics and text describes the drawing tools that you can use to create text and a variety of graphic shapes that can be added to your design.

Chapter 8: Changing your view of a schematic describes how to view specific areas of a schematic page using the Zoom command. It also describes jumping to different locations within a schematic using the Location, Reference, and Bookmark commands.

Chapter 9: Printing or plotting describes how to print or plot your design.

Part Two



Chapter 5

Setting up your design

Capture provides different levels of configuration. Using commands on the Options menu, you can:

- Customize the working environment specific to your system (using *preferences*).
- Create default settings for new designs (using the *design template*). These settings stay with the design even if it is moved to another system with different preferences.
- Override design template settings in individual designs (using *design properties*) or individual schematic pages (using *schematic page properties*).

No matter which window in Capture is active, the Options menu always has a Preferences command and a Design Template command. In addition, the Options menu contains commands specific to the current active window. For example, the design manager's Options menu contains the Design Properties command, while the schematic page editor's Options menu contains the Schematic Page Properties command.

The settings on the Preferences dialog box determine how Capture works on your system, and persist from one Capture session to the next because they are stored in the CAPTURE.INI file on your system. If you pass designs to others, they won't inherit your Preferences settings. This means that you can set colors, grid display options, pan and zoom options, and so on to your liking and be assured that your settings will remain, even if you work on a design created on another system.

The Design Template dialog box determines the default characteristics of all the designs created on your system. Because a new design inherits characteristics from the current design template, it's a good idea to check the design template before you create a new design.

Once you begin working on a design, you can customize its particular characteristics by choosing Design Properties from the Options menu when you are in the design manager, or Schematic Page Properties when you are in the schematic page editor.

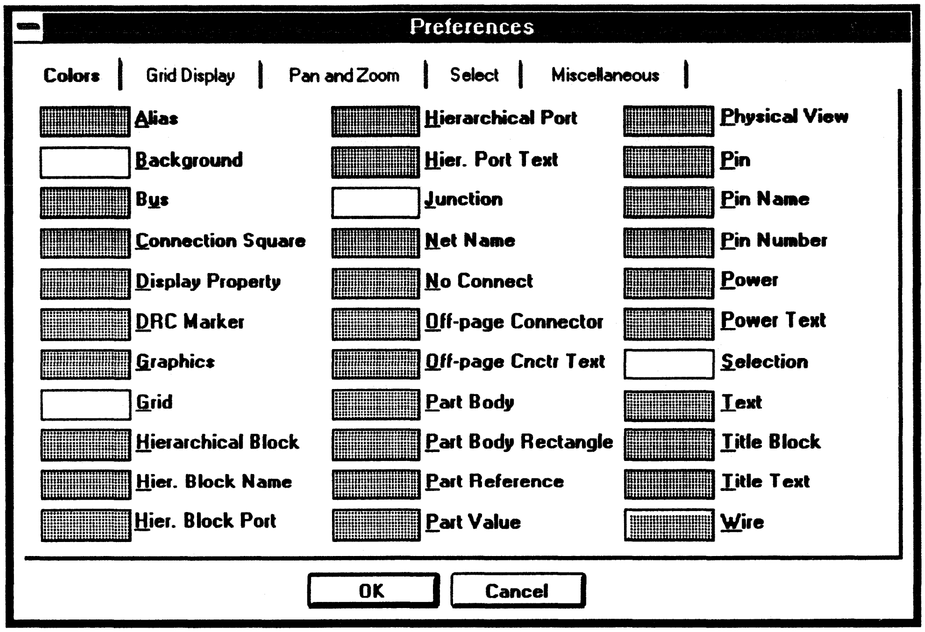
Defining your preferences

The options that you define on the Preferences dialog box affect how Capture works on all the designs that you work on. These are some of the things you can define on the Preferences dialog box:

- **Colors**. You can set up colors for particular objects such as off-page connectors, hierarchical blocks and ports, text, title blocks, and so on. You can also change the background color and the color of the grid.
- **Grid Display**. You can select dots or lines for your grid, and whether to display or print your grid. You can select whether to have your pointer snap to grid as you place objects. You can set these options independently for the schematic page editor and the part editor.
- **Pan and Zoom**. You can define how you want autoscrolling to work, and what the zoom factor should be. You can set these options independently for the schematic page editor and the part editor.
- **Select**. You can define whether you want to select objects enclosed by a selection rectangle or objects intersecting a selection rectangle, the maximum number of objects to display at high resolution while dragging, and whether to show the tool palette. You can set these options independently for the schematic page editor and the part editor.
- **Miscellaneous**. You can define the default fill, line style and width, color for graphic objects, and the font used in the design manager and session log.

Defining colors

You can control the color in which different schematic page objects display by using the Colors tab on the Preferences dialog box. Note that the color you select for Title Block is also the color used for borders and grid references.



To define an object's color

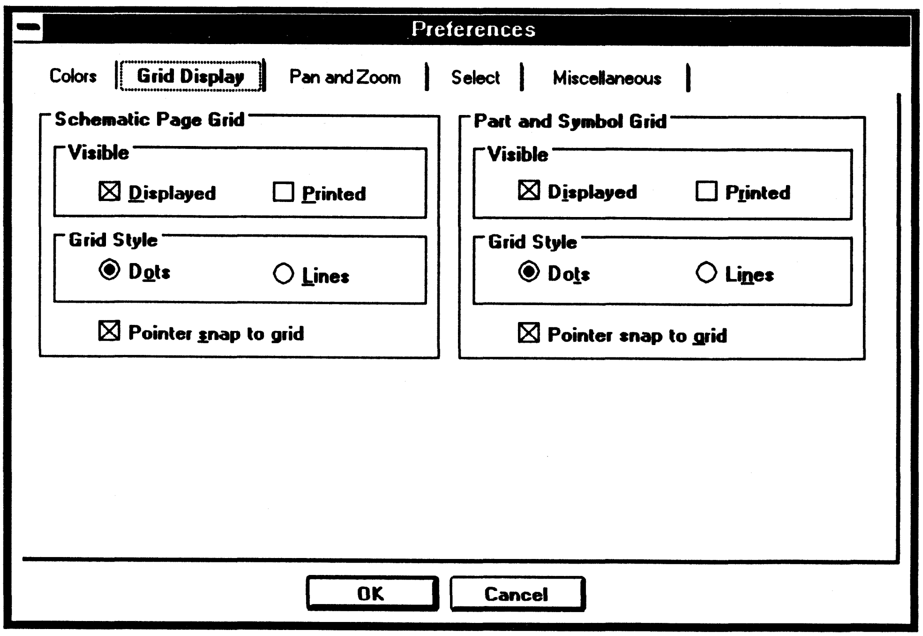
1. From the Options menu, choose Preferences (ALT, O, P), then choose the Colors tab.
2. Click the left mouse button on the color of an item. The color palette window opens.
3. Select the new color. Choose the OK button to dismiss the color palette.
4. Choose the OK button.

Note The color that you select for Graphics defines the color for lines, polylines, and arcs drawn in the schematic page editor, and for all graphics drawn in the part editor. It also becomes the default color for the schematic page editor on the Miscellaneous tab of the Preferences dialog box. If you change the Color option on the Miscellaneous tab, then rectangles, ellipses, and closed shapes that you draw in the schematic page editor are created in that color. Lines, polylines, and arcs, however, are drawn in the color selected for Graphics on the Colors tab.

Controlling the grid

You can control whether Capture displays or prints a grid independently in the schematic page editor and the part editor, and whether the grid uses dots or lines. You can also specify whether the pointer snaps to grid in each editor.

Caution If you disable the Pointer snap to grid option while you are drawing, be sure to enable it before you place electrical objects.



To control the grid

1. From the Options menu, choose Preferences (ALT, O, P), then choose the Grid Display tab.
2. For the schematic page editor and the part editor, specify:
   - Whether to display or print the grid.
   - Whether the grid uses dots or lines.
   - Whether the pointer snaps to grid as you place objects.
3. Choose the OK button.

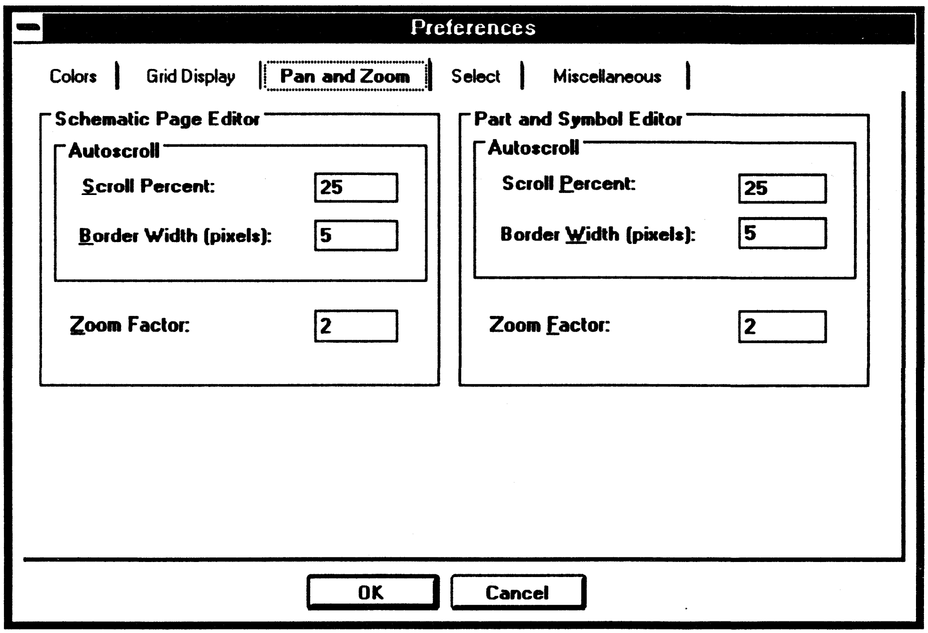
Tip You can also show or hide the grid using the Grid command on the View menu in the schematic page editor or the part editor.

Setting pan and zoom

When you position the pointer within a certain distance of the window's edge, the display changes so that you are viewing a different region of the drawing board. This change is called *panning*. You can configure the distance by which the display changes and the location at which the pointer triggers the change. The distance by which the display changes is the *panning distance*; the distance from the window's edge at which the pointer triggers the change is the *panning border*.

When you zoom in or zoom out, the view is changed by the *zoom factor* that you set.

You can define independent pan and zoom settings for the schematic page editor and the part editor.

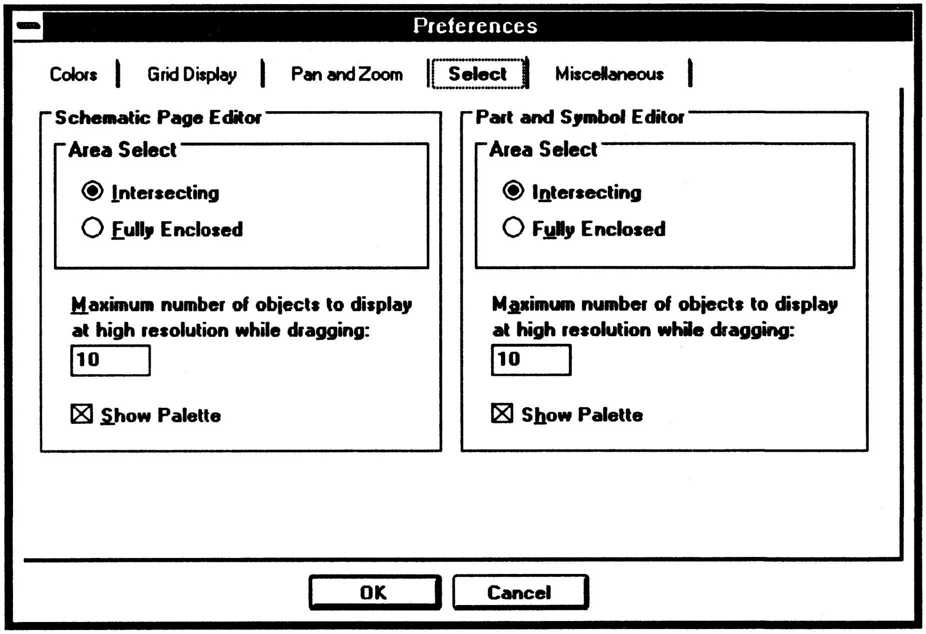


To configure panning distance, panning border, and zoom factor

1. From the Options menu, choose Preferences (ALT, O, P), then choose the Pan and Zoom tab.
2. For the schematic page editor and the part editor, set these options:
   - **Scroll Percent**. Enter the percent of the window's horizontal or vertical dimension by which the display will scroll.
   - **Border Width**. Enter the distance—in pixels—that the pointer must be from the window's edge before the display pans.
   - **Zoom Factor**. Enter a number to indicate the magnification or reduction of the objects shown in the window when you zoom in or zoom out.
3. Choose the OK button.

Defining selection options

You can specify whether objects are selected when the selection border intersects them or if the objects are selected only when they are completely enclosed in the selection area. You can also change the maximum number of objects displayed at high resolution while dragging, and set tool palette visibility in both the schematic page editor and the part editor.



To define selection options

1 From the Options menu, choose Preferences (ALT, O, P), then choose the Select tab.

2 For the schematic page editor and the part editor, set these options:

- **Area Select**. Specify whether to select objects that are intersecting the selection border or only objects that are fully enclosed by the selection border.
- **Maximum number of objects to display at high resolution while dragging**. If you drag more objects than you specify here, you will see rectangular placeholders for the objects as you drag them.
- **Show Palette**. Select this check box to make the tool palette visible; deselect it to make the tool palette invisible.

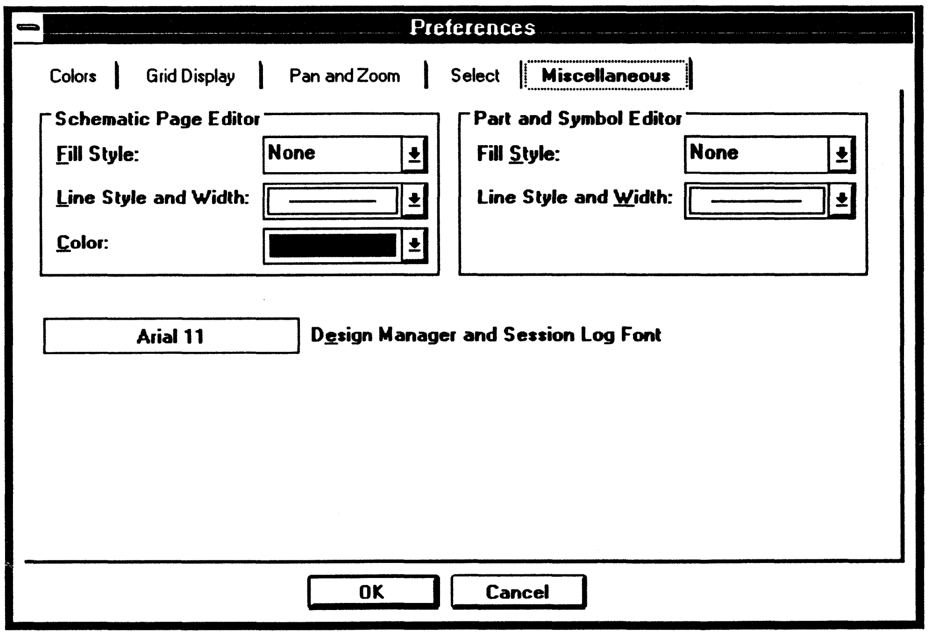
3 Choose the OK button.

Setting miscellaneous options

You can specify default fill style, line style, and line width for graphics you create in the schematic page editor and the part editor. You can also specify a default line color for the schematic page editor.

Note You can change the fill style, line style and width, and color on graphic objects on an individual basis once they are drawn on a schematic page or part. Select the object, then from the Edit menu, choose Properties. For specific instructions, see *Chapter 7: Adding and editing graphics and text.*

You can also specify the font that is used to display text in the design manager and the session log.



To set miscellaneous options

1 From the Options menu, choose Preferences (ALT, O, P), then choose the Miscellaneous tab.

2 For the schematic page editor and the part editor, set these options:

- **Fill Style**. Select the fill pattern to be used when drawing rectangles, ellipses, and closed shapes drawn with the polyline tool.
- **Line Style and Width**. Select the line style and width used for lines, polylines, rectangles, ellipses, and arcs.

3 For the schematic page editor, select the color used for rectangles, ellipses, and closed shapes.

Note The color Default is the color defined in the Graphics box on the Colors tab of the Preferences dialog box.

The color for lines, polylines, and arcs drawn in the schematic page editor is defined in the Graphics box on the Colors tab of the Preferences dialog box.

---

4 Select a font to display text in the design manager and session log. If you click on the box for the Session Log Font option, a standard Windows dialog box for font selection displays. Select a font, style, and size from this dialog box and choose the OK button.

5 Choose the OK button.

Setting up your design template

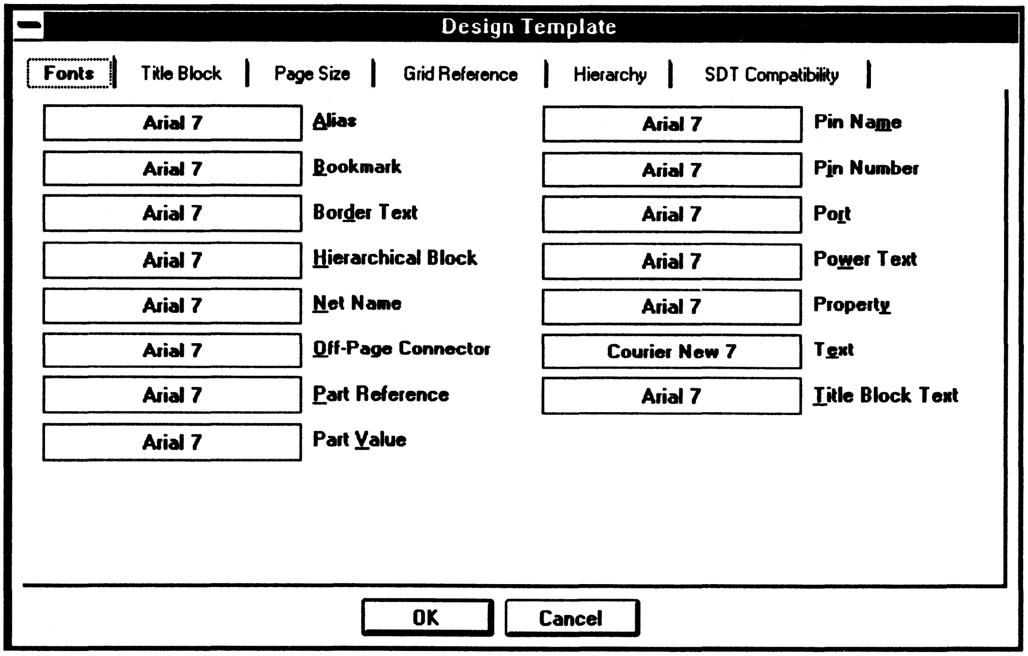
The options that you define on the Design Template dialog box are the default settings for all new designs, and for schematics and schematic pages you add to an existing design. You can override some of these options for individual designs or schematic pages. Some of the things you can define on the Design Template dialog box are:

- **Fonts**. You can define the fonts for schematic objects that contain text, such as part references and values.
- **Title Block**. You can specify the text to appear in title block fields, as well as the path and filename of the library containing the title block. This affects new designs, as well as new schematic pages in existing designs.
- **Page Size**. You can specify whether inches or millimeters are used as the unit of measure, the width and height of a schematic page, and the spacing between pins.
- **Grid Reference**. For horizontal and vertical grid references, you can set the number of grid references to display in either direction, whether the grid references are alphabetic or numeric, whether they increment or decrement across the schematic page, and how wide grid reference cells are. You can also make the border, grid references, and title block visible or invisible. This affects new designs, as well as new schematic pages in existing designs.
- **Hierarchy**. For hierarchical blocks and part instances that have their Primitive property set to Default, you can specify if you want Capture to treat each as primitive (cannot descend into attached schematics) or nonprimitive (can descend into attached schematics).
- **SDT Compatibility**. You can specify which Capture properties map to which SDT part fields when saving a design in SDT format.

Setting up fonts for new designs

You can define the fonts assigned to the text associated with different schematic page objects in new designs. The fonts specified here do not affect existing designs.

Note To change the fonts for an existing design, use the Fonts tab on the Design Properties dialog box. You can access this dialog box by choosing Design Properties from the design manager's Options menu.



To assign fonts used for new designs

1. From the Options menu, choose Design Template (ALT, O, D), then choose the Fonts tab.
2. Click the left mouse button on the font of an item. A standard Windows font dialog box displays.
3. Select a font, font style, and size. Choose the OK button to dismiss the font dialog box.
4. Choose the OK button.

Note The default fonts were selected for optimal compatibility with SDT. Changing these fonts may result in less optimal text sizes for translated designs.

Defining title block information

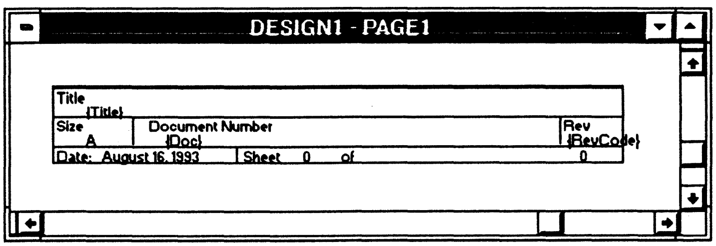
There are two types of title blocks: default and optional. The default title block is specified on the Title Block tab of the Design Template dialog box. Capture places one default title block in the lower right corner of each schematic page if a library and title block name are specified here, and places the text you enter in the Title, Organization Name, Organization Address, Document Number, Revision, and CAGE Code fields in the title block. This information is also used in reports created by the commands on the Tools menu.

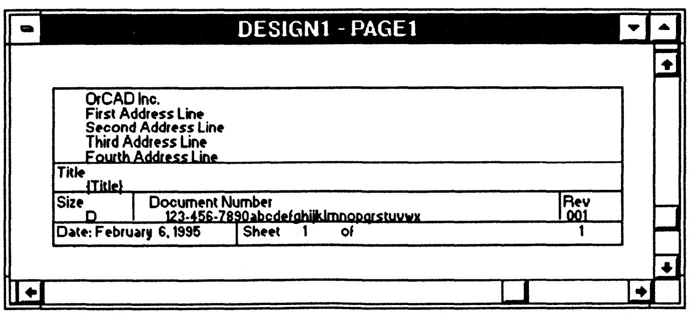
This affects new designs, as well as new schematic pages in existing designs. You can set the default title block to be visible or invisible on an existing schematic page by changing the setting on the Grid References tab window in the Schematic Page Properties dialog box.

Note You can place any number of optional title blocks anywhere on the schematic page using the Title Block command on the Place menu. Optional title blocks display information that you define as property values for the title block symbol.

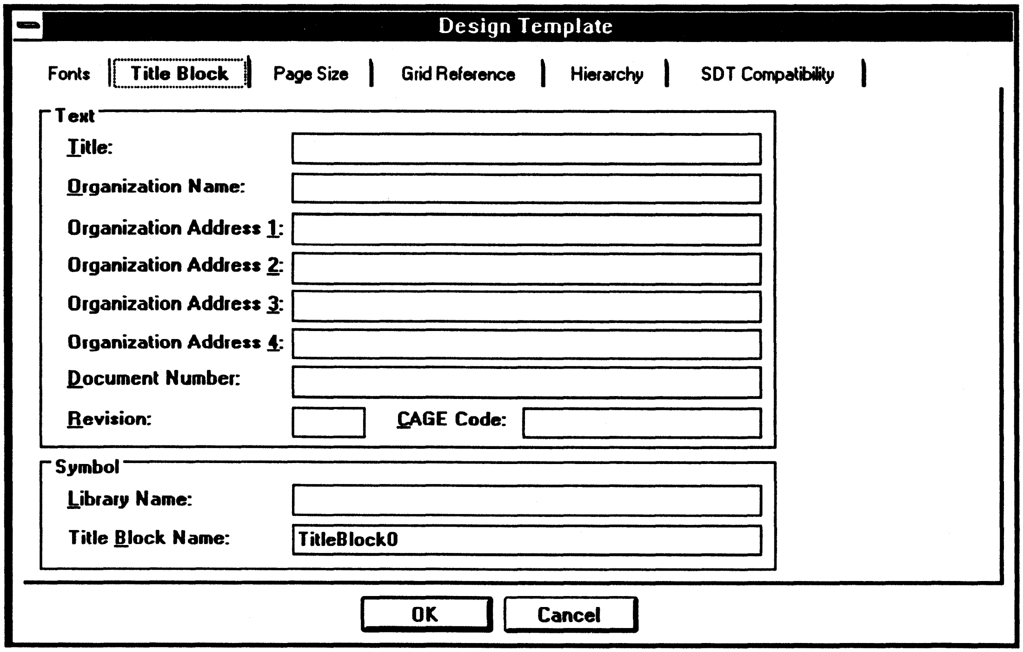
Capture provides default title block symbols in the CAPSYM.OLB library. Two such title blocks are shown below.

Note The text shown in curly braces is a property text placeholder. The value can be specified by double-clicking on the text itself and supplying a value. You can control the visibility by selecting or deselecting the Visible check box on the Display Properties dialog box.





Tip You can create custom title blocks and store them in a library. If you specify the name of the custom library and title block in the Symbol area of the Design Template's Title Block tab, then the custom title block appears in the lower right corner of each schematic page. If the custom title block contains the properties required for a default title block (Document Number, Revision, CAGE Code, Title, Organization Name, and Organization Address 1–Organization Address 4), then the text you enter in these fields on the Design Template's Title Block tab displays in the title block. See Capture's online help for specific instructions.



To choose a title block and define the text it contains

1. From the Options menu, choose Design Template (ALT, O, D), then choose the Title Block tab.
2. In the Text area, enter the information you want to appear in the title block.
3. In the Symbol area, enter the path and filename of the library containing the title block, and enter the name of the title block.
4. Choose the OK button.

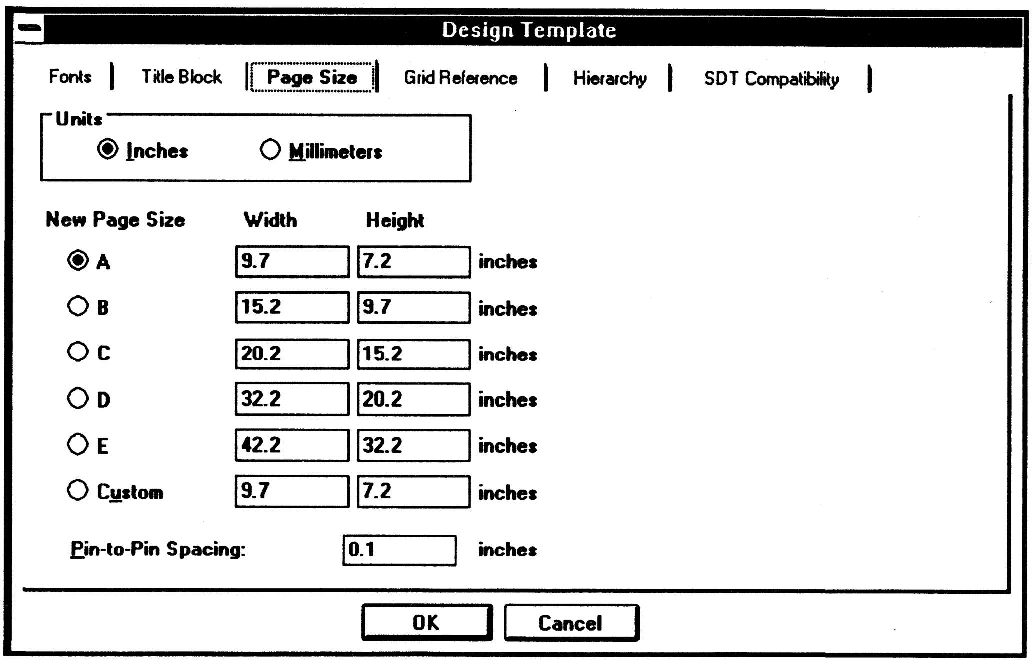
Note You can create custom power, ground, and other symbols for hierarchical ports, off-page connectors, title blocks, and power objects by using the New Symbol command from the Design menu in the design manager window. For information on how to use this command, see Capture's online help.

Setting up the schematic page size for new designs

For new designs, you can specify the default unit of measure—inches or millimeters—and the default width and height of schematic pages.

You can also define the default spacing between pins in the Pin-to-Pin Spacing text box. The value you enter in the text box defines how close together pins are when you place them on a part in the part editor. It also defines the grid spacing (the space between grid dots or grid lines).

Note You can select a different unit of measure or page size (A, B, C, D, E, and Custom if the unit of measure is Inches; or A4, A3, A2, A1, A0, and Custom if the unit of measure is Millimeters) for individual schematic pages in existing designs using the Page Size tab on the Schematic Page Properties dialog box. You can access this dialog box by choosing Schematic Page Properties from the schematic page editor's Options menu. Note that you can select a different page size, but you can't change the dimensions of that page size or change the pin-to-pin spacing.



To set up the schematic page size

1 From the Options menu, choose Design Template (ALT, O, D), then choose the Page Size tab.

2 In the Units area, select the default unit of measure for new designs as either Inches or Millimeters.

Note Changing from Inches to Millimeters resets the page sizes to their defaults.

3 Select the default schematic page size for new designs. For each schematic page size (A, B, C, D, E, and Custom if the unit of measure is Inches; or A4, A3, A2, A1, A0, and Custom if the unit of measure is Millimeters) you can specify the width and height. The values that you enter in the Width and Height text boxes become the dimensions for each page size. You cannot change these dimensions for individual schematic pages, although you can select a different page size.

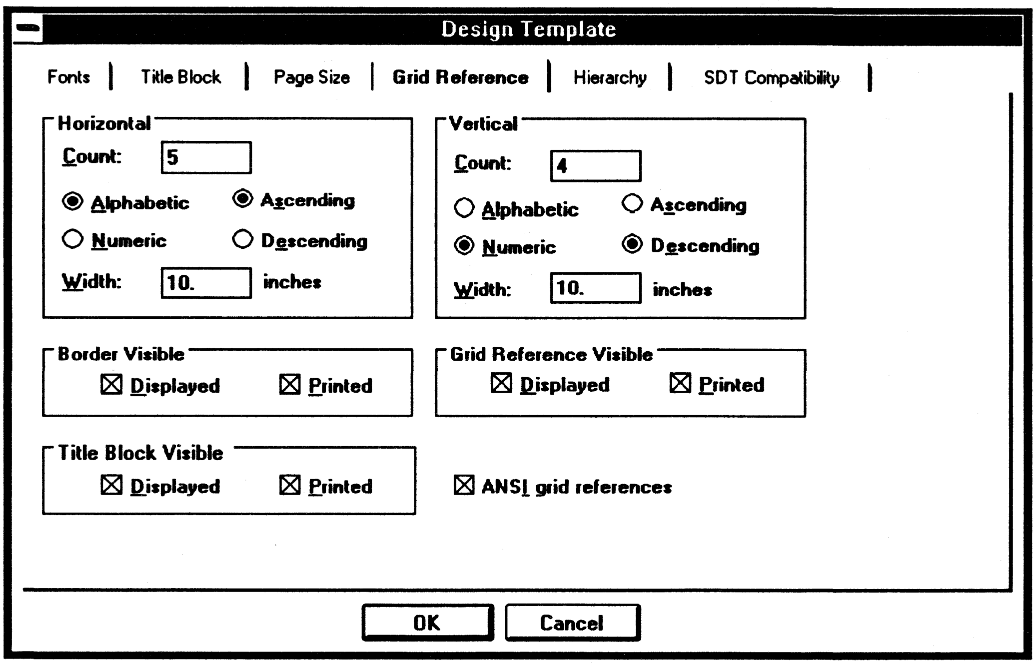
4 In the Pin-to-Pin text box, specify the default spacing between pins. The value you enter in this text box defines how close together pins are when you place them on a part in the part editor. It also defines the grid spacing (the space between grid dots or grid lines). You cannot change this value for existing designs or individual schematic pages.

5 Choose the OK button.

Defining the grid reference

For horizontal and vertical grid references, you can set the number of grid references to display in either direction, whether the grid references are alphabetic or numeric, whether they increment or decrement across the schematic page, and how wide grid reference cells are. You can also make the border, grid references, and title block visible or invisible on the screen and on schematic pages you print. The settings on this tab affect new designs and new schematic pages in existing designs.

Note You can change these settings for existing schematic pages using the Page Size tab on the Schematic Page Properties dialog box by choosing Schematic Page Properties from the schematic page editor's Options menu.



To define the grid reference

1 From the Options menu, choose Design Template (ALT, O, D), then choose the Grid Reference tab.

2 For horizontal and vertical grid references, specify:

- The number of grid references, and whether they are alphabetic or numeric.
- Whether the grid references increment (Ascending) or decrement (Descending) across the schematic page, and how tall the grid reference cells are.

