Orcad® Component Information System

User's Guide

Contents

Chapter 3   Working with database parts   57

Chapter 4   Finalizing and documenting designs   97

Glossary   115

Index   125

Contents

Tables

Before you begin

Welcome

Orcad family products offer a total solution for your core design tasks: schematic- and VHDL-based design entry; FPGA and CPLD design synthesis; digital, analog, and mixed-signal simulation; and printed circuit board layout. What's more, Orcad family products are a suite of applications built around an engineer's design flow—not just a collection of independently developed point tools. Orcad Component Information System (CIS) is just one element in our total solution design flow.

Orcad CIS is a part management system that is available as an option for use with Orcad Capture. Orcad CIS helps you manage part properties (including part information required at each step in the printed circuit board design process, from implementation through manufacturing) within your schematic designs.

How to use this guide

This guide is designed so you can quickly find the information you need to use Orcad Component Information System (CIS). To help you learn and use Orcad CIS efficiently, this manual is separated into the following sections:

Symbols and conventions

Our printed documentation uses a few special symbols and conventions.

Notation	Examples	Description
Ctrl + R	Press Ctrl + R	Means to hold down the Ctrl key while pressing R.
Alt, F, O	From the File menu, choose Open (Alt, F, O)	Means that you have two options. You can use the mouse to choose the Open command from the File menu, or you can press each of the keys in parentheses in order: first Alt, then F, then O.
Monospace font	In the Part Name text box, type PARAM.	Text that you type is shown in monospace font. In the example, you type the characters P, A, R, A, and M.
UPPERCASE	In Capture, open CLIPPERA.DSN.	Path and filenames are shown in uppercase. In the example, you open the design file named CLIPPERA.DSN.
Italics	In Capture, save *design_name*.DSN.	Information that you are to provide is shown in italics. In the example, you save the design with a name of your choice, but it must have an extension of .DSN.

Related documentation

In addition to this guide, you can find technical product information in the online help, the online interactive tutorial, online books, and our technical web site, as well as in other books. The table below describes the types of technical documentation provided with Orcad Component Information System (CIS).

This documentation component . . .	Provides this . . .
This guide— *Orcad Component Information Sytem User's Guide*	A comprehensive guide for understanding and using the features available in Orcad CIS.
Online help	Comprehensive information for understanding and using the features available in Orcad CIS.
	You can access help from the Help menu in Orcad CIS, by choosing the Help button in a dialog box, or by pressing F1. Topics include:
	• Explanations and instructions for common tasks.
	• Descriptions of menu commands, dialog boxes, tools on the toolbar and tool palettes, and the status bar.
	• Error messages and glossary terms.
	• Reference information.
	• Product support information.
	You can get context-sensitive help for a error message by placing your cursor in the error message line in the session log and pressing F1.
Online interactive tutorial	A series of self-paced interactive lessons. You can practice what you've learned by going through the tutorial's specially designed exercises that interact directly with Orcad CIS. You can start the tutorial by choosing Learning Orcad CIS from the Help menu.
Online *Orcad Component Information Sytem User's Guide*	An online, searchable version of this guide, available when choosing Online Manuals from the Orcad family program group (on the Start menu).
Online Orcad CIS quick reference	Concise descriptions of the commands, shortcuts, and tools available in Orcad CIS, available when choosing Online Manuals from the Orcad family program group (on the Start menu).

This documentation component . . .	Provides this . . .
Orcad family customer support at www.orcad.com/technical/technical.asp	An Internet-based support service available to customers with current support options. A few of the technical solutions within the customer support area are:
	• The Knowledge Base, which is a searchable database containing thousands of articles on topics ranging from schematic design entry and VHDL-based PLD design to PCB layout methodologies. It also contains answers to frequently asked questions.
	• The Knowledge Exchange, which enables you to share information and ideas with other users and with our technical experts in a real-time online forum. You can submit issues or questions for open discussion, search the Knowledge Exchange for information, or send email to another participant for one-on-one communication. A list of new postings will appear each time you visit the Knowledge Exchange, providing you with a quick update of what's been discussed since your last visit.
	• The Technical Library, which contains online customer support information that you can search through by category or product. You can find product manuals, product literature, technical notes, articles, samples, books, and other technical information. Additionally, technical information can be obtained through SourceLink, which is an online customer support information service for users of Cadence software other than Capture, Component Information System (CIS), Express, Layout, or PSpice.
	• The Support Connection, which allows you to choose to either view and update existing incidents, or create new incidents. The information is delivered directly to us via our internal database. This service is only available to customers with current maintenance or Extended Support Options (ESOs) in the United States and Canada.
	• The Live Connection, which enables you to open access to your computer to a Customer Support person, who can then view your actions on your computer monitor as you demonstrate the problem you're having. Live Connection's two-way transmission can also let you view the actions on the Customer Support person's computer monitor, as he or she demonstrates a method or procedure to help you solve your problem. To participate in Live Connection, you need to contact a Customer Support person, in order to obtain a support number to grant you access to the Live Connection site, and to set up a time to "meet online" using Live Connection.

What is the Orcad Component Information System?

Overview

Orcad Component Information System (CIS) is a part management system that is available as an option for use with Orcad Capture. Orcad CIS helps you manage part properties (including part information required at each step in the printed circuit board design process, from implementation through manufacturing) within your schematic designs.

CIS provides access to local (preferred parts database) and remote part databases that contain all relevant information for the parts used in your designs. This information may include company part numbers, part descriptions, PCB layout footprints, technical parameters (such as speeds, tolerances, and ratings), and purchasing information.

With CIS, you can select a part from your preferred parts database or a remote database and place it directly in your schematic design. You can configure CIS to transfer any or all properties associated with that part to the schematic

when you place the part. CIS maintains a link to the engineering database part so that you can retrieve other part properties at any time. Linking placed parts to your preferred parts database gives you access to complete part information during the schematic design process.

If you need a part for your design that is not yet in the parts database, you can create the part in the design and add the part to your part database immediately or at a later time. You can also link a non-database part you've created before to a database part at any time.

Local Preferred Part Data

External New Part Data

UNC, URLs, path to browsable files

Orcad Layout Libraries

Orcad Capture Libraries

Orcad PSpice Libraries

Combined or separate databases

Corporate Database (MRP/ERP/PDM)

Upload BOM to corporate system

Preferred Parts Database

ODBC

Orcad Capture

Orcad CIS

ICA

Internet

Activeparts Database

Orcad Layout Libraries

Orcad Capture Libraries

Supplier Pricing & Availa-bility

URLs to Mfr. Spec Sheets

Reports

Tab delimited ASCII

Comma delimited ASCII

MS Access database format

Seagate Crystal Reports

To File

To Exchange Folder

To Lotus Notes Database

To MS Mail

CIS product history

Orcad CIS Release 9 is the latest version of the Orcad Enterprise CIS product offering. With the release of version 7.2 in April 1998, the Orcad Enterprise CIS family was the evolution and final merging of two popular Orcad CIS product lines—Enterprise Edition and EDA-Bridge.

Enterprise Edition evolved from the DDL product originally developed by an Orcad Industry Partner called Q-Point. Q-Point was acquired by Orcad in June 1997. Orcad Enterprise Edition was packaged and shipped under the product names Orcad Capture EE and Orcad Express EE. Version 7.1 was the last release of both these products lines. They were replaced by Orcad Enterprise CIS v7.2.

EDA-Bridge was acquired as part of The T.E.A.M. Corporation merger in April 1997. The product was sold as a stand-alone option and was available until the release of Orcad Capture v7.1 and Orcad Express v7.1. Subsequently, EDA-bridge was replaced by Orcad Enterprise CIS version 7.2.

Enterprise CIS v7.2 merged features of EDA-Bridge and Enterprise Edition, and can be used with both Orcad Capture v7.2 and Orcad Express v7.2.

With the Release 9.0, the name Orcad Enterprise CIS has been shortened to Orcad CIS. Orcad Capture is available with CIS as a product called Orcad Capture CIS. Orcad Capture CIS has additional options that can be purchased, including digital simulation and programmable logic functionality from the Orcad Express family and analog simulation from the Orcad PSpice family.

As part of an effort by Cadence to simplify packaging and licensing of the Design Desktop for Windows product line, the architecture of Express has changed with Release 9.0 to be a *plug-in* for Capture and Capture CIS.

The internet component assistant (ICA)

The CIS internet component assistant (ICA) allows you to access new components over the Internet by providing a link to the part database at the activeparts web site. After finding an activepart, you can download the part's component information to create temporary parts in your local preferred part database (PPD). You can immediately place the temporary part in your schematic designs. Temporary parts can then be elevated through your enterprise approval process, assigned corporate part numbers, and accepted as preferred parts within the database.

CIS in the PCB design process

To use CIS in your printed circuit board design flow, you or your system administrator must perform the following setup tasks.

1 Create the preferred part database. This involves creating and entering the part data appropriate for the database.

2 Set up the Open Database Connectivity (ODBC) data source to point to the preferred part database.

3 Configure the part management system.

Each of these tasks is described in **Chapter 2,** Setting up Orcad CIS.

Once the setup is complete, you can use CIS in Capture to:

• Select parts with associated properties from the database and place them on schematic pages in your design.

• Check the status of placed parts and update placed part properties to reflect changes to the database.

• Generate bills of materials and other reports using both design and part database information.

Setup

Create the part database	Set the ODBC data source	Configure CIS

Place database parts	Update part status	Create reports

Design
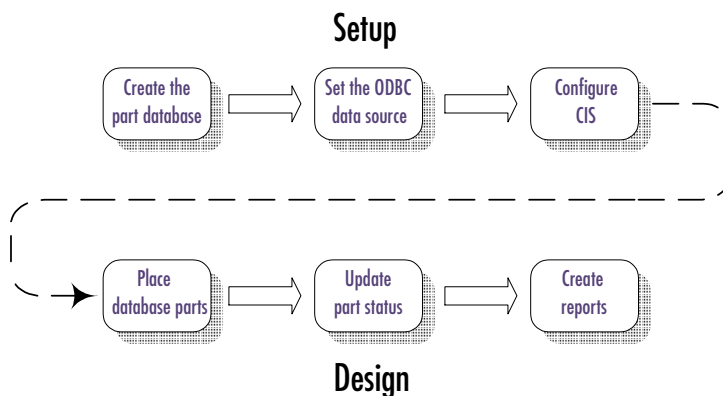
The CIS work environment

For descriptions of how to use the CIS Windows interface, see Using the CIS interface on page 58.

The CIS work environment includes two specialized windows that allow you to manage your local part database and explore sources of part data—the CIS explorer and the part manager. All the CIS functionality for these windows is integrated into Capture menus.

The CIS explorer window
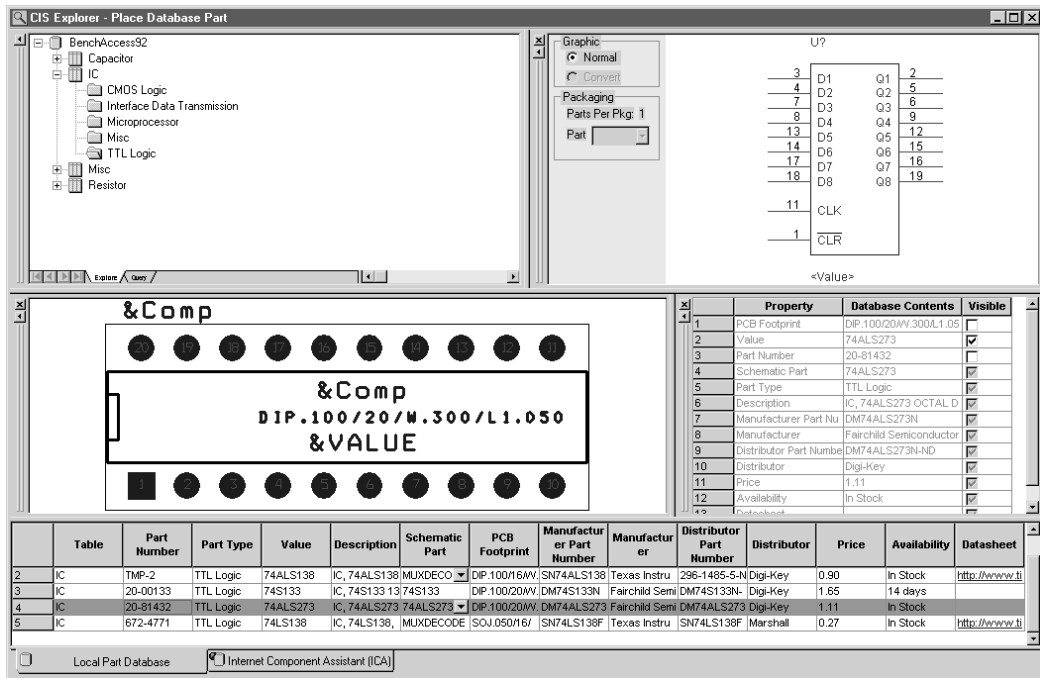
The CIS explorer window allows you to search for and retrieve a variety of part information. The main window contains two tabbed windows—the local part database and the internet component assistant (ICA). The following table lists the windows available for each tabbed window.

This window...	Is available in this tabbed window...
Footprint	Both
Part	Both
Visibility	Both
Explorer	Local part database
Database parts	Local part database
ICA home page	Internet component assistant (ICA)

The CIS explorer window displays in one of the following three modes, depending on which of the following CIS features you are using:

- Placing a database part
- Linking a database part to a placed part
- Viewing the database properties of a placed part

The mode is displayed as part of the window title bar.

The footprint window

Note   *The footprint window is only available if you chose to view Orcad Layout-generated footprint libraries during installation.*

The footprint window displays the Layout-generated PCB footprint associated with the currently selected database part.
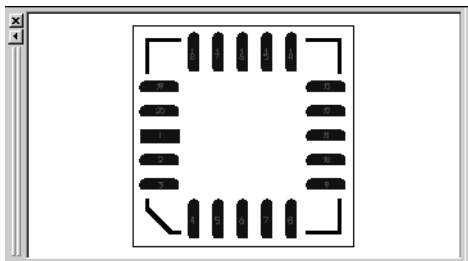
The part window

The part window displays the Capture library part associated with the currently selected database part. If the part has a convert (such as a DeMorgan equivalent), you can select it. For a multiple-part package, you can select the specific part in the package.
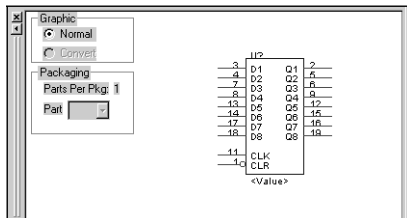
The visibility window

You can also use the visibility window to display a compact summary of the part properties and their contents for the part you have selected in the database parts window. You can see more of the properties and contents in this view because the visibility window displays them in rows rather than columns.

The visibility window displays the default settings for which part properties are visible on your schematic page. You can use the visibility window to override these default settings. You can also set custom visibility settings for the current part selection.

The explorer window

The explorer window allows you to search for parts using local data from your preferred part database. The window contains two tabbed sections—explore and query. In the explore tab, you can search for parts using a hierarchical tree organized by part type. The query tab allows you to further filter your selection based on parametric or field data.
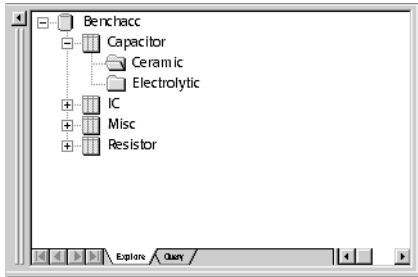
The database parts window

The database parts window displays the results of your part browsing and database queries.

	Table	Manufacturer	Manufacturer Part Number	Part Number	Price	Availability	Distributer Part Number	Part Type	Description	Value	Voltage	Distributer	Schematic Part
1	Capacitor	PHILIPS COMPON	A104K29X7RFV	A104K29X7RFVVWN	0.12	In Stock	1109PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
2	Capacitor	PHILIPS COMPON	A104M15Z5UFV	A104M15Z5UFVVWN	0.04	In Stock	1210PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
3	Capacitor	PHILIPS COMPON	A104Z15Z5UFV	A104Z15Z5UFVVWN	0.04	In Stock	1203PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
4	Capacitor	Panasonic - SCD	ECK-F1E104ZV	ECK-F1E104ZV	0.20	In Stock	P4201-ND	Ceramic	CAP CERAMIC	0.1UF	25V	Digi-Key	bench\SM
5	Capacitor	Panasonic - SCD	ECU-V1H104KB	ECU-V1H104KBW	0.11	In Stock	PCC104BTR-ND	Ceramic	.1 UF/50 V CE	0.1UF	50V	Digi-Key	bench\SM
6	Capacitor	PHILIPS COMPON	A100J15C0GHV	A100J15C0GHVVWA	0.06	In Stock	1011PHTR-ND	Ceramic	CAP 100V 10P	10PF	100V	Digi-Key	bench\SM
7	Capacitor	PHILIPS COMPON	D100D20COGHA	D100D20COGHAAAC	0.07	In Stock	1326PH-ND	Ceramic	CAP 10PF 100	10PF	100V	Digi-Key	bench\SM
8	Capacitor	PHILIPS COMPON	D100D20COGLA	D100D20COGLAAAA	0.14	In Stock	1355PH-ND	Ceramic	CAP 10PF 500	10PF	500V	Digi-Key	bench\SM
9	Capacitor	PHILIPS COMPON	D100D20U2MHA	D100D20U2MHAAAC	0.07	In Stock	1301PH-ND	Ceramic	CAP 10PF 100	10PF	100V	Digi-Key	bench\SM
10	Capacitor	PHILIPS COMPON	D100K20U2ML A	D100K20U2ML A A A	0.14	In Stock	1344PH-ND	Ceramic	CAP 10PF 500	10PF	500V	Digi-Key	bench\SM

The ICA home page

The ICA home page window provides access to the activeparts web site. This site is continuously updated to make sure you have the latest, most complete Internet part information.

When you select the activeparts web site from the ICA home page, CIS will replace the home page with the activeparts search page. Then, you can use the activeparts interface to search its part database. Because this is a live link to the database, you always have access to the most recent part information.

When you find a part that you want, you can place it, along with its component information, directly onto a schematic page. Component information associated with activeparts may include:

When placing parts from the activeparts web site, CIS users have access to Orcad schematic representations of parts and PCB footprints over the Internet.

9

- Part manufacturers and manufacturer part numbers

- Part values

- Orcad schematic parts and PCB footprints

- Supplier pricing and availability information

- URLs of manufacturer specification sheets

The part manager window

The part manager window summarizes the status of all the parts in your design.

The part manager displays information for each part in your design including:

- Schematic page on which the part is placed

- Part reference designator

- Part value

- Part number

- Status of the placed part relative to the part database

- Database table that contains the placed part's associated database part

- Capture source library from which the part was placed

- Capture source package to which the part belongs

Setting up Orcad CIS

Overview

This chapter describes the four tasks necessary to set up Orcad Component Information System (CIS), including:

1   Create a part database. See <u>Creating a part database on page 12</u>.

2   Set up the Open Database Connectivity (ODBC) data source name. See <u>Setting up the ODBC data source on page 23</u>.

3   Create a configuration file to enable CIS to access and use the part database. See <u>Creating a configuration file on page 26</u>.

4   Edit your configuration file to customize how CIS interacts with your part database and your work environment. See <u>Editing a configuration file on page 36</u>.

Also included are procedures for setting the up the CIS work environment, including <u>Setting up the ICA to access external part data on page 53</u> and <u>Creating Crystal Reports templates on page 54</u>.

Creating a part database

The part database is the entity that contains all the relevant information for the parts you use in your designs. You can think of a part database as one or more tables, with each row in a table representing a part and each column representing a part property.

Before you create your part database, you need to choose whether to use a database or spreadsheet application. Then, when you create your part database, you need to carefully setup its structure and organization, including:

You can create a database from an existing Capture design. For more information, see Creating a part database from an existing design on page 2-20.

You can also extract information for your part database from an MRP database. For more information, see Extracting ERP or MRP database part information on page 2-22.

- Part property assignments. See Determining part properties on page 2-14.

- Field formats for properties. See Setting the field format on page 2-20.

- Number of tables used in the database implementation. See Using more than one table on page 2-20.

Using ODBC, CIS interfaces directly with your ODBC compliant database or spreadsheet. This means you can use your preferred application to create and maintain your part database.

Choosing a database format

To take full advantage of the speed and power of CIS, you may want to use a database application rather than a spreadsheet application. As a general rule, you should use a database application for databases with more than 1,000 parts. Database applications offer the following advantages over spreadsheets:

Note   *Cadence developed, wrote, and tested Orcad CIS in the Access 97 and SQL Server 6.5 environment. Access offers high performance queries with databases of several thousand parts. If your database outgrows Access, moving to SQL Server is a seamless process. Using the upsize wizard, Access can easily transfer the database to SQL Server and still serve as an interface for maintaining data.*

- Comprehensive data management features

- Better performance for part searching

- Form-based entry for entering part information

- Safeguard against duplicate part numbers

The disadvantages of spreadsheets and delimited text files include:

Lack of structure. During data entry, spreadsheets allow you to configure every cell in a different format. This makes querying fields (columns) very difficult because CIS expects queried fields to be in a consistent format.

Lack of ANSI SQL compliance. Because true databases are ANSI SQL compliant, their drivers do very little work in relaying SQL commands to connected data sources. In contrast, with non-SQL compliant spreadsheets, the ODBC driver is responsible for translating the SQL commands into functions that the spreadsheet can understand. This results in a dramatic increase in the time your queries can be processed. Query time increases are even more dramatic with text files because the driver must search for and compile a matching list entirely on its own. An optimized, SQL-compliant database format is several times faster than a spreadsheet with more than 100 parts or a text file of over 20 parts.

Lack of replication support. If you have users in more than one location accessing your database, you have either a single, central database or a replicated database. Replication is a method of storing a single database in more than one location. True databases support scheduled synchronization: they maintain data integrity by checking separate database transaction logs and replicating modifications on multiple systems. If you are using a spreadsheet or text program, you must check manually because your format does not support replication.

Structured query language (SQL) is used in querying, updating, and managing relational databases.

You may want to look at the sample part databases provided with CIS. Two samples are provided in the SAMPLES folder. The first part database, BENCH.MDB, is an Access 97 database. The second, BENCH.XLS, is an Excel spreadsheet. As you read this section, refer to these samples to get a better understanding of how to set up your part database.

Example: You can name the Part Number property My Company Part Number.

Example: you may call a property Tolerance in the database and Tol on the placed part.

Note   *Your database can also contain mechanical (non-electrical) parts. However, you must not allow users to add mechanical parts to the database whose schematic representations (symbols) have pins. If mechanical parts with pins are placed in a design, they will invalidate netlists generated from that design.*

Determining part properties

The first step in creating a part database is to determine the properties to include for each part. Typical properties in a part database include part number, part description, tolerance, rating, speed, timing parameters, PCB footprint, manufacturer, and cost. CIS supports an unlimited number of properties, so you can include as much information in your part database as you want.

There are no restrictions on database table property names. Also, the names you use in the database can be different than the property names you assign to the placed parts. Database property types and placed-part property names are defined during database configuration. For more information, see

Note   *Do not use the same property name more than once. For example, if you have two manufacturer columns in your database, call them Manufacturer 1 and Manufacturer 2.*

When you transfer a property, that property is included in the schematic as an attribute of the placed part. Normally, you transfer properties that are required by CIS (such as Part Number and Schematic Part), used in the design process (such as Value, Tolerance, and Rating), or needed for use by other software products (such as PCB Footprint). Properties that aren't transferred can still be included in a bill of materials report.

Parts in your database must include all of the properties in Table 1. Any of the optional properties in Table 2 may also be added. Recommendations on whether to transfer properties to your design are included in both tables. Properties recommended for transfer to the design are either likely to be required for netlisting or are generally needed on printouts of the design.

Table 1 *Required part properties*

Property	Description	Transfer to design?
Part Number	Required to identify the part in association with the CIS Part_Number property type. This property is required by the part manager and the bill of materials report. CIS lets you enter more than one database part with the same number in the database. However, you should use a unique part number for each part and have no duplicate part entries in your database.	Required
Part Type	Identifies the part type. The Part Database Explorer uses this property to define the part database folder hierarchy. Use this property to facilitate part searches. Typical contents are resistor, resistor\fixed, capacitor, capacitor\electrolytic, IC, IC\Memory\SRAM, connector, and so on. The levels of the hierarchy are defined using the backslash (\) character (or any character you define in the configuration). You can define any number of levels in the hierarchy. This property is also used by the Part Reference Associations option. (See Defining part reference associations on page 43.) This property is case-sensitive. Make sure you use uppercase-lowercase conventions consistently when entering values for this property.	No

Table 1   Required part properties (continued)

Property	Description	Transfer to design?
Schematic Part (Symbol)	The part name. This property is required to use the Place Database Part command. Only the part name is necessary if the part is stored in the same directory as the custom schematic part libraries (.OLB files) that you configure in Capture. (See the *Orcad Capture User's Guide* for more information about configuring part libraries.)	Automatic
	However, to make sure that you don't accidentally place an identically named part, you should include the library name, a backslash (\), and then the part name. Examples include:	
	DISCRETE\CAP DISCRETE\DIODE OPAMP\CMP01	
	In addition, you can use the explicit path so CIS can locate an unconfigured library. For example:	
	C:\MYLIB\DISCRETE\CAP	
	Caution *Do not directly reference the schematic part libraries that are supplied with Capture CIS (resource libraries). Because the resource libraries in your installations of Capture CIS are often changed during upgrades to new software versions, the library names and paths in your database can be made invalid. Instead, create your own custom libraries by renaming the resource library files or copying individual parts from the resource libraries to your custom libraries.*	
	You can also assign multiple schematic part names to a single part. Use the same format as above for each name and separate each one with the multi-value delimiter (by default, a comma). For example:	
	DISCRETE\CAPACITOR NON-POL, DISCRETE\CAP NP, DISCRETE\SMALL CAP	
	Tip *The default value of the multi-value delimiter is a comma. However, when setting administrative preferences during database configuration, you can change the character CIS recognizes as the delimiter to a colon, semi-colon, question mark, or vertical bar. For information, see* <u>Setting administrative preferences on page 2-47</u>.	
	Then, when you update your design's part status, CIS can approve and make current a schematic part which has several acceptable names. Also, if a database part has several different valid schematic parts, you will be able to choose	

Table 1   Required part properties (continued)

Property	Description	Transfer to design?
	any one of them when you are placing parts from the database parts window. All the configured schematic parts for the database part will be available from a drop-down list under the Schematic Part property name. CIS locates the Capture library using the following set of prioritized rules: **1** Search the library at the explicit path, if provided. **2** Search the first library listed in CAPTURE.INI that has a matching library filename. **3** Search all directories that contain configured libraries. **4** If no libraries are included specifically in your Capture design, CIS searches the LIBRARY directory in your Capture installation directory.	
Value	The part value. Examples are 1.2K, 10.0uF, and 74ALS374. CIS supports the use of common magnitude identifiers (such as K and uF). The database query uses intelligent unit conversion to interpret common magnitude identifiers in part definitions, since entries in part databases are often inconsistent (for example, the Value for a 2.7K resistor can be 2.7K, 2,700, 2.70K, 2.700K, 2700.0, 0.0027M, etc.). Unit suffixes (such as F for farads or H for henries) are ignored in translation. The magnitude identifiers supported in CIS include:	Required

Identifier	Represents	Magnitude
f	femto	10^{-15}
p	pico	10^{-12}
n	nano	10^{-9}
u	micro	10^{-6}
m	milli	10^{-3}
K	kilo	10^{3}
M	mega	10^{6}
G	giga	10^{9}
T	tera	10^{12}

Table 2   *Optional part properties*

Property	Description	Transfer to design?
Availability	Number of the parts in stock at your company.	Not recommended
Data Sheet	The name of the detailed datasheet for this part. Select the Browsable check box in the configuration for CIS to automatically launch the appropriate browser for this property. Examples include: 74ALS374.PDF, http://www.chipmaker.com/specs/74ALS374, and RES1K1/4W.DOC. You can browse any format you want. CIS uses the application assigned to that file extension in your Windows registry. (Extension assignments are managed in the Windows Explorer. See your MS Windows documentation for more information.) For example, a .DOC entry might cause CIS to launch Microsoft Word, and a URL entry might cause it to launch your default web browser. Note CIS uses the PATH environment variable and the current working directory to find the specified document.	Not recommended
Description	A brief description of the part.	Not recommended
Distributor	Name of part distributor.	Not recommended
Distributor Part Number	Part number used to order from the distributor.	Not recommended
Manufacturer	Name of part manufacturer.	Not recommended
Manufacturer Part Number	Part number used to order from the manufacturer.	Not recommended

Table 2   Optional part properties (continued)

Property	Description	Transfer to design?
PCB Footprint	The PCB footprint name (from the footprint library) assigned to a part.	Recommended
	Note *Unlike Schematic Part property names, you cannot use directory paths for libraries with PCB footprint names. If you use a path with a PCB footprint name, the footprint will not display in the CIS explorer's footprint window.*	
	You can assign multiple PCB footprints to a single part by entering them in the part's footprint field in your part database and separating each footprint name with the multi-value delimiter.	
	Tip *The default value of the multi-value delimiter is a comma. However, when setting administrative preferences during database configuration, you can change the character CIS recognizes as the delimiter to a colon, semi-colon, question mark, or vertical bar. For information, see* Setting administrative preferences on page 2-47.	
	Then, when you update your design's part status, CIS can approve and make current a schematic part which has several acceptable footprint names. Also, if a database part has several different valid PCB footprints, you will be able to choose any one of them when you are placing parts from the database parts window. All the configured PCB footprints for the database part will be available from a drop-down list under the PCB Footprint property name.	
Price	Part price. Use this information when selecting parts to design for cost. Include this property in reports to get a cost roll-up of your design.	Not recommended
Rating	The maximum rating for the part (for example, maximum voltage or power dissipation).	Optional
Tolerance	The percent tolerance specified for the part.	Optional
Voltage	The voltage requirement of the part.	Optional

Setting the field format

CIS can work with database text and number format fields (known as the cell format in spreadsheets). As a general rule, set each database field (that is, each table column) to text format. CIS converts other database field formats, such as float, to text format when properties are transferred to placed parts.