3 For the border, title block, and grid reference, select Displayed to have the item display on the screen or Printed to have the item appear on schematic pages you print.

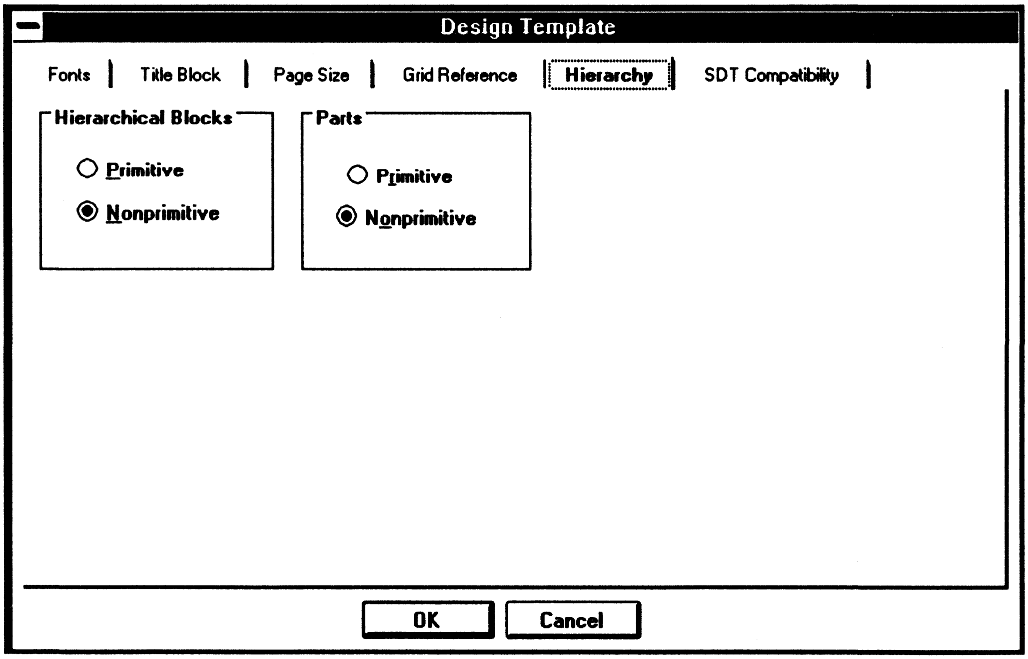
4 Choose the OK button.

Defining the default hierarchy option for new designs

For hierarchical blocks and part instances that have their Primitive property set to Default, you can specify if you want Capture to treat each as primitive (cannot descend into attached schematics) or nonprimitive (can descend into attached schematics). The Primitive and Nonprimitive options on the Hierarchy tab of the Design Template dialog box only affect new designs.

Note You can change this option for existing designs using the Hierarchy tab on the Design Properties dialog box. You can access this dialog box by choosing Design Properties from the design manager's Options menu.

Note that this setting affects how the options on the Tools menu process designs.



To define the default hierarchy option

1. From the Options menu, choose Design Template (ALT, O, D), then choose the Hierarchy tab.
2. For hierarchical blocks and parts, select Primitive or Nonprimitive. All hierarchical blocks and part instances that have their Primitive property set to Default will use the setting selected here.
3. Choose the OK button.

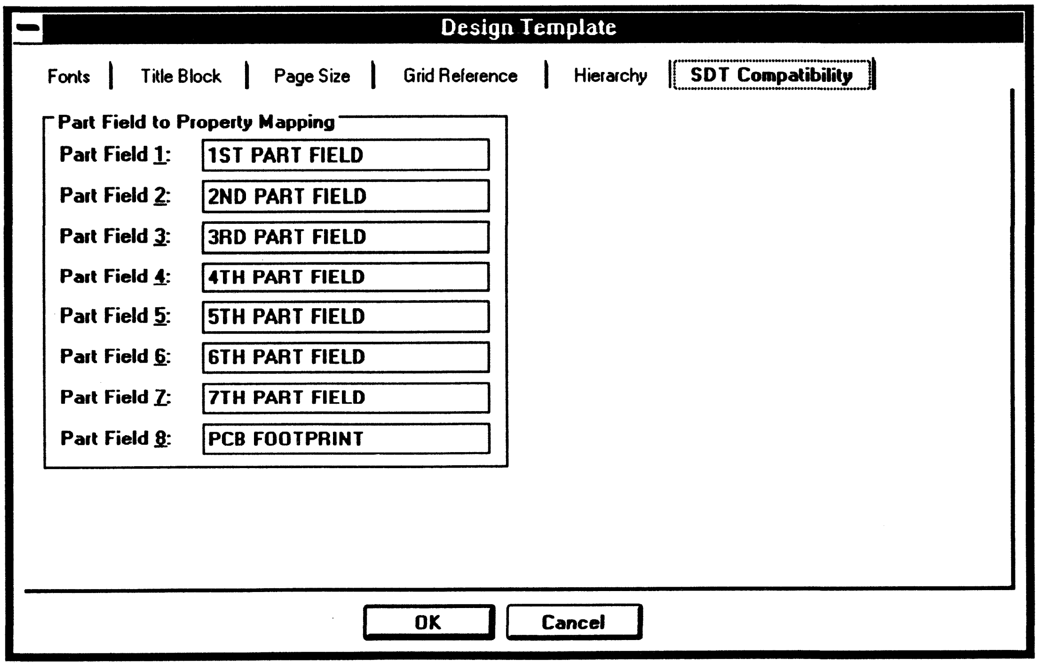
Setting up compatibility with OrCAD's Schematic Design Tools

You can specify which properties Capture stores in the eight SDT part fields when saving a design in SDT format.

Note You can also use the fields for mapping netlists that use part field information. For information on creating these types of netlists and the combined property strings they require, see Capture's online help.

In the dialog box shown below, the part fields listed on the left are SDT's part fields. The text boxes on the right are used to specify which of Capture's properties map to the part fields in SDT. The options on the SDT Compatibility tab of the Design Template dialog box only affect new designs.

Note To change the part field to property mapping for existing designs, use the SDT Compatibility tab on the Design Properties dialog box (from the design manager's Options menu, choose Design Properties).



To set up compatibility with OrCAD's Schematic Design Tools

1. From the Options menu, choose Design Template (ALT, O, D), then choose the SDT Compatibility tab.
2. For each SDT part field, specify the Capture property to be placed in the part field when you save a Capture design in SDT format.
3. Choose the OK button.

Changing properties of individual designs

When you create a new design, it inherits the options defined on the Design Template dialog box. You can override these options on individual designs:

- **Fonts**. You can define the fonts for schematic objects that contain text, such as part references and values.
- **Hierarchy**. For hierarchical blocks and part instances that have their Primitive property set to Default, you can specify if you want Capture to treat each as primitive (cannot descend into attached schematics) or nonprimitive (can descend into attached schematics).
- **SDT Compatibility**. You can specify which Capture properties map to which SDT part fields when saving a design in SDT format.
- **Miscellaneous**. If you need to see the power pins on a design for documentation or debugging purposes, you can display them on the screen.

See You can override some Design Template options (page size and grid reference) using the Schematic Page Properties dialog box. You can access this dialog box by choosing Schematic Page Properties from the schematic page editor's Options menu.

Assigning fonts

Fonts are assigned to new designs using the Fonts tab on the Design Template dialog box. You can change fonts for individual designs using the Fonts tab on the Design Properties dialog box. You can access this dialog box by choosing Design Properties from the design manager's Options menu. See *Setting up fonts for new designs* in *Setting up your design template* for a picture of this dialog box.

Defining hierarchy

The behavior for hierarchical blocks and part instances that have their Primitive property set to Default (that is, whether to act as primitive or nonprimitive) is defined for new designs using the Hierarchy tab on the Design Template dialog box. You can change this behavior for individual designs using the Hierarchy tab on the Design Properties dialog box. You can access this dialog box by choosing Design Properties from the design manager's Options menu. See *Defining the default hierarchy option for new designs* in *Setting up your design template* for a picture of this dialog box.

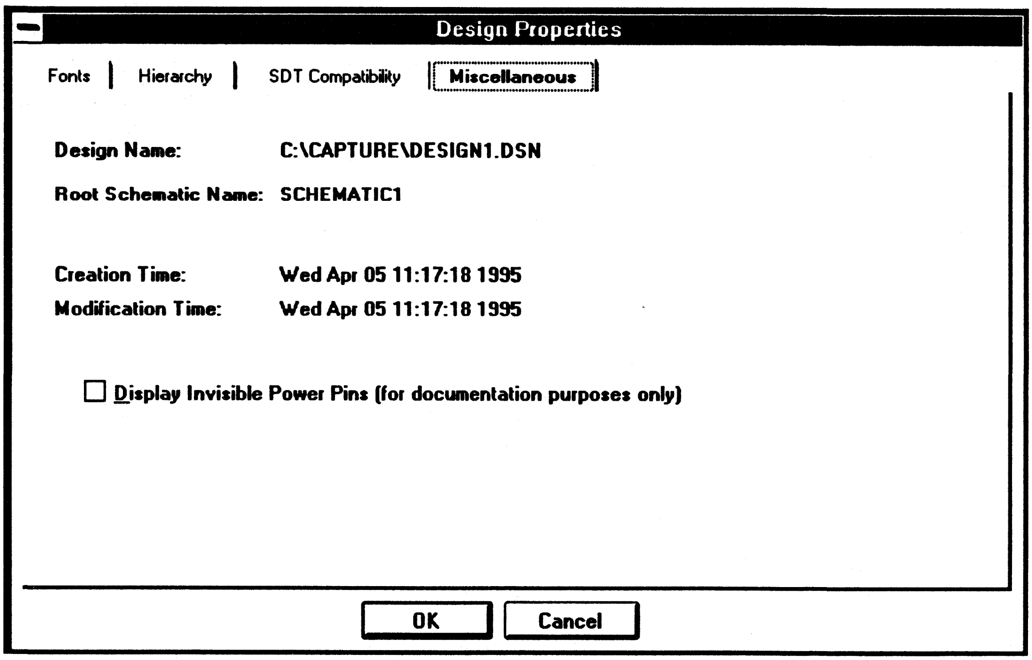
Setting up compatibility with OrCAD's Schematic Design Tools

The mapping of Schematic Design Tools to Capture properties for new designs is defined using the SDT Compatibility tab on the Design Template box. You can change this mapping for individual designs using the SDT Compatibility tab on the Design Properties dialog box. You can access this dialog box by choosing Design Properties from the design manager's Options menu. See *Setting up compatibility with OrCAD's Schematic Design Tools* in *Setting up your design template* for a picture of this dialog box.

Viewing invisible power pins without isolating them

Normally, power pins are invisible, and thus global. Selecting the Display Invisible Power Pins (for documentation purposes only) option on the Miscellaneous tab will display the pins on the screen, and they will still be considered global. However, you can only view the power pins—you cannot connect to them.

Note To be able to connect wires and other electrical objects to power pins, you must make them visible on the part or instance. Select the part and then, from the Edit menu, choose Properties. Select the Power Pins Visible option and choose the OK button. If you connect a wire or other electrical object to a power pin made visible by this method, that pin is isolated from the design-wide power net.



To view invisible power pins without isolating them

1. From the design manager's Options menu, choose Design Properties (ALT, O, R), then choose the Miscellaneous tab.
2. Select the Display Invisible Power Pins option.
3. Choose the OK button.

Changing properties of individual schematic pages

When you create a new design, it inherits the options defined on the Design Template dialog box. You can override these options on individual schematic pages:

- **Page size**. You can define the fonts for schematic objects that contain text, such as part references and values.
- **Grid reference.** For hierarchical blocks and part instances that have their Primitive property set to Default, you can specify whether you want Capture to treat each as primitive (cannot descend into attached schematics) or nonprimitive (can descend into attached schematics).
- **Miscellaneous.** You can view information about the schematic page, such as the creation time, modification time, and the page number.

See You can override some Design Template options (fonts, hierarchy, and SDT compatibility) using the Design Properties dialog box. You can access this dialog box by choosing Design Properties from the design manager's Options menu.

Changing page size

The unit of measure (inches or millimeters), page size, and pin-to-pin spacing for new designs are defined using the Page Size tab on the Design Template dialog box.

For individual schematic pages, you can change the unit of measure from Inches to Millimeters or select a different page size (A, B, C, D, E, or Custom if the unit of measure is Inches; or A4, A3, A2, A1, A0, or Custom if the unit of measure is Millimeters). You can't change the dimensions of that page size or change the pin-to-pin spacing for individual schematic pages.

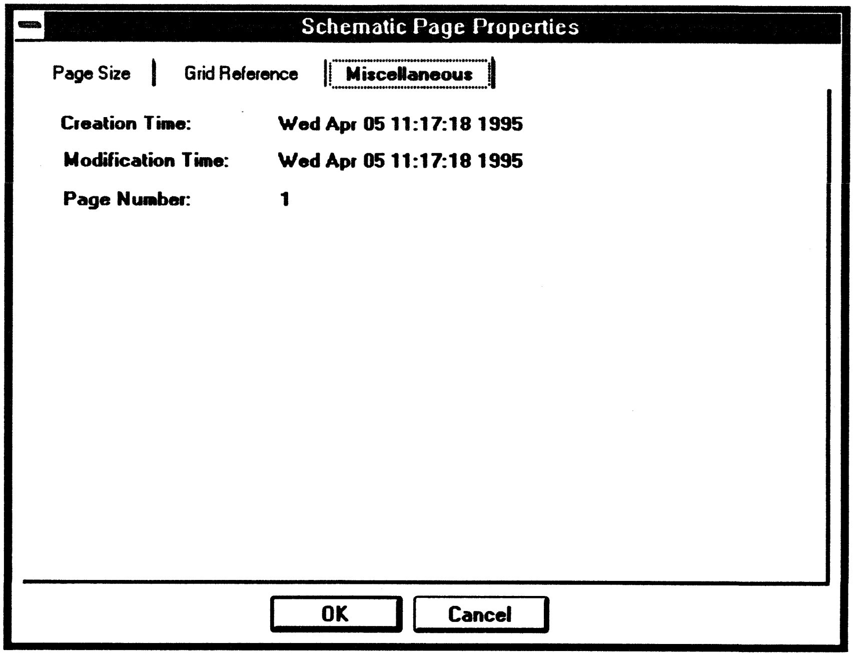
You can change these settings for individual schematic pages using the Page Size tab on the Schematic Page Properties dialog box. You can access this dialog box by choosing Schematic Page Properties from the schematic page editor's Options menu. See *Setting up the schematic page size for new designs* in *Setting up your design template* for a picture of this dialog box.

Setting up new grid references

Horizontal and vertical grid references for new designs are set up on the Grid Reference tab of the Design Template dialog box. You can change these settings for individual schematic pages using the Grid Reference tab on the Schematic Page Properties dialog box. You can access this dialog box by choosing Schematic Page Properties from the schematic page editor's Options menu. See *Defining the grid reference* in *Setting up your design template* for a picture of this dialog box.

Viewing miscellaneous schematic page properties

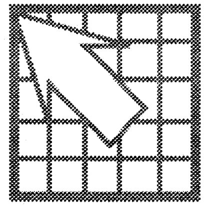
The Miscellaneous tab on the Schematic Page Properties dialog box displays the creation time and the last modification time of the schematic page, as well as the page number.



To view miscellaneous schematic page properties

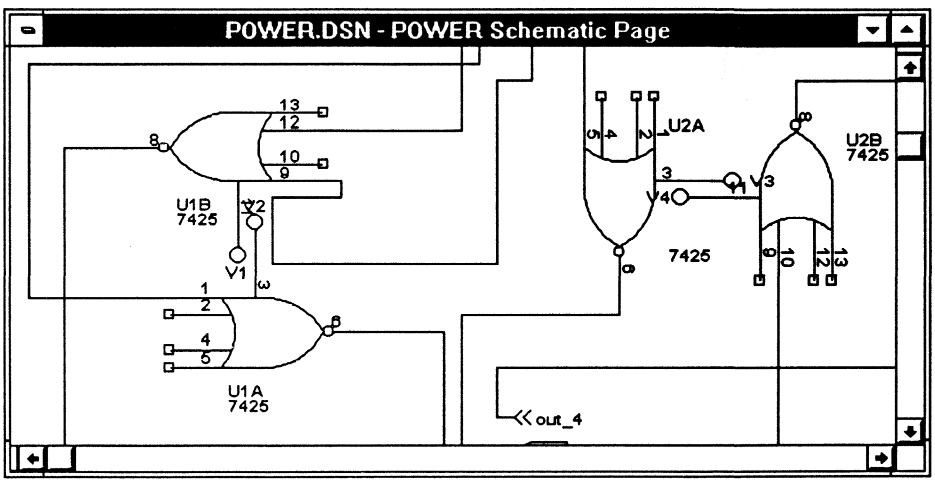
1. From the schematic page editor's Options menu, choose Schematic Page Properties (ALT, O, E), then choose the Miscellaneous tab.
2. When you are done viewing the information, choose the OK button.

Chapter 6



Placing, editing, and connecting parts and electrical symbols

Capture includes libraries containing parts, power symbols, and ground symbols. You can place instances of these objects on your schematic page. Once you have placed a part, you can edit its appearance, properties, or location. Once you have placed a power or ground symbol, you can rotate it or edit its name.



Capture also includes symbols used to establish connectivity between other schematic pages. You use off-page connectors to connect signals between schematic pages within a schematic. To connect signals from one schematic to another, you use hierarchical blocks and hierarchical ports.

Wires and buses are used to conduct signals between parts and electrical objects. A wire represents one net; a bus represents multiple wires.

This chapter contains information about placing and editing all of these objects. It also explains how to connect these objects using wires and buses.

Placing and editing parts

A part is a graphical representation of a single electrical object. Capture includes libraries with over 20,000 parts that you can place on your schematics. In addition, you can create your own parts.

See For information about creating your own parts, see *Chapter 11: Creating and editing parts.*

A part defines how an electrical object is packaged. For example, a part can consist of several logical components that are placed on a schematic page. This type of part is called a *package*. The different parts that make up a package can be identical in their graphic appearance and electrical connectivity (in which case it is called a *homogeneous* package) or they can be dissimilar in their graphic appearance or electrical connectivity (in which case it is called a *heterogeneous* package).

Each part has a set of properties that contain information—such as part value and reference designator—used by layout or simulation tools. In addition, you can create your own unique part properties to carry information important to your application.

Parts have pins that define the part's electrical connectivity. Pins carry information in properties that define the characteristics of each pin. This information includes the pin's name, number, shape (clock, dot, dot-clock, line, short, or zero length), type (3 state, bidirectional, input, open collector, open emitter, output, passive, or power), width, and visibility. The pin type is used by the Design Rules Check command on the Tools menu to check conformance to basic electrical rules.

Tip A part doesn't have to have pins. If a part doesn't have pins, it is listed in a bill of materials report, but doesn't appear in a netlist. This is useful if you want to show hardware—such as screws, nuts, or washers—in a bill of materials report.

A primitive part is a basic part without any underlying hierarchy. A nonprimitive part is a part that has an underlying hierarchy, such as an attached schematic. Placing a nonprimitive part adds all the schematic pages that are part of the attached schematic to your design, making it easy to add levels of hierarchy to your design.

Tip When placing objects of any kind in Capture—whether parts, electrical symbols, or graphics—that are slightly different, you can quickly place multiple copies of the object with minor changes. Just click the right mouse button and choose Edit from the pop-up menu. You can modify the object, then place additional copies of the object.

Placing parts

Instances of parts are selected from part libraries and placed on a schematic page using the Part command from the Place menu, or using the part selector tool on the tool palette.

To place a part

1. From the schematic page editor's Place menu, choose Part (ALT, P, P).
   or
   Choose the part selector tool on the schematic page editor's tool palette.

   The Place Part dialog box displays. You use this dialog box to choose a part to place.
2. Select a part from the list that displays.
   or
   In the Part text box, type the name of the part to place. If you aren't sure of the exact name of the part, you can enter wildcard characters to constrain the list of parts. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters.

   After you type the name of the part to place, press TAB. All parts in the libraries (listed in the Libraries list box) that match the part name are listed in the box below the Part text box. When you select a part from this box, its graphic image displays.

Tips You can add more libraries to the Libraries list box by choosing the Add Library button. Capture displays a standard Open dialog box that you can use to locate a library to add to the list.

You can remove a library from the Libraries list box by selecting it and choosing the Remove Library button.

You can switch between the Normal view of a part and the Convert view of a part by choosing the appropriate radio button in the Graphic section.

If your part is a package that contains multiple parts, you can use the Part drop-down list in the Packaging area to select which part in the package to view.

All of the options on the Place Part dialog box are described later in this section.

---

3. When you have located the part you want to place, choose the OK button.

   An image of the selected part is attached to your pointer. You can press the right mouse button to display a pop-up menu with commands that you can use to change the attributes of the part before you place it. You can mirror the part horizontally or vertically, rotate the part, edit the part's properties, and change between the normal and convert view of the part. If the part is a package, you can view a different part in a package.

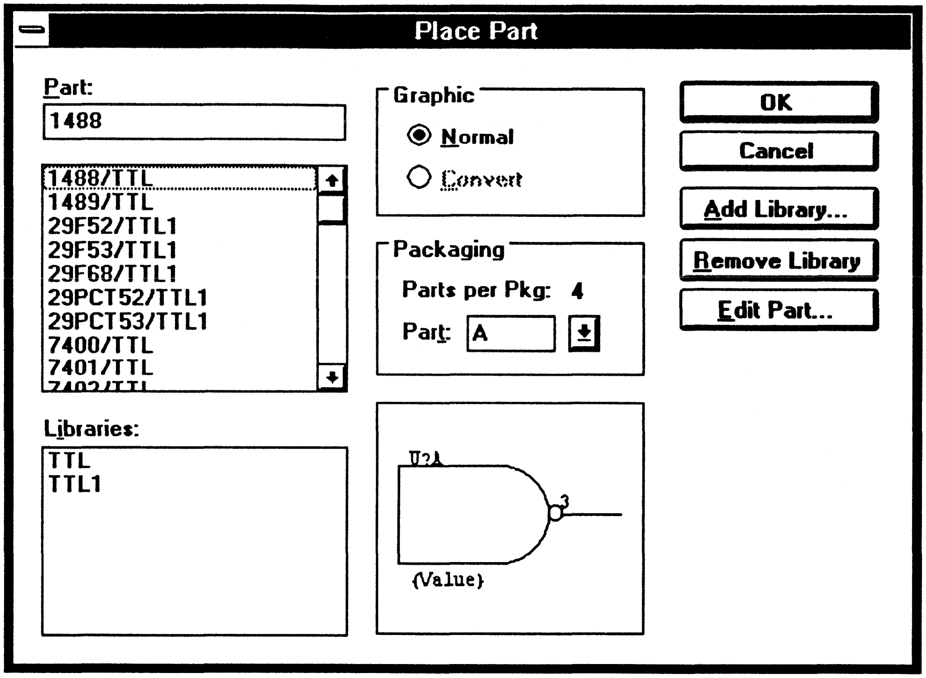
Note All objects that you can place on a schematic page have right mouse button pop-up menus. These menus are context sensitive, meaning they display commands that are appropriate for the selected object. For information about the commands on a pop-up menu, see Capture's online help.

4. Move the pointer to the location on your schematic page where you want the part, and click the left mouse button. This places an instance of the part on your schematic page.

   You can place multiple instances of the part by clicking the left mouse button each place you want an instance of the part.

5. When you are done placing instances of the selected part, choose the selection tool or press ESC to dismiss the part selector tool.

Place Part dialog box



Part Specifies the name of the part. If you aren't sure of the exact name of the part, you can enter wildcard characters to constrain the list of parts. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters. The names of all parts in the selected libraries that match the wildcard appear in the Part list box.

Part list Lists the names of all parts in the selected libraries that match the name entered in the Part text box. If more than one library is selected, the part name is followed by a slash (/) and a library name. When you select a part in this list, its name displays in the Part text box, and its graphic displays in the preview box.

Libraries Lists the library names currently available. All parts in the selected libraries that match the Part text box display in the Part list. To select more than one library, press CTRL while you click the mouse.

Graphic You can choose the view of the selected part: Normal or Convert. Some parts have a Convert view that is used for things such as a DeMorgan equivalent view of a part.

Packaging If a part contains two or more parts, it is called a *package*. The Parts per Pkg field displays the number of parts in the selected package. The Part field is used to select which part in the package you wish to view or place.

Preview box Displays the graphic of the selected part.

Add Library Displays a standard Open dialog box that you can use to locate a library and add it to the Libraries list.

Note Whenever you open a library in Capture, it is automatically added to the library list. These library names are stored in the CAPTURE.INI file, allowing you to configure and save the list of libraries you commonly use.

Note If you select an SDT 386+ or SDT Release IV library from the dialog box that displays when you choose Add Library, Capture automatically translates the file. You must specify the name of the new Capture library before the translation begins. A progress indicator displays in Capture's status bar.

Remove Library Removes the selected libraries from the Libraries list.

Edit Part Opens a part editor window for the selected part, and a design manager window for the part's library.

Editing parts

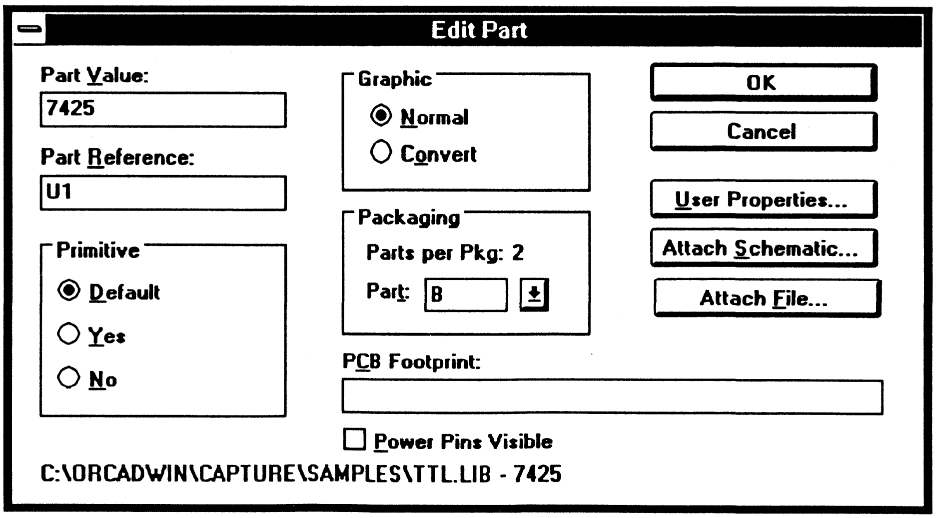
Once you have placed a part on a schematic page, you can move it by selecting it and dragging it to a new location. You can rotate a part using the Rotate command from the Edit menu, or you can mirror a part using the Mirror command from the Edit menu. You can invoke the part editor to change the part's physical appearance, and you can edit the part's properties. When you edit a part on a schematic page, you make a local part that differs from the part in the library and exists only in that design; the only way to place another copy of this part is to use the Copy command from the Edit menu.

See also For more information about editing parts, see *Chapter 11: Creating and editing parts.*

To edit the physical appearance of a part, select the part on the schematic page, and either choose Part from the Edit menu or choose Edit Part from the right mouse button pop-up menu. This opens the part in a part editor window. After you finish editing the part and choose Save, you're given a choice of updating the single instance, or updating all instances in the design. If you update only the single instance, Capture creates a new part in the design cache. If you update all instances, Capture replaces the library part in the design cache with your edited part.

To edit the properties of a part, select the part on the schematic page, and either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the part. This displays the Edit Part dialog box, which is described below.

Edit Part dialog box



Part Value Specifies the part value name. By default, the part value is set to the name of the part if you don't specify a part value in the library.

Part Reference Specifies the part reference.

Primitive Default indicates that the part uses the default setting that is set in the Hierarchy tab of the Design Template dialog box. Yes indicates that the part is primitive. No indicates that the part is nonprimitive.

Graphic Indicates whether the part you are editing is in Normal view or in Convert view.

Packaging Parts per package indicates the number of parts in the package you are editing. Part indicates which part of a multiple-part package you are editing.

PCB Footprint The PCB physical package name to be included for this part in the netlist.

Power Pins Visible Specifies the visibility of the part's power pins. See Capture's online help for information on power pin visibility and how it affects a global power net.

Note The path and filename of the library that contains the part are displayed at the bottom left corner of the dialog box.

User Properties Displays a dialog box that you can use to edit the part's property names and their respective property values.

Attach Schematic You can attach a schematic to create a hierarchy. When you choose this button, a dialog box displays that you use to specify the name of a schematic and the library or design that contains it.

Note If you specify a library or design that you haven't yet saved to disk, Capture creates the library or design in the directory specified by your TEMP environment variable.

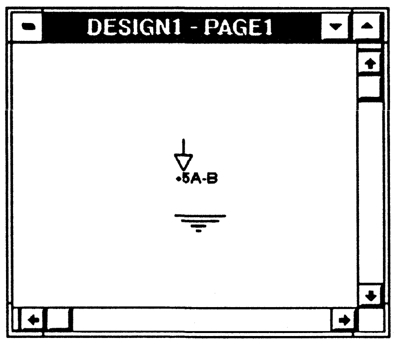
Attach File You can attach a text file that contains related information—such as PLD source code. There are no format restrictions for attached files. When you choose Attach File, a dialog box displays that you use to specify the name of the file.

Caution An attached schematic or other file is not stored with the design or library. If you copy or move the design or library to a new location, you must also move or copy the attached file to keep the two files together. In addition, you may need to edit the path to the attached schematic or file if you move the design to a new location with a different directory structure.

Placing and editing power and ground symbols

You can place power and ground symbols, and once they're placed, you can edit their names. You can also edit the text associated with the symbols. The name of a power symbol is the name of the global net that gets created.

Placing power and ground symbols



Power and ground symbols are placed on a schematic page using the Power command or Ground command on the Place menu, or using the power tool or ground tool on the tool palette. Power and ground symbols are selected from symbol libraries in a manner similar to selecting parts from part libraries.

To place a power symbol

1 From the schematic page editor's Place menu, choose Power (ALT, P, O).
or
Choose the power tool on the schematic page editor's tool palette.

The Place Power dialog box displays. You use this dialog box to choose a power symbol to place.

2 In the Symbol text box, type the name of the symbol to place. If you aren't sure of the exact name of the symbol, you can enter wildcard characters to constrain the list of symbols. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters.

After you type the name of the power symbol to place, press TAB. All power symbols in the libraries listed in the Libraries list box that match the name of the power symbol are listed in the box below the Power Symbol text box. When you select a symbol from this box, its graphic image displays.

Tips You can add more libraries to the Libraries list box by choosing the Browse button. Capture displays a standard Open dialog box that you can use to locate a library to add to the list.

You can remove a library from the Libraries list box by selecting it and choosing the Remove Library button.

You can assign a name (such as +5, GND, +5VDC, -12VDC, VSS, or VEE) by typing it in the Name text box. You can also assign a name after the power symbol is placed.

All of the options on the Place Power dialog box are described later in this section.

3 When you have located the power symbol you wish to place, choose the OK button.

An image of the power symbol is attached to your pointer. You can press the right mouse button to display a pop-up menu with commands that you can use to change the attributes of the power symbol before you place it. You can mirror the power symbol horizontally or vertically, rotate the power symbol, and edit the power symbol's properties.

Note All objects that you can place on a schematic page have right mouse button pop-up menus. These menus are context sensitive, meaning they display commands that are appropriate for the selected object. For information about the commands on a pop-up menu, see Capture's online help.

4 Move the pointer to the location on your schematic page where you want the power symbol and click the left mouse button. This places the power symbol on your schematic page.

   You can place multiple instances of the power symbol by clicking the left mouse button each place you want an instance of the symbol.

5 When you are done placing power symbols, choose the selection tool or press ESC to dismiss the power tool.

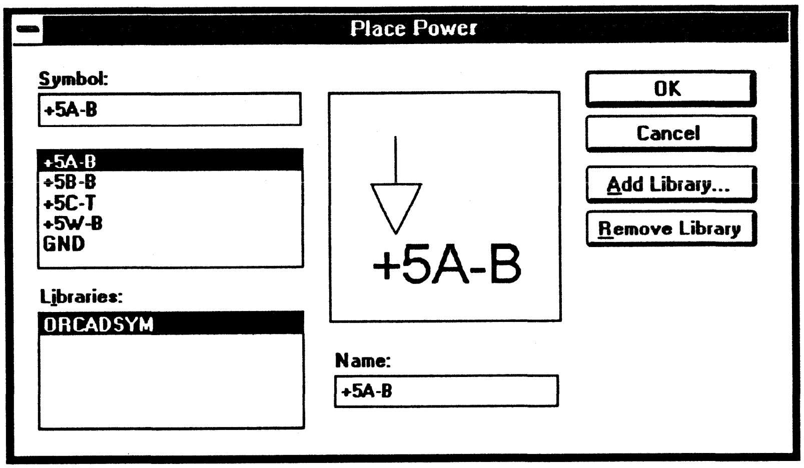
Note You can create custom power, ground, and other symbols for hierarchical ports, off-page connectors, title blocks, and power objects by using the New Symbol command from the Design menu in the design manager window. For information on how to use this command, see Capture's online help.