Note   *Use only ANSI SQL-92 compliant data types for your field formats. If you use non-compliant data types, CIS may misinterpret property values.*

Using more than one table

CIS can reference more than one table or worksheet in your part database. For example, you can group your parts into several tables according to type. That is, you can place resistors in one table, ICs in another, capacitors in another, and the remaining parts in a fourth table. When you're viewing database parts with CIS, each table has a unique view. You can arrange the column settings for each table independently, allowing you to view different properties for each.

Creating a part database from an existing design

You can obtain a wizard from the Orcad web site that automates this process. The wizard also automatically derives schematic part and source library information for the placed parts in your Capture schematic. If you use the manual procedure detailed here, you will have to manually reference your placed parts to your libraries.

To download the CIS wizard, visit the Orcad web site at www.orcad.com.

You can also create a part database by extracting part properties from an existing Capture design using the CIS Bill of Materials command.

Tip   *If your designs are created in another schematic capture program that creates bills of materials for part properties, you can still create a part database from an existing design. Follow the instructions supplied with the program to create the appropriate bill of materials file from your design. Then, use the bill of materials file to create a part database based on that design.*

Consider the example in the following illustration. The Standard Bill of Materials dialog box lists a typical set of properties to extract from the design.
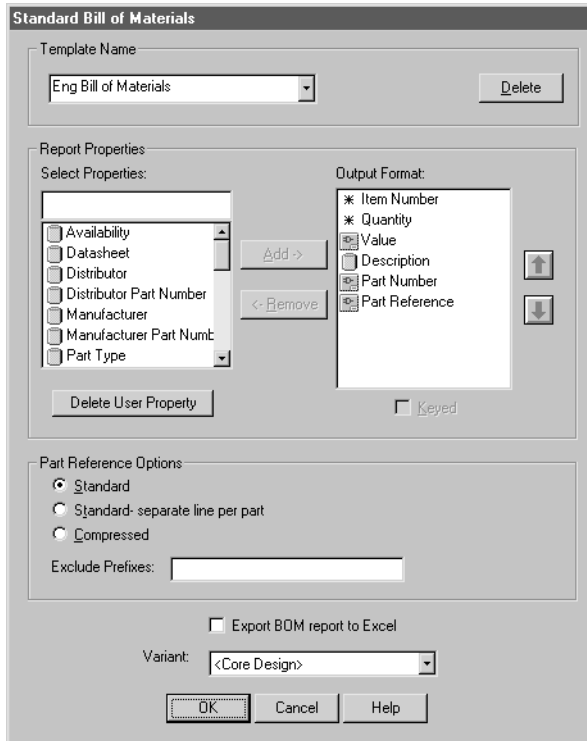
In the Output Format list, CIS lists a number of properties that will be extracted to the bill of materials. You can add to this list by selecting properties from the Select Properties list and clicking the Add button. You can remove properties from the Output Format list by selecting them and clicking the Remove button.

When you click OK in the Standard Bill of Materials dialog box, CIS creates a file with the property information for all the parts in the design. You can directly import this file into your database.

Note *Your database should have a unique part number for each part and no duplicate part entries. Although some database applications will not import more than one occurrence of a keyed property, you may need to eliminate duplicate part entries manually. To do this, sort the database by part number, then delete duplicate entries using your database or spreadsheet program.*

Extracting ERP or MRP database part information

You can also extract information from an existing enterprise or manufacturing resource planning (ERP or MRP) database, to create a part database. Save the ERP or MRP database as an ASCII file, then use that file as your part database file or as a source for a true part database file (for example, an MS Access format file).

Note   *If your ERP database is ODBC compliant, you may not need to extract part information. Instead, you should be able to directly connect your preferred part database (PPD) to the ERP database— you should not have to set up and run regular batch routines to update the data in your PPD. For information about connecting these databases, see the product documentation for both database applications.*

You may need to enhance the information in an ERP or MRP database, since it generally will not include everything required for schematic design. However, an ERP or MRP database usually contains a complete list of your company's parts, part numbers, part descriptions, costs, quantities in stock, lead times, manufacturers, and alternate manufacturers.

Often, the data in an ERP or MRP database exists in a format that does not lend itself to ease of use. For example, the part value may be merged with the description, such as:

```
RES 1K 5% 1/4W
```

This makes searching for parts more difficult. If you can, split this information out in the ERP or MRP database into separate database properties, such as Part Type, Value, Tolerance, and Rating.

You can also contact the Cadence Methodology Services group for assistance in converting your data into a more convenient format.

To take advantage of updated information in an ERP or MRP database, you need to set up a means of extracting data on a regular basis and using it to update your part database. That way, your part database will include information that changes frequently, such as quantity in stock and lead time.

Setting up the ODBC data source

Before you can set the CIS configuration, you must define the open database connectivity (ODBC) data source for your database. CIS interfaces to your part database using a defined ODBC data source name. A data source consists of a database filename and an associated ODBC driver with which to access it. If you are setting up a client-server database, the data source also references the database server. You define the data source name, assign the database file name, and specify the ODBC driver using the 32-bit ODBC Windows control panel.

You must define the data source on each user's system. When you do so, use the same data source name. This allows users to share the same configuration file.

Note   *If your workgroup is sharing a configuration file, make sure to write-protect the file.*

To set the data source for the local database

1   From the Windows Settings menu, choose Control Panel.

Note   *If you are using the Windows 2000 operating system, double-click the Administrative Tools folder to access the ODBC icon.*

2   Double-click on the ODBC icon. Depending on your system configuration, Windows displays the ODBC Data Source Administrator or a Data Sources dialog box.

Note   *If you do not see an icon for 32-bit ODBC, run the Capture CIS installation program to install ODBC.*

If the driver for your application is not present, you must install it. Check the CIS installation disk or contact the database or spreadsheet program supplier.

3   Click the ODBC Drivers tab and make sure the driver appropriate to your database or spreadsheet is installed on your system.
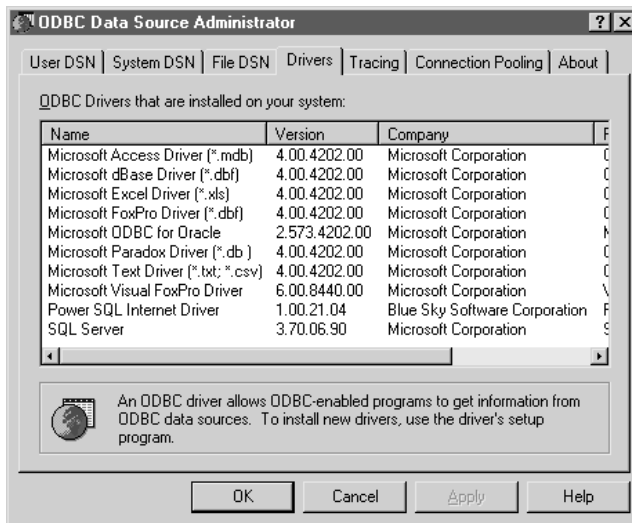
If you are using Windows NT, you should click the System DSN tab to create new data sources. If you do not use the System DSN tab, users with different logins will not be able to use the ODBC source.

4   Click the User DSN tab then choose the Add button. Windows displays the Create New Data Source dialog box.

Setting up the ODBC data sourcesegment>
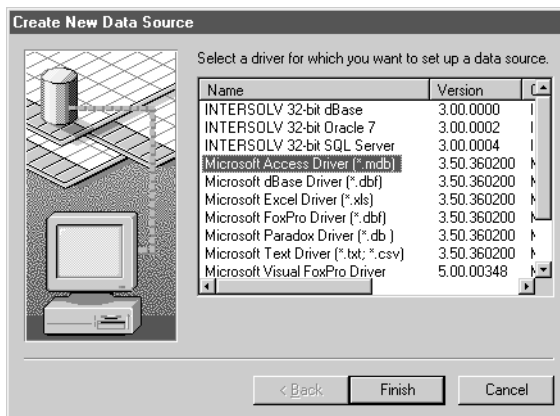
5   Choose the appropriate driver for your program
     (Microsoft Access, in this example), then click the
     Finish button.

```
┌─────────────────────────────────────────────────────────┐
│ ODBC Microsoft Access 97 Setup                      [X]  │
│                                                          │
│ Data Source Name:  [MyDatabase            ]   ┌────────┐ │
│                                               │   OK   │ │
│ Description:       [Access part database  ]   └────────┘ │
│ ┌─Database──────────────────────────────┐    ┌────────┐ │
│ │  Database:                             │    │ Cancel │ │
│ │  ┌──────┐ ┌──────┐ ┌──────┐ ┌────────┐ │    └────────┘ │
│ │  │Select│ │Create│ │Repair│ │Compact.│ │    ┌────────┐ │
│ │  └──────┘ └──────┘ └──────┘ └────────┘ │    │  Help  │ │
│ │                                        │    └────────┘ │
│ │ ┌─System Database──────────────────┐   │    ┌────────┐ │
│ │ │ (•) None                         │   │    │Advanced│ │
│ │ │ ( ) Database:                    │   │    └────────┘ │
│ │ │        ┌────────────────┐        │   │              │
│ │ │        │ System Database│        │   │    ┌────────┐ │
│ │ │        └────────────────┘        │   │    │Options>>│ │
│ │ └──────────────────────────────────┘   │    └────────┘ │
│ │ ┌─Driver───────────────────────────┐   │              │
│ │ │ Page Timeout: [5]  Buffer Size: [512] │              │
│ │ │ [ ] Exclusive   [ ] Read Only    │   │              │
│ │ └──────────────────────────────────┘   │              │
│ └────────────────────────────────────────┘              │
└─────────────────────────────────────────────────────────┘
```
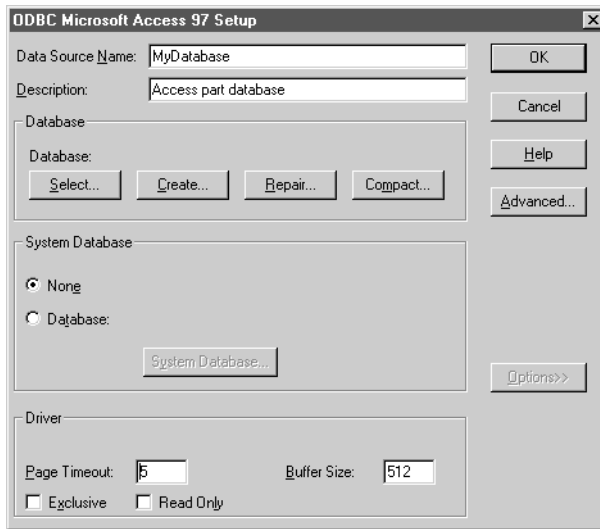
6   Assign a name for the data source. If desired, enter a
     description for the data source name.

Note   *Step 7 may be different depending on the type of program you
        used to create your database.*

7   Under Database, click the Select button, and locate the
     database.

8   Click the Options button, and clear the Read Only
     check box so users can create new parts and add them
     to the database using CIS.

9   Click OK to set the data source.

See Setting administrative
preferences on page 47 for more
information on temporary parts.

Creating a configuration file

Keep the configuration (.DBC) file in a read-only directory that is accessible to all CIS users. You should make the directory read-only to prevent users from inadvertently changing the configuration.

CIS requires a configuration (.DBC) file to make use of your part database. The configuration file:

- Identifies the ODBC data source to use as the part database and specifies the tables to use within that database.

- Identifies the part properties that are transferred to your design when you place or link a database part.

- Sets the visibility for each of the transferred properties.

- Contains the part type associations.

Note   *If you use a CIS v7.2 configuration file with CIS Release 9, the file will be updated to support the new features of this release. After CIS updates the file, though, you cannot use it again with CIS v7.2. If you are using a configuration file from before CIS v7.2, you must recreate the file because the configuration file format was changed for CIS v7.2.*

When creating a configuration file, you should use the database configuration wizard. Completing the wizard guarantees that your part database can be used with most CIS features. For information about using the wizard, see Using the database configuration wizard below.

However, if you need to use the manual method of creating a configuration file, you can reference Creating a configuration file manually on page 2-32.

Using the database configuration wizard

You must set up a new ODBC data source before you can create a new configuration file. If you haven't set up a new ODBC data source for your database, see Setting up the ODBC data source on page 2-23.

You can use the database configuration wizard each time you want to create a new database configuration file. The wizard is designed to make sure that you set at least the minimum table and property configuration that is required for CIS to work with your part database.

However, the wizard does not include all the options available to you, including options to optimize the performance of CIS and customize your CIS work environment. For this reason, when you have finished running the wizard, you should read through the last two parts of this section, <u>Setting other part database options on page 2-30</u> and <u>Setting other configuration options on page 2-32</u>.
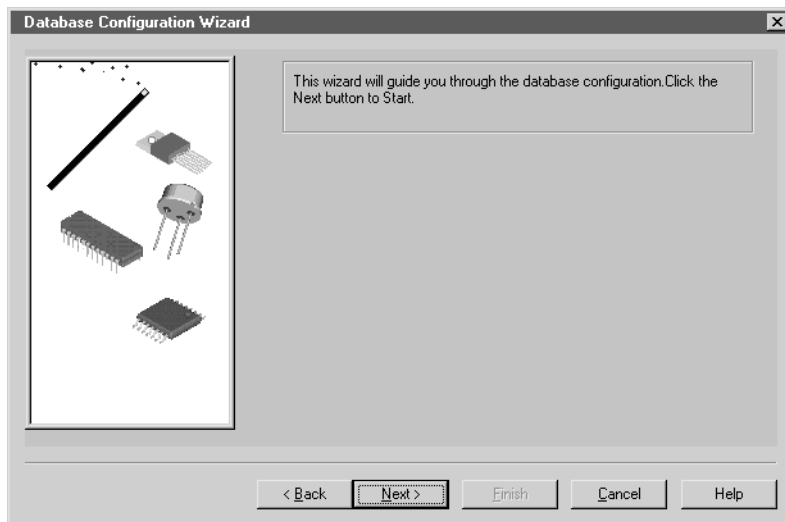
Starting the database configuration wizard

You can start the database configuration wizard anytime you want to create a new database configuration file.

To start the database configuration wizard

1 Open a new or existing schematic design in Capture.

2 From the project manager's Options menu, choose CIS Configuration. CIS displays the CIS Configuration File dialog box.
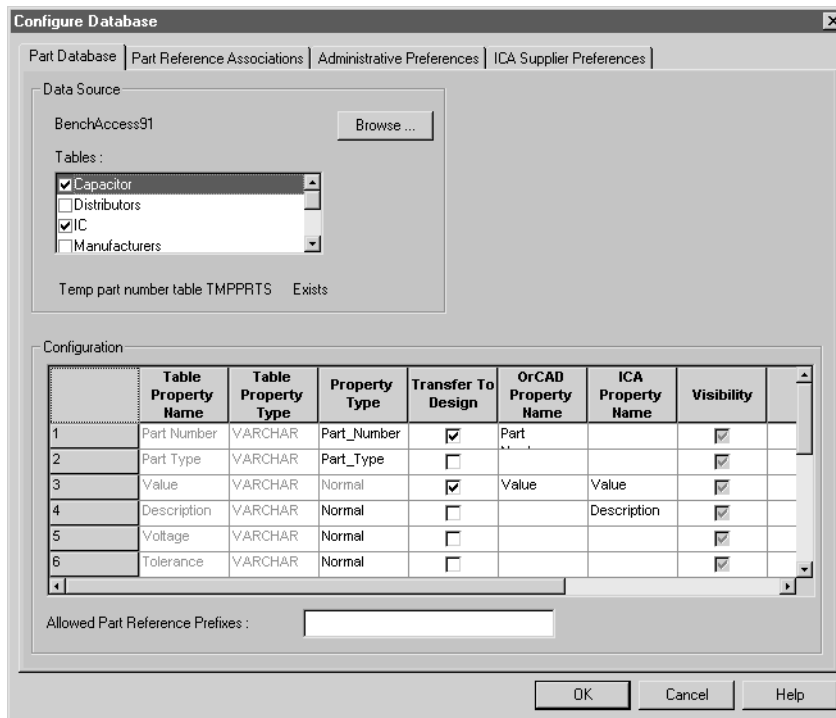
CIS Configuration File	
Configuration File:	
C:\ORCAD\CAPTURE\SAMPLES\BENCHACC.DBC	Setup...
	Browse...
	New...
	Save As...
	Help
OK Cancel	

27
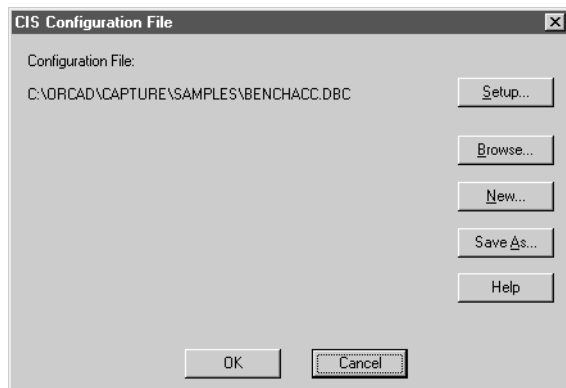
3   Click the New button. CIS displays the Database
     Configuration Wizard.

4   Follow the instructions in the wizard to create your
     database configuration file. If you need more
     information about any of the wizard steps, click the
     Help button.

5   Click the Finish button. CIS displays the Configure
    Database dialog box and the wizard is complete.

6   If you want to set other part database options,
    complete the next section, <u>Setting other part database
    options</u>.

7   If you do not want to set any other part database or
    configuration options, click OK to save the new
    database configuration file.

Setting other part database options

You can set other part database options that are not available in the database configuration wizard, including:

- The part properties to be checked against the part database when you update the part status of a design. CIS sets which properties are checked by default but, if you have a special situation that requires different properties to be checked, you are allowed to change the defaults.

- The part tables that CIS will search when you are linking placed parts to database parts. This is useful when your database consists of several part tables that are organized by device type (for example, capacitors in one table, resistors in another, and so on). When you are linking database parts to placed parts, CIS uses allowed part reference prefixes to limit your search to the appropriate tables. The result is that your search takes less time.

The procedure below describes how to set these options.

To set other part database options

1   In the Configure Database dialog box, select the Part Database tab.
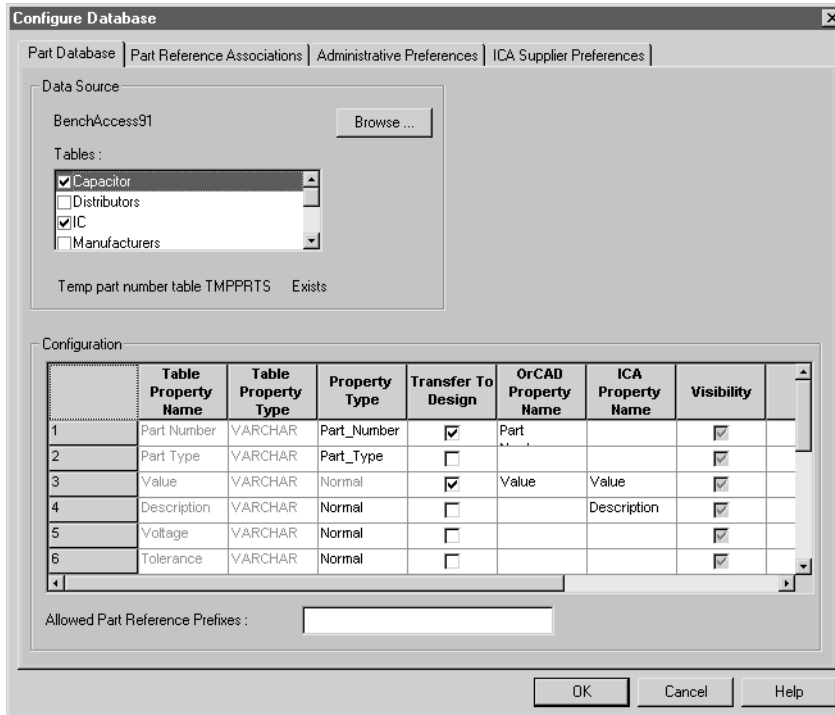
2   In the Tables list, select the database table for which you want to set options.

3   In Update Property column of the Configuration area, select the check box for each property that you want to be checked when you update the part status of your designs.

4   In the Allowed Part Reference Prefixes text box, type the part reference prefixes you want CIS to use to limit searches on the database table.

5   Repeat this procedure on each database table for which you want to set these options.

6   If you do not want to set any other part database or configuration options, click OK to save the new database configuration file.

For example, if a table contains only capacitors, you could enter C as the allowed part reference prefix. Then, when you choose the Link Database Part command, CIS searches that table only when the placed part you are linking has a part reference prefix of C.

Note *Leave this box blank if you want the table to be searched regardless of the part reference prefix.*

31

7   If you want to set other configuration options, go to the next section, <u>Setting other configuration options</u>.

Setting other configuration options

You can set other options in the database configuration that customize how CIS interacts with your part database and how you use CIS in your work environment. These options include:

- Defining part reference associations to improve the speed and accuracy of the search for database parts to link to placed parts. For more information, see <u>Defining part reference associations on page 2-43</u>.

- Setting administrative preferences to customize some CIS features for your work environment. For more information, see <u>Setting administrative preferences on page 2-47</u>.

- Setting ICA supplier preferences to specify which component suppliers you want CIS to search for part pricing and availability. For more information, see <u>Setting ICA supplier preferences on page 2-50</u>.
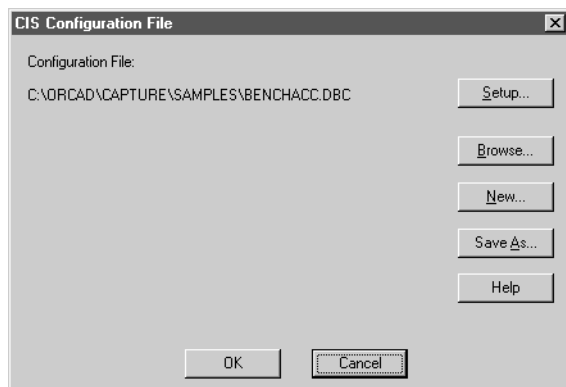
Creating a configuration file manually

You must set up a new ODBC data source before you can create a new configuration file. If you haven't set up a new ODBC data source for your database, see <u>Setting up the ODBC data source on page 2-23</u>.

You can create a configuration file using the manual procedure below. When you are finished creating the file, you will need to set the options for the file so that CIS knows how to handle your database table part properties. For information about setting or editing configuration file options, see <u>Editing a configuration file on page 2-36</u>.

To create a configuration file manually

1 Open a new or existing schematic design in Capture.

2 From the project manager's Options menu, choose CIS
Configuration. CIS displays the CIS Configuration File
dialog box.

```
CIS Configuration File                              [X]

Configuration File:

C:\ORCAD\CAPTURE\SAMPLES\BENCHACC.DBC       [  Setup...  ]

                                            [  Browse...  ]

                                            [   New...    ]

                                            [  Save As... ]

                                            [    Help     ]


            [    OK    ]    [   Cancel   ]
```

3   Click the Setup button. CIS displays the Configure
     Database dialog box.
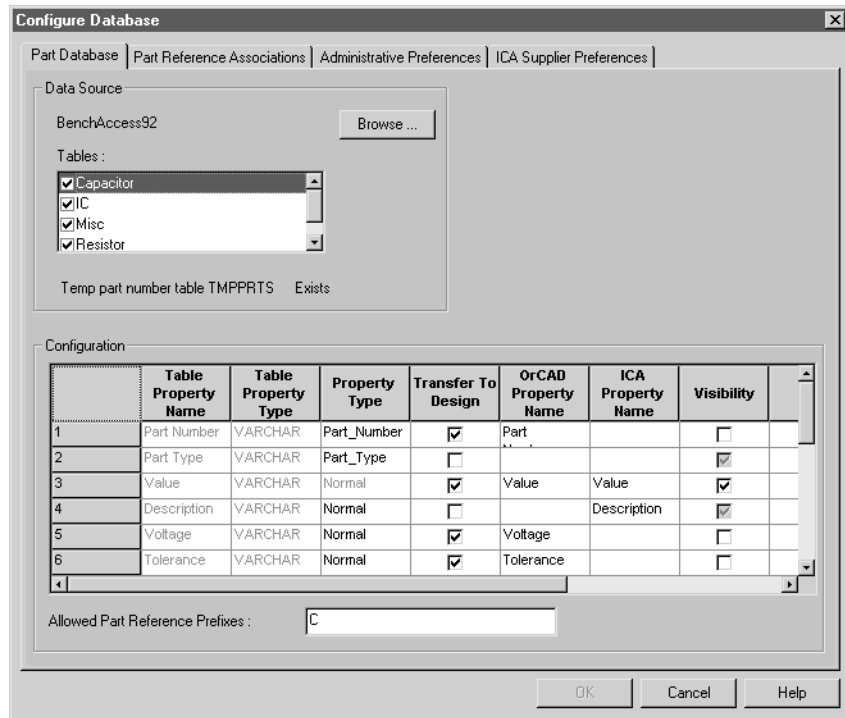
4   Click the Browse button. CIS displays the Browse Data
     Source dialog box.
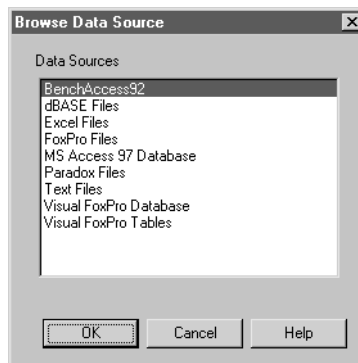
5 Select the data source name you defined for your
ODBC driver, then click OK. The Configure Database
dialog box lists the tables found in your data source.
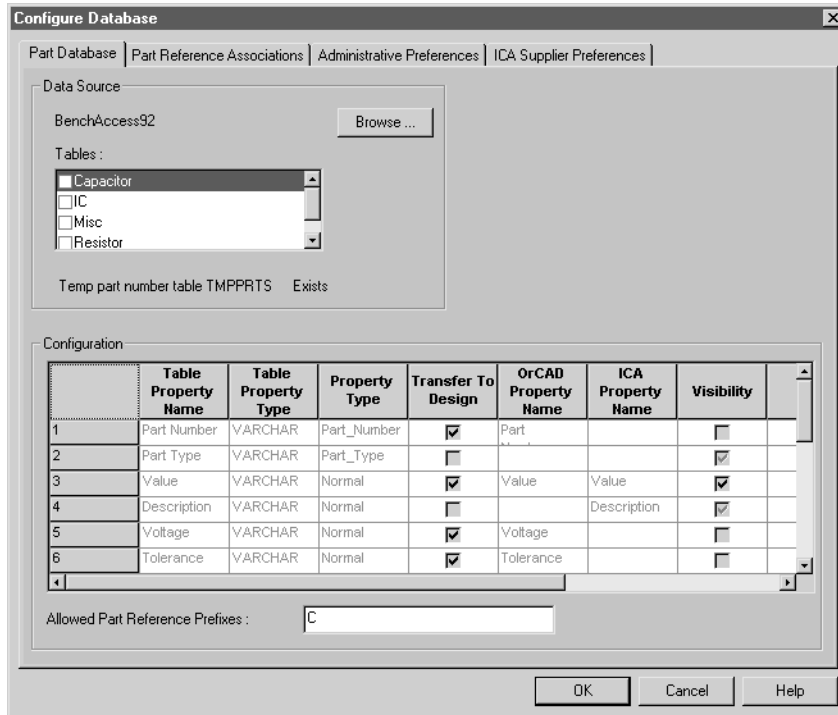
6 Set the configuration file options as desired. For
information about setting configuration file options,
see .

Editing a configuration file

For information about creating a configuration file, see <u>Creating a configuration file on page 2-26</u>.