To place a ground symbol

Follow the instruction in the previous section, *To place a power symbol*, but substitute the Ground command or the ground tool in the appropriate places.

Place Power *or* Place Ground dialog box

The Place Power and Place Ground dialog boxes are identical, except that each displays the last power or ground symbol you placed. This figure shows the Place Power dialog box.



Symbol Specifies the name of the power or ground symbol in the library. If you aren't sure of the exact name of the symbol, you can enter wildcard characters to constrain the list of symbols. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters. The names of all symbols in the selected libraries that match the wildcard appear in the Symbol list box.

Symbol list Lists the names of all symbols in the selected libraries that match the name entered in the Symbol text box. If more than one library is selected, the symbol name is followed by a slash (/) and a library name. When you select a symbol in this list, its name displays in the Symbol text box, and its graphic displays in the preview box.

Libraries Lists the library names currently available. Select the libraries from which to select power or ground symbols. All symbols in the selected libraries that match the Symbol text box display in the Symbol list. To select more than one library, press CTRL while you click the mouse.

Preview box Displays the graphic of the selected symbol.

Add Library Displays a standard Open dialog box that you can use to locate a library and add it to the Libraries list.

Remove Library Removes the selected libraries from the Libraries list.

Name Assigns a name—such as +5, GND, +5VDC, -12 VDC, VSS, or VEE—to the symbol.

Editing power and ground symbols

You can change the name of a power or ground symbol by selecting the symbol on the schematic page, and either choosing Properties from the Edit menu, or choosing Edit from the right mouse button pop-up menu. You can also double-click on the symbol. This displays the Rename Global dialog box. Once you have finished editing the name, choose the OK button.

You can also edit the display properties of the text associated with the power or ground symbol. Select only the text of the symbol, then either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the symbol. This displays a Display Properties dialog box. Once you have finished editing the properties, choose the OK button. You can also use this dialog box to edit the name of the symbol.

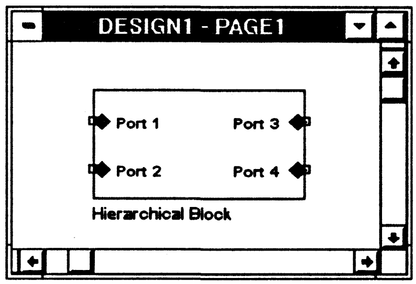
You cannot assign user properties to power or ground symbols.

Placing and editing hierarchical blocks

A hierarchical block is a representation of a schematic, which is attached to the hierarchical block. It provides vertical (downward-pointing) connection only. The hierarchical ports in a hierarchical block act as points of attachment for electrical connections between the hierarchical block and other electrical objects on the schematic page. Each hierarchical port in a hierarchical block corresponds to at least one hierarchical port in the attached schematic. A hierarchical block functions just like a part with an attached schematic.

Placing hierarchical blocks

You create hierarchical designs using hierarchical blocks to represent child schematics. When you create a hierarchical block, you specify the name of the child schematic that the hierarchical block represents. Once you've created the hierarchical block, you place hierarchical ports inside it to connect it to hierarchical ports on the child schematic.



See also For information about how hierarchical designs are connected using hierarchical blocks and hierarchical ports, see *Connecting designs* in *Chapter 8: Design structure*.

To place a hierarchical block

1. From the schematic page editor's Place menu, choose Hierarchical Block (ALT, P, H).
   or
   Choose the hierarchical block tool on the schematic page editor's tool palette.

   The Place Hierarchical Block dialog box displays. You use this dialog box to define characteristics of the hierarchical block.
2. In the Name text box, type the name of the hierarchical block.
3. Accept the Primitive setting of Default, or choose Yes or No.
4. Choose the User Properties button to display a dialog box where you can add or change property names and their associated values. When you've finished editing the hierarchical block's properties, choose the OK button. The Place Hierarchical Block dialog box displays.
5. Choose the Attach Schematic button to display a dialog box. Use this dialog box to locate the schematic that the hierarchical block points to. When you've located the schematic, choose the OK button. The Place Hierarchical Block dialog box displays.
6. Choose the Attach File button to display a dialog box where you can specify the text file, such as PLD source code, that defines this part.

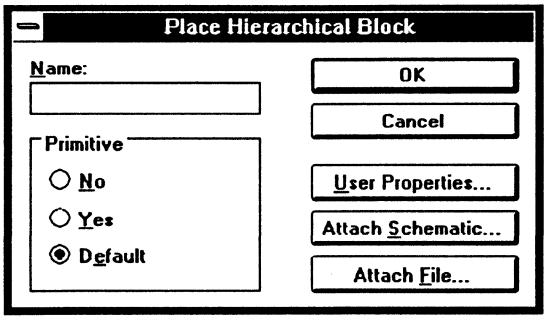
7 When you have specified the characteristics of the hierarchical block, choose the OK button. The Place Hierarchical Block dialog box closes.

8 Draw the hierarchical block. Press the left mouse button, drag the mouse to draw a rectangle, and release the mouse button when the rectangle is the desired size.

Tip Once you've placed a hierarchical block, you must define hierarchical ports to connect to hierarchical ports in the child schematic. For the hierarchical ports to be part of the hierarchical block, the hierarchical block must be selected when you place the hierarchical ports. Placing hierarchical ports is described in the next section.

After you draw the hierarchical block, you can click anywhere in open space to deselect it. Notice that the name of the hierarchical block displays below the hierarchical block.

Place Hierarchical Block dialog box



Name The name of the hierarchical block.

Primitive Default indicates that the part uses the default setting that is set on the Hierarchy tab of the Design Template dialog box. Yes indicates that the part is primitive. No indicates that the part is nonprimitive.

User Properties Displays a dialog box that you can use to edit the part's property names and their respective property values.

Attach Schematic You can attach a schematic to create a hierarchy. When you choose this button, a dialog box displays that you use to specify the name of a schematic and the library or design that contains it.

Note If you specify a library or design that you haven't yet saved to disk, Capture creates the library or design in the directory specified by your TEMP environment variable.

Attach File You can attach a text file that contains related information—such as PLD source code. There are no format restrictions for attached files. When you choose Attach File, a dialog box displays that you use to specify the name of the text file.

Caution An attached schematic or other file is not stored with the design or library. If you copy or move the design or library to a new location, you must also move or copy the attached file to keep the two files together. In addition, you may need to edit the path to the attached schematic or file if you move the design to a new location with a different directory structure.

Editing hierarchical blocks

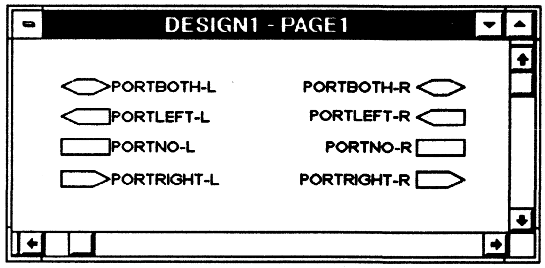
You can edit the hierarchical block after it is placed by selecting the block on the schematic page, and either choosing Properties from the Edit menu, or choosing Edit from the right mouse button pop-up menu. You can also double-click on the block. This displays the Edit Hierarchical Block dialog box, which lets you change the information for Name, Primitive, User Properties, Attach Schematic, and Attach File. Once you have finished editing the information in the Edit Hierarchical Block dialog box, choose the OK button.

You can also edit the display properties of the text associated with the hierarchical block. Select only the text of the block, then either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the block. This displays a Display Properties dialog box. Once you have finished editing the properties, choose the OK button.

You can click on a hierarchical block and move it to another location, or you can drag its selection rectangles to change its size. You can also use both the Mirror and Rotate commands from the Edit menu to change the appearance of the block.

Placing and editing hierarchical ports

Placing hierarchical ports



Hierarchical ports are used to connect signals to hierarchical ports on other schematics and schematic pages. On a parent schematic, hierarchical ports inside a hierarchical block correspond to hierarchical ports on the schematic specified by the hierarchical block. On a child schematic, hierarchical ports outside a hierarchical block correspond to hierarchical ports inside a hierarchical block on the parent schematic. Outside a hierarchical block, hierarchical ports also connect to hierarchical ports and off-page connectors (with the same name or alias) on other schematic pages within the same schematic.

Caution There are two ways you can place an off-grid hierarchical port on a hierarchical block:

- If the Pointer snap to grid option on the Grid Display tab of the Preferences dialog box is not selected.
- If the hierarchical block is off grid, any hierarchical ports you place on the hierarchical block will not snap to grid, regardless of the setting of the Pointer snap to grid option.

See For information about how hierarchical designs are connected using hierarchical blocks and hierarchical ports, see *Connecting designs* in *Chapter 3: Design structure.*

To place a hierarchical port inside a hierarchical block

Tip If you have a hierarchical block selected, you can only place the hierarchical port within the boundaries of the hierarchical block. This means that the hierarchical port connects to all other free-standing, like-named hierarchical ports in the schematic attached to the hierarchical block.

1 Select a hierarchical block.

2 From the schematic page editor's Place menu, choose Hierarchical Port (ALT, P, I).
or
Choose the hierarchical port tool on the schematic page editor's tool palette.

The Place Hierarchical Port dialog box displays. You use this dialog box to define characteristics of the hierarchical port.

3 In the Name text box, type in the name for the hierarchical port. This name, which is also the net name, is used to determine which like-named hierarchical ports the port will connect to.

4 From the drop-down list under Type, select the pin type for the hierarchical port.

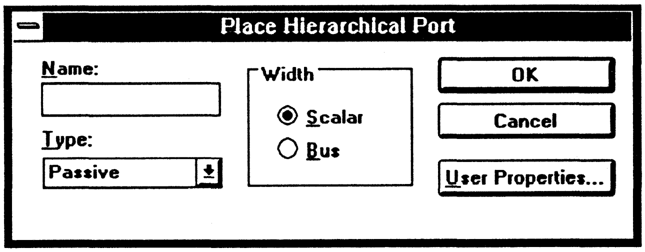
5 In the Width area, select either Scalar or Bus.

6 Choose the User Properties button to display a dialog box where you can add or change property names and their associated values. When you've finished editing the hierarchical block's properties, choose the OK button. The Place Hierarchical Port dialog box displays.

7 When you have specified the characteristics of the hierarchical port, choose the OK button. The Place Hierarchical Port dialog box closes.

8 You can now place the hierarchical port within the selected hierarchical block.

Place Hierarchical Port dialog box (inside a hierarchical block)



Name Specifies the hierarchical port's name.

Type Specifies the type of port.

Width Specifies whether the port is Scalar or Bus.

User Properties Displays a dialog box that you can use to edit the port's property names and their respective property values.

To place a hierarchical port outside a hierarchical block

Tip If you don't have a hierarchical block selected, you can place the hierarchical port anywhere on the schematic page. This means that the hierarchical port connects to all other like-named hierarchical ports and off-page connectors on schematic pages in the same schematic and in the parent schematic.

---

1 From the schematic page editor's Place menu, choose Hierarchical Port (ALT, P, I).
or
Choose the hierarchical port tool on the schematic page editor's tool palette.

The Place Hierarchical Port dialog box displays. You use this dialog box to define characteristics of the hierarchical port.

2 In the list below the Symbol text box, select the type of the hierarchical port. The selection's name appears in the Symbol text box and its graphic displays in the preview box.

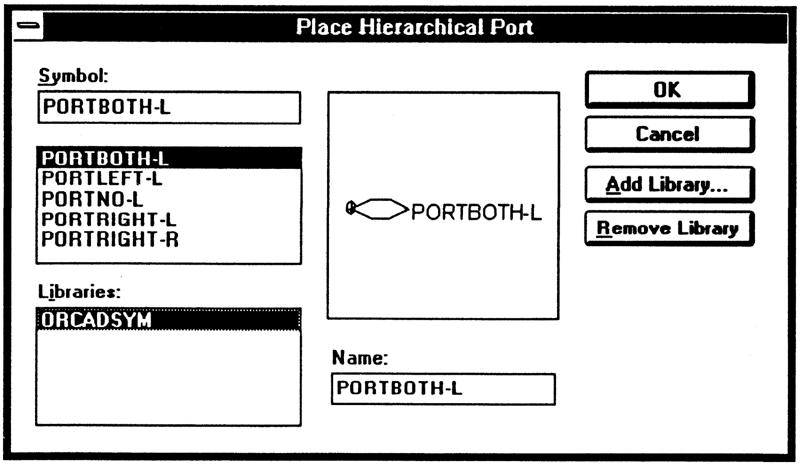
3 Select one or more libraries from those listed in the Libraries field, or use the Add Library and Remove Library buttons to change what is listed in the Libraries field, then select one or more libraries.

4 Type in the name for the hierarchical port. This name, which is also the net name, is used to determine which like-named hierarchical ports the port will connect to.

5 When you have specified the characteristics of the hierarchical port, choose the OK button. The Place Hierarchical Port dialog box closes.

6 You can now place the hierarchical port anywhere on the schematic page.

Place Hierarchical Port dialog box (outside a hierarchical block)



Symbol Specifies the hierarchical port symbol to use. If you aren't sure of the exact name of the symbol, you can enter wildcard characters to constrain the list of symbols. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters. The names of all symbols in the selected libraries that match the wildcard appear in the Symbol list box.

Symbol list Lists the names of all symbols in the selected libraries that match the text entered in the Symbol text box. If more than one library is selected, the symbol name is followed by a slash (/) and a library name. When you select a part in this list, its name displays in the Symbol text box, and its graphic displays in the preview box.

Libraries Lists the library names currently available. Select the libraries from which to select symbols. All symbols in the selected libraries that match the Symbol text box display in the Symbol list. To select more than one library, press CTRL while you click the mouse.

Preview box Displays the graphic of the selected symbol.

Name Specifies the symbol's name. This name is used to determine which like-named hierarchical ports the port will connect to.

Add Library Displays a standard Open dialog box that you can use to locate a library and add it to the Libraries list.

Remove Library Removes the selected libraries from the Libraries list.

Editing hierarchical ports

Note You can create custom power, ground, and other symbols for hierarchical ports, off-page connectors, title blocks, and power objects by using the New Symbol command from the Design menu in the design manager window. For information on how to use this command, see Capture's online help.

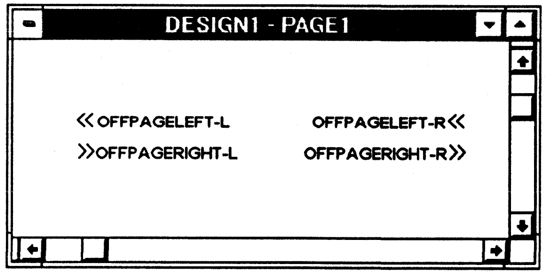
You can edit a hierarchical port after it is placed by selecting the port and either choosing Properties from the Edit menu, or choosing Edit from the right mouse button pop-up menu. You can also double-click on the port. This displays the Edit Port dialog box, which lets you change the information for Name and Type. Once you have finished editing the information in the Edit Port dialog box, choose the OK button.

You can also edit the display properties of the text associated with the hierarchical port. Select only the text of the port, then either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the port. This displays a Display Properties dialog box. Once you have finished editing the properties, choose the OK button.

You can drag a hierarchical port to another location.

Placing and editing off-page connectors

Off-page connectors are used to connect signals to like-named off-page connectors and hierarchical ports on other schematic pages within the same schematic.



See also For more information about how designs are connected using off-page connectors, see *Connecting designs* in *Chapter 8: Design structure.*

Placing off-page connectors

To place an off-page connector, you use the Off-Page Connector command on the Place menu or the off-page connector tool on the tool palette.

To place an off-page connector

1 From the schematic page editor's Place menu, choose Off-Page Connector (ALT, P, F).
or
Choose the off-page connector tool on the schematic page editor's tool palette.

The Place Off-Page Connector dialog box displays. You use this dialog box to choose an off-page connector symbol to place.

2 In the Symbol text box, type the name of the symbol to place. If you aren't sure of the exact name of the symbol, you can enter wildcard characters to constrain the list of symbols. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters.

After you type the name of the symbol to place, press TAB. All symbols in the libraries listed in the Libraries list box that match the symbol name are listed in the box below the Symbol text box. When you select a symbol from this box, its graphic image displays.

Tips You can add more libraries to the Libraries list box by choosing the Add Library button. Capture displays a standard Open dialog box that you can use to locate a library to add to the list.

You can remove a library from the Libraries list box by selecting it and choosing the Remove Library button.

You can assign a name by typing it in the Name text box. The name is used to connect to other off-page connectors in the same schematic. You can also assign a name after the symbol is placed.

All of the options on the Place Off-Page Connector dialog box are described later in this section.

3. When you have located the symbol you wish to place, choose the OK button.

   An image of the symbol is attached to your pointer. You can press the right mouse button to display a pop-up menu with commands that you can use to change the appearance of the symbol before you place it. You can mirror the symbol horizontally or vertically, rotate the symbol, edit the symbol in a part editor window, and edit the symbol's properties.

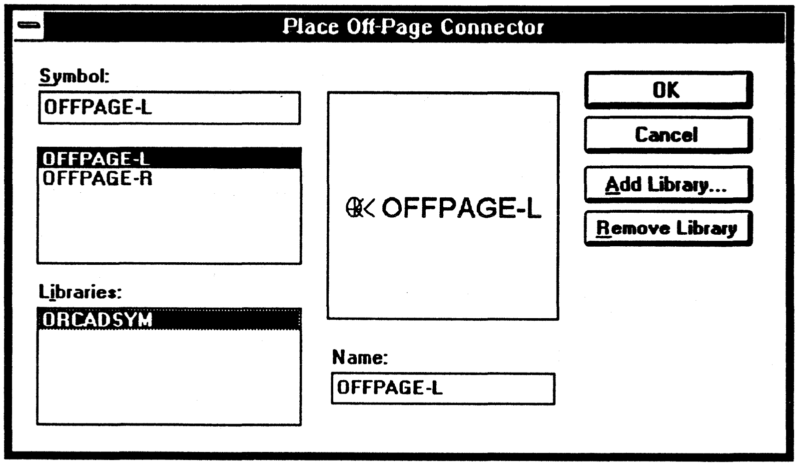
Note All objects that you can place on a schematic page have right mouse button pop-up menus. These menus are context sensitive, meaning they display commands that are appropriate for the selected object. For information about the commands on a pop-up menu, see Capture's online help.

4. Move the pointer to the location on your schematic page where you want the symbol and click the left mouse button. This places the symbol on your schematic page.

   You can place multiple instances of the symbol by clicking the left mouse button each place you want an instance of the symbol.

5. When you are done placing symbols, choose the selection tool or press ESC to dismiss the off-page connector tool.

Place Off-Page Connector dialog box



Symbol Specifies the off-page connector symbol to use. If you aren't sure of the exact name of the symbol, you can enter wildcard characters to constrain the list of symbols. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters. The names of all symbols in the selected libraries that match the wildcard appear in the Symbol list box.

Symbol list Lists the names of all symbols in the selected libraries that match the text entered in the Symbol text box. If more than one library is selected, the symbol name is followed by a slash (/) and a library name. When you select a part in this list, its name displays in the Symbol text box, and its graphic displays in the preview box.

Libraries Lists the library names currently available. Select the libraries from which to select symbols. All symbols in the selected libraries that match the Symbol text box display in the Symbol list. To select more than one library, press CTRL while you click the mouse.

Preview box Displays the graphic of the selected symbol.

Name The name of the symbol. Other off-page connectors in this schematic that have this name are connected to this off-page connector.

Add Library Displays a standard Open dialog box that you can use to locate a library and add it to the Libraries list.

Remove Library Removes the selected libraries from the Libraries list.

Editing off-page connectors

You can edit an off-page connector after it is placed by selecting it and either choosing Properties from the Edit menu, or choosing Edit from the right mouse button pop-up menu. You can also double-click on the off-page connector. This displays the Edit Off-Page Connector dialog box, which lets you change the Name. Once you have finished editing the information in the Edit Off-Page Connector dialog box, choose the OK button.

You can also edit the display properties of the text associated with the off-page connector. Select only the text of the off-page connector, then either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the off-page connector. This displays a Display Properties dialog box. Once you have finished editing the properties, choose the OK button.

You can click on an off-page connector and move it to another location. You can also use both the Mirror and Rotate commands from the Edit menu to change the appearance of the off-page connector.

Note You can create custom power, ground, and other symbols for hierarchical ports, off-page connectors, title blocks, and power objects by using the New Symbol command from the Design menu in the design manager window. For information on how to use this command, see Capture's online help.

Placing and editing wires and buses

As you place wires and buses, remember the following points:

- A wire and a bus can be connected only by name.

  If you begin or end a wire segment on a segment of a bus, a vertex is added to the bus, but no junction displays—the wire and bus are *not* connected.

  If you begin or end a bus segment on a segment of a wire, a vertex is added to the wire, but no junction displays—the bus and wire are *not* connected.

- Two wires or two buses can be connected physically.

  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 displays—the wires are connected.

  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 displays—the buses are connected.

Wires and buses, along with other parts and symbols in the design that are logically connected via net names, form a net. When you place a wire, it is assigned a system-generated netname, which you can replace by an alias or a different netname. Once a bus acquires a valid name or alias, then that name or alias defines the signals carried by the bus and connects those signals to the corresponding nets. For example, the alias A[0:3] defines a four-signal bus and connects the four signals it carries—A[0], A[1], A[2], and A[3]—with nets A0, A1, A2, and A3.

Like wires, buses can acquire names and aliases by two means:

- Direct application of a valid bus name
- Electrical connection to a hierarchical port, off-page connector, or global bus pin with a valid bus name or alias

See also For more information about how placing and connecting wires and buses, see Capture's online help.

Placing wires

When you connect a wire to a pin, Capture provides visual confirmation of the connection by removing the no-connect symbol on the pin. If two wires cross at 90°, they are not electrically connected, unless you create a junction by clicking the left mouse button on one wire as you draw the other across it.

Note If you place parts so that two pins meet end to end, the pins are connected.

Tip You can find out the name of the net a pin is on. Double-click on the pin, then choose the User Properties button in the Pin Properties dialog box. The Net Name property is the name of the net.

To place a wire

1. From the schematic page editor's Place menu, choose Wire (ALT, P, W).
   or
   Choose the wire tool on the schematic page editor's tool palette.
2. Click the left mouse button to start the wire.
3. Move the mouse to draw the wire. Click the left mouse button if you want to place a vertex and change directions, or connect to another wire as you pass over it. The wire is constrained to multiples of 90° unless you hold down the SHIFT key while you draw the wire.
4. Double-click to end the wire. The wire displays in the selection color.

Tip If you wish to place more than one wire, do not double-click. Instead, click the left mouse button, press ESC to end the wire, then begin at step 2.

5. When you are done placing wires, choose the selection tool or press ESC to dismiss the wire tool.

Editing wires

Select the wire and either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the wire. This displays the Net Properties dialog box, which lets you add or change the net's properties. Once you have finished editing the information in the Net Properties dialog box, choose the OK button.

Note When you click on a wire, all the graphical handles (vertices) on the wire are highlighted, but only the segment on which you click is actually selected. To select the entire wire, drag the mouse to select the area enclosing the net, then select one vertex. To select the entire net, click on the wire, then click the right mouse button and choose the Select Entire Net command from the pop-up menu.

You can also edit the display properties of the text associated with the wire. Select only the text of the wire, then either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the text. This displays a Display Properties dialog box. Once you have finished editing the properties, choose the OK button.

Tip To edit a net alias, double-click on the alias itself and update it.

To move a wire, select it and drag it to a new location; the wire stretches to maintain its connectivity. To break the wire's connectivity, press ALT while you move it. To move a vertex, select a wire segment next to the vertex and drag the vertex to the new location. Capture adds segments and vertices to the wire as needed to reach the new location.

See also For more information about editing wires and nets, see Capture's online help.

Placing buses

To place a bus

1 From the schematic page editor's Place menu, choose Bus (ALT, P, B).
or
Choose the bus tool on the schematic page editor's tool palette.

2 Click the left mouse button to start the bus.

3 Move the mouse to draw the bus. Click the left mouse button if you want to place a vertex and change directions, or to connect to another bus as you pass over it. The bus is constrained to multiples of 90° unless you hold down the SHIFT key while you draw the bus.

4 Double-click to end the bus.

5 Enter the alias for the bus in the Place Net Alias dialog box that displays, then choose the OK button. The bus displays in the selection color.

Note Bus names and aliases have the form *X*[*m..n*]. *X* represents the "base name." The portion *m..n* represents the range of signals carried by the bus. Note that *m* may be less than or greater than *n*. In other words, 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*.

---

6 When you are done placing buses, choose the selection tool or press ESC to dismiss the bus tool.

Editing buses

Select the bus and either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the bus. This displays the Net Properties dialog box, which lets you which lets you add or change the bus's properties. Once you have finished editing the information in the Net Properties dialog box, choose the OK button.

You can also edit the display properties of the text associated with the bus. Select only the text of the bus, then either choose Properties from the Edit menu, or choose Edit from the right mouse button pop-up menu. You can also double-click on the text. This displays the Edit Net Alias dialog box, which allows you to change the Alias, Color, Rotation, or Font. Once you have finished editing the properties, choose the OK button.

To move a bus, select it and drag it to a new location; the bus stretches to maintain its connectivity. To break the bus connectivity, press ALT while you move it.

Placing bus entries

Bus entries are used to bring a net into a bus. They are optional. The only distinction between a bus entry and a wire segment is that two bus entries that touch are not connected as two wires are.

To place a bus entry

1. Select the bus to which the bus entry will connect.
2. From the schematic page editor's Place menu, choose Bus Entry (ALT, P, E).
   or
   Choose the bus entry tool on the schematic page editor's tool palette.

   The bus entry is attached to the pointer.
3. From the Edit menu, choose Rotate (ALT, E, O) to rotate the bus entry 90° counterclockwise if the bus entry is not at the angle you need.
4. Use the mouse to position one end of the bus entry on the bus, then click the left mouse button to place the bus entry.
5. Repeat step 4 until all of the bus entries are placed.
6. When you are done placing bus entries, choose the selection tool or press ESC to dismiss the bus entry tool.

To connect multiple bus entries

1. Place a wire to connect the first bus entry to one net.
2. Place an alias. Be sure to assign the first bus entry the lowest value in the bus range.

Tip To place an alias, choose Net Alias from the Place menu. Enter the net alias text, following the naming conventions for buses and bus members, then choose the OK button. A rectangle representing the alias text is attached to the pointer. Click the left mouse button on the bus or net. The alias text displays in the selection color. Choose the selection tool or press ESC to dismiss the Net Alias tool.

---

3. Select both the wire and the alias text, then press CTRL while you drag a copy a specific distance so that it connects the next net to the bus. The alias value is increased by one.
4. From the Edit menu, choose the Repeat (ALT, E, R) command. The wire and the incremented alias are placed at the specified distance from the previous set.
5. Repeat step 4 for every net in the bus, or repeat steps 3 and 4 as needed.
6. When you are done connecting bus entries, choose the selection tool or press ESC.

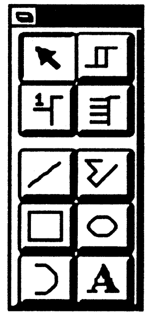
Editing bus entries

To move a bus entry's text, select it and drag it to a new location. To rotate a bus entry, select it and choose Rotate from the Edit menu.

Adding and editing graphics and text

You can create a wide variety of graphic shapes to add to your schematic pages. You can work with the snap-to-grid option turned on or turned off. For close work, you may want to zoom in on your graphic using the Go To command on the View menu.

Before you begin drawing, you may want to specify default line and fill styles, because all lines and shapes you draw will use the current line style, and closed shapes that you draw will use the current fill style. You can use a variety of line or fill styles for any schematic page or part. You can specify these styles on the Miscellaneous tab on the Preferences dialog box.



Part editor tool palette.

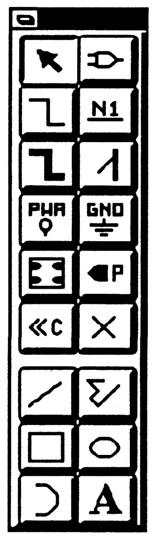
Drawing tools

You use tools on the tool palette to draw objects. There are two tool palettes: one for the schematic page editor window, and one for the part editor window. While many of the tools on these two tool palettes are the same, each tool palette has some unique tools.

The tool palettes are each divided into two groups of tools. The top tools are electrical. The bottom tools are drawing tools. The specific function of each tool is described in chapter 2.

General rules for all drawing tools:

- If you choose a drawing tool and press ESC before you begin drawing an object, the drawing tool changes to the selection tool.
- If you choose a drawing tool, and click in an empty area before you begin drawing an object, the drawing tool changes to the selection tool. Note that this doesn't work when drawing arcs or polylines/polygons, or placing text or bitmaps.
- If you choose a drawing tool, start drawing an object, and then press ESC the unfinished object goes away, but the drawing tool is still selected.
- If you choose a drawing tool, start drawing an object, and then press ESC twice, the unfinished object goes away, and the drawing tool changes to the selection tool.



Schematic page editor tool palette.

Drawing lines

You use the line tool to draw a single line. The line you draw adopts the current line style. If you wish to draw a line with multiple contiguous segments, use the polyline tool.

To draw a line segment

1. From the Place menu, choose Line (ALT, P, L).
   or
   Choose the line tool on the tool palette.
2. Move the pointer to the line's beginning.
3. Press and hold the left mouse button while moving the mouse to draw the line. To constrain the line angle to multiples of 90°, press SHIFT while you draw the line.
4. Release the left mouse button to end the line. The line displays in the selection color.
5. Click an area where there are no parts or objects to deselect the line.
6. Choose the selection tool or press ESC to dismiss the line tool.

To resize a line

Use the selection tool to select a line. Edit handles appear at either end of the line. With the pointer on one of the two edit handles, press the left mouse button and drag the line. You can change the angle of the line in the process of resizing it. Press the SHIFT key to constrain the line angle to multiples of 90°.

Drawing rectangles and squares

You use the rectangle tool to create orthogonal shapes; if you wish to create a polygon, use the polyline tool. To create a square, hold down the SHIFT key while you draw. Any rectangles or squares you create will have the current fill style and line style.

To draw a rectangle or a square

1. From the Place menu, choose Rectangle (ALT, P, R).
   or
   Choose the rectangle tool on the tool palette.
2. Move the pointer to one corner of the intended rectangle.
3. Press and hold the left mouse button while you drag the mouse. The rectangle changes shape as you move the mouse. Release the left mouse button when you have the correct shape. To draw a square, hold the SHIFT key while you perform this step. The rectangle displays in the selection color.
4. Click an area where there are no parts or objects to deselect the rectangle.
5. Choose the selection tool or press ESC to dismiss the rectangle tool.

To resize a rectangle or square

Use the selection tool to select a rectangle or square. Edit handles appear on the four corners. With the pointer on one of the handles, press the mouse and drag. Press SHIFT to constrain the rectangle to a square.

Drawing circles and ellipses

You use the ellipse tool to draw a closed ellipse; if you wish to draw an arc, use the arc tool. To draw a circle, hold down the SHIFT key while you drag the mouse. Because they are closed shapes, circles and ellipses will have the current fill style. They will also have the current line style.

To draw an ellipse or a circle

1. From the Place menu, choose Ellipse (ALT, P, S).
   or
   Choose the ellipse tool on the tool palette.
2. Move the pointer to an edge of the intended ellipse.
3. Press and hold the left mouse button while you drag the mouse. The ellipse changes shape as you move the mouse. If you wish to draw a circle, hold down the SHIFT key while you perform this step. Release the left mouse button when you have the correct shape. The ellipse appears in the selection color.
4. Click an area where there are no parts or objects to deselect the ellipse.
5. Choose the selection tool or press ESC to dismiss the ellipse tool.

To resize an ellipse or circle

Use the selection tool to select an ellipse or a circle. Edit handles appear on all four corners of the rectangle that encloses it. With the pointer on one of the handles, press the mouse and drag. Press SHIFT to constrain the ellipse to a circle.

Drawing arcs

You create an arc of any angle using the arc tool. If you wish to create a full circle, you can use the ellipse tool. Because it is a line, the arc adopts the current line style. Drawing an arc is done in three stages: you establish the center of the arc with the first mouse click, you establish the radius of the arc with the second mouse click, and you establish the arc segment endpoint with the third mouse click.