As you set up and work with CIS, you may discover performance issues or elements of the CIS work environment that are not optimal for your work group. This section describes in detail all the configuration options that are available and how to set them optimally for your situation. The four categories for these options include:

- <u>Setting database table property options on page 2-36</u>.

- <u>Defining part reference associations on page 2-43</u>.

- <u>Setting administrative preferences on page 2-47</u>.

- <u>Setting ICA supplier preferences on page 2-50</u>.

Setting database table property options

Once you have created a database configuration file, you need to set the options for the file so that CIS knows how to handle your database table part properties.

To set database table property options

1   Open a new or existing schematic design in Capture.

2   From the project manager's Options menu, choose CIS
    Configuration. CIS displays the CIS Configuration File
    dialog box.
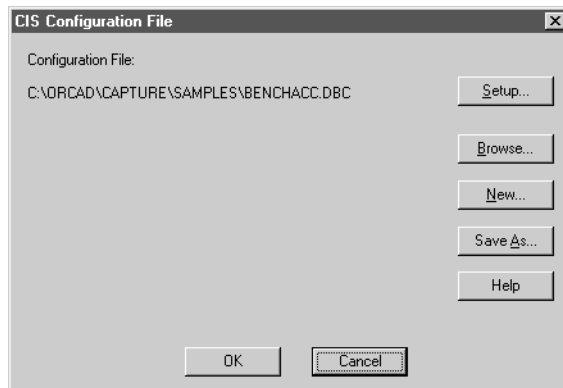
3   Choose a configuration file to edit by doing one of the
    following:

    • To edit the current configuration file, click the
      Setup button.

    • To edit a different configuration file, click the
      Browse button, locate and open the file, then click
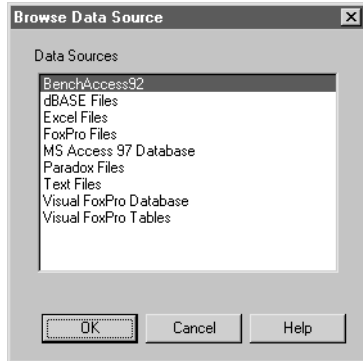   the Setup button.

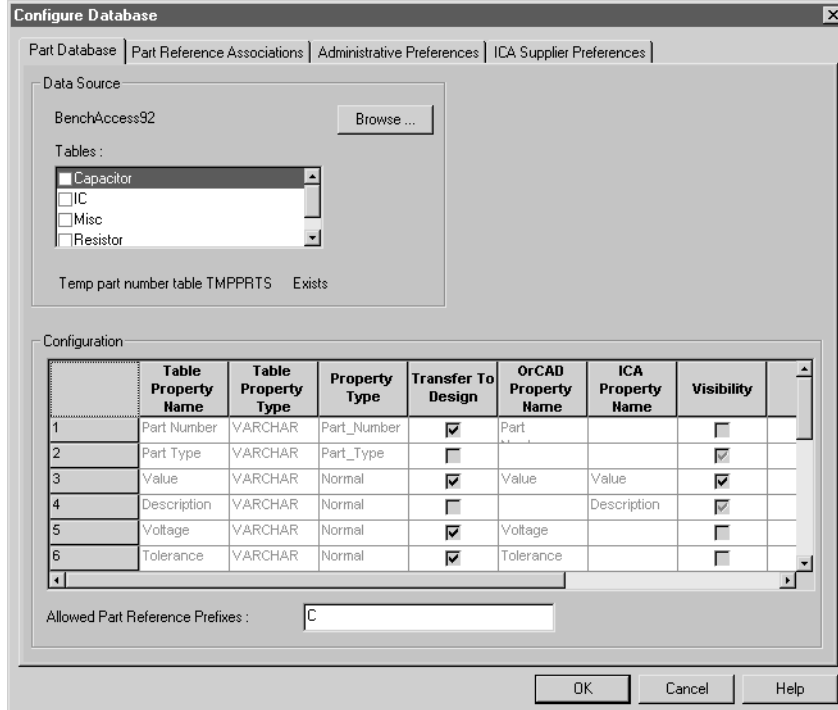CIS displays the Configure Database dialog box.
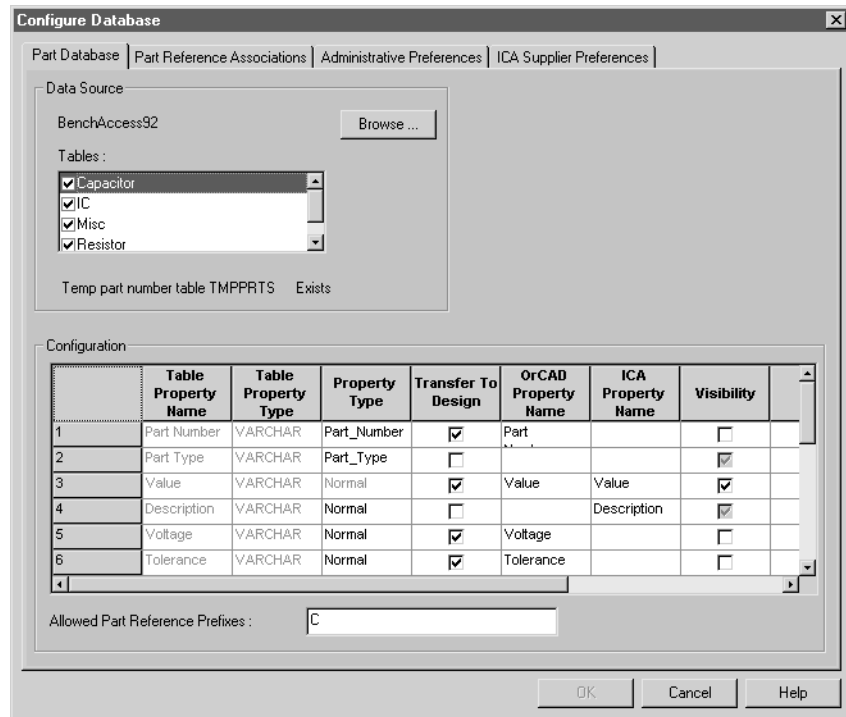
4   If you want to choose a new data source for your configuration, do the following:

Caution   **If you choose a new data source name, all of the settings made with the previously selected data source will be lost for the current configuration (.DBC) file. If you want to keep the settings in the current .DBC file, use the database configuration wizard to create a new configuration file instead. For more information, see** **.**

a Click the Browse button. CIS displays the Browse
Data Source dialog box.

Browse Data Source

Data Sources

BenchAccess92
dBASE Files
Excel Files
FoxPro Files
MS Access 97 Database
Paradox Files
Text Files
Visual FoxPro Database
Visual FoxPro Tables

OK   Cancel   Help

b Select the data source name you defined for your
ODBC driver, then click OK. The Configure
Database dialog box lists the tables found in your
data source.

Configure Database

Part Database | Part Reference Associations | Administrative Preferences | ICA Supplier Preferences

Data Source

BenchAccess92   Browse ...

Tables :

☐ Capacitor
☐ IC
☐ Misc
☐ Resistor

Temp part number table TMPPRTS   Exists

Configuration

	Table Property Name	Table Property Type	Property Type	Transfer To Design	OrCAD Property Name	ICA Property Name	Visibility	
1	Part Number	VARCHAR	Part_Number	☑	Part Number		☐	
2	Part Type	VARCHAR	Part_Type	☐			☑	
3	Value	VARCHAR	Normal	☑	Value	Value	☑	
4	Description	VARCHAR	Normal	☐		Description	☑	
5	Voltage	VARCHAR	Normal	☑	Voltage		☐	
6	Tolerance	VARCHAR	Normal	☑	Tolerance		☐	

Allowed Part Reference Prefixes :   C

OK   Cancel   Help

5   If you want to configure table properties, select the table that contains the properties. In the graphic below, the Capacitor table is selected.
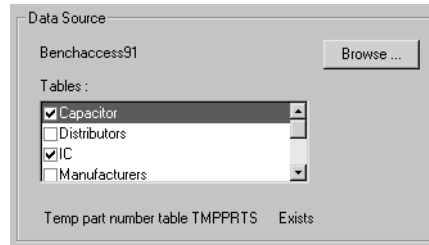
When you select a table, the Configuration area lists the properties it contains. Each row represents a part property, and each property has the following characteristics:

Table Property Name. This is the name of the property as it is defined in the part database.

Table Property Type. This is the data type for the property. Most properties are type Text, but there may be other data types.

Property Type. The Property Type determines how CIS interprets the property. Your database must include a property of type Part_Number in every table.

Set the following property types:

- The Part_Type type for the database property that defines the part type. (This defines the field for folder view in CIS explorer.)

- The Schematic_Part type for the database property that contains the schematic part (symbol) name.

- The PCB_Footprint type for the database property that contains your Layout footprint name. If you want the database to include footprints not generated by Orcad Layout, set the property type to Normal. This prevents the footprint viewer from trying to interpret them.
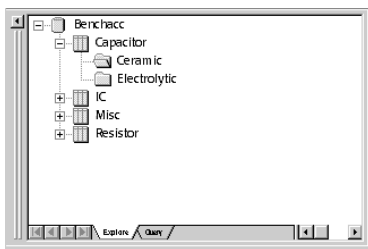
folder view in CIS explorer

- The PSpice_Model type if you have a database property that contains PSpice Models. When you set this property type, CIS activates Pspice functionality for placed parts with a value for the database property.

- The Activepart_ID type if you have a database property to store the unique identifiers for parts that are placed using the activeparts web site through the ICA. When you set this property type, you can create reports about parts using the latest information from the activeparts database. In addition, you can view the latest information for a part directly from an instance on a schematic page.

Transfer To Design. Setting this option (by selecting the appropriate check box) causes CIS to transfer the property along with the part when the part is placed or linked to a schematic part.

Caution   *CIS will not work unless you transfer at least the table property names for the part number and value properties.*

Orcad Property Name. This defines the name of the property when it is transferred to the placed part. By default, it uses the name of the property as it appears in the database. Note that the Orcad Property Name applies only to properties for which you have enabled the Transfer to Design option.

Example: If the property name in the database is Component Tolerance and you want to call it Tolerance on the placed part, type `Tolerance` or choose Tolerance from the drop-down list.

ICA Property Name. This maps property names on new parts created with the internet component assistant (ICA) to corresponding database property names.

Example: You can map the ICA property name Mfr to your database property name Manufacturer.

Visibility. This sets the default visibility of the property when it is copied to the placed part. There are four visibility modes:

    ☑ **Visible.** CIS displays the property with the part on the schematic page.

    ☐ **Invisible.** CIS does not display the property with the part on the schematic page.

No Change. CIS does not modify the property visibility. If the property does not exist, it is set to invisible. You can override the default visibility for specific parts when you place or link database parts. Cells that you cannot change are shown with a light gray background in this column.

Non-adjustable. CIS does not allow this property to be set as visible on schematic pages.

Key. This sets the property as a key during the initial part search. The key is used when you are linking a database part to a previously placed part. Normally, you set only the Value property as a key so that, when you want to link a part, CIS searches the part database for parts with a specific value. If you don't have a Value property in your database, do not set a key.

Browsable. This sets browse capability for the property. It allows you to put references to datasheets, drawings, and documents in your part database. For example, you can reference Adobe Acrobat (.PDF) files, Microsoft Word (.DOC) files, and even worldwide web addresses (URLs). You can then view these items online in CIS when browsing the part database. They are also browsable when you are viewing standard CIS bills of materials.

You can browse any format you want. CIS uses the application assigned to that file extension in your Windows registry. (Extension assignments are managed in the Windows Explorer.) For example, a .DOC entry might cause CIS to launch Microsoft Word, and a URL entry might cause it to launch your default web browser.

Note   *CIS uses the PATH environment variable and the current working directory to find the specified document.*

Update Part Property. Select this if you want the value of this property for placed parts to be checked against the database part's value when you update the part status of your design.

6 (Optional) Type the part reference prefixes in the Allowed Part Reference Prefixes text box.

This is useful when your database consists of several part tables that are organized by device type (for example, capacitors in one table, resistors in another, and so on). When you are linking database parts to placed parts, CIS uses allowed part reference prefixes to limit your search to the appropriate tables. The result is that your search takes less time.

For example, if a table contains only capacitors, you could enter C as the allowed part reference prefix. Then, when you choose the Link Database Part command, CIS searches that table only when the placed part you are linking has a part reference prefix of C.

Note   *Leave this box blank if you want the table to be searched regardless of the part reference prefix.*

7 Repeat steps **5** and **6** for each table in the database.

8 When you are finished configuring table properties, click OK or choose another tab to continue configuring your database.

Defining part reference associations

Part reference associations are used to improve the speed and accuracy of the search for database parts to link to placed parts. You create associations between a particular part type and the prefixes in the part database for that part type. For example, you can create a part reference association for resistors such that all resistors in the part database use the R prefix.

Note   *Defining part reference associations only improves part search speeds for true databases (for example, Microsoft Access)—there is no speed improvement from setting up associations if you are using a spreadsheet or a text file for your database.*

Once you've defined part reference associations for your database, when you choose the Link Database Part command, CIS displays database parts of the appropriate type. Without defined part reference associations, CIS displays all parts in the database that match the keyed property value, regardless of the part type. By defining an appropriate reference association, you can limit the number of parts that CIS displays.

Example: If you select an inductor, L1, on the schematic page with a value of 100uH, then choose the Link Database Part command, CIS displays all parts in the database having a value of 100 x 10^{-6}. This may include capacitors, inductors, or other parts with a similar value.

By defining a reference association between inductors and the L prefix, you can limit the parts that CIS displays to inductors only.

Note   *Defining part reference associations is optional. You can define associations later if you need to improve the search performance during database part linking.*

When you define part reference associations for your part database, keep the following points in mind:

- Part reference associations apply to all database tables.

- The Part Type Property Contents value in the dialog box is case-sensitive.

- You can associate one prefix with several different part types.

Example: An association between the part reference prefix C and the part type Capacitor applies to part types of Capacitor, Capacitor\Electrolytic, and Capacitor\Ceramic\Fixed. Note that a part reference prefix associated with the part type Capacitor\Ceramic does not apply to part types of Capacitor or Capacitor\Electrolytic.

- An association applies to its level in the part type hierarchy and all *lower* levels.

- A part reference prefix without a defined part type association can be matched to any part type in the database.

- If a particular part in the database does not have its Part Type property contents defined, that part will be matched only to part reference prefixes with no defined part type associations.

To define part reference associations

1   In your local preferred parts database, make sure that the Part Type property field is indexed.

2   If you are not already in the Configure Database dialog box, do the following:

   a   Open a new or existing schematic design in Capture.
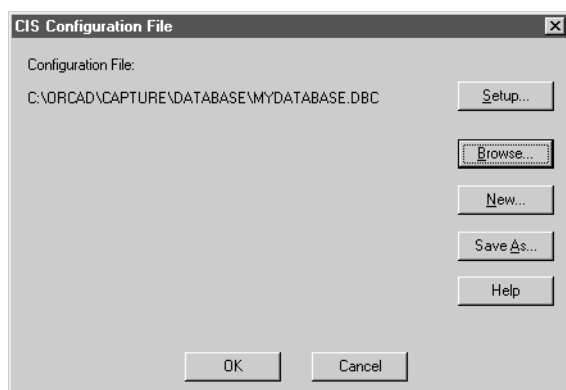
   b   From the project manager's Options menu, choose CIS Configuration. CIS displays the CIS Configuration File dialog box.

You can add icons for custom part references that will automatically display in the Part Reference column of the part manager window. Add your custom icons to the standard icons used for the Part Reference column located in the directory:

ORCAD\CAPTURE\VENDOR

Each icon's filename corresponds to a part reference prefix (for example, the icon for the part reference prefix R is stored in R.BMP).

Caution   *You can use a bitmap editor (such as Microsoft Paint) to modify the bitmaps, but be careful not to change the image size.*

```
+-----------------------------------------------------+
| CIS Configuration File                        [x]   |
|                                                     |
| Configuration File:                                 |
|                                         +--------+  |
| C:\ORCAD\CAPTURE\DATABASE\MYDATABASE.DBC | Setup..|  |
|                                         +--------+  |
|                                         +--------+  |
|                                         | Browse.|  |
|                                         +--------+  |
|                                         +--------+  |
|                                         |  New...|  |
|                                         +--------+  |
|                                         +--------+  |
|                                         | Save As|  |
|                                         +--------+  |
|                                         +--------+  |
|                                         |  Help  |  |
|                                         +--------+  |
|        +--------+      +--------+                    |
|        |  OK    |      | Cancel |                    |
|        +--------+      +--------+                    |
+-----------------------------------------------------+
```

   c   If necessary, click Browse to locate the database configuration file you want to setup.

   d   Click Setup. CIS displays the Configure Database dialog box.

3   In the Configure Database dialog box, choose the Part Reference Associations tab.
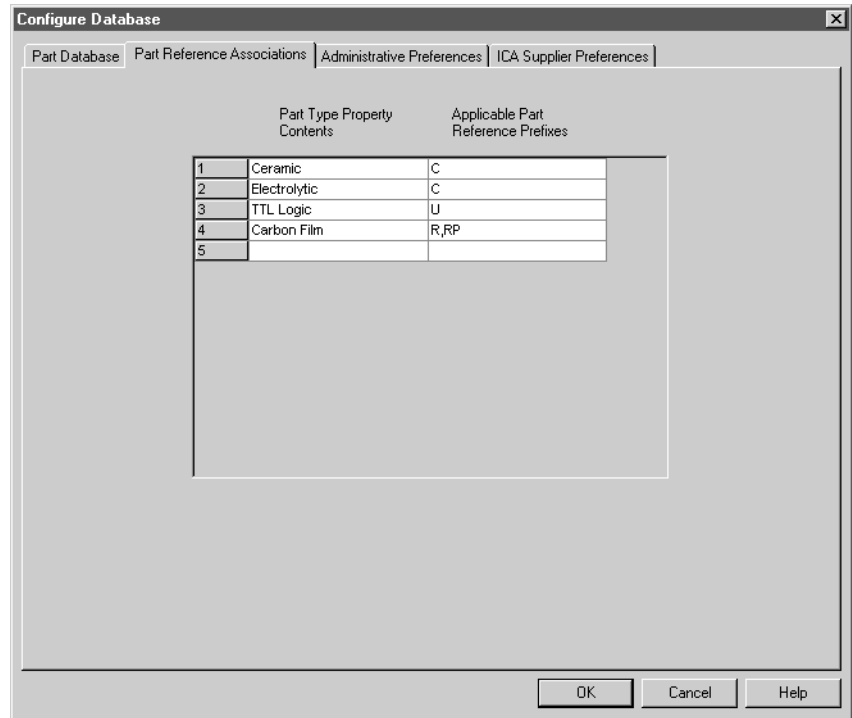
4   Enter a part type in the Part Type Property Contents column and a corresponding part reference prefix (or set of prefixes) for that part type in the Applicable Part Reference Prefixes column. Separate prefix entries with commas.

Note   *Part prefixes need not be unique to a particular part type. That is, you can make an association between one prefix and several different part types.*

5   Click OK or choose another tab to continue configuring your database.

Setting administrative preferences

Administrative preferences allow you to customize some CIS features for your work environment.

To set administrative preferences

1   If you are not already in the Configure Database dialog box, do the following:

  a   Open a new or existing schematic design in Capture.

  b   From the project manager's Options menu, choose CIS Configuration. CIS displays the CIS Configuration File dialog box.
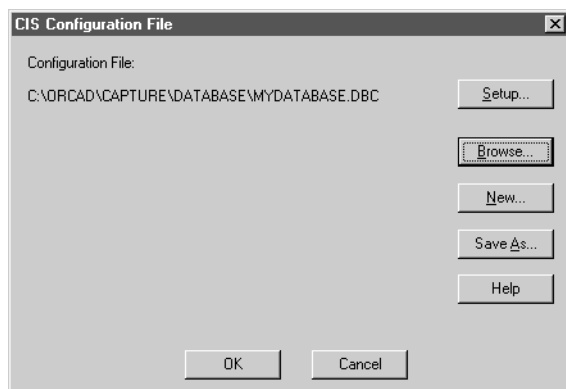
  c   If necessary, click Browse to locate the database configuration file you want to setup.

  d   Click Setup. CIS displays the Configure Database dialog box.

2   In the Configure Database dialog box, choose the
     Administrative Preferences tab.
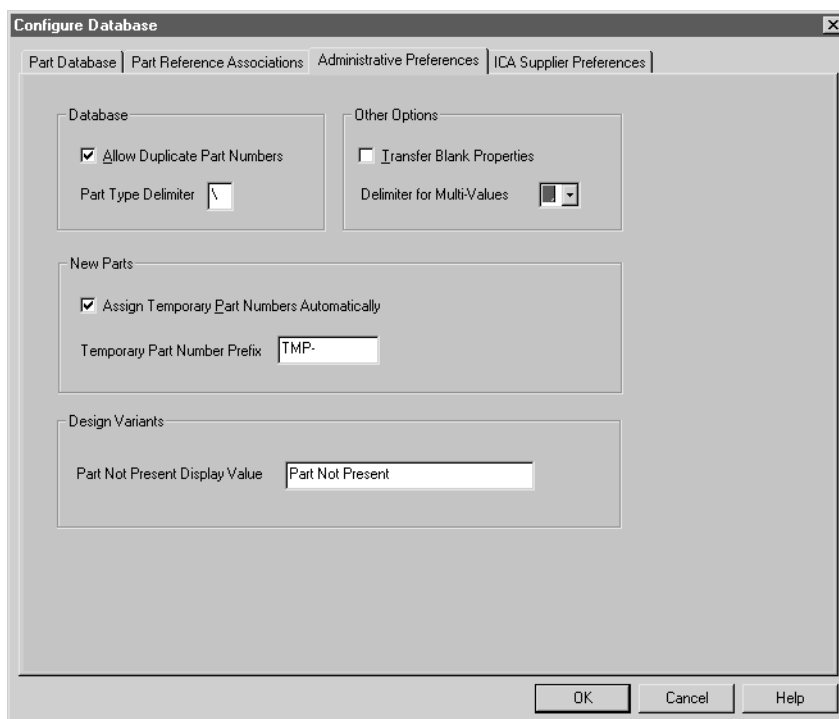
If you want a database part to work with
two or more different layout footprints, you
can enter multiple PCB footprint names in
your part database. For more information,
see the PCB Footprint property entry in
Table 2 on page 19.

Example: The database could have part
types Capacitor\Electrolytic and
Capacitor\Ceramic.

Example: Speed Grade does not apply to
resistors.

3   Select the Allow Duplicate Part Numbers check box to
     allow the same part number to appear more than once
     in the database.

4   In the Part Type Delimiter text box, type the character
     that indicates a hierarchical level within a path in the
     database. Normally, the delimiter is a backslash (\)
     character.

5   Select the Transfer Blank Properties check box to
     create a property on the placed part even if the
     database part property does not have a specific value.
     This is useful if all your database parts are in a single
     table since, in that case, you will have properties in the
     table which are not relevant to certain types of parts.

6   If you want to use a character other than a comma to
     separate multiple field values in your database,

choose another character from the Delimiters for Multi-Values list. For information about entering multiple values for part properties in your part database, see the **Schematic Part (Symbol)** property description in the **Required part properties** table on page 16.

7   Select the Assign Temporary Part Numbers Automatically check box so that CIS will create and track temporary part numbers for you. That way when you create a new part, CIS automatically assigns a temporary part number to that part and enters the part number into the part record in the database as well as in a special table named TMPPRTS.

   Caution   *CIS automatically creates the TMPPRTS table. Do not remove, rename, or modify the structure of this table or temporary part number tracking will not operate properly. Also, never remove temporary part records, even after you have assigned them approved part numbers. If you do, designs that have not yet been updated with the new part numbers will have to be updated manually.*

8   In the Temporary Part Number Prefix text box, enter the prefix to use for temporary part numbers. CIS automatically increments the temporary part number each time you create a new part. The temporary part number is then appended to this supplied prefix.

9   In the Part Not Present Display Value text box, enter the text description that you want CIS to use for variant parts set to Not Present. The property is displayed in the following locations:

   •   Part Number and Value fields in the part manager

   •   Design variant columns in variant reports

   •   Variant parts on schematic page previews and printouts. For information about design variants, see the CIS online help.

10   Click OK or choose another tab to continue configuring your database.

To promote a temporary part to an approved part, do the following:

1   Using your database application, enter the approved part number in the TMPPRTS table next to the corresponding temporary part number.

2   Replace the temporary part number with the approved part number in the part table.

CIS automatically monitors the TMPPRTS table and notifies you if a temporary part in the design has been promoted to an approved part.

If your workgroup is using a shared, read-only database configuration file, all users must use the same temporary part prefix.

The Part Not Present Display Value does not display in Capture's schematic page editor. This property also cannot be repositioned or edited in the schematic page editor. For this reason, you will have to print preview or print a schematic page to make sure that the value you assign the property does not overlap another part or property display.

Because a long value is more likely to overlap a display, you should use a fairly short text equivalent for the default Not Present value.

Setting ICA supplier preferences

ICA supplier preferences allow you to specify which component suppliers you wish CIS to search for part pricing and availability.

To set ICA supplier preferences

1   If you are not already in the Configure Database dialog box, do the following:

a   Open a new or existing schematic design in Capture.

b   From the project manager's Options menu, choose CIS Configuration. CIS displays the CIS Configuration File dialog box.
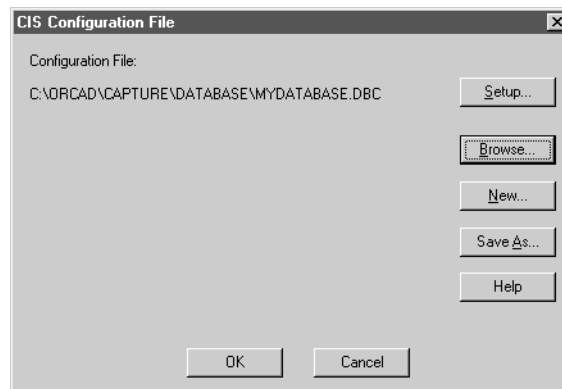
c   If necessary, click Browse to locate the database configuration file you want to setup.

d   Click Setup. CIS displays the Configure Database dialog box.

2   In the Configure Database dialog box, choose the ICA
    Supplier Preferences tab.
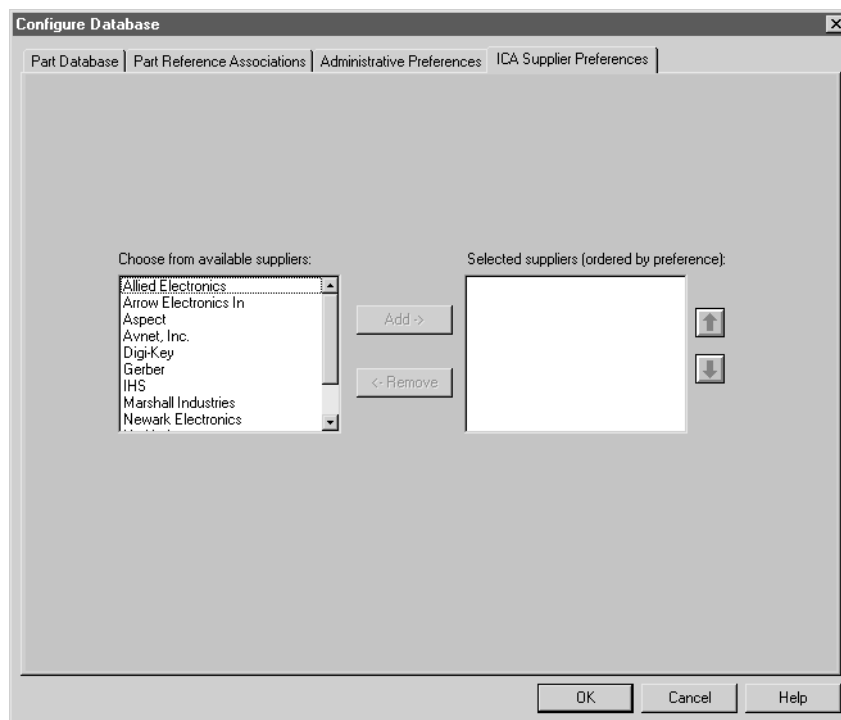
3   In the list on the left, select the name of a supplier to be
    added to the list on the right, and click the Add button.

        or

    In the list on the right, select the name of a supplier to
    remove, and click the Remove button.

4   In the list on the right, select a supplier name, and use
    the arrow buttons to move the supplier higher or
    lower in the order of preference.

    Note   *The internet component assistant only shows information for
            the first supplier that has the part you want, in the order of the
            preference you set here.*

5   Click OK or choose another tab to continue
    configuring your database.

You should select as many part distributors
as possible in case your preferred
distributor does not carry a part you need.

51

Saving the configuration file

1   (Optional) In the CIS Configuration File dialog box, choose Save As to save the configuration for future use.