To draw an arc

1. From the Place menu, choose Arc (ALT, P, A).
   or
   Choose the arc tool on the tool palette.
2. Move the pointer to the center of the arc, and press and hold the left mouse button.
3. Drag the mouse to establish the radius of the arc and click the left mouse button to establish the location of one end of the arc.
4. Use the mouse to establish the other end of the arc and click the left mouse button. The arc is drawn counterclockwise from the end point, and displays in the selection color.
5. Click an area where there are no parts or objects to deselect the arc.
6. Choose the selection tool or press ESC to dismiss the arc tool.

To resize an arc

Use the selection tool to select an arc. Edit handles appear at the ends of the arc. With the pointer on one of the handles, press the mouse and drag. The center remains the same. The other arc endpoint uses the new radius. Press SHIFT to constrain the arc to the same radius at which it was drawn.

Drawing polylines and polygons

To draw a line with multiple contiguous segments, use the polyline tool. The line you draw adopts the current line style. Polygons can be created with the polyline tool, and they adopt the current fill style. To create an orthogonal polyline, hold down the SHIFT key while you draw.

To draw a polyline

1. From the Place menu, choose Polyline (ALT, P, Y).
   or
   Choose the polyline tool on the tool palette.
2. Click the left mouse button to begin drawing, click to change directions, and double-click to end the final segment. To constrain the direction changes to multiples of 90°, press SHIFT. After you double-click, the polyline displays in the selection color.
3. Click an area where there are no parts or objects to deselect the polyline.
4. Choose the selection tool or press ESC to dismiss the polyline tool.

To draw a polygon

➥ Follow the instructions above, ending the line with a single mouse-button click at the beginning point. The polygon adopts the current line and fill style.

To resize a polyline or polygon

Use the selection tool to select a polyline or polygon. Edit handles appear at the ends of all the lines in the polyline/polygon. With the pointer on a handle, press the left mouse button and drag. For polylines: you can change the line angle, and SHIFT constrains the line angle to multiples of 90°.

Adding fill to an object

To add fill to an object, select the object, then from the Edit menu, choose Properties. Select a fill style from the Fill Style drop-down box, then choose the OK button.

You can have closed shapes automatically filled in after you finish drawing them by defining a default fill. From the Options menu, choose the Preferences command, then choose the Miscellaneous tab. Click on the Fill Style drop-down box to display the options. Note that you can specify separate options for the schematic page editor and the part editor. Select one of the options and choose the OK button.

Mirroring an object

You can mirror objects horizontally, vertically, or both horizontally and vertically. Some objects, such as text and bitmaps, cannot be mirrored. If the Mirror command appears dimmed on the Edit menu, the object cannot be mirrored.

To mirror an object, select the object. From the Edit menu, choose Mirror, then choose Horizontally, Vertically or Both from the pop-up menu. The object flips in the chosen direction.

Rotating an object

You can rotate objects by 90° increments. Some objects, such as bitmaps, cannot be rotated. If the Rotate command appears dimmed on the Edit menu, the object cannot be rotated.

To rotate an object, select the object. From the Edit menu, choose Rotate. The selection rotates 90° counterclockwise.

Cutting an object

Select the object. From the Edit menu, choose Cut. The object is removed from the schematic and placed on the Clipboard.

Copying an object

There are two ways to copy an object:

- Select the object. From the Edit menu, choose Copy. The object remains on the schematic and a copy of it is placed on the Clipboard. It can be pasted in Capture or in other Windows programs.
- Position the pointer on the object. Press the left mouse button, hold down the CTRL key, and drag a copy of the object to the new location.

Tip After you copy an object using the second method above, you can use the Repeat command (from the Edit menu) to place multiple copies of the object using the same spacing. This is a quick way to create an array of aligned objects.

Pasting an object

To use the Paste command, an object must already be on the Clipboard via the Cut or Copy command.

Place the pointer where you want the object to be pasted. From the Edit menu, choose Paste. The object is removed from the Clipboard and placed on the schematic.

Note Capture cannot paste data from other Windows applications other than text in dialog boxes.

Deleting an object

There are three ways to delete a selected object:

- From the Edit menu, choose Delete.
- Press the DELETE key.
- Press the BACKSPACE key.

Placing bitmaps

You can create a bitmap in another application and place it on a schematic page or library part, or in a custom title block.

To place a bitmap

1. From the Place menu, choose Picture. A standard Open dialog box displays.
2. Select the bitmap file. If the file is not listed in the File Name box, do one or more of the following:
   - In the Drives box, select a new drive.
   - In the Directories box, select a new directory.
   - In the List Files of Type box, select the type of file you wish to open.
3. Choose the OK button. A rectangle representing the bitmap image is attached to the pointer.
4. Click the left mouse button to place the bitmap at the desired location. If you wish to place multiple copies of the bitmap, repeat this step.
5. Press ESC or choose the selection tool.

To resize a bitmap

To resize a bitmap, select it so that it displays in the selection color with edit handles at the four corners. Position the pointer over an edit handle and drag the edit handle. The bitmap's size and shape change to accommodate the new dimensions.

To resize a bitmap proportionally, hold the SHIFT key down, then drag the edit handle.

Placing text

You can place text, in the font of your choice, on a schematic page or a part. Use the text tool to document your schematic or to place the logic definition for a programmable logic device.

To place text on a schematic page

1. From the Place menu, choose Text (ALT, P, T).
   or
   Choose the text tool on the tool palette.
2. Enter the text in the dialog box that displays. To type numbers using the numeric keypad on your keyboard, you must first press the NUM LOCK key.
3. Complete the dialog box selections by specifying the font, color, and rotation.
4. Choose the OK button to dismiss the dialog box. A rectangle representing the text is attached to the pointer.
5. Click the left mouse button to place the text at the desired location.

   You can place multiple copies of the text by clicking the left mouse button at each location where you would like text. When you are done placing text, press ESC or choose the selection tool.

Tip If you have text in another Windows application, you can copy it to the Clipboard and paste it into the text dialog box using the CTRL+V shortcut keys.

To move text

1. Select the text so that it displays in the selection color with edit handles at the four corners.
2. Position the pointer over the text—not an edit handle—and drag the text to the new location.
3. Click an area where there are no parts or objects to deselect the text.

To move or copy text using the Clipboard

1. Select the text.
2. From the Edit menu, choose Cut (ALT, E, T). The text is placed on the Clipboard.
   or
   From the Edit menu, choose Copy (ALT, E, C). A copy of the text is placed on the Clipboard.
3. From the Edit menu, choose Paste (ALT, E, P). The text is attached to the pointer.

4 Move the pointer to the location where you wish to place the text and click the left mouse button. The text is placed and displays in the selection color.

5 Click an area where there are no parts or objects to deselect the text.

To rotate text

1 Select the text so that it displays in the selection color with edit handles at the four corners.

2 From the Edit menu, choose Rotate (ALT, E, O). The text rotates 90° counterclockwise.

3 Repeat step 2 as necessary.

4 Click an area where there are no parts or objects to deselect the text.

The text bounding box

Text that you place wraps according to the dimensions of its bounding box. To change how the text wraps, select it so that it displays in the selection color with edit handles at the four corners. Position the pointer over an edit handle and drag the edit handle. The text inside the bounding box rewraps within the new dimensions.

Deleting text

To delete the text and its bounding box, select the text so that it displays in the selection color with edit handles at the four corners. Press either the DELETE key or the BACKSPACE key.

To delete text when it is highlighted in the Edit Text dialog box, press the DELETE key, the BACKSPACE key, or begin typing new text.

To delete individual words within the text, double-click on the text, or select the text and choose Properties from the Edit menu. The Edit Text dialog box displays, with the text highlighted. Press one of the four arrow keys to remove the text highlighting. Double-click on the word you want to remove. Press the DELETE key or the BACKSPACE key.

Adding to text you've already typed

To add more text to text you have already placed, double-click on the text, or select the text and choose Properties from the Edit menu. The Edit Text dialog box displays, with the text highlighted. Press one of the four arrow keys to remove the text highlighting. Type the additional text. When you are done, choose the OK button.

Finding text

You can use the Find command to search an entire design, selected schematic pages, one schematic page, or the part editor for text.

To find text

1 In the design manager's design structure pane, select the root schematic (to search the entire design) or select schematic pages.
or
Make the schematic page editor window the active window.
or
Make the part editor window the active window.

2 From the Edit menu, choose Find (ALT, E, F). The Find dialog box displays.

3 Leave the asterisk in the Find What text box to locate all occurrences of all text.
or
Narrow the text search by entering specific text in the Find What text box. If you aren't sure of the exact text, you can enter wildcard characters to constrain the search list. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters.

4 Verify that the Match Case option is as you want it.

5 Select Text from the object types in the Scope area.

6 Choose the OK button.

If you search the entire design or selected schematic pages from the design manager, the search results are listed on the right side of the design manager's design structure pane.

If you search with either the schematic page editor window or the part editor window as the active window, the results of the search display in the selection color in either the schematic page editor window or the part editor window.

Replacing text

The text that you place in the schematic page editor or the part editor can easily be replaced. You can enter the replacement text using the keyboard, or you can copy the replacement text from another application.

To replace text

1 Select the text so that it displays in the selection color with edit handles at the four corners.

2 From the Edit menu, choose Properties (ALT, E, S). The Edit Text dialog box displays, with the text highlighted.

3 Enter the replacement text or press CTRL+V to paste text from the Clipboard, then choose the OK button.

Importing text

You can import text from any Windows program that copies text to the Clipboard. This is especially useful for simplifying the creation of a programmable logic device.

To import text from other Windows applications

1. In the other Windows application, copy the text to the Clipboard using that program's Copy command.
2. Activate the Capture schematic page editor or part editor.
3. From the Place menu, choose Text (ALT, E, T). The Place Text dialog box displays.
4. Press CTRL+V to paste the text into the text box, then verify that the color, font, and rotation are as you want them and choose the OK button. A rectangle representing the text is attached to the pointer.
5. Click the left mouse button to place the text at the desired location.
6. When you are done placing text, press ESC or choose the selection tool.

Exporting text

You can export Capture text to any Windows program that uses the Clipboard.

To export text to other Windows applications

1. In Capture, select the text you wish to export.
2. From the Edit menu, choose Cut (ALT, E, T) or Copy (ALT, E, C). The text is placed on the Clipboard.
3. Activate the other Windows application and use that application's Paste command to place the text.

Character formatting

You may want the text to have a distinctive appearance, or to fit a specific space. Capture supports TrueType® fonts. You can preview a sample of the selected font before you choose it. You can also select the default font that you set up in the Fonts tab of the Design Template dialog box (from the Options menu).

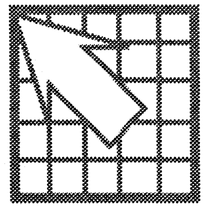
To change fonts and point sizes

1. If you are placing the text, choose Text from the Place menu (ALT, P, T). The Place Text dialog box displays.
   or
   If the text has already been placed, double-click on the text. The Edit Text dialog box displays.
2. In the Font group box, choose the Change button. The Font dialog box displays.
3. Select a font, style, and size. Sample text displays in the Sample group box.
4. Choose the OK button twice.

About screen fonts

See For information about setting up your screen fonts, see *Chapter 5: Setting up your design.*

Chapter 8



Changing your view of a schematic

There are several ways to change your view of a schematic page. They include zooming to a smaller or larger view of the schematic page; centering a view on a particular position; and moving to a different location. You can also choose whether or not to display a grid or grid references.

Zooming

In the schematic page editor and in the part editor, you can look closely at a particular area by using the Zoom In command on the View menu. Conversely, you can change your viewing perspective to increase the portion of the drawing board that is visible by using the Zoom Out command. When you zoom in or out, Capture centers your view on the current pointer position, if possible. If the pointer is outside the window, or if you choose the Zoom In or Zoom Out toolbar button, Capture centers your view on any selected objects. Otherwise, Capture zooms in or out on the center of the active window.

Note If you choose an editing function, then choose a command to change the view, your next mouse click implements the editing function. For example, if you have the circle tool selected and then zoom in or out, your next mouse click will start a circle.

To zoom in 

➥ From the View menu, choose Zoom, then choose In (ALT, V, Z, I).
or
Choose the Zoom In tool on the toolbar.

The current zoom scale is multiplied by the zoom factor. With a zoom factor of two, zooming in makes the image twice as large and displays half the area of the previous view.

To zoom out

➥ From the View menu, choose Zoom, then choose Out (ALT, V, Z, O).
or
Choose the Zoom Out tool on the toolbar.

The current zoom scale is divided by the zoom factor. With a zoom factor of two, zooming out halves the image size and shows twice the area of the previous view.

To change the zoom factor

1 From the Options menu, choose Preferences (ALT, O, P), then choose the Pan and Zoom tab.

2 In the Zoom Factor text box, enter the new zoom factor. Note that you can specify separate values for the schematic page editor and the part editor.

3 Choose the OK button.

Zooming to a specified scale

The Zoom Scale dialog box provides predefined scales (25%, 50%, 100%, 200%, 300%, and 400%). You can also type in a custom scale value (as a percentage). The scaling limits the size of the schematic page that can be displayed (for example, if 1 unit = .01", then you can display: 320" at 100%, 160" at 200%, 80" at 400%, and so on).

Tip You can view the current scale in the status bar at the bottom of the schematic page editor window. It displays to the left of the X and Y coordinates.

To zoom to a specific scale

1 From the View menu, choose Zoom, then choose Scale (ALT, V, Z, S).

2 Select one of the preset scales, or enter a custom scale.

3 Choose the OK button.

Other viewing options

You can view a selected area or the entire page, or you can center your view.

To view a selected area 

1 From the View menu, choose Zoom, then choose Area (ALT, V, Z, A).
or
Choose the Zoom Area tool on the toolbar.

The pointer displays as a magnifying glass.

2 Move the pointer to one corner of the area to view.

3 Press and hold the left mouse button as you move the pointer to the opposite corner of the area to view.

4 Release the mouse button. The area is enlarged to fill the window.

To view the entire page or part 

➥ From the View menu, choose Zoom, then choose All (ALT, V, Z, L).
or
Choose the Zoom All tool on the toolbar.

The entire schematic page or part is reduced to fit the window.

To center the view on an object or area

1 Select objects or an area.

2 From the View menu, choose Zoom, then choose Selection (ALT, V, Z, E).

The display scrolls so that the selected objects or selected area is in the center of the window. The zoom scale does not change.

To center the view on your pointer

➥ Press SHIFT+C.

The display scrolls so that the pointer's selection is in the center of the window. The zoom scale does not change.

Jumping to a new location

The X and Y coordinates of your pointer's current location are displayed at the right of the status bar. Grid references are marked on the left and upper edges of the schematic page in the schematic page editor. From the View menu, choose Go To to display the Go To dialog box, which has three tabs labeled Location, Grid Reference, and Bookmark, shown below and on the next page.

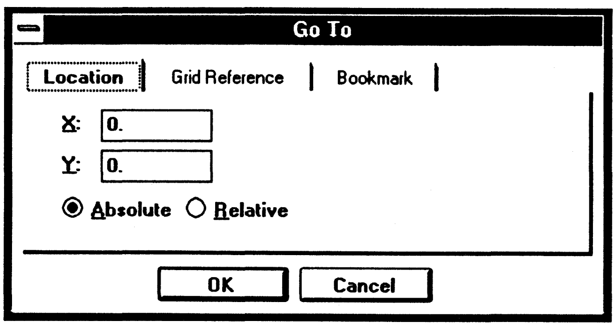
To move to a specific location

1. From the View menu, choose Go To (ALT, V, G).
2. Choose the Location tab.
3. Enter the X and Y values, select the Absolute option, then choose the OK button. The coordinates are measured in inches or millimeters, depending on what you have configured in the Page Size tab on the Schematic Page Properties dialog box. Your pointer moves to the new coordinates.

To move a specific distance

1. From the View menu, choose Go To (ALT, V, G).
2. Choose the Location tab.
3. Enter the X and Y values that you want the pointer to move, select the Relative option, then choose the OK button. The jump distance is measured in inches or millimeters, depending on what you have configured in the Page Size tab on the Schematic Page Properties dialog box. Your pointer moves the specified distance.

Go To dialog box, Location tab



X The X field specifies the X-axis coordinate for the jump.

Y The Y field specifies the Y-axis coordinate for the jump.

Absolute and **Relative** The Absolute and Relative options specify whether the jump is absolute (to the indicated coordinates) or relative (using the coordinates as an offset to the pointer's current position).

Jumping to a specific grid reference

To jump to a reference area

1 From the View menu, choose Go To (ALT, V, G).

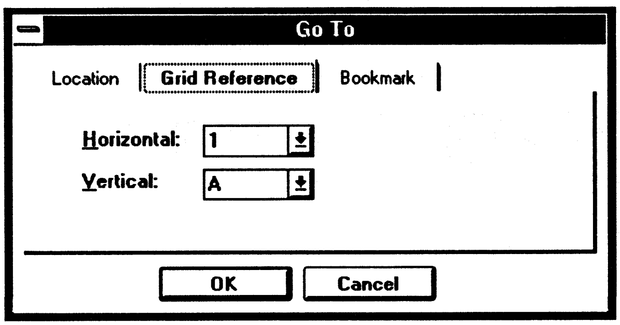
2 Choose the Grid Reference tab.

3 Select a horizontal grid reference from the Horizontal drop-down list.

4 Select a vertical grid reference from the Vertical drop-down list.

5 Choose the OK button.

Go To dialog box, Grid Reference tab



Horizontal The Horizontal field specifies a horizontal grid reference for the jump.

Vertical The Vertical field specifies a vertical grid reference for the jump.

Jumping to a marked location

If you find that you need to return repeatedly to a specific area of a schematic page, or if you need to direct attention to a particular location, a *bookmark* is very convenient. To set a bookmark, you assign it a name, then use the Go To command to go to the marked location.

Note Bookmarks are not saved with your design.

To place a bookmark

1 From the Place menu, choose Bookmark (ALT, P, M).

2 Enter the name of the bookmark, then choose the OK button to dismiss the bookmark dialog box.

3 Position the pointer where you want the bookmark and click the left mouse button. The bookmark is placed, and displays in the selection color.

4 Click an area where there are no parts or objects to deselect the bookmark.

To rename a bookmark

1 Select the bookmark.

2 From the Edit menu, choose Properties (ALT, E, S). The Rename Bookmark dialog box displays.

3 Enter a new name in the text box.

4 Choose the OK button.

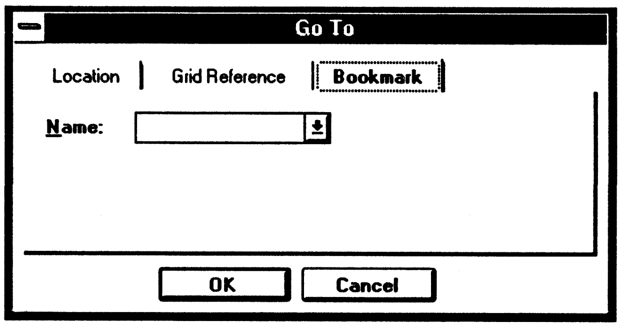
To move to a specific bookmark

1 From the View menu, choose Go To (ALT, V, G).

2 Choose the Bookmark tab.

3 Enter the name of the bookmark and choose the OK button.

Go To dialog box, Bookmark tab



Name The Name field specifies a name of a bookmark for the jump.

Displaying the grid and grid references

Like the toolbar, you can hide the grid display and grid references, then display them again later when you need them.

To display or hide the grid

➥ From the View menu, choose Grid (ALT, V, I).

To display or hide the grid references

➥ From the View menu, choose Grid References (ALT, V, R).

Note The settings in the Grid Display tab in the Preferences dialog box (from the Options menu) control whether the grid appears as grid dots or lines. The Grid Display tab setting also controls whether the pointer snaps to grid.

Finding an object

Finding parts in a design

Using the Find command and a part property value, you can locate a part in a schematic or on a page. In the Find dialog box, you enter a property value string and specify that you want to find a part. Capture searches all the parts to find those with a property value that matches the string. If you aren't sure of the exact property value string, you can enter wildcard characters to constrain the search list. Valid wildcard characters are an asterisk (*) to match multiple characters and a question mark (?) to match individual characters.

To find a part on a schematic page

1. Open the schematic page.
2. From the Edit menu, choose the Find command (ALT, E, F).
3. Enter the property value string that defines the part you want to search for.
4. Select Parts from the object types in the Scope area.
5. Choose the OK button to start the search. Parts that have a property value matching the property value string of step 3 are selected on the schematic page.

Tip You can edit the properties of multiple parts when they are selected using the spreadsheet editor. From the Edit menu, choose the Properties command (ALT, E, S), or choose Edit from the right mouse button's pop-up menu. For information on using the spreadsheet editor, see the section *Using the spreadsheet editor to edit properties* in *Chapter 2: The Capture work environment.*

To find a part in a design

1. In the design structure pane of the design manager, select the schematic or schematic pages you want to search.
2. From the Edit menu, choose the Find command (ALT, E, F).
3. Enter the property value string that defines the part you want to search for.
4. Select Parts from the object types in the Scope area.
5. Choose the OK button to start the search. Parts that have a property value matching the property value string of step 3 are listed in the browse pane.
6. Double-click on the part in the browse pane list to open the schematic page editor with the found part displayed and selected.

Chapter 9

Printing and plotting

To send output to a printer, a plotter, or an encapsulated PostScript® file, you use the standard Windows Print Setup, Print Preview, and Print dialog boxes.

Note Capture can send output to any print driver that Windows supports. For additional information on print drivers, see the Windows documentation.

The printing commands can be accessed from the File menu in the design manager, the schematic page editor, or the part editor. You can print a schematic, a schematic page, a part, or a package.

To configure the output device

- From the File menu, choose the Printer Setup button. Select an appropriate printer or plotter, or change the printer settings, then choose the OK button.

See To install and remove printers and plotters, and to set additional printing options, see the Windows documentation regarding the Windows Control Panel.

Printing or plotting schematics or schematic pages

You can print or plot a schematic or several schematics from the design manager. You can also print selected schematic pages from the design manager. With the schematic page editor active and open to a specific schematic page, you can create a print or a plot of that schematic page.

To print or plot a schematic or schematics

1. In the design structure pane of the design manager, select the schematic or schematics you wish to print.
2. From the File menu, choose Print (ALT, F, P). The Print dialog box opens.
3. Select the scale, print quality, and number of copies.
4. Choose the OK button to send the image to the output device.

To print or plot a schematic page or schematic pages

1. Activate the schematic page editor window for the page you wish to print.
   or
   In the design manager window, select the schematic page or pages in the design structure pane.
2. From the File menu, choose Print (ALT, F, P). The Print dialog box opens.
3. Select the scale, print quality, and number of copies.
4. Choose the OK button to send the image to the output device.

Printing or plotting a part or package

With the part editor active and open to a specific part or package, you can create a print or a plot of that part or package. You can also print a library part from the design manager.

To print or plot a part or package

1. Select the part or package you wish to print in schematic page editor.
   or
   Select the library part in the design manager.
2. From the right mouse button pop-up menu in the schematic page editor, choose Edit Part. The part appears in a part editor window.
3. From the View menu in the part editor, choose Part if you wish to print a part and choose Package if you wish to print a package.
4. From the File menu, choose Print (ALT, F, P). The Print dialog box opens.
5. Select the scale, the print quality, and the number of copies.
6. Choose the OK button to send the image to the output device.

Previewing print output

Using the Print Preview command, you can preview your schematic, schematic page, part, or package output to check its appearance before you commit it to paper.

To preview a schematic page

1. From the File menu, choose Print Preview (ALT, F, V). The Print Preview dialog box displays.
2. Specify the values on the dialog box.
3. Choose the OK button to begin. The Print Preview display window opens with a display of your schematic, schematic page, part, or package.
4. Use the Previous page and Next page buttons to look at other printer pages.
5. To zoom in, move the magnifier pointer to a specific area and click the left mouse button.
6. Choose the Close button to dismiss the Print Preview window.

Scaling a print or plot

You can manually scale, or have Capture automatically scale, printer output and plots to fit the paper size you choose.

To scale a print or a plot

1. From the File menu, choose Print (ALT, F, P). The Print dialog box displays.
2. Select one of the three radio buttons in the Scale box.
   - The Scale to paper size option scales each schematic page to fit a single sheet of paper (as configured in the printer driver).
   - The Scale to page size option scales each schematic page to the sheet size you select in the Page size area. The sheet size is configured in the Page Size tab in the Design Template dialog box.
   - The Scaling option scales your schematic page to a factor of your choice. The acceptable range of factors is 0.1 to 10.0; up to three decimal places are allowed.
3. If you select the Scale to page size option above, the Page size area becomes available. Select a sheet size. This results in multiple sheets of paper if you select a sheet size larger than your printer paper.
4. Choose the OK button to send the image to the output device.

Plotter pen colors

The plotter driver maps your color choice to the closest available pen color as established in your plotter driver configuration. See your plotter's driver setup and documentation for more details.

Special considerations for plotting

Many plotters do not support bitmaps directly. If you are sending Capture output to a plotter, your bitmaps may not be plotted.

See The plotter setup dialog boxes are only accessible from the Windows Control Panel. See the Windows documentation regarding the Windows Control Panel.

Many plotters do not have drivers that ship with Windows. If you do not see the plotter you are looking for in the list of available drivers, contact your plotter manufacturer and ask for a Windows driver. If your plotter emulates HPGL, and you are using Windows 3.1 or Windows 95, an alternative solution is to use the HPGL driver.

Libraries and parts

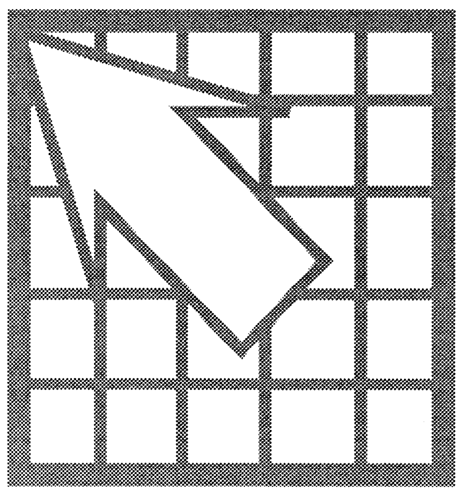
Part Three provides information about libraries and parts. A library is a file that stores parts, symbols, title blocks, and schematics. Capture provides over 20,000 parts contained in more than 80 libraries. You can create additional custom libraries to store any combination of items.

Part Three includes these chapters:

Chapter 10: About libraries and parts describes libraries, parts, and part instances. It also describes how the parts in a design are stored in the design cache, and how you can replace or update parts in the design cache.

Chapter 11: Creating and editing parts describes how to create new parts and store them in a library, and how to edit parts in a library, as well as after they are placed on a schematic page.

Part Three



Chapter 10

About libraries and parts

Capture's libraries contain more than 20,000 parts. This chapter describes Capture's libraries, and explains how parts, packages, electrical symbols, and schematics are stored in libraries.

Libraries

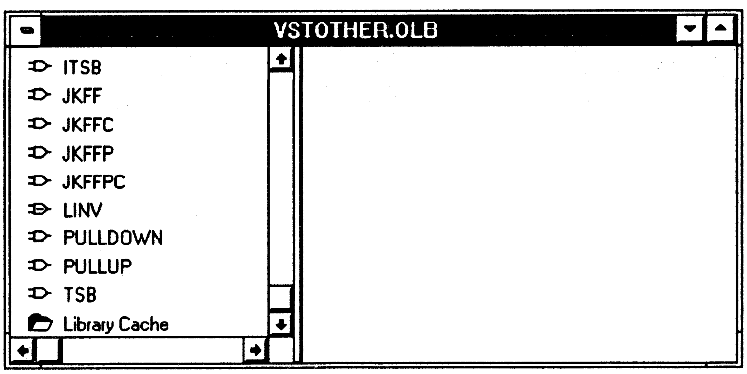
Libraries are files that contain reusable design data. They contain parts that you can place as instances on schematic pages. Libraries may also contain a variety of net symbols and title blocks that you can reuse in your designs.

The relationship between the library and the parts and symbols it contains is similar to the relationship between a design and its contents. The contents of the library move with the library and are deleted with the library.

You can create custom libraries to store any combination of items. You can, for example, create a library to hold all your programmable logic devices, or to hold schematics that you use often. There is no need to create a library for a particular design, because the design cache holds all the parts and symbols used in the design.

Caution If you edit a library provided by Capture, you should give it a new and unique name so that you will not copy over your changes when you receive updated libraries.

When you work with a library in Capture, you use the design manager. The design manager lists the parts and symbols contained in the library in its design structure pane.



To edit a part, double-click on it. The part opens in a part editor window.

To move a part to a different library, open the source library and the destination library in separate design manager windows. Select the part and drag it from one library to the other.

To copy a part to a different library, follow the same procedure but hold the CTRL key down while you drag the part.

See For information about printing a part, see *Chapter 9: Printing and plotting.*

Tip You can also use the Cut, Copy, and Paste commands on the Edit menu to move or copy parts between libraries.

See For general information about using the design manager, see *Chapter 2: The Capture work environment.*

Because a library is a file, you can work with it in the Windows File Manager as well as in Capture. When you need to back up a library, use the File Manager to create a copy.

Parts

Parts are the basic building blocks of a design. A part may represent one or more physical components, or it may represent a function, a simulation model, or a text description for use by an external application. The part's behavior is described by a PCB footprint, an attached schematic, HDL statements, or other means.

Parts usually correspond to physical objects—gates, chips, connectors, and so on—that come in packages of one or more parts. You can think of these packages as physical parts, and the parts you place on a schematic page as logical parts. Physical parts that comprise more than one logical part are sometimes referred to as *multiple-part packages*. For simplicity, Capture usually refers to both as *parts*.

Each logical part has graphics, pins, and properties that describe it. As you place the logical parts in a package to suit your design requirements, Capture maintains the identity of the single physical part—the package—for back annotation, netlisting, bills of materials, and processes that require it. The logical part inherits this information from the physical package.

You specify physical packaging information when you create a part. You can also change it in the part editor (from the View menu, choose Package; then, from the Options menu, choose Package Properties).

The logical parts in a package may have different pin assignments, graphics, and user properties. If all the logical parts in a package are identical except for the pin names and numbers, the package is *homogeneous*. If the logical parts in a package have different graphics, numbers of pins, or properties, the package is *heterogeneous*. For example, a hex inverter is homogeneous: the six inverters are identical, except for their pin numbers. A relay, which has a normally opened switch, a normally closed switch, and a coil, is heterogeneous: the three physical parts differ in graphics, number of pins, and properties.

When you place a part on a schematic page, you actually create an instance of the part. A part instance is like a "snapshot" of the part in the library; that is, it inherits all the properties of the library part. Once a part instance is on the schematic page, you can edit the properties of that part instance without changing the properties of any other part instance. The instance values of those properties supersede the values of any identical properties that exist on the library part.

When you look at a schematic page in physical view, you see occurrences of the part instances you placed in logical view. You can edit or add properties to part occurrences, and the occurrence values of those properties supersede the values of any identical properties that exist on the part instance.

Note Don't confuse physical parts and logical parts with part occurrences and part instances. A physical part is a package, and a logical part is one device in that package. By contrast, a part instance is a part placed on a schematic page as seen in logical view, and a part occurrence is the same part on the schematic page as seen in physical view.

Part instances

A part instance is a part you have placed on a schematic page. You place part instances in logical view. If you change to physical view, you see occurrences of the instances.

If you edit a part in a library, your changes don't affect instances of the part in any design until you want them to. You use the Update Cache or Replace Cache commands to bring library changes into a design.

You can also edit part instances in the schematic page editor by selecting a part instance and choosing the Part command from the Edit menu. Once you finish editing the part instance, you can apply the changes to every instance in the design or just the single (current) part instance. If you update all instances of the part, the new part replaces the old in the design cache, and the link with the original library is broken. If you update only the current part instance, you create a new part in the design cache, and the new part has no link to the original library.

The design cache

When you place the first instance of a part in a design, a copy of the part is created in the design cache. The design cache stores one copy of every part used in the design—you can think of it as an "embedded library." Normally, all instances of the part refer to this copy in the design cache.

A cache part also retains a link to the library part on which it is based, so you can update all of the parts in the design cache to synchronize them with the parts in the libraries. For more information, see *About part instances.*

Once you edit a part instance, the link to the original library no longer exists. This means that:

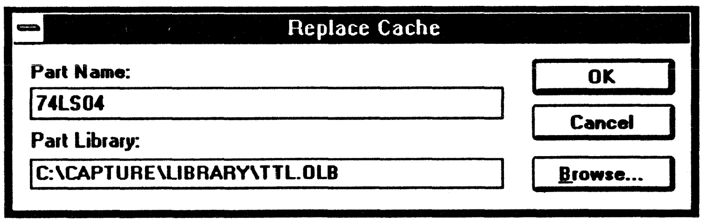
- Because it doesn't exist in a library, the only way to place a copy of the part is to use the Copy and Paste commands on the schematic page editor's Edit menu.
- The part instance is not affected by the Update Cache command.
- To restore its link with the original library, choose the Replace Cache command from the design manager's Design menu.