2   Choose OK to set the configuration for the current session and close the CIS Configuration File dialog box.

Setting up the ICA to access external part data

Note   *Internet Explorer 4 or higher must be installed on your system for the internet component assistant (ICA) to run.*

Setup of the ICA is handled during the Capture CIS installation process. CIS uses the Cadence product registration number to control your access to the ICA. You enter the registration number when you install Capture CIS.

If you need assistance accessing the ICA, contact us by email at ica@orcad.com.

Creating Crystal Reports templates

Crystal Reports is a widely-used report design software produced by Seagate Technology, Inc.

If you want to make a standard CIS bill of materials template, see <u>Creating a standard CIS bill of materials on page 4-104</u>.

You can use Crystal Reports in conjunction with CIS to make customized report templates with more advanced features than the standard CIS bill of materials, including precision formatting and formulas that total, filter, and analyze data for highly specific results. When you create customized Crystal Reports templates, you can use all of the part properties included in your preferred parts database and your individual design projects.

Note   *You must have Crystal Reports Designer installed on your system with CIS to create Crystal Reports templates. If you do not own a copy of Crystal Reports Version 6 or 7, visit Orcad's web site at www.orcad.com for reseller information.*

Once you have created Crystal Reports templates, any CIS user with access to them can generate, preview, print, and export reports with their design's database part information from within CIS. They will not need to have copies of Crystal Reports installed locally on their workstations to use these templates.

This is the process you will complete to create your Crystal Reports templates:

1   Create an Access database bill of materials (.MDB) file using a standard CIS bill of materials.

You only need Crystal Reports software to create templates, not to use them. For this reason, you only need to buy one copy of the software for your entire workgroup.

2   Create report template (.RPT) files in Crystal Reports using the Access database file.

3   Store the Crystal Reports template files where CIS can use them.

To create an Access database (.MDB) file for use by Crystal Reports

You should use a schematic design that has all the non-database part properties defined that you want to use when designing your Crystal Reports templates.

1   Open a new or existing schematic design in Capture.

2   From the project manager's Reports menu, point to
     CIS Bill of Materials and choose Standard. CIS
     displays the Standard Bill of Materials dialog box.
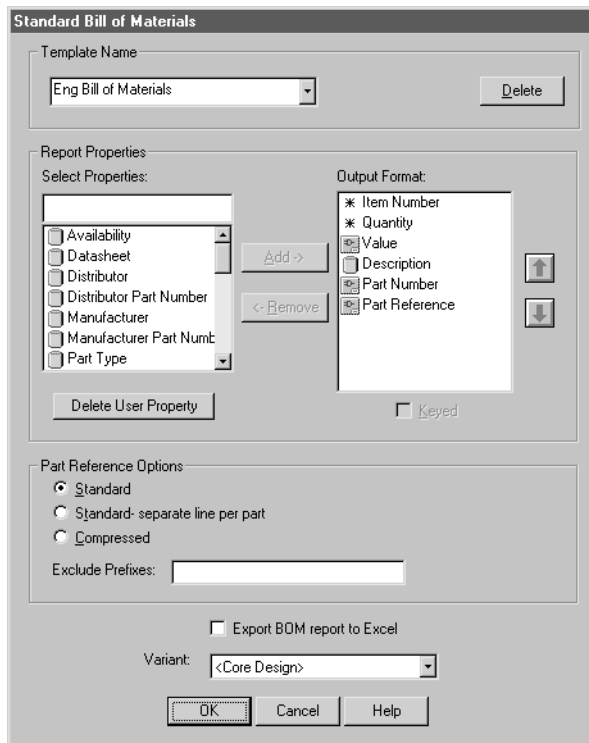
3   In the Template Name text box, type in a name for the
     report (for example, Report for Crystal Reports
     template).

4   In the Select Properties list on the left, double-click
     each of the part properties until all of them are moved
     to the Output Format list on the right.

5   Click OK. The report may take several minutes to
     generate.

6   From the File menu, choose Save As. CIS displays the
     Save As dialog box.

7   In the Save As Type list, select Access format (*.MDB).

8   Specify the file name and location and click OK.

Do not try to be selective about the part properties you include at this time. Generating this report with all the properties in your database allows you to create a single database file with all the headings you might want to use to design your templates in Crystal Reports. This way, you will not have to repeat this procedure again unless the part properties used in your database change.

To create a Crystal Reports template

Note   *You must have Crystal Reports Designer installed on your system with CIS to create Crystal Reports templates. If you do not own a copy of Crystal Reports Version 6 or 7, visit Orcad's web site at www.orcad.com for reseller information.*

1   Using Crystal Report Designer, create a custom report using the Access database bill of materials (.MDB) file as your data file.

To avoid having to browse for your template, store the file in the same directory as your database configuration (.DBC) file. The template will automatically appear in the Crystal Reports Bill of Materials dialog box.

You only need Crystal Reports software to create templates, not to use them. For this reason, you only need to buy one copy of the software for your entire workgroup.

To store a template file for use by CIS

1   Store your Crystal Reports template (.RPT) file where all users in your workgroup have access to it.

Working with database parts

3

Overview

This chapter describes the various procedures you use when working with database parts, including the following:
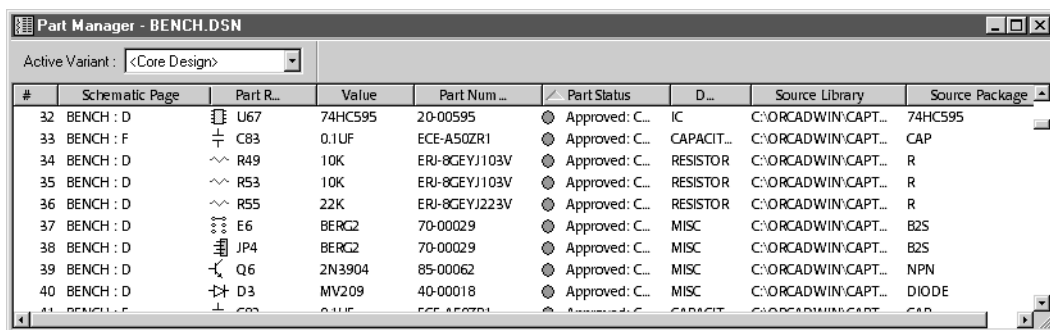
For descriptions of the functions of the windows in the CIS interface, see The CIS work environment on page 1-6.

Tip   *Right mouse button access for the most commonly used CIS commands is available in the project manager, schematic page editor, part manager, and CIS explorer windows.*

Using the CIS interface

The CIS user interface includes standard Windows interface functionality along with a few additional features to make your work environment easy to use. You can also customize the layout of CIS windows for each of your Capture projects. The main CIS interface is comprised of the part manager and CIS explorer windows. Additional CIS menu commands are interspersed throughout the menus in Capture.

Using the part manager window

To open the part manager window, do the following:

1   From the project manager's Tools menu, point to Part Manager and choose Open.

For information about how to use the part database management features of the part manager, see Viewing and updating part status on page 4-98.

You can adjust the work environment for the part manager in the following ways:

•   Sort the part manager data by any one of the data columns.

•   Adjust the part manager display by changing data column width and splitting the window into panes.

•   Display the toolbar.

To sort information in the part manager

1 Click the heading of the column you want to sort by. To reverse the sort order, click the same column heading again.
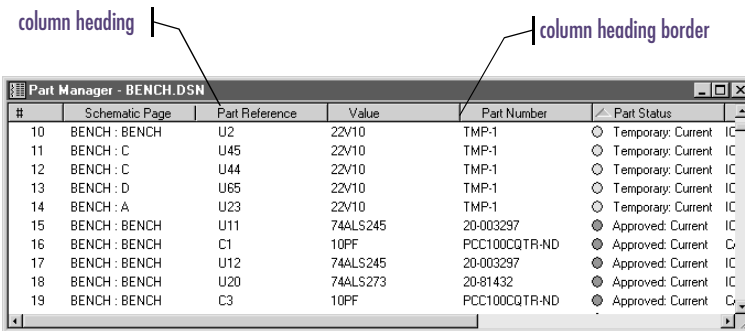
The column by which the data in the part manager is currently sorted is indicated by a triangle icon. If the triangle points upward, the data is sorted in ascending order. If it points downward, the data is sorted in descending order.

column heading

column heading border

To adjust the part manager display

1 To adjust the width of a column, select the right border of the column heading and drag it to the left or right.

2 To split the part manager into panes, choose Split from the part manager's Window menu.
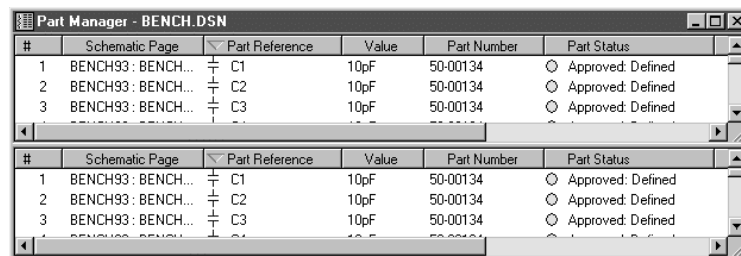
To display the part manager toolbar

1 Move or dock the toolbar as desired.

2 To display the name of the button command, point at the button briefly.

3 To display the toolbar, choose Toolbar from part manager's View menu.

The tasks that you use these tools to perform are described later in this section.

4   To hide the toolbar, click the hide button in the top corner of the toolbar.

Using the CIS explorer window
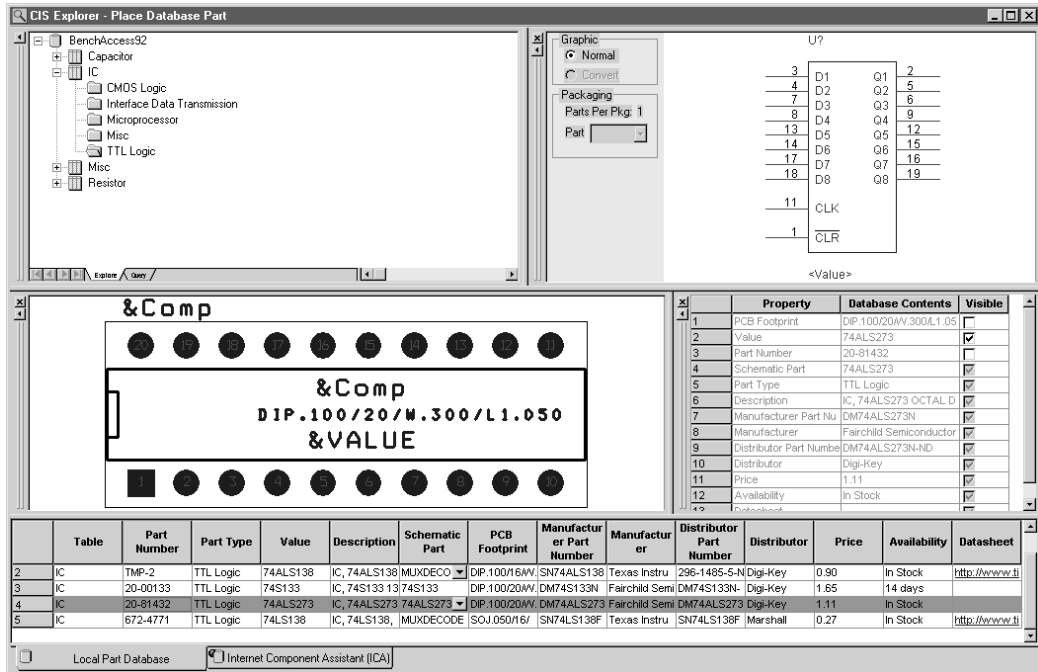
CIS displays the CIS explorer when you choose Place Database Part or Link Database Part from a schematic page in Capture's schematic page editor. Information about how to use the database part features of the CIS explorer is covered in the following sections of this chapter.

The CIS explorer contains two tabbed windows—the Local Part Database and the Internet Component Assistant (ICA). You can navigate between these windows by clicking either one of the tabs at the bottom of the CIS explorer. Each window consists of a set of docking windows that can be redisplayed as floating windows or hidden. The Local Part Database window also contains a non-docking part selection window that cannot be redisplayed as floating or hidden (the database parts window).

Using docking windows

Docking windows can be positioned and sized independently like standard windows. But, they allow you added control over your workspace in relation to the other windows inside the CIS explorer. Docking windows have these unique features:

- Docking to the main window

- Expanding or contracting relative to other windows

- Floating over other docking windows

Depending on the function of the window, you can also choose to show or hide some docking windows.

docking window

CIS retains the size, position, and state of the windows between Capture sessions.

To dock a window in the CIS explorer

1   Drag the window by the double bar over another border of the CIS explorer. The other windows automatically adjust to allow the window you are moving to dock on that border.

To expand or contract a window relative to other windows

1   Click the arrow above the window's double bar to expand or contract the window in the direction of the arrow. The bordering window automatically expands or contracts to fit the new window size.

Since the expand/contract feature only functions horizontally, the arrow is dimmed if the window is not bordered by another window on the left or right.

To change a docked window to a floating window

1 Double-click in the window border. The docking window redisplays with a title bar instead of the double bar.
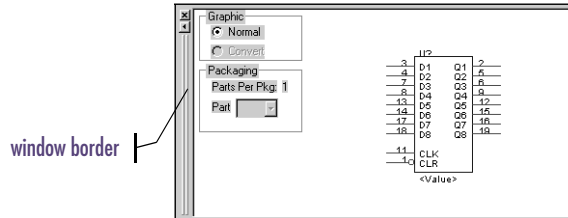
window border

To dock a floating window

1 Double-click the window title bar to return the window to its previous docked location, or drag the window by the title bar to a different docking border.