To replace a part instance in the design cache with a different library part

1 If it's not already open, open the design containing the part instance you wish to replace.

2 Open the design cache and select the part instance you wish to replace.

3 From the Design menu, choose Replace Cache (ALT, D, C). The Replace Cache dialog box displays.



Notice that the text boxes in this dialog box contain the name of the part instance you are replacing and its original source library. You can leave these fields unchanged if you would like to update the part instance with its original version. This is useful if you have changed the part instance in the design cache and would like to use its original version, or if you have received an update to a library and want to use the new version of the library part.

4 In the Part Name text box, type the name of the library part you want to use to replace the selected part instance.

5 In the Part Library text box, type the path and filename of the library containing the part.

6 Choose the OK button when you are ready to replace the part instance.

Capture replaces the part instance you selected in step 2 with the library part you specified in step 4.

To update selected parts in the design cache so they match their corresponding library parts

When you update parts in the cache, any user-defined properties on the parts being updated are preserved. User-defined properties on pins, however, are deleted.

1 If it's not already open, open the design containing the parts you wish to update.

2 Open the design cache and select the parts you wish to update.

3 From the Design menu, choose Update Cache (ALT, D, U). Capture displays a message warning you that you are about to update the selected parts and symbols with parts and symbols from their original libraries.

4 Choose Yes when you are ready to update the parts.

Capture updates the parts you selected in step 2 with their corresponding library parts. Other designs that use these parts are not affected.

Primitive and nonprimitive parts

A *primitive* part contains no hierarchy. A nonprimitive part has an underlying hierarchical description, such as an attached schematic. In Capture, this characteristic is defined in a property, called Primitive, on every part instance. When a part is marked as primitive, all of Capture's tools treat it as such. You cannot descend into a primitive part, even if it has an attached schematic.

Note If you attach a schematic to a part in a homogeneous package in a library, the schematic is attached to each part in the package. Once the part is placed on a schematic page, you can attach different schematics to each part in the package. You cannot attach a schematic to a part in a heterogeneous package.

You can change the Primitive property as often as you like during the design process. For example, you might create a part and attach a schematic that describes its gates and wiring, and then attach schematics to some of those parts to describe their transistors. Before you create a netlist for simulation, you would specify those parts as nonprimitive, so that Create Netlist can descend far enough to find the transistor-level descriptions. Before you create a netlist for board layout, you would specify the parts as primitive, so that Create Netlist stops at the gate-level descriptions.

For part instances that have their Primitive property set to Default, you can tell Capture to treat them as primitive or nonprimitive on a design basis using the Design Template or Design Properties commands on the Options menu. This is useful when you are describing and simulating your design at varying levels of abstraction (as in top-down design).

Chapter 11

Creating and editing parts

In Capture you can create parts and add them to a new or existing library. You can also edit existing parts in a library or on a schematic page. All of these processes are described in this chapter.

To create or edit a part, you use the part editor. There are many different ways to access the part editor:

- To create a new part, open a new or existing library in the design manager. From the Design menu, choose the New Part command.
- To edit an existing part, open a library in the design manager, then double-click on the part.
- To edit a part instance on a schematic page, select it. From the Edit menu, choose Part.

Tip—Parts and packages: homogeneous or heterogeneous

A part may be divided into several *logical* parts all contained in a single *physical* package. You can distribute the individual logical parts throughout your design, while maintaining the part's identity as a single physical part.

Each logical part has graphics, pins, and properties that describe it. If you define a package in which all the logical parts are identical except the pins, the package is homogeneous. For example, a hex inverter is a homogeneous package. If the logical parts vary in graphics, number of pins, or properties, the package is heterogeneous. An example of a heterogeneous package is a relay with a normally open switch, a normally closed switch, and a coil.

Both homogeneous and heterogeneous packages may have shared pins, such as supply pins that are used by every logical part in the package. Often, these pins are invisible, but are connected by name to a power or ground net.

Creating a new part

You can create your own custom parts and save them in a library. A custom part can be a single part, or it can be a package that contains multiple parts. It can contain pins, graphics, text, and IEEE symbols. Graphics must be within the part's body, while text and IEEE symbols can be either inside or outside the part's body. Pins are attached to the part at the *part body border*, which defines the size and shape of the region in which you create the part body.

To create a part, you complete three processes: you define the part, you add graphics to the part, and you place pins on the part. This section describes these processes.

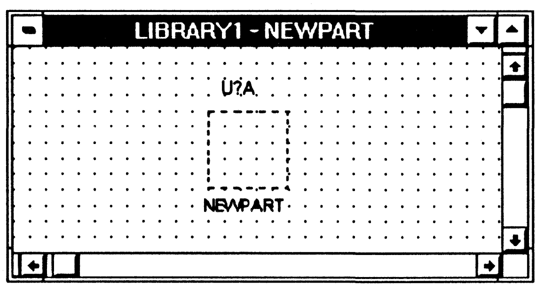
Defining a part

Before you begin drawing a part, you must provide Capture with specific information about the part, such as the part's name. If the part is a multiple-part package, you can specify how many parts are in the package and whether the part is homogeneous or heterogeneous. Once you provide this information, you can draw the part, place graphics, and place pins.

To define a new part

1. From the File menu, choose New or Open, then choose Library.
2. From the Design menu, choose New Part (ALT, D, T).

   The New Part Properties dialog box displays. You use this dialog box to provide basic information about the part you are creating.
3. In the Name text box, type a name for the part you are creating. You can use the default settings for the other options on this dialog box or you can change them to fit your requirements. For example, if this part is a multiple-part package, enter the number of parts in the package in the Packaging area. All fields on the New Part Properties dialog box are described later in this section.
4. When the part is specified to your requirements, choose the OK button.

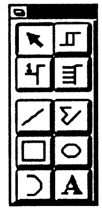
   The part editor window opens, with the dashed outline of the part's body shown in the middle. The dashed outline is the part body border. Pins will be placed on the part outside of this region, touching the part body border. The part's name displays below the part, and the part's reference displays above the part. Notice that the title bar shows the name of the library followed by the name of the part you are creating.

Note If you're creating a multiple-part package, the part editor window contains the first part in the package at this point. If you are creating a homogeneous part, all edits you make to this part (with the exception of changes to pin numbers) apply to all parts in the package. If you are creating a heterogeneous part, edits you make to this part apply to *this part only*. To edit a different part in the package, choose Next Part or Previous Part from the View menu.

To view all the parts in the package in a window, choose Package from the View menu. You can edit a different part by double-clicking on it from the package view.

5 To change the size and shape of the part body border, select the border and drag the selection handles until the part body border is the size you need.

6 Once you have established the part's border, you can use the drawing tools on the part editor's tool palette to draw the part and place text on the part. All graphics that make up the part must fit within the boundaries of the part body border—with the exception of IEEE symbols and text. If you draw or place something other than IEEE symbols or text outside of the part body border, the part's border expands to encompass the graphics.



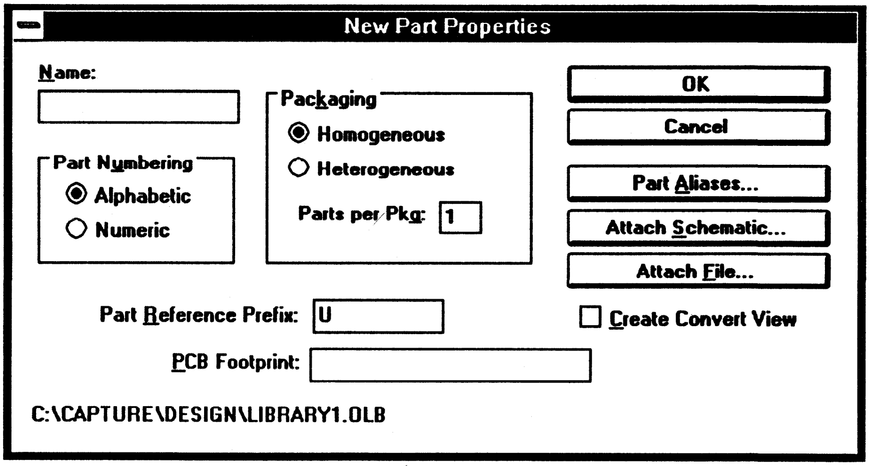
See also For general information about adding graphics and text, and detailed information about placing IEEE symbols, see *Adding graphics, text, and IEEE symbols to a part* in this chapter. For details about each of the graphic and text tools, see *Chapter 7: Adding and editing graphics and text.*

7 You can add pins to the part using the pin tool or the pin array tool. This is described in *Placing pins on a part* later in this chapter.

8 When you are done creating the part, you must save it. From the File menu, choose Save (ALT, F, S).

If you are creating this part in a new library that hasn't yet been saved, the Save As dialog box displays, giving you the opportunity to name the library file. If you are creating this part in a library that already exists on disk, the part is written to that file.

New Part Properties dialog box



Tip Once you have defined the fields on this dialog box, you can display the dialog box again later. Change to Package view (from the View menu, choose Package), then from the Options menu, choose Package Properties.

Name The name of the part. This is used as the default part value when the part is placed on a schematic page.

Part Numbering If the part is a multiple-part package, this specifies whether parts in the package are identified by letter or number. For example:

- U?A (alphabetic)
- U?-1 (numeric—usually used for connectors)

Packaging If the part is a multiple-part package, this specifies whether all the parts in the package have the same graphical representation (homogeneous) or different graphical representations (heterogeneous). The **Parts per Pkg** text box specifies the number of parts in the package.

Part Reference Prefix Specifies the part reference prefix, such as "C" for capacitor or "R" for resistor. For example:

- C? (capacitor)
- R? (resistor)

PCB Footprint The PCB physical package name to be included for this part in the netlist.

Note The path and filename of the library that contains the part are displayed at the bottom left corner of the dialog box.

Part Aliases Displays a dialog box that you use to add or remove *part aliases*, which are duplicate copies of parts that use different names in a library. Part aliases use the same graphics, attached schematics, and properties as the originals, with the exception of the part values.

Tip When you view the list of parts in the design cache, parts that are placed via part alias display with a line through the center.

Attach Schematic You can attach a schematic to create a hierarchy. When you choose this button, a dialog box displays that you use to specify the name of a schematic and the library or design that contains it. For more information, see the next section, *Attaching a schematic.*

Note If you specify a library or design that you haven't yet saved to disk, Capture creates the library or design in the directory specified by your TEMP environment variable.

Attach File You can attach a text file that contains related information, such as PLD source code. There are no format restrictions for attached files. When you choose Attach File, a dialog box displays that you use to specify the name of the text file.

Tips Once you've attached a file, you can use the Descend Hierarchy command on the schematic page editor's View menu to open the file—if you first use the Windows File Manager to create an association with that file extension. For information on associating files with applications, see the File Manager's online help.

If you have an attached schematic as well as an attached file, Descend Hierarchy opens the schematic and not the file.

Caution An attached schematic or other file is not stored with the design or library. If you copy or move the design or library to a new location, you must also move or copy the attached file to keep the two files together. In addition, you may need to edit the path to the attached schematic or file if you move the design to a new location with a different directory structure.

Create Convert View Some library parts have a second form, such as a DeMorgan equivalent, as well as the standard representation. Select this check box to give the part a convert. If a part has a convert, you can switch between the part's normal and convert view once it is placed on a schematic page.

Tip Once you've specified that the part has a convert, you can change to the convert view. From the part editor's View menu, choose Convert.

Attaching a schematic to a part

Note Library parts, part instances, part occurrences, and hierarchical blocks allow attached schematics. This discussion provides information about attaching a schematic to a library part. This information also applies, however, to part instances, part occurrences, and hierarchical blocks.

Attaching a schematic to a part creates hierarchy in Capture. When you attach a schematic to a part, you must specify the schematic's name. You may optionally specify a library or design containing the schematic. If you don't, Capture assumes the schematic is contained in the current design or library. If you attach a schematic to a part, you can move or copy the child schematic into the same library as the part after the schematic is created. This permits the part and attached schematic to be reused in other designs. If an attached schematic is specified for a part, but the schematic has not been created, Capture creates the schematic when you descend the hierarchy on the part. To descend the hierarchy, you must define the part instance as nonprimitive.

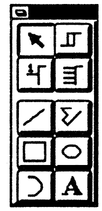
Tip To define a part instance as nonprimitive, double-click on the part on a schematic page. On the Edit Part dialog box, set Primitive to No. In physical view, you can also define part occurrences as nonprimitive. This lets you control whether or not to descend the hierarchy on an occurrence-by-occurrence basis.

You can change the Primitive property as often as you like. For example, you can create a part and attach a schematic that describes its gates and wiring, then attach schematics to some of those parts to describe their transistors. Before you create a netlist for simulation, you would specify those parts as nonprimitive, so that Create Netlist can descend far enough to find the transistor-level descriptions. Before you create a netlist for board layout, you would specify the parts as primitive, so that Create Netlist stops at the gate-level descriptions. To make all parts whose Primitive property is set to Default behave as nonprimitive parts, choose Design Template from the Options menu, and choose the Hierarchy tab. Set Parts to Nonprimitive.

To attach a schematic to a part

1. From the design manager's Design menu, choose New Part (ALT, D, T).
   or
   From the part editor's View menu, choose Package (ALT, V, K). From the Options menu choose Package Properties (ALT, O, R).
   or
   Select a part instance on a schematic page. From the schematic page editor's Edit menu, choose Properties (ALT, E, P).
2. Choose the Attach Schematic button. The Attach Schematic dialog box displays.
3. In the Schematic text box, enter the name of the child schematic.
4. If the child schematic is not in the current library or design, specify the path and library or design where the schematic is located.
5. Choose the OK button twice.

Adding graphics, text, and IEEE symbols to a part



Once you have defined a part, you can draw the part. You can draw an outline to reflect the part's shape, and you can add graphics to add detail to the part. To add graphics to the part, you use the Line, Rectangle, Ellipse, Arc, Polyline, and Picture commands on the Place menu or the corresponding tools on the tool palette. You can also add text to the part using the Text command on the Place menu or the text tool on the tool palette.

See also For details about each of the graphic and text tools, see *Chapter 7: Adding and editing graphics and text.*

When you place graphics in a part, they must be within the part's body. If they don't fit within the part's body, the part body border expands to enclose the graphics.

Tip Like the schematic page editor, the part editor can display the part you are working on at several levels of detail. Just use the Zoom command on the View menu or the right mouse button pop-up menu.

A part can also include IEEE symbols (shown at right). Unlike graphics, IEEE symbols do not have to be within the part's body.

To place an IEEE symbol

1 From the part editor's Place menu, choose IEEE Symbol (ALT, P, E).
or
Choose the IEEE symbol tool from the schematic page editor's tool palette. 

2 In the Place IEEE Symbol dialog box, select a symbol from the Symbol list box. The symbol displays in the preview box. When you have selected the symbol you want to place, choose the OK button. The IEEE Symbol dialog box closes.

An image of the IEEE symbol is attached to your pointer. You can press the right mouse button to display a pop-up menu with commands that you can use to change the appearance of the symbol before you place it. You can mirror the symbol horizontally or vertically, rotate the symbol, or choose another IEEE symbol to place.

3 Move the pointer to the location on your part where you want the symbol and click the left mouse button to place the symbol. You can place multiple instances of the symbol by clicking the left mouse button each place you want the symbol.

4 When you are done placing symbols, choose the selection tool or press ESC to dismiss the IEEE symbol tool.

Tip Once you've placed an IEEE symbol, you can change its size and shape by selecting it and dragging its selection handles.

Placing pins on the part

There are several ways you can place pins on a part. You can use the Pin command on the part editor's Place menu or the pin tool on the tool palette to place individual pins; or you can use the Pin Array command on the Place menu or the pin array tool on the tool palette to place several pins at once.

Pins must connect to the part body border (the dotted line around the part). If the edge of a part body coincides with this border, pins can connect directly to the part body. However, if the part body is inside this border, you can draw a line between the part body and the pin on the part body border to make the pin look as though it connects to the part body.

To place a pin

1. From the part editor's Place menu, choose Pin (ALT, P, P).
   or
   From the part editor's tool palette, choose the pin tool.

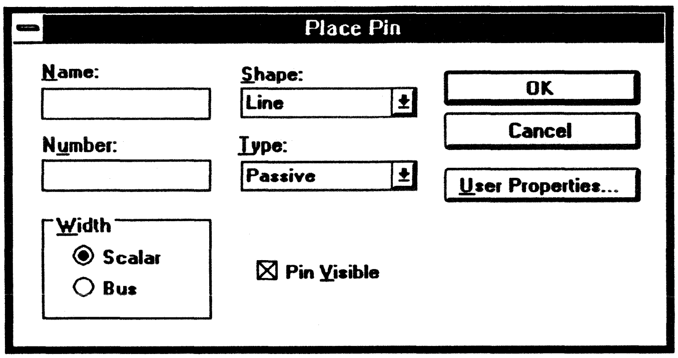
   The Place Pin dialog box displays. You must provide information about the pin you are placing.
2. In the Name text box, type a name and pin number for the pin you are placing. You can use the default settings for the other options on this dialog box or change them to fit your requirements.
3. When the pin is specified to your requirements, choose the OK button.
4. Using the pointer, drag the pin to the desired location along the part body border.
5. Click the left mouse button to place the pin. You can place multiple instances of the pin by clicking the left mouse button each time you want to place an instance of the pin. The pin number and name increment automatically.

Tip To place multiple pins that are slightly different, you can click the right mouse button and choose the Edit command.

---

6. When you are done placing pins, choose the selection tool or press ESC to dismiss the pin tool.

Place Pin dialog box



Name The name of the pin. To enter a name with a bar over it (indicating negation), type a backslash character after each letter. For example, type **R\E\S\E\T** to define the name:

$\overline{\text{RESET}}$

Tip You can make each part's pin names visible or invisible. From the part editor's Options menu, choose Part Properties (ALT, O, R). When the User Properties dialog box appears, set Pin Names Visible to either True or False.

Number The pin's number.

Shape The shape of the pin, as described in the table at right.

Type The type of the pin, as described in the table on the next page.

Width If the pin connects to a wire, select Scalar. If the pin connects to a bus, select Bus.

If the pin connects to a bus, the pin should be named in the format *busname* [*range*]. Bus pins are expanded into separate pins in a netlist, just as a bus is separated into separate signals. Each pin is named *basename x* where *basename* is the name of the bus and *x* is the signal index.

Shape	*Description*
Dot	An inversion bubble.
Clock	A clock symbol.
Dot-Clock	A clock symbol with an inversion bubble.
Zero	A normal pin with a lead zero grid units in length.
Short	A normal pin with a lead one grid unit in length.
Line	A normal pin with a lead three grid units in length.

Pin shapes.

Pin Visible If this option is selected, the pin is visible when the part is placed on the schematic page. Otherwise, the pin is not visible on the schematic page.

User Properties Displays the User Properties dialog box. You can use this dialog box to define additional properties for the pin.

Pin type	*Description*
3-state	A 3-state pin has three possible states: low, high, and high impedance. In its high impedance state, a 3-state pin looks like an open circuit. For example, the 74LS373 latch has 3-state pins.
Bidirectional	A bidirectional pin acts as both input and output. For example, pin 2 on the 74LS245 bus transceiver is a bidirectional pin. The value at pin 1 (an input) determines the activity of pin 2, as well as others.
Input	An input pin is one to which you apply a signal. For example, pins 1 and 2 on the 74LS00 NAND gate are input pins.
Open collector	An open collector gate omits the collector pull-up. Use an open collector to make "wired-OR" connections between the collectors of several gates and to connect with a single pull-up resistor. For example, pin 1 on the 74LS01 NAND gate is an open collector gate.
Open emitter	An open emitter gate omits the emitter pull-down. The proper resistance is added externally. ECL logic uses an open emitter gate and is analogous to an open collector gate. For example, the MC10100 has an open emitter gate.
Output	An output pin is one to which the part applies a signal. For example, pin 3 on the 74LS00 NAND gate is an output pin.
Passive	A passive pin is typically connected to a passive device. A passive device does not have a source of energy. For example, a resistor lead is a passive pin.
Power	A power pin expects either supply voltage or ground. For example, on the 74LS00 NAND gate, pin 14 is VCC and pin 7 is GND.

Pin types.

Note Pins that are invisible (usually power) are not connected using wires and buses, but instead are connected globally via name.

To place several pins at once

1 From the part editor's Place menu, choose Pin Array (ALT, P, I).
or
From the part editor's tool palette, choose the pin array tool.

The Place Pin Array dialog box displays. You must provide information about the array of pins you are placing. This dialog box is similar to the Place Pin dialog box, with these exceptions:

- The Starting Name and Starting Number text boxes are used to specify a name and pin number that is incremented for each pin that is placed.
- The Number of Pins, Increment, and Pin Spacing text boxes are used to specify how the pins are placed.

2 In the Starting Name text box, enter the name of the first pin. If the pin name ends in a digit (0–9), subsequent pin names in the array will be incremented by the value in the Increment text box (if there is no value in the Increment text box, the pin names are incremented by 1).

In the Starting Number text box, enter the number of the first pin. Subsequent pin numbers will be incremented by the value in the Increment text box (if there is no value in the Increment text box, the pin names are incremented by 1).

In the Number of Pins text box, specify how many pins you want to place.

In the Increment text box, specify the number by which you wish to increment the pin name (if it ends in a digit) and pin number for each pin in the array.

In the Pin Spacing text box, specify the number of grid units you would like between each pin.

You can use the default settings for the Shape, Type, and Width options, or you can change them to fit your requirements.

3 When the pin is specified to your requirements, choose the OK button.

4 Using the pointer, drag the pin array to the desired location along the part body border.

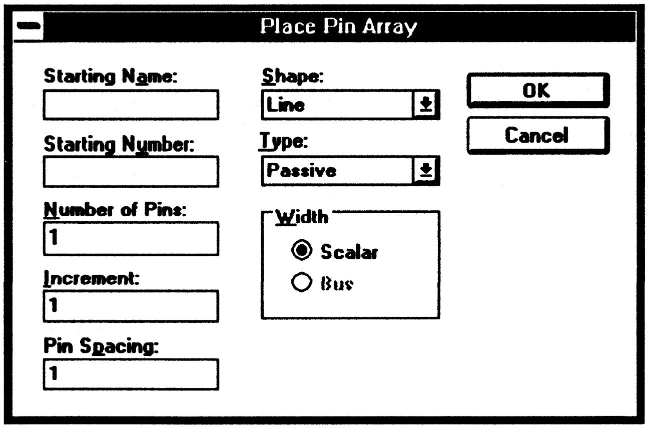
5 Click the left mouse button to place the array of pins. You can place multiple copies of the pin array by clicking the left mouse button each time you want to place the pins. Each time you place the pin array, the pin names and pin numbers are incremented based on the number of the last pin placed.

If the pin array is longer than the edge of the part body, the part body border expands to accommodate the extra pins.

6 When you are done placing pins, choose the selection tool or press ESC to dismiss the pin array tool.

Tip Once you place an array of pins, you can edit their properties by selecting the individual pins and choosing Properties from the Edit menu. This opens the spreadsheet editor, which you can use to edit the pins as a group.

Place Pin Array dialog box



Starting Name The name of the first pin in the array. If the name ends with a digit (0–9), each pin in the array is incremented by the value specified in the Increment text box.

For a name with a bar over it (indicating negation), type a backslash character after each letter. For example, type **R\E\S\E\T** to define the name:

$\overline{\text{RESET}}$

Starting Number The number of the first pin in the array. Each pin in the array is incremented by the value specified in the Increment text box.

Number of Pins The number of pins in the array.

Increment The number by which to increment the pin name (if it ends in a digit) and pin number for each pin in the array. This can be a negative number.

Pin Spacing The number of grid units between each pin in the array.

Shape The shape of the pins in the array, as described in *Place Pin dialog box*, earlier in this chapter.

Type The type of the pins in the array, as described in *Place Pin dialog box*, earlier in this chapter.

About power and ground pins

Both homogeneous and heterogeneous parts may have shared pins. A common use of shared pins is for supply pins (power or ground), which are referred to in Capture as *power pins*. Normally, power pins are invisible, and thus *global*—that is, they are connected to like-named power pins, power objects, and power nets throughout the design.

If you create a part with visible power pins, they are not global. You must connect them to a net using a hierarchical port, an off-page connector, or a power or ground object.

On heterogeneous parts, power pins may not appear on every part in the package. If the pins are visible, they must be placed on at least one part in the package, and that part must be placed in the design in order for the power connections to appear in the netlist.

On homogeneous parts, power pins appear on every part in the package. The pin names are filled in automatically, but you must set the pin numbers. For the pins to be shared, make sure that both the pin names and the numbers are the same for every part in the package.

Caution If you place the same pin on multiple parts in the package, you can inadvertently short two nets. Use caution, and always run Design Rules Check before creating a netlist to avoid this problem.

To display invisible power pins

You can display power pins throughout a design, or on individual part instances. Merely displaying invisible power pins does not change their global nature. The method you choose to display power pins determines whether you can connect to them.

- **On a part instance:** Select the part; then, from the Edit menu, choose Part. Select the Power Pins Visible option, and choose the OK button.

  If you connect to an invisible power pin that is displayed by this method, the pin is isolated from the design-wide power net.

- **Throughout a design:** From the design manager's Options menu, choose Design Properties, and then choose the Miscellaneous tab. Select the option Display Invisible Power Pins (for documentation purposes only). Choose the OK button.

  You cannot connect to an invisible power pin that is displayed by this method.

Invisible power pins are always displayed in the part editor.

Editing an existing part

You can edit an existing part in the library in which it resides, or you can edit a part after it is placed on a schematic page.

In a library

Once you edit a part in a library, you can update existing designs with the new part using the Update Cache or Replace Cache commands on the design manager's Design menu.

To edit a part in a library

1. From the File menu, choose Open, then choose Library (ALT, F, O, L).

   A standard Open dialog box displays. Choose the library containing the part you wish to edit.

2. The library opens, showing all its parts in the design structure pane. Double-click the part you want to edit.

   The selected part appears in the part editor.

3. Edit the part.

   You can resize it, add or delete graphics or symbols, and add or delete pins. These processes are all described in *Creating a new part* earlier in this chapter. You can also edit the part's properties. Editing properties is described in *Chapter 2: The Capture work environment.*

4. When you are done editing the part, you must save it. From the File menu, choose Save (ALT, F, S).

   The part is saved in the library.

Tip In the design manager, you can create a new part from an existing part by dragging a copy of the part (press the CTRL key while you drag the part) to a new library and then editing the part.

You can also create an alias of the part by changing to package view in the part editor, choosing Package Properties from the Options menu, and then choosing the Part Alias button. Once you make a part alias, you must save the part to have the alias show up in the design manager. In the design manager, part aliases display with a line through their part icons.

On a schematic page

Once you edit a part on a schematic page, you can apply the edits to all instances of the same part in the design, or you can apply the edits only to the particular part instance you edit.

Caution Once you edit a part instance on a schematic page, it is no longer linked to its corresponding library part. In addition, a new part (with "-*n*" appended to the original part name) appears in the design cache. This means that you can't perform a cache update on an edited part, since there is no link to the original library.

To edit a part instance on a schematic page

1 Select a part instance on a schematic page.

2 From the Edit menu, choose Part (ALT, E, T).

The library part that was used to define the part instance appears in the part editor.

3 Edit the part.

You can resize it, add graphics or symbols, and add or delete pins. These processes are all described in *Creating a new part* earlier in this chapter. You can also edit the part's properties. Editing properties is described in *Chapter 2: The Capture work environment.*

4 When you are done editing the part, you must close it and save the changes on the schematic page. From the File menu, choose Close (ALT, F, C).

A message displays asking if you would like to:

- Update only the part instance being edited (Update Current)
- Update all instances of the part in the design (Update All)
- Discard the edits to the part and return to the schematic page editor (Discard)
- Cancel the close operation and return to the part editor to continue making changes to the part (Cancel)

After your respond to the message, the part editor window closes. Depending on your response to the message, the change is reflected in the selected part or in all instances of the selected part.

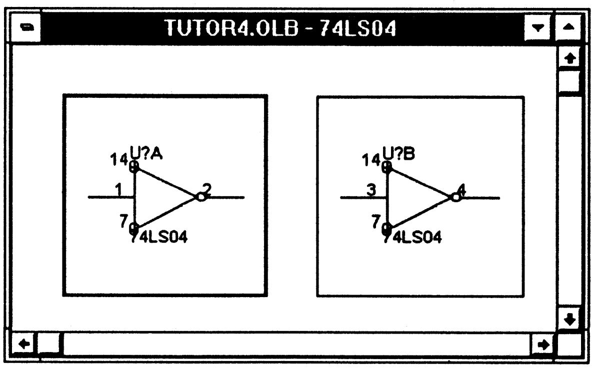
A new part (with "-*n*" appended to the part name) appears in the design cache, since the link to the library part no longer exists.

Viewing parts in a package

For a package containing multiple parts, you can use the Package command on the View menu to view all the parts in the package at once. You can then choose which part in the package to edit. If the package is homogeneous, you can only edit pin names and pin numbers, since each part in a package must have the same graphic representation. If the package is heterogeneous, you can make each part in the package graphically distinct.

To view a package

1 From the part editor's View menu, choose Package (ALT, V, K). The package view window (shown below) replaces the part editor window.



2 You can move from part to part using the ARROW and TAB keys.

If the package is heterogeneous, you can select which part to edit by double-clicking on it. The package view window closes and the selected part opens in the part editor window.

If the part is homogeneous, double-clicking on any of the parts in the package closes the package view window and opens the part in the part editor window.

Tip When editing a heterogeneous package, a quick way to move from one part in a package to another is to use the Next Part (ALT, V, X) and Previous Part (ALT, V, V) commands on the View menu.

Viewing a part's convert

A convert view is an alternate view of a part. It can be used for things such as a DeMorgan equivalent of a part. If a part has a convert view, you can easily switch between the normal view and convert view in the part editor.

To view a part's convert

1 From the part editor's View menu, choose Convert (ALT, V, C).

2 To go back to the normal view of the part, choose Normal from the View menu (ALT, V, N).

Note If you're editing a part that doesn't have a convert view, the Convert command is not available.

Processing your design

Part Four provides details about processing your design once you have created it in the schematic page editor. It gives an overview of the design process, including when to use logical or physical view as you process your design. It also provides detailed information about each of Capture's tools—located on the design manager's Tools menu—that you use to process your design.

Part Four contains these chapters:

Chapter 12: About the design process provides general guidelines for processing your design. It describes when to use Capture's different processing tools, and explains when to use logical view and when to use physical view.

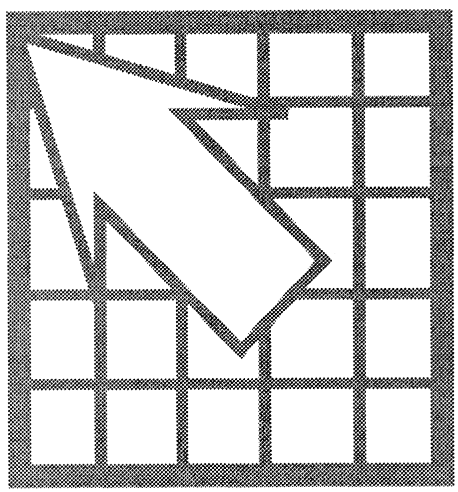
Chapter 13: Preparing to create a netlist describes the tools you use to prepare a design for creating a netlist. It describes the Update Part References, Update Properties, Design Rules Check, and Gate and Pin Swap tools.

Chapter 14: Creating a netlist explains how to create a netlist using the Create Netlist tool.

Chapter 15: Creating reports explains how to create reports using the Bill of Materials and Cross Reference tools.

Chapter 16: Exporting Capture data describes the Export Properties, Import Properties, and Extract PLD tools used to move data into and out of Capture.

Part Four



Chapter 12

About the design process

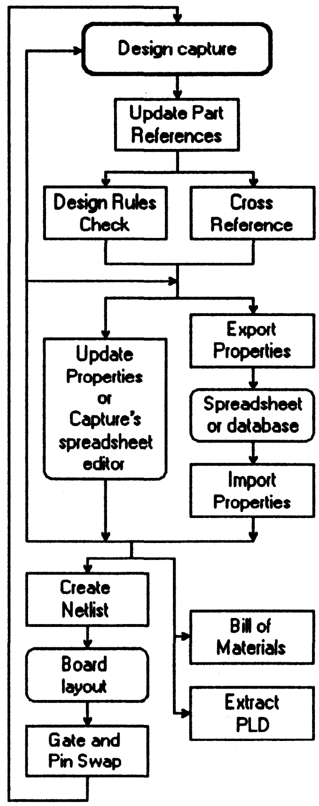
Your design process typically involves placing and connecting parts in the schematic page editor, then using Capture's tools to specify how parts are to be packaged and uniquely identified. You then add information for simulation, synthesis, board layout, purchasing, or other external functions, then create a netlist and incorporate "back annotation" information from external applications.

As shown in the figure at right, you use Update Part References, Design Rules Check, and Cross Reference to package the parts in your design and to check that there are no unconnected parts, unwanted connections, or other invalid design conditions. In practice, you might run these tools several times before moving on to the next phase.

You can add properties or change their values at any point, and there are several ways to do this. To change the value of one or two properties, edit them on the schematic page. To edit properties on many parts at the same time, use Update Properties or Capture's built-in spreadsheet editor (from the Edit menu, choose Browse and then Parts). If you prefer editing in a full-featured spreadsheet or database program, use Export Properties to write design data out and Import Properties to read the changes back in.

Once you're satisfied with your design, use Create Netlist to convey design information to your board layout program or other external application. This is often the point at which you use Bill of Materials to create a list of parts used in the design or Extract PLD to create device files for use with OrCAD's Programmable Logic Design Tools 386+.

Some external tools create back annotation files containing packaging changes necessary because of routing or manufacturing constraints. Use Gate and Pin Swap to incorporate this information. After you do this, you may need to make additional changes to your design, then repeat some or all phases of the design process.



Design process overview.

Tools overview

Once you finish a first pass at placing and connecting parts in the schematic page editor, Capture provides the following commands on the design manager's Tools menu to help you complete the design process:

Capture tool	*Description*	*Described in*
Update Part References	Packages parts by resolving part references and pin numbers, or removes packaging information by resetting part references to their unassigned values.	*Chapter 13: Preparing to create a netlist*
Gate and Pin Swap	Swaps pins or gates, or changes packaging, based on a swap file your board layout program or you create.	*Chapter 13: Preparing to create a netlist*
Update Properties	Adds properties, or changes the values of properties, based on an update file you create.	*Chapter 13: Preparing to create a netlist*
Design Rules Check	Reports and flags violations of electrical rules and other design constraints, including identical part references, unconnected electrical objects, part type mismatches, off-grid parts, and more. Starts by removing existing DRC markers.	*Chapter 13: Preparing to create a netlist*
Create Netlist	Creates a text file listing the logical interconnections between signals and pins—for use by a board layout program or other external tool—in one of more than thirty standard formats.	*Chapter 14: Creating a netlist*
Cross Reference	Reports the schematic page and location of every part, for use in developing or documenting the design.	*Chapter 15: Creating reports*
Bill of Materials	Creates a formatted list of electrical and other parts in the design. Optionally adds information, based on an include file you create.	*Chapter 15: Creating reports*
Extract PLD	Creates a source file—for use with OrCAD's PLD 386+—for each programmable logic device in the design, based on text descriptions placed on the schematic page. Optionally creates a list of the source files created.	*Chapter 16: Exporting Capture data*
Export Properties	Creates a tab-delimited list—for manipulation in a spreadsheet or database program—of properties and values for each part in the design.	*Chapter 16: Exporting Capture data*
Import Properties	Adds properties, or changes the values of properties, based on a tab-delimited list in the format created by the Export Properties command.	*Chapter 16: Exporting Capture data*

Choosing between logical and physical view

In logical view, you see only one instance of each schematic in your design; in physical view, you see every occurrence of those schematics. The effect of a processing command reflects the view—logical or physical—that's active when you choose the command from the Tools menu. In other words, the command processes what's visible in the design manager.

With a flat design or a simple hierarchical design, it may not matter which view is active, because there's only one occurrence of each schematic, and they're all visible in either view. With a complex hierarchy, though, most of the tools produce different results in different views.

Here are some guidelines:

- If you use one tool in a given view, use all the tools you need in that same view.
- Run Update Part References in the current view, and run it again if you change views.
- Back annotate a complex hierarchical design—using Gate and Pin Swap—in physical view only.
- If you work in both views, use Gate and Pin Swap in physical view only.

For example, consider a complex hierarchy in which you place a part at three locations in one schematic and use the schematic twice in your design. In physical view, the tools recognize and act upon all six occurrences of the part; but in logical view, they see only the three instances in the one visible instance of the schematic. If the parts are packaged in fours, then a bill of materials produced in physical view shows you need two packages, but a bill of materials produced in logical view shows you need only one.

Likewise, Cross Reference documents only the three instances of the part in logical view, but all six occurrences in physical view. The same is true for most of the other tools.

Notes Extract PLD produce the same information in either view. To reduce confusion, this command is available only in logical view.

For libraries, only the Update Properties, Export Properties, and Import Properties commands are available. The concept of view does not apply to libraries, so commands on the View menu are unavailable.

Working in both logical and physical view

You will probably find that you do most of your design processing in one view or the other. Some designs lend themselves to a complex hierarchy, others do not. Also, people think of similar designs in different ways. But there are some situations in which you might want to work on a single design in both views. For example, a design that reuses circuitry is suited for a complex hierarchy. Logical view allows you to easily edit re-used circuitry, while physical view allows you to map re-used circuitry to individual, unique physical parts.

Consider a circuit board you design as a complex hierarchy. You must be in physical view when you create a netlist for the board layout program. But most simulators accept complex hierarchies, so you can create a simulation netlist in logical view.

Here's how you might use this technique:

- Capture your design and update part references in logical view. Create a netlist in VHDL, Verilog, or EDIF 2 0 0 format to send to a simulator. Use the simulation results to refine the logic of your design. When you are satisfied with the basic logic, switch to physical view.
- In physical view, update part references again—you must have unique part references before you proceed. Next, create another netlist in PCB or one of the other flat netlist formats to send to your board layout program. After the board is laid out and routed, use Gate and Pin Swap and Update Properties to modify parts and pins and to add timing information.
- Still in physical view, create a new netlist in VHDL, Verilog, or EDIF 2 0 0 format to send back to the simulator. On this pass, the simulator tests the timing of the final board as well as the basic design logic.

Chapter 13

Preparing to create a netlist

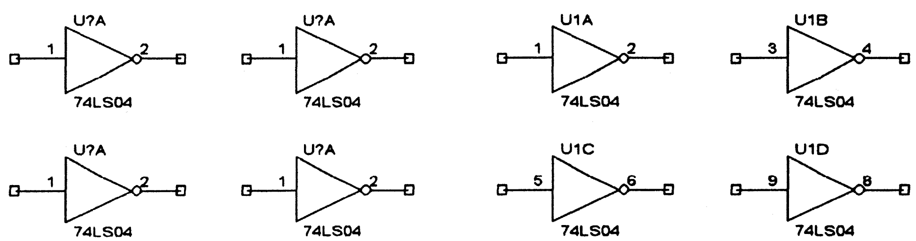
Updating part references

After you place parts on a schematic page, all parts need to be uniquely identified. You can manually edit each part and assign unique part references, but it is much easier to use the Update Part References command on the design manager's Tools menu. This tool assigns unique part references to each part in a design.

You use Update Part References after you've placed all parts and before you use other Capture tools. You can update references incrementally, so that previously assigned part references are not changed; or you can update part references unconditionally, changing all the parts across all the schematic pages processed.

Tip Update Part References also assigns individual parts to a multiple-part package, thereby assigning unique pin numbers to each part in a multiple-part package. This process is sometimes called *packaging*. For more information on controlling packaging in multiple-part packages, see Capture's online help.

Parts are updated in the order in which they appear on a schematic page, going from left to right and top to bottom, as shown in the figure below.



Before updating part references *After updating part references*

Parts are updated from left to right and top to bottom.

To update part references

1 From the design manager's View menu, choose Logical to update part references on part instances.
or
From the design manager's View menu, choose Physical to update part references on part occurrences.

2 In the design manager's design structure pane, select the schematic pages on which to update part references.

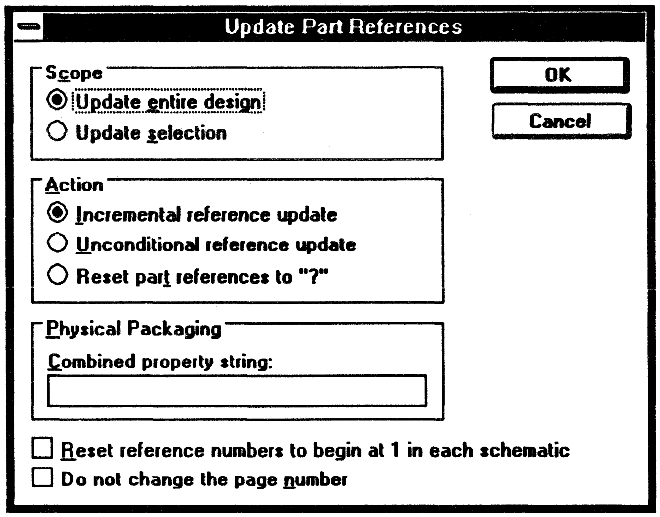
3 From the design manager's Tools menu, choose Update Part References (ALT, T, U).
or
Choose the update part references tool from the toolbar. 

The Update Part References dialog box displays.

4 Set the options on this dialog box as necessary. You must specify whether to update the entire design or just the schematic pages selected in the design manager, whether to update the part references that haven't yet been updated, update all part references, or reset part references to "?". These options are described in the section *Update Part References dialog box.*

5 When the Update Part References dialog box has the settings you want, choose the OK button to begin the update.

Update Part References dialog box



Scope Specifies whether to update all the reference designators in the design or just those on the selected schematic pages.

Incremental reference update or **Unconditional reference update** or **Reset part references to "?"** Specifies whether to incrementally update parts (those parts with a question mark in the part reference), unconditionally update all the parts in the selected schematic pages, or to reset all the part references to "?".

Physical Packaging Specifies the properties that must match for Capture to group parts in a single package. See Capture's online help for information on how the Update Part References tool packages individual parts into multiple-part packages, as well as information about combined property strings.

Reset reference numbers to begin at 1 in each schematic If this option is selected, Capture numbers part references beginning with 1 in each schematic. Otherwise, Capture looks at the selected schematic pages, finds the highest part reference number, and begins numbering part references from that number.

Do not change the page number If you are running Update Part References in logical view, the schematic pages are renumbered based on the order in which they appear in the design structure pane of the design manager. If you select this option, this does not happen.

Tip The properties that contain the page number are part of the title block. They are called "page number" and "page count." If you create your own custom title blocks, you can give them these properties so that they are updated when you run Update Part References.

Updating properties

If you need to edit the properties for a few parts or nets, you can do so in the schematic page editor. If, however, you want to make changes to a number of parts or nets, the Update Properties tool is much quicker and easier than hand-editing each property you'd like to change. You can use Update Properties to edit any properties except part value, part reference, and net name. You can also use Update Properties to add properties. In essence, it is a search-and-add-or-replace tool.

To update properties, you create a file, called an *update file*, with one line for each part or net to change. The leftmost column identifies the part or net (by specifying the property value to match), and the remaining columns provide the new property value. The format of the update file is described in *Update file format* later in this section.

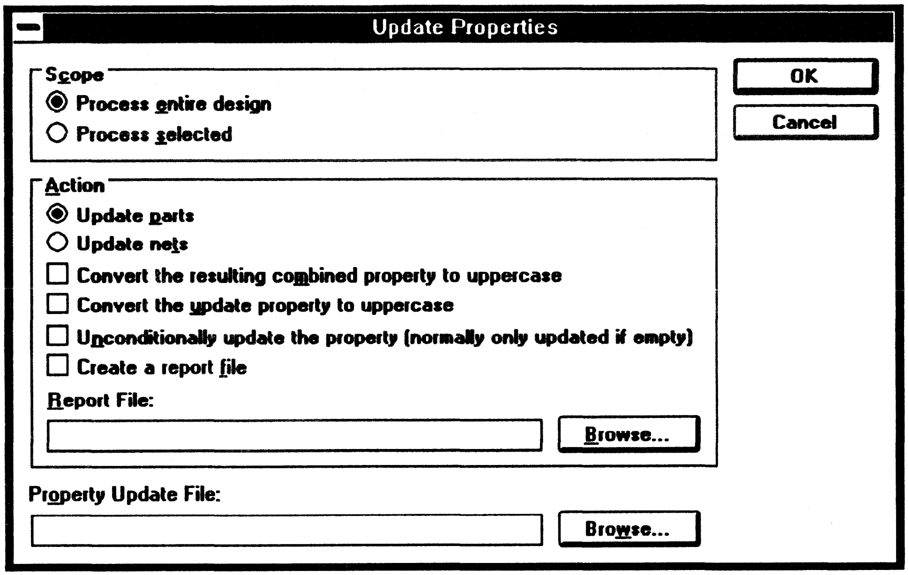
You can run Update Properties on designs in both logical and physical view. You can run Update Properties on libraries as well as designs.

To update part or net properties

1. Using a text editor, create an update file, as described in *Update file format* later in this section.
2. From the design manager's View menu, choose Logical to update properties on part instances.
   or
   From the design manager's View menu, choose Physical to update properties on part occurrences.
3. To process only part of your design, select the pages to process in the design manager's design structure pane.
4. From the design manager's Tools menu, choose Update Properties (ALT, T, P).

   The Update Properties dialog box displays.
5. Set the options on this dialog box as necessary. You must specify whether to process the entire design or just the schematic pages selected in the design manager, and whether to update parts or nets. You can customize your update further by specifying that your comparison and update strings be converted to uppercase. You can also have Capture create a report file listing the properties that it changes. You must specify the name of the update file containing the properties to match and the text to place in the specified properties. These options and others are described in the section *Update Properties dialog box.*
6. When the dialog box has the settings you want, choose the OK button. Capture updates the properties you specified. If you set up the dialog box to create a report file, you can use a text editor to view the file when the Update Properties tool is done.

Update Properties dialog box



Scope Specifies whether to process the entire design or just the selected schematic page or pages.

Update parts or **Update nets** Specifies whether to update the properties of parts or nets.

Convert the resulting combined property to uppercase Converts the combined property values to uppercase before a comparison is done.

Convert the update property to uppercase Converts the update property to uppercase before it is placed in an object's property.

Unconditionally update the property By default, a property is updated only if it is empty. That is, properties with values already in them are not updated. If you select this option, Capture unconditionally changes the specified property regardless of whether it's empty.

Create a report file Specifies whether or not Capture creates a report file. If you select this option, enter the name of the report file in the Report File text box.

Property Update File The name of the update file containing the properties to match, the properties to update, and the values to use to update the properties. This file usually has an extension of .UPD.

Update file format

The update file is an ASCII text file that you create to specify the properties to match, the properties to update, and the values to use to update the properties. The file can include comments—any text to the right of a semicolon is ignored by the Update Properties tool.

The first line of the update file has this format:

CombinedPropString *PropToUpdate1* *PropToUpdate2 ...*

The first field is a combined property string specifying which properties to combine into a match string (note that property names in a property string must be enclosed in braces). The remaining fields in the line are the properties to update on each part or net to be updated. The combined property string and the property names entered in these fields must be enclosed in quotation marks.

See For information about combined property strings, see Capture's online help.

Subsequent lines of the update file have this format:

MatchString1 *Update1* *Update2 ...*
MatchString2 *Update1* *Update2 ...*
.
.
.

The match string is the text to use to compare with the values of the properties specified by the combined property string in the first line. The update fields are the values to be placed in the properties specified in the first line, if the match string matches the combined property string. These values must also be enclosed in quotation marks.

For example,

```
"{Value}"       "PCB Footprint"
"74LS00"        "14DIP300"
"74LS138"       "16DIP300"
"74LS163"       "16DIP300"
"8259A"         "28DIP600"
```

This tells Capture that the property to use as a match string is Value. Each time an object's Value matches the value listed in the left column of the update file, the corresponding text listed in the right column is placed in the object's PCB Footprint property. In this example, each time a part has a value of 74LS00, the text 14DIP300 is placed in the part's PCB Footprint property; each time a part has a Value of 74LS138, the text 16DIP300 is placed in the part's PCB Footprint property, and so on.

Checking for design rules violations

The Design Rules Check tool scans a design and checks for conformance to basic design and electrical rules. The results of this check are marked on the schematic page with DRC markers and are also listed in a report. This makes it easy to locate and fix design or electrical errors. You can search for DRC markers using the Browse command on the design manager's Edit menu, and then double-click on any item in the resulting list to go immediately to the location of the marker on your schematic page. Once you are viewing the marker on the schematic page, you can display the marker's text by double-clicking on it.

You can specify the conditions that cause errors to be generated. Optional checks performed by the Design Rules Check tool include off-grid parts; unconnected nets, pins, ports, and off-page connectors; identical part references; type mismatch parts; and design elements that are not compatible with OrCAD's Schematic Design Tools product.

Note When Design Rules Check checks for unconnected nets, it looks for nets with less than two connection points. Thus, a net can still have unconnected endpoints that aren't reported by Design Rules Check.

The Design Rules Check is helpful in preparing your design for use with other tools. For example, you can use the Design Rules Check tool to catch problems such as bus contention or shorted power pins before you generate a netlist to be used by simulation or synthesis tools.

The Design Rules Check reports two categories of electrical rules violations:

- Errors that *must* be fixed.
- Warnings of situations that may or may not be acceptable in your design.

You can control whether electrical rules violations are reported as errors or warnings on the ERC Matrix tab of the Design Rules Check dialog box. Errors are always marked with DRC markers on the schematic page. Warnings are also marked with DRC markers if you select the Create DRC markers for warnings option on the Design Rules Check dialog box. In the report generated by Design Rules Check, however, the problems are categorized as "WARNING" or "ERROR" so that you can immediately identify the more critical problems.

Once the Design Rules Check begins, it first removes existing DRC markers from the schematic pages being processed. This means that each time you run this process, the error markers on your schematic pages reflect the current state of your design.

You can also use the Design Rules Check tool to remove DRC markers from schematic pages, but not do any further checking. Just select the Delete existing DRC markers option on the Design Rules Check dialog box.

Caution You should ALWAYS run Design Rules Check before you create a netlist.

To check for design rules violations

1. In the design manager's design structure pane, select the schematic pages on which to check for design rules violations.

2. From the design manager's Tools menu, choose Design Rules Check (ALT, T, D).
   or
   Choose the design rules check tool from the toolbar. 

   The Design Rules Check dialog box displays.

3. This dialog box has two tabs.

   The Design Rules Check tab contains options about things you want included in the report generated by Design Rules Check. Select the things you want included in the report and provide a name for the report file. You can also specify if you want to create DRC markers on the selected schematic pages for both errors and warnings, or just for errors; or you can delete existing DRC markers instead of adding new DRC markers. Note that if you select the option to delete existing DRC markers, the options that customize the DRC report become dimmed and aren't available for selection. These options are described in the section *Design Rules Check dialog box, Design Rules Check tab.*

   The ERC Matrix tab contains the ERC (electrical rules check) matrix. This matrix is used to set the electrical rules that Design Rules Check uses when testing connections between pins, hierarchical ports, and off-page connectors. The pins, hierarchical ports, and off-page connectors are headings in the columns and rows of the matrix. Each cell in the matrix refers to an electrical connection specified by the intersection of a row and a column. If a cell contains a "W," the Design Rules Check tool generates a warning message when it encounters the specified connection. An "E" in a cell generates an error message. If a cell is empty, the Design Rules Check tool issues no warnings or error messages for that type of connection. To change the setting of a cell, place the pointer over the cell and click the left mouse button to cycle from "W" to "E" to empty. These options are described in the section *Design Rules Check dialog box, ERC Matrix tab.*

4. When both tabs of the Design Rules Check dialog box have the settings you want, choose the OK button. As Capture checks your design, a dialog box displays giving status information about the check. This dialog box also has a Cancel button that you can press at any time to stop the design rules check. If you stop the design rules check midstream, the schematics that have already been processed will have DRC markers if any error situations were encountered.

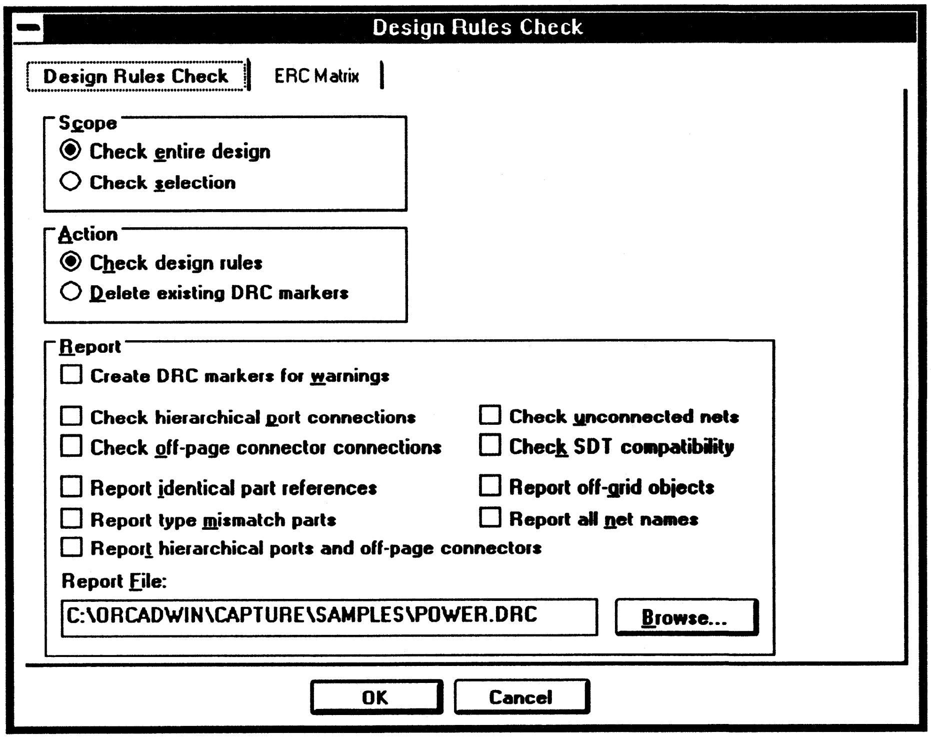
5. Once the design rules check is complete, there are two ways to view the results:

   - You can open the report file it creates using a text editor or word processor. This file has a default extension of .DRC. The session log also contains the same information.

   - You can use the Browse command on the design manager's Edit menu to display a list of all DRC markers in the design. This list gives information about each error and warning. Each DRC marker on a schematic page displays this same information. Once this list displays in the design manager's browse pane, you can double-click on an item to go directly to the item on its schematic page. Once you are viewing the marker on the schematic page, you can display the marker's text by double-clicking on it. You can also use the schematic page editor's Find command to find specific DRC markers. To do this, you must enter the text associated with the marker.

Tip To easily view the information contained in a .DRC report, set up a file association in the Windows File Manager. By doing this, you can quickly open the .DRC report by double-clicking on it in the File Manager.

Design Rules Check dialog box, Design Rules Check tab



Scope Specifies whether to process the entire design or just the selected schematic page or pages.

Check design rules or **Delete existing DRC markers** Specifies whether to check for design rules violations or just delete existing DRC markers. Note that if you select the Check design rules option, Capture deletes existing DRC markers before it begins the design rules check.

Report All the remaining options on the dialog box are report options, and leave messages and reports in the session log. Selected report options are also included in the specified report file.

Create DRC markers for warnings Design Rules Check always places DRC markers on the schematic page for errors defined on the ERC Matrix tab. If you select this option, it also places DRC symbols on the schematic page for warnings defined on the ERC Matrix tab.

Check hierarchical port connections Verifies that hierarchical ports in a parent schematic match those in the child schematic. Errors are generated if a hierarchical port specified on a parent schematic doesn't have a corresponding hierarchical port on the child schematic *with an identical name*; if the number of hierarchical ports are different between the parent and child schematics; and if the type of hierarchical ports is not identical between the two schematics.

Check off-page connector connections Verifies that off-page connector nets on a schematic page match those on other schematic pages.

Check unconnected nets Checks for these conditions: nets that aren't connected to at least two pins or ports; nets that don't have a driving signal; and two nets with the same name in a schematic, but no off-page connector or hierarchical port to connect them.

Check SDT compatibility Checks for compatibility with OrCAD's SDT product in case you plan on saving the design in SDT format. See Capture's online help for information about the rules you should follow if you are planning on using a Capture design in SDT.

Report identical part references Checks for unique part references, and reports parts that use the same part reference. For example, Capture considers two U1's to be identical, two U1A's to be identical, as well as U1 and U1A to be identical.

Report type mismatch parts Reports parts that are packaged into the same physical package, but whose package properties conflict (primarily source package and PCB footprint). This may happen if a part is edited on the schematic page, or if you manually package two parts that have different packages into the same physical package by making their part references the same.

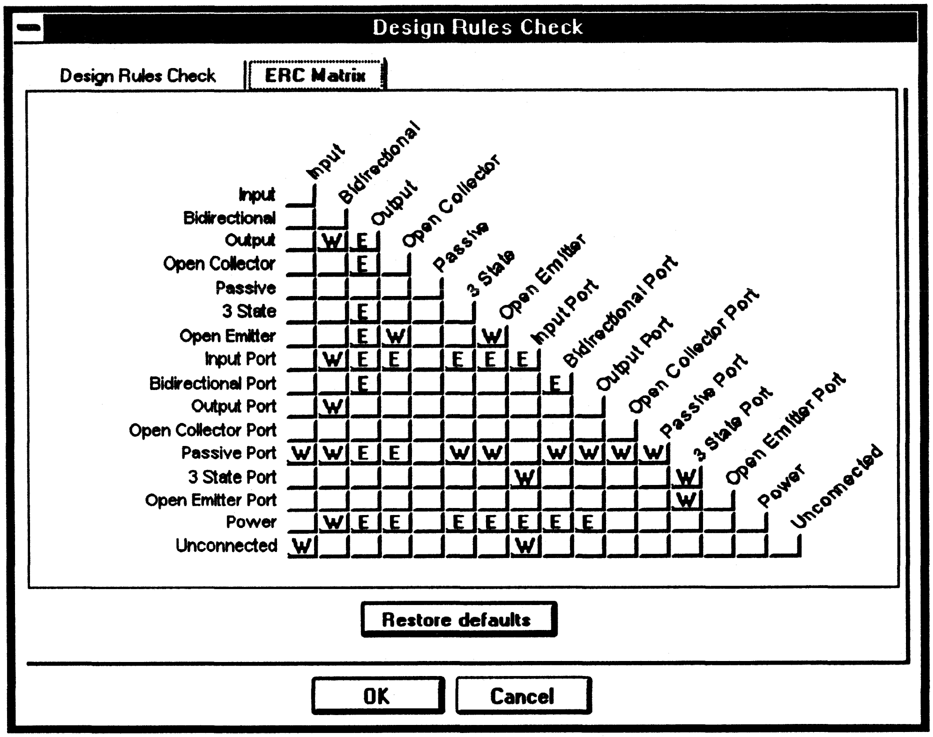
Report hierarchical ports and off-page connectors In the report file, lists all hierarchical ports and off-page connectors in the report file.

Report off-grid objects In the report file, lists the names and locations of objects that are off grid.

Report all net names In the report file, lists the names of all nets.

Report File The name of the report file containing the information prepared by Design Rules Check. This file usually has an extension of .DRC. For an example of the report generated by the Design Rules Check tool, see the next section *Sample Design Rules Check report.*

Design Rules Check dialog box, ERC Matrix tab



You use the ERC matrix to set the electrical rules that Design Rules Check uses when testing connections between pins, hierarchical ports, and off-page connectors. All types of pins, hierarchical ports, and off-page connectors are listed in the columns and rows in the table. A test is represented by the intersection of a row and column. Either the intersection is empty, or it contains a "W," or it contains an "E."

- An empty intersection represents a valid connection, in which case nothing is reported if you specify a report file.
- A "W" represents a warning.
- An "E" represents an error.

You can cycle through these three settings by pointing to an intersection and clicking the mouse button until the desired setting appears. For all rows except the Unconnected row, the ERC reports an error or warning for any net that has two connections as specified in the ERC matrix. For example, consider the dialog box above. If a net has an output pin and a bidirectional pin, a warning is issued based on the "W" in the intersection of the Output row and the Bidirectional column. For the Unconnected row, the ERC checks to see if the specified pin or port type is unconnected.

Tip You can also type `W` for warning, `E` for error and `N` for an empty intersection. In addition to these keys, you can use the arrow keys to select other intersections.

Sample Design Rules Check report

```
Design Rules Check
----------------------------------------
Checking Schematic: 4BIT
----------------------------------------
Checking Electrical Rules

Checking for Unconnected Wires

Checking Off-Page Connections

Checking Pin to Port Connections
WARNING:  [DRC0014]  Type of pin above does not match the pin type of
    corresponding port below fulladd_1,SUM
WARNING:  [DRC0014]  Type of pin above does not match the pin type of
    corresponding port below fulladd_2,SUM
WARNING:  [DRC0014]  Type of pin above does not match the pin type of
    corresponding port below fulladd_3,SUM
WARNING:  [DRC0014]  Type of pin above does not match the pin type of
    corresponding port below fulladd_4,SUM

Checking for Invalid References

Checking for Duplicate References

Checking for Compatibility with SDT

Reporting Off-Grid Objects

Reporting Ports
    S1
    S2
    S3
    CIN
    S[0..3]
    X[0..3]
    Y[0..3]
    Y0
    Y1
    X0
    COUT
    Y2
    X1
    Y3
    X2
    X3
    S0

Reporting Off-Page Connections

Reporting Globals

Reporting Net Names
    S0
    N00074
    N00072
    N00070
    COUT
    CIN
    S[0..3]
    X[0..3]
```

Sample Design Rules Check report for the 4BIT.DSN design (page 1 of 3).

```
    Y[0..3]
    Y3
    Y2
    Y1
    Y0
    X3
    X2
    X1
    X0
    S3
    S2
    S1
----------------------------------------
Checking Schematic: FULLADD
----------------------------------------
Checking Electrical Rules
ERROR:  [DRC0004]  Possible pin type conflict halfadd_A,SUM Output
    Connected to Bidirectional Port
Checking for Unconnected Wires
Checking Off-Page Connections
Checking Pin to Port Connections
Checking for Invalid References
Checking for Duplicate References
Checking for Compatibility with SDT
Reporting Off-Grid Objects
Reporting Ports
    X
    Y
    CARRY_IN
    CARRY_OUT
    SUM
Reporting Off-Page Connections
Reporting Globals
    VCC
    GND
Reporting Net Names
    VCC
    CARRY_OUT
    N00040
    N00038
    N00036
    Y
    X
    SUM
    CARRY_IN
    GND
----------------------------------------
Checking Schematic: HALFADD
----------------------------------------
Checking Electrical Rules
```

Sample Design Rules Check report for the 4BIT.DSN design (page 2 of 3).

```
Checking for Unconnected Wires

Checking Off-Page Connections

Checking Pin to Port Connections

Checking for Invalid References

Checking for Duplicate References

Checking for Compatibility with SDT

Reporting Off-Grid Objects

Reporting Ports
    X
    Y
    CARRY
    SUM

Reporting Off-Page Connections

Reporting Globals
    VCC
    GND

Reporting Net Names
    SUM
    N00039
    GND
    VCC
    N00033
    N00031
    X_BAR
    Y
    X
    CARRY
```

Sample Design Rules Check report for the 4BIT.DSN design (page 3 of 3).

Swapping gates and swapping pins

When you need to transfer packaging information to your schematic from other EDA tools, use the Gate and Pin Swap tool.

Note When you need to back annotate properties, use the Update Properties tool.

Using Gate and Pin Swap, you can import changes created by external tools such as a PCB layout application. Capture uses a simple file format to allow gate swapping, pin swapping, and changing part references. If the external tool creates a back annotation file, edit the file to match the format described in *Swap file format* in this section.

When should you use Gate and Pin Swap? After you've completed your schematic design and while you are routing a printed circuit board, you might discover that you can reduce via count, track length, or routing complexity by exchanging two gates of one part. You would use the PCB layout tool to rewire the board to exchange (or swap) the connections of U1A and U1B. To ensure that your schematic design reflects the rewired board, you create a swap file using the PCB layout tool and then run Capture's Gate and Pin Swap. When you look at the schematic page, you will see that U1A is where U1B was before, and vice versa.

Tip If you find you do not like the results of the Gate and Pin Swap, you can choose the Rebuild Physical View command from the design manager's Design menu. This command will delete the physical view's added properties, pin swaps, and gate swaps, then recreate the physical view from scratch.

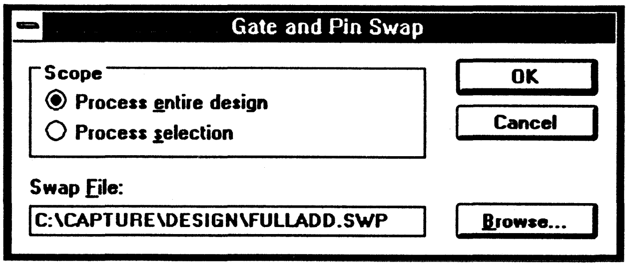
To swap gates and swap pins

1. Generally, a swap file is created by another application such as OrCAD's PCB 386+. Alternatively, you can create a swap file using a text editor, following the format described in *Swap file format* later in this section.
2. On the design manager's View menu, choose Logical to swap gates or pins on part instances.
   or
   On the design manager's View menu, choose Physical to swap gates or pins on part occurrences.
3. To process only part of your design, select the pages to process in the design manager's design structure pane.
4. From the design manager's Tools menu, choose Gate and Pin Swap.
   or
   Choose the gate and pin swap tool from the toolbar.

   The Gate and Pin Swap dialog box displays.
5. Set the options on this dialog box as necessary. You must specify whether to process the entire design or just the selected schematic pages. You must also specify the name of the swap file containing the gates and pins to swap. These options are described in the section *Gate and Pin Swap dialog box*.

6 When the dialog box has the settings you want, choose the OK button. Capture swaps the gates and pins you specified.

Gate and Pin Swap dialog box



Scope Specifies whether to process the entire design or just the selected schematic page or pages.

File Specifies the swap file. For more information, see *Swap file format* in this section.

Swap file format

A swap file is a text file containing old and new part references for use with the Gate and Pin Swap command on the design manager's Tools menu. A swap file is typically created by another application, such as OrCAD's PCB 386+. Or you can create a swap (.SWP) file using any text editor that allows you to save the file in ASCII format. The file can include comments; any text to the right of a semicolon is ignored by the Gate and Pin Swap tool.

Each line (unless preceded by a semicolon) causes one action. The elements of each line may be separated with any number of space or tab characters. In general, the first element of the line specifies the type of swap. If no swap type is specified, CHANGEREF is assumed. The other swap types are GATESWAP and PINSWAP.

When you are creating a swap file, include only the changes from the present state of the design to the state you want it to have. For example, you might place a part as U1 in the design, and change it in a PCB layout package first to U2, then to U3. The swap file should reflect the change from U1 to U3; it shouldn't include the intermediate step involving U2.

For gate swaps, make sure that the gates being swapped are identical. If they are not, you may get incorrect results.

For pin swaps, an additional element—the part reference—must be specified before the old and new values, as shown in the following example. Pin swaps are limited to pins of the same type and shape on the same part. For example, you can swap data pins on U5B, but you cannot swap a pin on U5B with a pin on U5C.

The following example illustrates a swap file. The comments to the right of the semicolons describe what the swap file will do.

```
CHANGEREF  U1   U2           ;Change part reference U1 to U2
CHANGEREF  U1A  U1B          ;Change part reference U1A to U1B
           U1C  U2C          ;Change part reference U1C to U2C
GATESWAP   U1   U2           ;Change part U1 to U2 and part U2 to U1
GATESWAP   U1A  U1B          ;Swap gates U1A and U1B
PINSWAP    U7   1     2      ;Swap pins 1 and 2 on U7
PINSWAP    U5B  "D0"  "D1"   ;Swap pins named D0 and D1 on U5B
PINSWAP    U3   5     6      ;Swap pins 5 and 6 on U3
```

Note Swap files created by OrCAD's PCB 386+ are called "was/is" files. These files contain no keyword identifiers, and therefore each line is assumed to be a CHANGEREF instruction. In the example above, the line without a keyword identifier (the third line from the top) is an example of how the changes are specified in a was/is file generated by PCB 386+.

Caution Gate and Pin Swap *does not* check to make sure that parts are the same type before a swap is performed. Therefore if you swap gates between dissimilar part types (as shown in the example below), odd results may occur in your design.

```
GATESWAP  U1C  U2B ;Swap gates U1C and U2B
```

Creating a netlist

After you create a design in Capture, you can create a netlist to exchange schematic information with other EDA tools. You can choose from more than 30 industry-recognized netlist formats. You can also create your own custom netlist format. Your choice of netlist depends on the destination application.

Using the Create Netlist tool

Before you create a netlist, be sure your design is complete, has been annotated (using Capture's Update Part References command), and is free from electrical rules violations. *Chapter 13: Preparing to create a netlist* describes how to use Capture's tools to prepare your design before you create a netlist.

To create a netlist

1 In the design manager, open the design for which you are going to create a netlist.

2 From the design manager's View menu, choose the appropriate view, depending on the type of netlist you are creating. The table below summarizes what type of netlist you will get (flat, complex hierarchical, or simple hierarchical) depending on the nature of your design, the view at the time you choose Create Netlist, and the netlist formatter you choose.

	EDIF 2 0 0, SPICE, VHDL, Verilog, or VST netlist format		*Layout, PCB, OHDL, or Other netlist format*	
	Logical view	*Physical view*	*Logical view*	*Physical view*
Flat design	Flat netlist	Flat netlist	--------------	Flat netlist
Hierarchical design	Hierarchical netlist	Hierarchical netlist	--------------	Flat netlist

Creating a flat or hierarchical netlist.

3 From the design manager's Tools menu, choose Create Netlist. The Create Netlist dialog box displays.

4 Choose a netlist format. Refer to the table in step 2 for guidelines on the format to select.

5 In the Netlist File entry box, enter a name for the output file. If the selected format creates an additional file (such as a map file or pinlist file), enter the filename in the second entry box.

6 Set the Part Value and PCB Footprint combined property strings to reflect the information you want in the netlist.

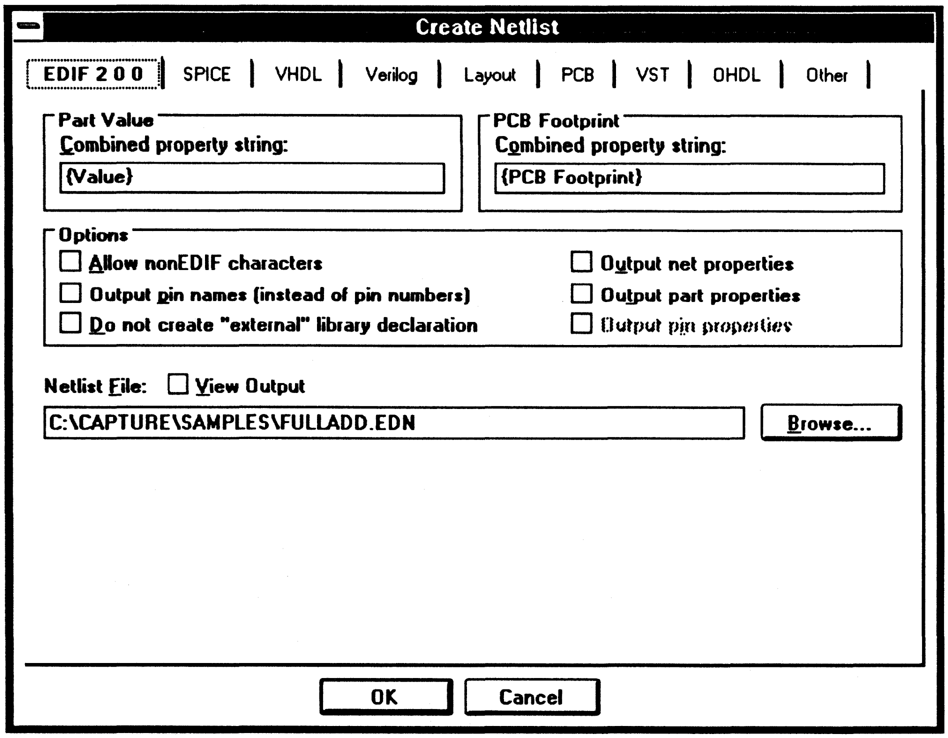
See For information about using combined property strings, see Capture's online help.

7 If necessary, choose the Options button, set the format-specific options, and choose the OK button to close the Netlist Options dialog box.

See For information about the options for individual netlist formats, see Capture's online help.

8 Choose the OK button to create the netlist.

Create Netlist dialog box



See For information about the Create Netlist dialog box, see Capture's online help.

Netlist format files

Capture includes over 30 netlist format files. They include:

Algorex	EEDESIGNER	PCB
Allegro	FutureNet	PLD
AlteraADF	HiLo	RacalRedac
AppliconBRAVO	IntelADF	Scicards
AppliconLEAP	Intergraph	SPICE
Cadnetix	Layout	Tango
Calay	Mentor	Telesis
Calay 90	MultiWire	Vectron
Case	OHDL	Verilog
CBDS	PADS 2000	VHDL
ComputerVision	PADS-PCB	VST
DUMP	PCAD	VST Model
EDIF 2 0 0	PCADnlt	WireList

See For information about the characteristics, formatting options, and an example of each netlist format, see Capture's online help.

Types of netlist format files

Capture includes two versions of each netlist format file: a compiled (.EXE) version and an uncompiled (.C) version written in the C programming language. You can view and edit uncompiled netlist format files with any text editor.

The uncompiled versions are useful as a basis for customized netlist format files. Use the MKNETFMT batch program to compile custom netlist format files, as described in *Creating custom netlist format files*. Once you compile a custom netlist format file, you can use it as described in *Using the Create Netlist tool.*

Each uncompiled netlist format file includes a comment section at the top that lists the restrictions imposed by that particular netlist format. The same information is included in Capture's online help.

Creating custom netlist format files

Using the uncompiled netlist format files included with Capture as a guide, you can write a custom netlist format file in the C programming language, then use the MKNETFMT batch program to compile your custom netlist format file. To do this, you must have Microsoft's C compiler, version 6.0 or 7.0; or Borland's C compiler, version 4.0. For more information about using MKNETFMT, type **`MKNETFMT ?`** at the DOS prompt.

Netname resolution

When you design schematics, you can assign a variety of netnames and aliases to signals that are connected. A netlist, however, needs exactly one name for each net.

If the Create Netlist tool encounters multiple names for a single net, higher priority netnames override lower priority netnames. Priority is determined by the source of the name, ranked from highest to lowest, as follows:

Named nets
Hierarchical port names
Off-page connectors
Power object names
Aliases
System-generated names

If there are any conflicts at any level of the comparison (if, for example, there are two power objects on a bus), they are resolved according to these rules:

- Between netnames of equal precedence, priority follows alphabetical order.
- If the net is a bus, the net alias assigned to the greatest number of bus members has highest priority.

In addition, if you are creating a flat netlist, the netname closest to the "root" of the design takes precedence over those further away.

A net may change names several times as Create Netlist works. For example, the net may start with an alias of Battery on one page, be renamed ToBattery from an off-page connector, change again to become DC as a port is encountered, and finally change to BatteryBackup when Create Netlist finds a named net closer to the root schematic.

Creating a netlist for use with OrCAD's PLD 386+

There are two methods for using Capture to design a programmable logic device for OrCAD's Programmable Logic Design Tools 386+. One is to draft the internal logic of a programmable logic device as a schematic, then use the Create Netlist tool to convert the schematic to an OrCAD Hardware Description Language (OHDL) file that is used by the PLD logic compiler. This method is described in this section.

The other method is to place the logic definition directly on a schematic, then extract signal placement and source statements from the schematic to create one or more unique OHDL (.PLD) files.

See For instructions on how to use the Extract PLD tool, see *Extracting PLD source code* in *Chapter 16: Exporting Capture data.*

For information about using OrCAD's PLD 386+, see the *Overview* chapter of the *Programmable Logic Design Tools User's Guide.*

To create a PLD 386+ schematic

1. Create a schematic of the logic design using the TTL.OLB or PLDGATES.OLB schematic symbol libraries.
2. Use pad symbols from the PLDGATES.OLB library to declare external signals. Create Netlist derives external signal names from the label adjacent to the pad symbol.
3. Place hierarchical ports or off-page connectors to transmit signals across the boundaries of schematic pages.
4. Place pipe-PLD or VECTORS text (or both) on the schematic page. Pipe-PLD and VECTORS text is used to document OHDL source code that cannot be expressed by the schematic circuit. These might include the architecture specification, Open-PLA property keywords, and test vector commands.

See For information about placing pipe-PLD or VECTORS text on a schematic page, see the *Programmable Logic Design Tools User's Guide.*

---

5. From the design manager's Tools menu, use the Update Part References command to assign unique part references to all components of the design.
6. From the design manager's Tools menu, choose the Create Netlist command. Choose the OHDL tab, and set the options to fit your requirements.
7. Choose the OK button to create an OHDL netlist.

See For information about the OHDL tab in the Create Netlist dialog box, as well as an example of a netlist created in the OHDL format, see Capture's online help.

Chapter 15

Creating reports

Capture provides two report tools that you can use to produce lists of the things contained in your design: Bill of Materials and Cross Reference.

Creating a bill of materials

You can use the Bill of Materials command from the design manager's Tools menu to create a bill of materials in a file which you can then print using a word processor or text editor. The bill of materials includes the properties item, quantity, reference, and part value. You can customize the report to include other properties.

Tip A bill of materials includes parts that don't have pins. This makes it possible to include non-electrical parts such as screws, washers, and other hardware that you may have in your design. These parts won't, however, appear in a netlist because they don't have pins.

To create a bill of materials

1 If desired, use a text editor to create an include file, as described in *Include file format* later in this section.

2 From the design manager's View menu, choose Physical.

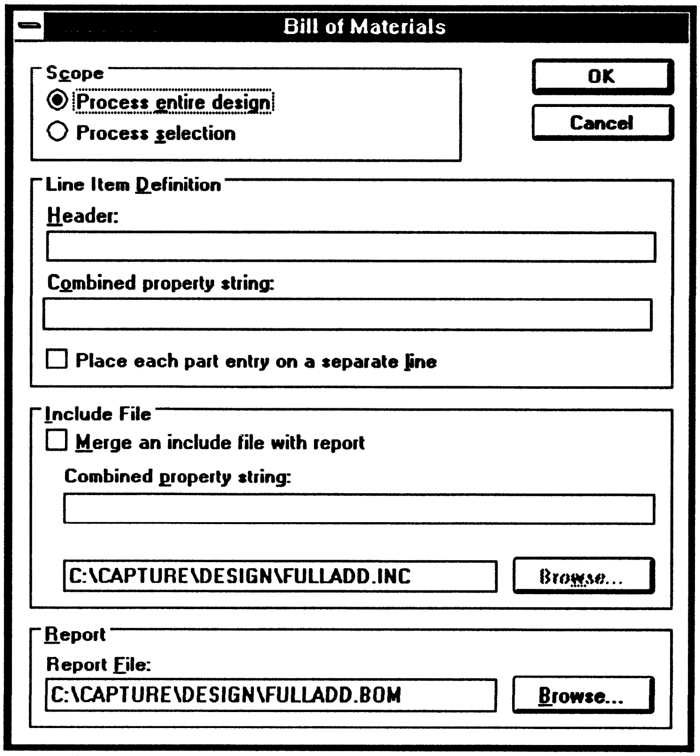
3 From the design manager's Tools menu, choose Bill of Materials.

   The Bill of Materials dialog box displays.

4 Fill in this dialog box as desired. If you want to customize the information contained in the bill of materials report, fill in the information in the Line Item Definition area. If you are using an include file, be sure to check the Merge an include file with report check box, enter the combined property string, and specify the name of the include file.

5 Choose the OK button when you are ready to create the report.

Bill of Materials dialog box



Scope Specifies whether to process the entire design or just the selected schematic page or pages.

Line Item Definition The items in the Line Item Definition area are used to customize the information included in the bill of materials.

Header Text that Capture places at the top of the first page. If this text box is left blank, there is no header on the first page. You can use this to specify column headers to match the data reported as defined by the combined property string.

Combined property string Specifies which properties Capture reports in the bill of materials. When you specify the combined property string, you enclose property names in curly braces. The properties in the curly braces are substituted with property values for the part (or left empty if the property is empty or doesn't exist on the part). The properties are left justified and separated by the characters you type outside the curly brackets . 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 define a left-justified, fixed-width column, use `\%`*n* where *n* is the width of the column. You can do the same in the header to line up the header with the data fields.

Place each part entry on a separate line Select this item if you want each part to be listed on a separate line.

Include File The items in the Include File area are used if you wish to use an include file to provide additional information for the bill of materials report. See the next section, *Include file format,* for information about creating an include file.

> **Merge an include file with report** Select this item if you want to use an include file.
>
> **Combined property string** Specify the property to use to match the property value specified in quotes on each line of the include file. This is the *search string*, and is compared with *match strings* specified in the include file.

Report File The name of the file to contain the bill of materials report.

Include file format

You can use an include file to have Bill of Materials add additional information to each line of the bill of materials. You create an include (.INC) file using any text editor that saves text in ASCII format.

The first line of an include file is a header. The bill of materials is always keyed to the part value, so the first line begins with a pair of single quotes with no spaces or other characters between them. The rest of the first line contains any information you want to include to make the file and the bill of materials more readable—this usually consists of headers for the values in the rest of the file.

The rest of the file contains a separate line for each part. Each line must begin with the property value (as specified in the Include File Combined property string on the Bill of Materials dialog box) enclosed in single quotes. This is the *match string*, and is compared with the *search string* specified in the Include File combined property string on the Bill of Materials dialog box. Following the part value (and on the same line) is the information that you want to add to the bill of materials. You can separate the part value from the additional information by any number of spaces or tab characters—Capture will align the first nonblank character in each line when it creates the bill of materials.

The following example illustrates an include file.

```
''       DESCRIPTION               PART ORDER CODE
'1K'     Resistor 1/4 Watt 5%      10000111003
'4.7K'   Resistor 1/4 Watt 5%      10000114703
'22K'    Resistor 1/4 Watt 5%      10000112204
'1uF'    Capacitor Ceramic Disk    10000211006
'.1uF'   Capacitor Ceramic Disk    10000211007
```

In the example above, the first column contains the match strings. If a match string matches the Include File Combined property string for the current part, the rest of the line (in the example above, the second and third columns) is tacked on to the end of the line item in the bill of materials.

Tip You can use an include file from OrCAD's Schematic Design Tools without modifying it.

Creating a cross reference report

The Cross Reference tool creates a report of all parts with their part references and part names. You may specify that the report also give the coordinates of each part.

To create a cross reference report

1 From the design manager's View menu, choose the desired view, Logical or Physical.

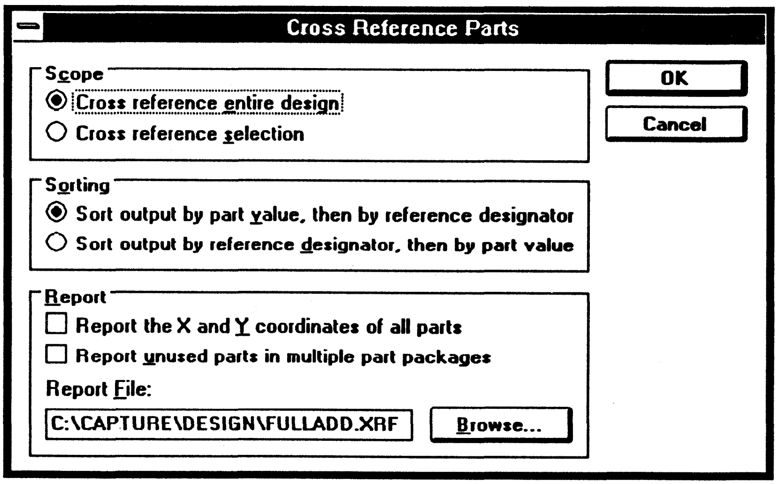
2 From the design manager's Tools menu, choose Cross Reference.

  The Cross Reference Parts dialog box displays.

3 Fill in this dialog box as desired. If you want to customize the information contained in the cross reference report, fill in the information in the Report area.

4 Choose the OK button when you are ready to create the report.

Cross Reference Parts dialog box



Scope Specifies whether to process the entire design or just the selected schematic page or pages.

Sorting Specifies whether to sort output by part value or part reference.

Report the X and Y coordinates of all parts If you check this option, the report includes the X and Y coordinates of all parts.

Report unused parts in multiple part packages If you check this option, the report identifies unused parts in multiple part packages.

Report File The name of the file to contain the cross reference report.

Chapter 16

Exporting Capture data

Capture provides two different ways to export data.

You can use Capture's Export Properties and Import Properties commands to edit properties of parts and pins in a spreadsheet or database program, or in a text editor that preserves tab characters. First you export the properties to a property file, then you edit the property file in the application of your choice, and finally you import the edited properties.

You can also extract PLD source code from a schematic, creating OrCAD Hardware Description Language files for use by OrCAD's PLD compiler.

Exporting properties to a tab-delimited file

You can export properties from a design or a library.

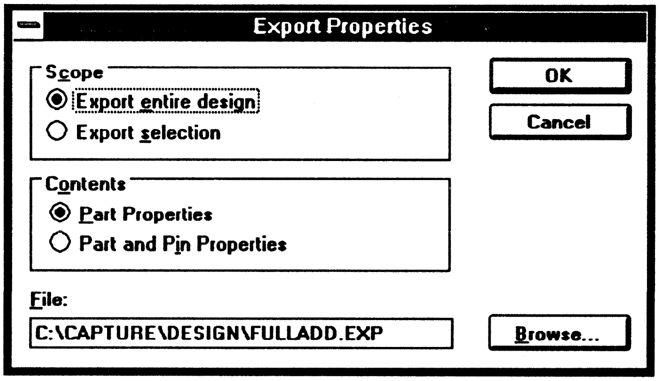
Note When you use the Export Properties tool, only the non-aliased parts are included in the property file, regardless of what you've selected; however, once you run Import Properties, the aliased parts change to match the non-aliased parts.

To export properties

1. In the design manager, open the library or design containing the parts to export.
2. If you are exporting properties from a design, select the schematics or schematic pages containing the properties to export.
   or
   If you are exporting properties from a library, select the parts to export.
3. From the design manager's Tools menu, choose Export Properties.

   The Export Properties dialog box displays.
4. Fill in this dialog box as desired. You can specify whether the property file is to include all documents in the file, or just the documents you selected in step 2. You can also specify whether you want to export properties for parts only, or for parts *and* pins.
5. Choose the OK button when you are ready to export the properties.

Export Properties dialog box



Scope Specifies whether to process the entire design or just the selected documents.

Contents Specifies whether to export only part properties or both part and pin properties.

File The name of the file to contain the exported properties.

Properties file format

When you export properties, Capture creates a tab-delimited list of keywords, identifiers, and properties, each of which is enclosed in double quotation marks. The first line of a property file begins with either the keyword "DESIGN" or the keyword "LIBRARY," to identify the document as being from either a design or a library. Subsequent lines of the property file begin with one of the following keywords: "PAGE," "HEADER," "PART," "PIN," or "SYMBOL."

If you export multiple pages in logical view, a PAGE and a HEADER line are listed for each page; if you export in physical view, only one HEADER line, but no PAGE lines, are listed. If you export both parts and pins, each PART line is followed by a PIN line. If you export symbols, there are no PIN lines following the SYMBOL lines, since symbols have no pins.

The HEADER lines are compiled from a superset of the property names found on parts (and pins, if applicable) from the whole page (for logical view), from the whole design (for physical view), or from the whole library. This means that (for a logical view) if part "1" has properties named "A," "B," and "C," part "2" has properties named "D," "E" and "F," and a pin has properties named "G," "H," and "I," then the HEADER line will have (after the first two columns) nine columns titled "A" through "I."

Editing a property file

You can edit the property file in a spreadsheet or database program, or even in a text editor (as long as the editor doesn't convert the tabs to spaces). Depending on which tool you use, you may see the property file as rows and columns of cells or fields, or as lines of text.

There are a few restrictions on the changes you can make in the property file:

- You *must not* change or delete the first line.
- You *must not* change or delete the first two fields in any line.
- In logical view, you *must not* change the sequence or number of lines; it is also a good idea *not* to change the sequence or number of lines in physical view or for libraries.
- Do not delete a field from a HEADER line without also deleting the corresponding fields from subsequent lines.

Caution If you add, delete, or reorder lines in a design property file created in logical view, the file cannot be imported.

If you move a PART line in a design property file (created in physical view) or 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 design or library.

In every case, it is much safer to make changes without adding, deleting, or reordering the lines in a property file.

Keeping these restrictions in mind, you can generally make the following changes:

- Add a field to a HEADER line and subsequent lines (add a column). This adds a property to parts and pins with a value in this field. The name of the property is the string in the HEADER line, and the value assigned to the part or pin is the string in the corresponding field. If the corresponding field is empty, Capture adds a property with no value and displays the property name as a place holder.
- Delete a field from a HEADER line and subsequent lines (delete a column). This has no effect on any part or pin. Deleting columns for properties you don't want to change may make the property file easier to edit. If you delete a field from a HEADER line without also deleting the corresponding fields from subsequent lines, Capture reports an error when you import the property file.

Caution Column deletion must be done with care. Columns may be deleted only from HEADER, PART, PIN, and SYMBOL lines. If you want to delete, for example, column three from a properties file, but accidentally include column three of the DESIGN line, Capture reports an error when you import the property file.

- Change the value of a field. This resets the value of the property on all parts or pins with that property.

Caution Be sure the same view is active when you import properties as when you export them.

Tip It is a good idea to update part references in the active view (logical or physical) before you export properties.

Because various popular spreadsheet and database applications behave differently, Capture can import properties with or without 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 You can change part references by editing the References column of the property file, but in a multiple-part package, the final element of the reference does not change. That is, changing U1A to U2B will result in the part reference U2A. This means that the designation of the part within the package does not change.

Importing properties

You can use the Import Properties command to import a file that you created with the Export Properties command and edited using a spreadsheet, database, or word processing application.

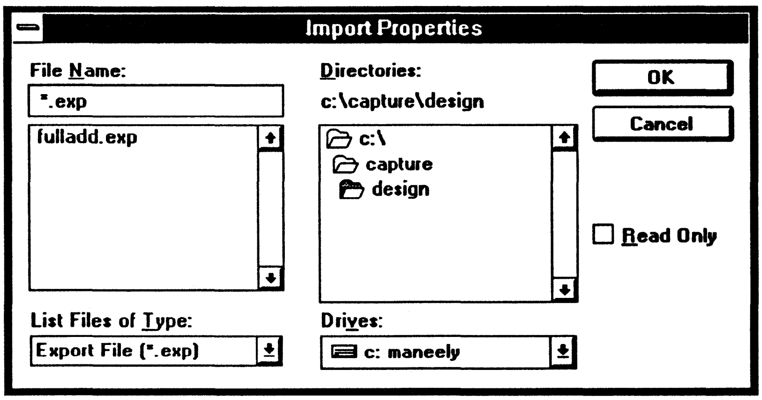
Caution *Do not* edit the design or library from which the properties were exported until after you import the changed properties. If you do, the Import Properties command may fail, and you will have to export and edit the properties again.

To import properties

1. In the design manager, open the library or design containing the parts to import.
2. From the design manager's Tools menu, choose Import Properties.

   The Import Properties dialog box displays.
3. Select the file containing the properties.
4. Choose the OK button when you are ready to import the properties.

Import Properties dialog box



This dialog box is a standard Windows dialog box for opening files.

Extracting PLD source code

There are two methods for using Capture to design a programmable logic device for OrCAD's Programmable Logic Design Tools 386+. One is to draft the internal logic of a programmable logic device as a schematic, then use the Create Netlist tool to convert the schematic to an OrCAD Hardware Description Language (OHDL) file that is used by the PLD logic compiler.

See For instructions on how to use Create Netlist to create a programmable logic device, see *Creating a netlist for use with OrCAD's PLD 386+* in *Chapter 14: Creating a netlist.*

The other method is to place the logic definition directly on a schematic, then extract signal placement and source statements from the schematic to create one or more unique OHDL (.PLD) files. This method is described in this section.

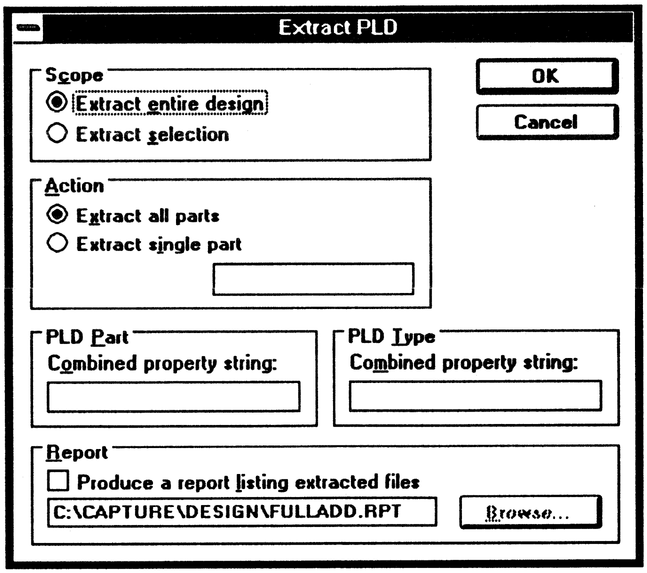
See For information about using OrCAD's PLD 386+, see the *Overview* chapter of the *Programmable Logic Design Tools User's Guide.*

To extract PLD source code from a schematic

1. Create a schematic that contains the PLD logic to extract. Use the MEMORY.OLB library or your own custom libraries to place symbols of programmable devices.
2. Edit the part name and pin names of each programmable logic device to suit the logic design.
3. On the Place menu, use the Text command to place PLD source statements directly on the schematic page. Follow these guidelines:
   - Each programmable device on a schematic must have a set of source statements.
   - For each programmable device, the first line of text associates the OHDL source statements with the part and has the form |PLD *part name*.
   - Each source statement must be placed as a separate text object.
   - The pipe character (|) precedes each source statement; comment lines do not start with the pipe character and are ignored by Extract PLD.
   - For each programmable device, the pipe characters that precede the source statements must be vertically aligned.
4. From the design manager's Tools menu, choose Extract PLD.

   If you choose the Extract all parts option, Extract PLD creates an OHDL (.PLD) compiler source file for each programmable device. If you choose the Produce a report listing extracted files option, Extract PLD creates a file (.RPT) that lists the extracted files. When the Extract PLD dialog box has the settings you want, choose the OK button.

Extract PLD dialog box



Scope Specifies whether to process the entire design or just the selected schematic page or pages.

Extract all parts or **Extract single part** Extract PLD extracts OrCAD Hardware Description Language (OHDL) source code from the design and produces one .PLD file for each device. If you select the Extract single part option, Extract PLD only creates one .PLD file for the device whose part value you enter in the text box.

PLD Part Specifies the property (typically the part value and speed rating) to place after the `Part:` keyword in the .PLD file. For information about combined property strings, see Capture's online help.

PLD Type Specifies the property (typically the device architecture) to place after the `Type:` keyword in the .PLD file. For information about combined property strings, see Capture's online help.

Produce a report listing extracted files Specifies whether to create a report listing the extracted files, and specifies the name of the file containing the report. A report of this type will have the format:

```
FILE1.PLD
FILE2.PLD
FILE3.PLD
```

You can use the report with the "Source is a Text File of Filenames" option on the Compile Source local configuration screen in Programmable Logic Design Tools 386+.

Sample Extract PLD reports

The following is an example of an Extract PLD report file (.RPT file extension) generated from the file:

`C:\CAPTURE\DESIGN\T1_1800C.pld`

The next several pages show the contents of the T1_1800C report file.

```
|"T1_1800C" 14:I1,
|           15:I2,
|           16:I3,
|           20:I4,
|           21:I5,
|           22:I6,
|           56:I7,
|           55:I8,
|           2:AIO1,
|           3:AIO2,
|           4:AIO3,
|           5:AIO4,
|           6:AIO5,
|           7:AIO6,
|           8:AIO7,
|           9:AIO8,
|           10:AIO9,
|           11:AIO10,
|           12:AIO11,
|           13:AIO12,
|           23:BIO13,
|           24:BIO14,
|           25:BIO15,
|           26:BIO16,
|           27:BIO17,
|           28:BIO18,
|           29:BIO19,
|           30:BIO20,
|           31:BIO21,
|           32:BIO22,
|           33:BIO23,
|           34:BIO24,
|           36:CIO25,
|           37:CIO26,
|           38:CIO27,
|           39:CIO28,
|           40:CIO29,
|           41:CIO30,
|           42:CIO31,
|           44:CIO33,
|           45:CIO34,
|           46:CIO35\,
|           47:CIO36,
|           57:DIO37,
|           58:DIO38,
|           59:DIO39,
|           60:DIO40,
|           61:DIO41,
```

Sample Extract PLD report for the T1_1800C.pld file (page 1 of 3).

```
|           62:DIO42,
|           63:DIO43,
|           64:DIO44,
|           65:DIO45,
|           66:DIO46,
|           67:DIO47,
|           68:DIO48,
|           43:CIO32,
|           54:I9,
|           50:I10,
|           49:I11,
|           48:I12,
|           17:CLK1,
|           19:CLK2,
|           51:CLK3,
|           53:CLK4
|
|Value:    "T1_1800C"
|Type:     "T1_1800C"
|Part:     "T1_1800C"
|Library:  ""
|
|Title:    "Test Equations for the EP1800 EPLD"
|Title:    "      May 28, 1990"
|Title:    "900513-A"
|Title:    "Revision 1"
|Title:    "Testing and Special Projects"
|Title:    "OrCAD, Inc."
|Title:    "9300 S.W. Nimbus Avenue"
|Title:    "Beaverton, OR 97008-7137"

This PLD file provides a Basic Function test
for COLUMN placement for the INPUT pins.
This PLD file should also provide a test for
automatic REGISTER BYPASS and THREE-STATE
buffering, as well as disabling the ASYNC. CLEAR
product term.

|Active-HIGH: I[1~12], CLK[1~4]
| Active-LOW: AIO[1~12],
|             BIO[13~24],
|             CIO[25~36],
|             DIO[37~48]

|AIO1  = CLK1 ## I1
|AIO2  = CLK1 ## I2
|AIO3  = CLK1 ## I3
|AIO4  = CLK1 ## I4
|AIO5  = CLK1 ## I5
|AIO6  = CLK1 ## I6
|AIO7  = CLK1 ## I7
|AIO8  = CLK1 ## I8
|AIO9  = CLK1 ## I9
|AIO10 = CLK1 ## I10
|AIO11 = CLK1 ## I11
|AIO12 = CLK1 ## I12
```

Sample Extract PLD report for the T1_1800C.pld file (page 2 of 3).

```
|BIO13 = CLK2 ## I1
|BIO14 = CLK2 ## I2
|BIO15 = CLK2 ## I3
|BIO16 = CLK2 ## I4
|BIO17 = CLK2 ## I5
|BIO18 = CLK2 ## I6
|BIO19 = CLK2 ## I7
|BIO20 = CLK2 ## I8
|BIO21 = CLK2 ## I9
|BIO22 = CLK2 ## I10
|BIO23 = CLK2 ## I11
|BIO24 = CLK2 ## I12

|CIO25 = CLK3 ## I1
|CIO26 = CLK3 ## I2
|CIO27 = CLK3 ## I3
|CIO28 = CLK3 ## I4
|CIO29 = CLK3 ## I5
|CIO30 = CLK3 ## I6
|CIO31 = CLK3 ## I7
|CIO32 = CLK3 ## I8
|CIO33 = CLK3 ## I9
|CIO34 = CLK3 ## I10
|CIO35 = CLK3 ## I11
|CIO36 = CLK3 ## I12

|DIO37 = CLK4 ## I1
|DIO38 = CLK4 ## I2
|DIO39 = CLK4 ## I3
|DIO40 = CLK4 ## I4
|DIO41 = CLK4 ## I5
|DIO42 = CLK4 ## I6
|DIO43 = CLK4 ## I7
|DIO44 = CLK4 ## I8
|DIO45 = CLK4 ## I9
|DIO46 = CLK4 ## I10
|DIO47 = CLK4 ## I11
|DIO48 = CLK4 ## I12
```

Sample Extract PLD report for the T1_1800C.pld file (page 3 of 3).

Glossary

A

alias *See* net alias, part alias.

ANSI Pronounced "annsee." Acronym for *American National Standards Institute*, an organization formed by industry and the U.S. government to develop trade and communication standards. Internationally, ANSI is the American representative to the ISO (International Standards Organization). *See also* ASCII.

arrow keys On your computer keyboard, the keys you use to navigate around your screen. Each key is marked with an arrow and is named for the direction in which the arrow points. There is an UP ARROW, DOWN ARROW, LEFT ARROW, and RIGHT ARROW key. Also known as *direction keys*.

ascend In a hierarchical design, to move from a child schematic page to its parent schematic page. This is done in the schematic page editor using the Ascend Hierarchy command on the View menu. *See also* child, descend, parent.

ASCII Pronounced "askee." Acronym for *American Standard Code for Information Interchange;* a seven-bit code—based on the first 128 characters of the ANSI character set—that assigns numeric values to letters of the alphabet, the ten decimal digits, punctuation marks, and other characters such as Backspace, Carriage Return, and Line Feed. ASCII is the most widely used character-coding set, and as such enables different applications and computers to exchange information. *See also* ANSI.

B

back annotate To apply modifications to part properties in a schematic, such as updating part references and pin numbers, swapping gates, or swapping pins. Parts are back annotated in the design manager, using the Gate and Pin Swap command or the Update Properties command on the Tools menu.

BBS *See* bulletin board system.

bookmark Just as you can place bookmarks in a book to mark a specific place, you can place bookmarks on a schematic page to indicate a location you would like to return to frequently. To place a bookmark, use the Bookmark command on the Place menu in the schematic page editor. To go to a bookmark when in the schematic page editor, use the Go To command on the View menu. To go to a bookmark when in the design manager, use the Browse command on the Edit menu to display bookmarks in the browse pane, and then choose the bookmark.

browse pane The right pane of the design manager window. This pane displays the results of queries done using the Browse command on the Edit menu. You can double-click on an object in the browse pane to go to that item on a schematic page. *See also* design structure pane.

bulletin board system A computer system equipped with one or more modems that serves as an information and message-passing center for dial-in users. Abbreviated *BBS*.

C

CAGE code Abbreviation for *Commercial and Government Entity Code*. A number—provided by the federal government to its suppliers—that can be present in the title block of a schematic page.

child In a hierarchical design, a schematic whose circuitry is represented by a hierarchical block in the parent schematic page. To move from parent to child is to descend the hierarchy. This is done in the schematic page editor by selecting the hierarchical block representing the child, and then choosing the Descend Hierarchy command on the View menu. A child contains circuitry referenced by its parent. The child may contain hierarchical ports that connect its signals to signals on the parent or to signals on other pages of the schematic. *See also* ascend, descend, parent.

Clipboard A temporary storage location used to transfer data between files and between applications. You transfer data to the Clipboard by using the Copy or Cut command on the Edit menu, and you insert data from the Clipboard by using the Paste command on the Edit menu.

complex hierarchy A design in which two or more hierarchical blocks (or parts with attached schematics) reference the same schematic. In the design manager, you can view a complex hierarchy two ways: in the logical view, you see one schematic that represents all references to that schematic, while in the physical view, you see a separate schematic for each reference to that schematic. *See also* hierarchical design, logical view, physical view, simple hierarchy.

convert An alternate form—such as a DeMorgan equivalent—that can be stored with each part. *See also* DeMorgan equivalent.

D

DeMorgan equivalent An electrically-equivalent part based on the DeMorgan rules of equivalence. These rules represent the duality of AND and OR in Boolean expressions: if all AND operations are changed to OR operations, all OR operations are changed to AND operations, and all variables and constants are negated, then the value of the expression remains unchanged. A DeMorgan equivalent can be stored in the convert of a part. *See also* convert.

descend In a hierarchical design, to open and view the child schematic represented by a hierarchical block in the parent schematic. To descend a hierarchical design, you select a hierarchical block in the schematic page editor, and then choose the Descend Hierarchy command from the View menu. *See also* ascend, child, parent.

design In Capture, a single file that includes all of the schematics, schematic pages, parts, and symbols that make up a design. You can view these design elements in the design manager. A basic design contains one schematic and one schematic page, while a complicated design may contain a virtually unlimited number of schematics, each with many schematic pages.

design cache A local library contained in each design that contains all the parts and symbols used in the design.

design manager The Capture window used to perform design-wide tasks, such as locating groups of objects or specific objects, creating a netlist, or generating reports.

design structure pane The left pane of the design manager window. This pane displays the structure of the schematics and schematic pages contained in a design. You can view the design structure in logical view (which shows one schematic page that represents all references to that schematic page) or in physical view (which shows a separate schematic page for each reference to that schematic page). *See also* browse pane, hierarchical design, logical view, physical view, simple hierarchy.

document A design, schematic, schematic page, library, part, or symbol. Each of these is part of a design file or a library file.

DRC The abbreviation for *Design Rules Check*, a tool found on the Tools menu in the design manager. This tool checks a design (or a subset of the design) for conformance to a set of configurable design criteria and electrical rules for creating netlists. *See also* netlist.

E

equivalent *See* convert, DeMorgan equivalent.

ERC The abbreviation for *Electrical Rules Check*, a subset of the *Design Rules Check* tool found on the Tools menu in the design manager. The ERC matrix is the decision matrix that tells the Design Rules Check tool the conditions to check for when evaluating connections between pins, hierarchical ports, and off-page connectors.

F

flat design A schematic structure without hierarchy (no hierarchical blocks or ports; no parts with attached schematics). A flat design can include schematic pages in which output lines of one schematic page connect laterally to input lines of another schematic page through objects called *off-page connectors*. You place this object using the Off-Page Connector command on the Place menu in the schematic page editor. Flat designs are practical for small designs with few schematic pages. *See also* hierarchical design, complex hierarchy, simple hierarchy, off-page connector.

G

graphic object An object drawn or placed on a schematic page or part—such as an arc, line, rectangle, ellipse, polygon, bitmap, or text— that has no electrical connectivity.

grid reference The border around a schematic page, providing a visual reference to the grid. Grid references can be used as a destination for the Go To command on the View menu. Grid references can be set to visible or hidden in both the Design Template and Schematic Page Properties commands on the Options menu.

H

heterogeneous part A package with multiple parts that are graphically different or contain different numbers of pins (for example, a relay). *See also* homogeneous part.

hierarchical block A symbol that refers to a child schematic in a design. The connection points on a hierarchical block are called hierarchical ports. You place a hierarchical block using the Hierarchical Block command on the Place menu. *See also* hierarchical port.

hierarchical port A symbol that specifies that a signal on one schematic page connects to a signal on another schematic page. A hierarchical port includes a name and a type (either scalar or bus), and may be part of a hierarchical block. You place a hierarchical port using the Hierarchical Port command on the Place menu.

hierarchical design A design structure in which schematics are interconnected vertically with hierarchical blocks. At least one schematic page, the root, contains symbols representing other schematics. *See also* complex hierarchy, simple hierarchy, flat design.

homogeneous part A package with multiple parts that are graphically identical. *See also* heterogeneous part.

I–J

inherent property One of the set of properties required for a given object. Unlike user-defined properties, inherent properties cannot be removed.

instance A part or symbol placed on a schematic page. You place part instances in logical view. If you change to physical view, you see occurrences of the part instances. *See also* instance property, logical view, occurrence, physical view.

instance property A property that is attached to an instance, as opposed to a property that is attached to an occurrence or added to a part in a library. You edit instance properties in logical view. Instance properties can be overridden by occurrence properties, which are not reflected on the instance. *See also* instance, logical view, occurrence, physical view.

K

K The abbreviation for *kilobyte*. *See* kilobyte.

kilobyte 2^{10} (1,024) bytes. The prefix *kilo* is taken from the metric system, where it stands for "one thousand." Abbreviated *K*.

L

library A collection of often-used parts, graphics, schematic pages, and symbols.

location An X,Y coordinate on the schematic page or part. You can move to a location using the Go To command on the View menu. in the schematic page editor.

logical view A view that displays the "folded" view of a design, and thus part references and pin numbers on part instances. Note that logical view doesn't reflect any changes made to occurrences in physical view. To display the logical view of a design, use the Logical command on the View menu in the design manager. *See also* instance, occurrence, physical view, part instance.

M

MB The abbreviation for *megabyte*. *See* megabyte.

megabyte 2^{20} (1,048,576) bytes. The prefix *mega* is taken from the metric system, where it stands for "one million." Abbreviated *MB*.

mirror To flip along the X (horizontal) or Y (vertical) axis, or both.

N

net All of the wires, buses, parts, and symbols that are logically connected via net names, aliases, off-page connectors, and hierarchical ports.

net alias A name used to specify signal connections between unconnected wires or buses. For example, if you have wires in two remote locations in a schematic, you can assign each wire an alias such as "ABC" to connect the signals without physically drawing a wire between them.

netlist An ASCII file that lists the interconnections of a schematic by the names of the connected signals, parts, and pins.

nonprimitive A part with an underlying hierarchy, such as an attached schematic.

O

occurrence An instance, placed on a schematic page, as displayed in physical view. You can edit properties on occurrences, but you cannot edit an occurrence's physical appearance (such as the shape of a hierarchical block). *See also* instance.

occurrence property A property that is attached to an occurrence, as opposed to a property that is attached to an instance or added to a part in a library. You can edit occurrence properties in physical view. Occurrence properties override instance properties, but are not reflected on the instance.

off-page connector An object that conducts signals between schematic pages within a schematic. *See also* flat design.

P–Q

package A physical part that contains more than one logical part. For example, a 2N3905 transistor, a fuse, and a 74LS00 are packages. Each part in a package has a unique part reference comprised of a prefix common to all the parts in the package, and a letter unique to each part. For example, a 74LS00 whose part reference prefix is U15 would have four parts with part references of U15A, U15B, U15C, and U15D. *See also* homogeneous part, heterogeneous part.

pan To change the portion of the schematic page or part being viewed by dragging objects from one location to another. As you drag the object, the schematic page or part *pans* across the active window.

parent A schematic that contains a hierarchical block that references another schematic (called a *child*). *See also* child, hierarchical block.

part alias A duplicate copy of a part using a different name in a library. A part alias uses the same graphics, attached schematics, and properties as the original, with the exception of the part value.

part editor The editor used to create and edit parts and symbols.

part instance An instance of a part. *See also* instance.

part primitive *See* primitive.

PCB The abbreviation for Printed Circuit Board.

physical view A view that displays the "unfolded" view of a design, and thus part references, pin numbers, or properties on *occurrences* of part instances. Since changes made to part references and pin numbers on occurrences of part instances override the values on the original part instance, this view gives you an opportunity to customize specific part instance occurrences without affecting the original part instance. To display the physical view of a design, use the Physical command on the View menu in the design manager. *See also* occurrence, instance, logical view.

PLD The abbreviation for programmable logic device. *See* programmable logic device.

port *See* hierarchical port.

primitive A part or hierarchical block with no underlying hierarchy.

property A characteristic of an object that can be edited. A property consists of a name and a value. Examples of property names are part value and color. Their respective property values can be something such as capacitor and red.

programmable logic device A type of integrated circuit whose behavior can be determined by programming it. Abbreviated *PLD.*

R

RAM The acronym for random access memory. *See also* random access memory.

random access memory The memory that can be used by applications to perform necessary tasks while the computer is on. When you turn the computer off, all information in random access memory is lost.

root schematic The schematic at the top of a flat or hierarchical design. The root schematic contains a backslash in its icon in the design manager. A design has only one root schematic.

S

scalar A pin width that carries only one signal, as opposed to a bus pin that can carry multiple signals.

schematic A graphical representation of a circuit using a standard set of electronics symbols. In Capture, a collection of all schematic pages at the same level of hierarchy in a design. In the design manager, a schematic behaves like a container or DOS directory. *See also* flat design, hierarchical design, schematic page, root schematic.

schematic page The sheets of paper on which schematics are drawn. Schematic pages appear in a window, called the *schematic page editor*, in which you can place parts and draw wires.

schematic page editor The editor used to create and edit schematic pages.

session frame The Capture application window in which the various components of Capture—such as the session log, design manager, schematic page editor, and part editor—run.

session log A window that displays text messages generated by Capture, such as errors and informational messages. The session log starts empty with each new Capture session, but you can save its contents to a text file.

simple hierarchy A design in which there is a one-to-one correspondence between hierarchical blocks (or parts with attached schematics) and the schematic pages they reference. Each hierarchical block (or part with attached schematic) represents a unique schematic page. *See also* hierarchical design, complex hierarchy.

spreadsheet editor A window used to edit the properties of multiple objects at once.

T

tabbed dialog box A dialog box that has different views that can be displayed by clicking on tabs at the top of the dialog box.

U–Y

user-defined property A property you add to an object. Unlike inherent properties, user-defined properties can be removed. *See also* inherent property.

Z

zoom To change the view of a window, making the objects appear larger or smaller. When you zoom out, the objects are smaller and you see more of the schematic page or part. When you zoom in, the objects are larger, but you only see a small portion of the schematic page or part.

zoom factor The amount by which the zoom scale is multiplied or divided when you choose Zoom In or Zoom Out on the schematic page editor's View menu. The Zoom factor is normally 2, but you can change it using the Preferences command on the Options menu. For example, a zoom scale of two makes the image of the screen twice as large when you zoom in and half as large when you zoom out.

zoom scale The relative size of the image on the screen, as a percentage of the normal size. For example, a zoom scale of 250% means the image on the screen is two and one-half times as large as normal.

Index

A

Arc command 14, 129
arcs
  drawing 14, 93
  resizing 93
area selection 16
arrow keys xi
attached files 68, 74, 127
attached schematics 25, 26, 68, 122, 127, 128
  descending 122, 128
  hierarchical blocks 74
  homogeneous package 122, 128
  in complex hierarchical designs 24
  in simple hierarchical designs 24

B

backslash, in root schematic icon 6, 24
bidirectional, pin type 132
Bill of Materials
  command 62, 143, 144, 171-173
  dialog box 172
  include file format 173
  toolbar button 12
bitmaps
  placing 96
  plotting 114
  resizing 96
Bookmark command 107
bookmarks
  jumping to 108
  placing 107
  properties, editing 19
  renaming 108
border visibility
  defining for individual schematic pages 58
  defining for new designs 53
Browse command 8
browse pane 8
  design manager 6
  sort buttons 8
browsing a design or library 8
Bus command 13, 86
bus entries
  placing 14, 87
Bus Entry command 14, 87
buses
  drawing 13
  editing 84
  names 84, 86
  placing 84
  properties, editing 19
  user-defined properties 18, 20

C

CAPSYM.OLB 26, 49
CAPTURE.INI 39, 66
character formatting 101
circles
  drawing 14, 92
  resizing 92
Close command 35
colors
  defining for graphics in the schematic page editor 45
  defining preferences 41
complex hierarchy 24
  logical view 24
  physical view 24
configuration
Control key xi
Convert command 127, 139
convert view of a part 127, 139

Copy command *95, 96, 97, 100*
    toolbar button *11*
Create Netlist
    command *62, 122, 128, 143, 144, 165-169*
    dialog box *166*
    format files *167*
    toolbar button *12*
Cross Reference
    command *144, 145, 174*
    dialog box *174*
    toolbar button *12*
custom libraries *117*
custom parts *124*
Cut command *95, 100*
    toolbar button *11*

D

deleting an object *96*
DeMorgan equivalent *127*
Descend Hierarchy command *127*
descending into attached schematics *122, 128*
deselection *17*
design *31*
design cache *6, 120-121*
Design command *31, 32*
design manager *5*
    browse pane *6, 8*
    design structure pane *6, 117*
    font *45*
    libraries *117*
    toolbar button *12*
Design Properties *56-57*
    command *39*
    dialog box, Fonts tab *56*
    dialog box, Hierarchy tab *56, 122*
    dialog box, Miscellaneous tab *57*
    dialog box, SDT Compatibility tab *56*
Design Rules Check
    command *144, 153, 160*
    dialog box *155*
        ERC Matrix tab *157*
    sample report *158*
design rules violations, checking for *144, 153, 160*
design structure *6, 23-29*
    complex hierarchical *24*
    flat *23*
    hierarchical *24*
    simple hierarchical *24*
design structure pane, design manager *6*
Design Template *47-55*
    command *39*
    dialog box, Fonts tab *48*
    dialog box, Grid Reference tab *53*
    dialog box, Hierarchy tab *54*
    dialog box, Page Size tab *51*
    dialog box, SDT Compatibility tab *55*
    dialog box, Title Block tab *49*
Design Template dialog box, Hierarchy tab *122*
designs
    multiple *32*
    opening *31*
    saving *34*
displaying invisible power pins *135*
document *4*
drag, maximum number of objects displayed *44*
drawing *89-95*
    adding fill *95*
    arcs *14, 93*
    buses *13*
    circles *92*
    ellipses *14, 92*
    hierarchical blocks *14*
    lines *14, 90*
    polygons *94*
    polylines *14, 94*
    rectangles *14, 91*
    squares *91*
    wires *13*
DRC markers *144, 153, 154, 155*
    properties, editing *19*

E

Edit Part dialog box *67*
editing *18*
    ground symbols *69-72*
    hierarchical blocks *73-75*
    hierarchical ports *76-79*
    off-page connectors *80-83*
    parts *62-68*
    power symbols *69-72*
    properties *18*
    spreadsheet editor *19*
    user-defined properties *18, 20*
Ellipse command *14, 129*
ellipses
    drawing *14, 92*
    resizing *92*

Escape key *xi*
Exit command *35*
Export Properties
command *144, 145, 175-176*
dialog box *176*
properties file format *176, 177*
exporting properties to a file *144, 175-176*
exporting text *100*
Extract PLD
command *143, 144, 145, 180-181*
dialog box *181*
sample reports *182*

F

file folder icon *6*
file formats
include file, Bill of Materials command *173*
netlist format files *167*
netlist format files, custom *167*
properties file, Export Properties command *176, 177*
swap file, Gate and Pin Swap command *162*
update file, Update Properties command *152*
fill style *89*
defining for graphics *45*
fill, adding to a graphic object *95*
Find command *8*
finding
parts in a design *109*
parts on a schematic page *109*
flat designs *23*
off-page connectors *23*
fonts
changing placed text *101*
defining for objects in existing designs *56*
defining for objects in new designs *48*
defining for the design manager and session log *45*
screen *101*
format files, netlist *167*

G

Gate and Pin Swap
command *143, 144, 145, 161, 163*
dialog box *162*
swap file format *162*
toolbar button *12*
globals *57, 68, 132, 135*
Go To
command *106*
dialog box
Bookmark tab *108*
Grid Reference tab *107*
Location tab *106*
graphics
adding to a part *129*
Grid command *108*
grid display
dots or lines *42*
setting visibility *42*
snap-to-grid *42*
grid references
defining for individual schematic pages *58*
defining for new designs *53*
displaying or hiding *108*
Grid References command *108*
grid spacing, defining *51*
Ground command *14, 70*
ground pins *135*
ground symbols, placing *14, 69-72*
Group command *17*
grouping *17*

H

help, online *15*
heterogeneous package *119, 122, 123, 124, 126*
Hierarchical Block command *14, 74*
hierarchical blocks *25*
complex hierarchical designs *24*
drawing *14*
hierarchical designs *24*
nets between schematics and schematic pages *25*
placing *73-75*
simple hierarchical designs *24*
user-defined properties *18, 20*
hierarchical designs *24*
complex *24*
logical view *24*
physical view *24*
hierarchical blocks *24*
simple *24*
Hierarchical Port command *14, 76, 77*
hierarchical ports *26*
nets between schematics and schematic pages *25*
off-grid *76*

hierarchical ports (*continued*)
placing *14, 76-79*
placing inside a hierarchical block *76*
placing outside a hierarchical block *77*
properties, editing *19*
hierarchy *24, 122, 128*
complex *24*
defining default primitive behavior for existing designs *56*
defining default primitive behavior for new designs *54*
placing hierarchical blocks *73*
simple *24*
homogeneous package *119, 123, 124, 126*
attaching schematics *122*

I

icons
file folder *6*
page *6*
IEEE Symbol command *15, 129*
IEEE symbols
adding to a part *129*
placing *15*
Import Properties
command *144, 145, 179*
dialog box *179*
importing properties from a file *144, 179*
importing text *100*
include file format *173*
input, pin type *132*
instances *7, 9*

J

jumping
to a bookmark *108*
to a grid reference *107*
to a marked location *107*
to a new location *106*
junctions *84*

K

keyboard keys *xi*

L

libraries *117-118*
copying parts to a different library *118*
custom *117*
design manager, using *117*
moving parts to a different library *118*
opening *31*
Line command *14, 129*
line style *89*
defining for graphics *45*
line width, defining for graphics *45*
lines
drawing *14, 90*
resizing *90*
Logical command *7, 24*
logical view *7, 9, 120, 145, 146*
complex hierarchy *24*

M

Mirror command *95*
moving *21*
multiple objects, selecting *16*
multiple pins, placing *15*
multiple-part packages *119*
creating *124*

N

Net Alias command *14*
net aliases *84, 86*
editing *85*
placing *14*
net names *84*
netlist
creating *144, 165-169*
custom format files *167*
format files *167*
PLD 386+ *169*
netname resolution *168*
nets
hierarchical blocks, between schematics and schematic pages *25*
hierarchical ports, between schematics and schematic pages *25*
off-page connectors, between schematic pages in a single schematic *25*
properties, editing *19*
user-defined properties *18, 20*

New command *31*
  toolbar button *11*
New Part command *124*
New Part Properties dialog box *126*
Next Part command *124*, *138*
No Connect command *14*
no connect symbol *84*
no connect symbols
  placing *14*
non-orthogonal wires and buses *13*
nonprimitive parts *62*, *122*, *128*

O

occurrences *7*, *9*
Off-Page Connector command *14*, *80*
off-page connectors *26*
  in flat designs *23*
    example *27*
  nets between schematic pages in a single schematic *25*
  placing *14*, *80-83*
  properties, editing *19*
online
  help *15*
  tutorial *15*
open collector, pin type *132*
Open command *32*
  toolbar button *11*
open emitter, pin type *132*
output pin type *132*

P

Package command *124*, *138*
package view, part editor *10*
packages *62*, *119*
  heterogeneous *119*, *123*, *124*, *126*
  homogeneous *119*, *123*, *124*, *126*
    attaching schematics *122*
  user-defined properties *18*, *20*
  viewing *138*
page icon *6*
page size
  defining for individual schematic pages *58*
  defining for new designs *51*
pan settings, defining *43*
Part command *13*, *63*
part editor *10*
  tool palette *89*
part instances *7*, *119*, *120-121*
  editing *137*
  properties, editing *19*
  user-defined properties *18*, *20*
  viewing *9*
part occurrences *20*
part view, part editor *10*
parts *119*
  aliases *126*
  attached files *127*
  attached schematics *24*, *25*, *26*, *127*
  convert view *127*, *139*
  copying to a different library *118*
  creating *124*
  editing
    in a library *136*
    on a schematic page *137*
  finding *109*
  graphics, drawing *129*
  heterogeneous package *119*, *123*, *124*, *126*
  homogeneous package *119*, *123*, *124*, *126*
    attaching schematics *122*
  IEEE symbols, placing *129*
  instances *7*, *9*, *19*, *119*, *120-121*
  moving to a different library *118*
  multiple-part packages *119*
  numbering *126*
  occurrences *7*, *119*
  packages *119*
  part body *124*
  part body border *124*
  part reference *126*
  PCB footprint *126*
  pins, placing *130*
  placing *13*, *62-68*
  text, placing *129*
  user-defined properties *18*, *20*
passive, pin type *132*
Paste command *96*
  toolbar button *11*
PCB footprint property *68*
Physical command *7*, *24*
physical view *7*, *9*, *120*, *145*, *146*
  complex hierarchy *24*
Picture command *96*, *129*
Pin array command *15*, *133*
pin arrays, placing *15*

Pin command *15, 130*
pin-to-pin spacing, defining for new designs *51*
pins
  connecting to wires *84*
  ground *135*
  invisibile, displaying *135*
  name *131*
  number *131*
  placing *15*
  placing multiple *133*
  placing on a part *130*
  power *135*
  shape *131*
  shared *123, 135*
  types *131, 132*
  user-defined properties *18, 20*
  visibility *131, 135*
  width *131*
Place Ground dialog box *71*
Place Hierarchical Port dialog box
  inside a hierarchical block *77*
  outside a hierarchical block *78*
Place Off-Page Connector dialog box *82*
Place Part dialog box *65*
Place Pin Array dialog box *134*
Place Pin dialog box *131*
Place Power dialog box *71*
placing
  bitmaps *96*
  bus entries *14, 87*
  ground symbols *14, 69-72*
  hierarchical blocks *14, 73-75*
  hierarchical ports *14, 76-79*
    inside a hierarchical block *76*
    outside a hierarchical block *77*
  IEEE symbols *15*
  net aliases *14*
  no connect symbols *14*
  off-page connectors *14, 80-83*
  parts *13, 62-68*
  pin arrays *15*
  pins *15*
  power symbols *14, 69-72*
  text *14, 97*
PLD 386+
  creating a netlist *169*
  creating a schematic *169*
  extracting source code *144, 180-181*
plotter pen colors *114*
plotting *111*
  part or package *112*
  schematic pages *112*
  schematics *111*
polygons
  drawing *94*
  resizing *94*
Polyline command *14, 129*
polylines
  drawing *14, 94*
  resizing *94*
Power command *14, 69*
power pins *135*
  visibility *68*
power symbols, placing *14, 69-72*
power, pin type *132*
Preferences
  command *39*
  defining *40-46*
  dialog box, Colors tab *41*
  dialog box, Grid Display tab *42*
  dialog box, Miscellaneous tab *45*
  dialog box, Pan and Zoom tab *43, 104*
  dialog box, Select tab *16, 44*
previewing print output *113*
Previous Part command *124, 138*
primitive parts *62, 122*
Primitive property *68, 122, 128*
  Default option *122, 128*
  on hierarchical blocks *74*
Print command *111, 112, 113*
  toolbar button *11*
Print Preview command *113*
Print Setup command *111*
printer, setting up *111*
printing *111*
  part or package *112*
  previewing *113*
  schematic pages *112*
  schematics *111*
properties
  editing *18, 143*
  editing a property file *177*
  exporting to a file *144, 175-176*
  importing from a file *144, 179*
  spreadsheet editor *143*
  user-defined *18, 20*
Properties command *18, 19, 20*
properties file format *176, 177*

R

Rectangle command *14*, *129*
rectangles
drawing *14*, *91*
resizing *91*
Redo command *22*
toolbar button *11*
redoing actions *22*
Repeat command *22*
repeating actions *22*
Replace Cache command *120*, *121*
replacing a part instance in the design cache *121*
reports *171-174*
bill of materials *144*, *171-173*
cross reference *144*, *174*
extract PLD *182*
resizing *21*
root schematic *6*, *24*
Rotate command *95*

S

Save All command *35*
Save command *34*
toolbar button *11*
schematic page editor *9*
logical view *9*
part instances, viewing *9*
part occurrences, viewing *9*
physical view *9*
tool palette *89*
Schematic Page Properties *58-59*
command *39*
dialog box, Grid Reference tab *58*
dialog box, Miscellaneous tab *59*
dialog box, Page Size tab *58*
schematic page size
defining for individual schematic pages *58*
defining for new designs *51*
schematics *31*
attaching to a part *128*
copying *6*
dragging *6*
extracting PLD source code *180*
moving *6*
screen fonts *101*
SDT 386+ and Release IV
converting designs and libraries to Capture *15*, *31*
defining compatibility
of part fields for existing designs *56*
of part fields for new designs *55*
equivalent Capture objects, terms, and commands *15*
saving Capture designs *34*
Select All command *17*
selecting *16-17*
all objects *17*
area *16*
deselecting *17*
multiple objects *16*
one object *16*
selection border, intersect or enclosed *44*
session frame *4*
session log *10*
font *45*
shared pins *123*, *135*
simple hierarchy *24*
example *27*
snap-to-grid, defining *42*
spreadsheet editor *19*, *143*
squares
drawing *14*, *91*
resizing *91*
swap file format *162*
swapping gates or pins *144*, *161*, *163*

T

text
adding to a part *129*
bounding box *98*
character formatting *101*
deleting *98*
editing *98*
exporting *100*
finding *99*
importing *100*
moving *97*
placing *14*, *97*
replacing *99*
rotating *98*
typing *xii*
Text command *14*, *97*, *101*, *129*
three-state, pin type *132*

title blocks *117*
  setting up for new designs *49*
  visibility
    defining for individual schematic pages *58*
    defining for new designs *53*
tool palette
  part editor *13, 89*
  schematic page editor *13, 89*
  visibility *15*
tool palette visibility *44*
toolbar buttons *11, 12*
Toolbar command *12*
tutorial, online *15*

U

Undo command *22*
  toolbar button *11*
undoing actions *22*
Ungroup command *17*
unit of measure
  defining for individual schematic pages *58*
  defining for new designs *51*
Update Cache command *120, 121*
Update Part References
  command *144, 145, 147-149*
  dialog box *149*
  toolbar button *12*
Update Properties
  command *144, 150-152*
  dialog box *151*
  update file format *152*
updating
  part references *144, 147-149*
  properties *144, 150-152*
  selected parts in the design cache *121*
User Properties button *20*
user-defined properties *18, 20*

V

view
  physical and logical *7, 9, 120, 145, 146*
    complex hierarchy *24*
viewing
  centering *105*
  entire page or part *105*
  selected area *105*
visibility
  power pins *57, 68*
  tool palette *15, 44*
  toolbar *12*

W

window
  active *4*
  design manager *5*
  part editor *10*
  schematic page editor *9*
  session frame *4*
  session log *10*
Wire command *13, 85*
wires
  drawing *13*
  editing *84*
  placing *84*
  properties, editing *19*

Z

Zoom All command *105*
  toolbar button *12*
Zoom Area command *105*
  toolbar button *12*
zoom factor *104*
  defining *43*
Zoom In command *103*
  toolbar button *12*
Zoom Out command *103, 104*
  toolbar button *12*
zoom scale *104*
zoom settings, defining *43*
zooming *103*