To position a floating window over a dock (without docking it)

1 Press and hold down Ctrl and drag the window by the title bar.

To show or hide a window

1 To show a window, choose the window name (for example, Visibility) from the View menu.

2 To hide a window, click the hide button in the top corner of the window, or choose the window name from the View menu.

Using the database parts window

	Table	Manufacturer	Manufacturer Part Number	Part Number	Price	Availability	Distributer Part Number	Part Type	Description	Value	Voltage	Distributer	Schematic Part
1	Capacitor	PHILIPS COMPON	A104K29X7RFV	A104K29X7RFVVWN	0.12	In Stock	1109PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
2	Capacitor	PHILIPS COMPON	A104M15Z5UFV	A104M15Z5UFVVWN	0.04	In Stock	1210PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
3	Capacitor	PHILIPS COMPON	A104Z15Z5UFV	A104Z15Z5UFVVWN	0.04	In Stock	1203PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
4	Capacitor	Panasonic - SCD	ECK-F1E104ZV	ECK-F1E104ZV	0.20	In Stock	P4201-ND	Ceramic	CAP CERAMIC	0.1UF	25V	Digi-Key	bench\SM
5	Capacitor	Panasonic - SCD	ECU-V1H104KB	ECU-V1H104KBW	0.11	In Stock	PCC104BTR-ND	Ceramic	.1 UF/50 V CE	0.1UF	50V	Digi-Key	bench\SM
6	Capacitor	PHILIPS COMPON	A100J15C0GHV	A100J15C0GHVVWA	0.06	In Stock	1011PHTR-ND	Ceramic	CAP 100V 10P	10PF	100V	Digi-Key	bench\SM
7	Capacitor	PHILIPS COMPON	D100D20C0GHA	D100D20C0GHAAAC	0.07	In Stock	1326PH-ND	Ceramic	CAP 10PF 100	10PF	100V	Digi-Key	bench\SM
8	Capacitor	PHILIPS COMPON	D100D20C0GLA	D100D20C0GLAAAA	0.14	In Stock	1355PH-ND	Ceramic	CAP 10PF 500	10PF	500V	Digi-Key	bench\SM
9	Capacitor	PHILIPS COMPON	D100D20U2MHA	D100D20U2MHAAAC	0.07	In Stock	1301PH-ND	Ceramic	CAP 10PF 100	10PF	100V	Digi-Key	bench\SM
10	Capacitor	PHILIPS COMPON	D100K20U2MLA	D100K20U2MLAAAA	0.14	In Stock	1344PH-ND	Ceramic	CAP 10PF 500	10PF	500V	Digi-Key	bench\SM

You can use the spreadsheet format of the database parts window to do the following:

- Sort rows using column values

- Adjust column widths

- Change the order of the columns

- Hide or unhide columns

Note   *For a description of the database parts window, see* .

The database parts window is part of the main CIS explorer window—you cannot hide or dock it. This is so that the window will be present at all times to place and link database parts.

Once you set up the columns, CIS saves the new settings for your next session.

The column by which the data is currently sorted is indicated by a triangle icon. If the triangle points upward, the data is sorted in ascending order. If it points downward, the data is sorted in descending order.

To sort rows using column values

1 Click a column heading to select the column, then click the column heading again to sort by its values. Click the same column heading again to reverse the sort order.

column heading   column heading border

	Table	Part Type	Part Number	Value	Descriptic
1	ICs$	TTL	20-81432	74HC273	8 Bit Regist
2	ICs$	TTL	20-08042	74HC273	8 Bit Regist
3	ICs$	TTL	20-00133	74HC133	
4	ICs$	TTL	20-000244	74HC244	8 bit driver
5	ICs$	TTL	20-003296	74HC245	8 bit Bidirec
6	ICs$	TTL	20-00595	74HC595	
7	ICs$	TTL	20-00163	74HC163	8 bit Counte
8	ICs$	TTL	20-00165	74HC165	
9	ICs$	TTL	20-00176	75176	
10	ICs$	TTL	20-00374	74HC374	8 bit Regist
11	ICs$	TTL	20-00091	74HC573	

To adjust column widths

1 Select the right border of a column heading and drag it to the left or right.

To change the order of the columns

1 Click on a column heading to highlight it.

2 Drag the column heading to the left or right until the vertical red line is in the correct location.

CIS keeps a separate column display configuration for each part type folder. So, when you hide or unhide a column, CIS only hides or unhides the column for the part type folder whose parts are currently displayed in the database parts window.

To hide columns

1 Click on a column heading to highlight it.

2 Click the right mouse button on the column heading, and choose Hide Column.

To unhide columns

1 Click the right mouse button on any column heading, and choose Unhide Columns. CIS displays the Unhide Columns dialog box.
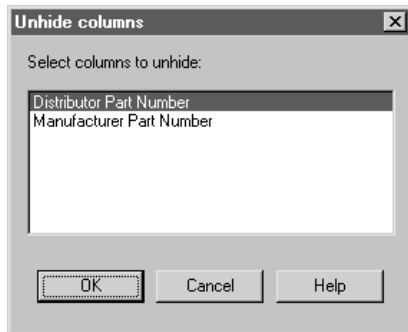
2 Select the names of the columns you want to unhide and click OK.

Placing a database part on a schematic page

This section describes how to do the following:

- Locate a part in your parts database.

- Place a database part on a schematic page.

When you place a database part, CIS includes all database part properties to be transferred.

Note   *The Place Database Part command will not work unless you have entered the schematic part name in your database and have set the Schematic_Part property type in the configuration.*

Using the explorer to locate a database part

You can use the explorer to locate a particular part in your database.

To locate a database part with the explorer

1   From the schematic page editor's Place menu, choose Database Part. CIS displays the explorer window.

2   In the explorer window, choose the Explore tab. The drum icon (labeled Benchacc in this example) represents the part database.

3   Choose ⊞ or double-click the drum icon, to expand the database into the configured part tables. Similarly, choose ⊞ on the table to expand the tree to display the first-level part type folders.

Tip   *If you want to display the whole part tree structure, from the View menu, choose Expand Part Tree.*

4   Locate the part you want to place. When you open a part type folder at any given level, the database parts at that level are displayed in the database parts window. For information about adjusting the display
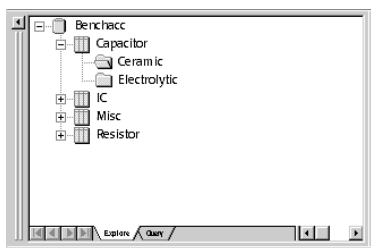
of part information and property columns in the database parts window, see <u>Using the database parts window on page 3-63</u>.

	Table	Manufacturer	Manufacturer Part Number	Part Number	Price	Availability	Distributer Part Number	Part Type	Description	Value	Voltage	Distributer	Schematic Part
1	Capacitor	PHILIPS COMPON	A104K29X7RFV	A104K29X7RFVVWN	0.12	In Stock	1109PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
2	Capacitor	PHILIPS COMPON	A104M15Z5UFV	A104M15Z5UFVVWN	0.04	In Stock	1210PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
3	Capacitor	PHILIPS COMPON	A104Z15Z5UFV	A104Z15Z5UFVVWN	0.04	In Stock	1203PHTR-ND	Ceramic	CAP 50V .1UF	0.1UF	50V	Digi-Key	bench\SM
4	Capacitor	Panasonic - SCD	ECK-F1E104ZV	ECK-F1E104ZV	0.20	In Stock	P4201-ND	Ceramic	CAP CERAMIC	0.1UF	25V	Digi-Key	bench\SM
5	Capacitor	Panasonic - SCD	ECU-V1H104KB	ECU-V1H104KBW	0.11	In Stock	PCC104BTR-ND	Ceramic	.1 UF/50 V CE	0.1UF	50V	Digi-Key	bench\SM
6	Capacitor	PHILIPS COMPON	A100J15C0GHV	A100J15C0GHVVWA	0.06	In Stock	1011PHTR-ND	Ceramic	CAP 100V 10P	10PF	100V	Digi-Key	bench\SM
7	Capacitor	PHILIPS COMPON	D100D20C0GHA	D100D20C0GHAAAC	0.07	In Stock	1326PH-ND	Ceramic	CAP 10PF 100	10PF	100V	Digi-Key	bench\SM
8	Capacitor	PHILIPS COMPON	D100D20C0GLA	D100D20C0GLAAAA	0.14	In Stock	1355PH-ND	Ceramic	CAP 10PF 500	10PF	500V	Digi-Key	bench\SM
9	Capacitor	PHILIPS COMPON	D100D20U2MHA	D100D20U2MHAAAC	0.07	In Stock	1301PH-ND	Ceramic	CAP 10PF 100	10PF	100V	Digi-Key	bench\SM
10	Capacitor	PHILIPS COMPON	D100K20U2ML A	D100K20U2MLAAAA	0.14	In Stock	1344PH-ND	Ceramic	CAP 10PE 500	10PE	500V	Digi-Key	bench\SM

Note   *CIS uses the part type property contents from your part database to determine the folder hierarchy. For a description of how to define your part database hierarchy, see* <u>Creating a part database on page 12</u>.

Using the query feature to locate database parts

You can also use the query feature in the CIS explorer window to locate all parts in the database that fit a set of criteria. This is useful when you want to view all the available parts that match a set of attributes. To use the query feature, you must first create the query criteria, then execute the database search.

Creating and executing a query

1   From the schematic page editor's Place menu, choose Database Part. CIS displays the CIS explorer window.

2   In the Explorer window, choose the Query tab.

3   In the first cell in the Property column, type in the
     name of a property or choose one from the cell's
     drop-down list.

	Property	Compare	Value
1	Value	>=	2.2k
2	Value	<=	3000
3			

4   In the first cell in the Compare column, type in a
     logical operator or choose one from the cell's
     drop-down list. The available logical operators
     include the following:

This operator...	Means this...
=	Equal to
<	Less than
>	Greater than
<=	Less than or equal to
>=	Greater than or equal to
!=	Not equal to
Contains	Contains

5   In the first cell in the Value column, enter a value. If
     you're using the equal to (=) logical operator, you can
     use wildcard characters for the value: an asterisk (*)
     matches a group of characters, and a question mark (?)
     matches any single character.

Since a database search identifies only
those parts that meet *all* the search criteria,
you can use additional rows of criteria to
pinpoint specific parts.

6   Repeat steps **3** through **5** as necessary to construct
     subsequent rows of search criteria. You can delete a
     query row by selecting the row and then
     pressing [Delete].

7   To perform the query, press ⌨️Enter↵ or choose Re-search Database from the Update menu. CIS displays the results in the database parts window.

To gain a better understanding of database queries and their results, see the examples in the CIS online help.

Placing a local database part on your schematic

After locating the database part from the local part database, you can place it on your schematic page. If you want to place a part from the ICA, see Using the ICA to locate and create a new database part on page 3-81.

Note   *If your database contains mechanical (non-electrical) parts, you can place them on your design. However, if you want the mechanical parts to be included in a CIS bill of materials, you must place them on the root schematic or on a schematic that is referenced by a hierarchical block on the root schematic. For information about root schematics and hierarchical blocks, see the Capture User's Guide or Capture online help.*

You can define property place holder positions in the schematic library for properties which you want pre-positioned. For example, you can define a Tolerance property in the resistor schematic part and position it so that when the part is placed, the property position is satisfactory. See the *Orcad Capture User's Guide* for more information.

Any properties specified during configuration as transferable are copied to the placed part and are present in the placed part properties.

CIS sets the appropriate property visibility. If the property is set to Visible and exists in the library, CIS uses the property position defined in the library; otherwise, CIS displays the property in a default position. If you change the position of a property in the part editor, any subsequent updates to the part (using the Link Database Part command) retain the new position.

You can also use the library convert feature to hold two different orientations of the part, one for the normal orientation and one for the 90-degree rotation. See the *Orcad Capture User's Guide* for more information.

To place a database part on your schematic

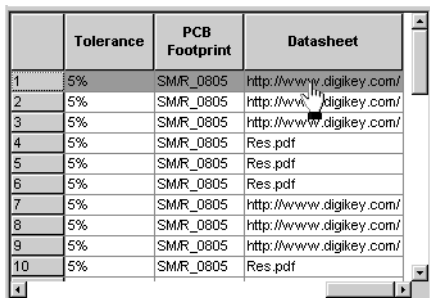1   In the database parts window, select the part you want to place.

2   If the database part has multiple schematic parts associated with it and you want to select a different schematic part name, select one from the drop-down list for the Schematic Part property name.

For information about associating multiple schematic parts with a part in your database, see the Schematic Part property entry in **Table 1** on page 16.

down arrow in value field indicates that multiple schematic parts are available in a drop-down list

For information about associating multiple PCB footprints with a part in your database, see the PCB Footprint property entry in **Table 2** on page 19.

3   If the database part has multiple PCB footprints associated with it and you want to select a different PCB footprint name, select one from the drop-down list for the PCB Footprint property name.

4   If you want to override the default visibility settings of one or more transferable part properties, change the settings in the visibility window.

There are four possible visibility settings:

☑   CIS displays the property with the part on the schematic page.

☐   CIS does not display the property with the part on the schematic page.

☑   CIS does not modify the property visibility. If the property does not exist, it is set to invisible.

▨   CIS does not allow this property to be set as visible on schematic pages.

You cannot change the visibility setting of properties that have not been configured as transferable in the database. The row of an unchangeable property has a light gray background in the Visible column.

5   Start placing the part by doing one of the following:

•   In the database parts window, double-click your selected part.

•   From the Update menu, choose Place Database Part.

6   Click to place the part. Press [Esc] when you're finished placing instances of this part.

Browsing part properties

Database parts may include browsable properties. These properties actually specify links to documents or web sites. CIS allows you to browse properties for which the Browsable option was set during configuration.

To browse a database part property

1   In the database parts window, click the database part property you want to browse. The cursor turns into a hand when you point at a browsable property.

Since the browsing feature is generic, you can view virtually any kind of data. You can include references to your component datasheets on your company's intranet, as Word files, Acrobat files, or any other format. For example, you may want to view package drawings, mechanical models, simulation models, and so on.

	Tolerance	PCB Footprint	Datasheet
1	5%	SM/R_0805	http://www.digikey.com/
2	5%	SM/R_0805	http://www.digikey.com/
3	5%	SM/R_0805	http://www.digikey.com/
4	5%	SM/R_0805	Res.pdf
5	5%	SM/R_0805	Res.pdf
6	5%	SM/R_0805	Res.pdf
7	5%	SM/R_0805	http://www.digikey.com/
8	5%	SM/R_0805	http://www.digikey.com/
9	5%	SM/R_0805	http://www.digikey.com/
10	5%	SM/R_0805	Res.pdf

CIS launches the appropriate browser based on the value of the part property.

Example: If the part property value is a URL such as:

```
http://www.chipmfgr.com/
datasheets/74ALS138.html
```

CIS starts your default web browser and displays the web page for that URL.

Creating a new database part

Almost every design will use some new parts that aren't in your part database and do not have a company part number. CIS lets you create a new part while you're working and save it to the database. This allows you to continue working without having to stop for the part approval process. When you save a temporary part to your database, CIS can automatically assign a temporary part number and track the part for you in the part manager. Later, if the part is approved and assigned a company part number, CIS can automatically update your design with the new part information.

For more information about updating part information on designs, see <u>Viewing and updating part status on page 98</u>.

You can create new database parts using one of the following methods:

- Derive a new database part from your local part database. See <u>Deriving a new database part from the local part database</u> below.

- Derive a new database part from a part placed on a schematic page. See <u>Deriving a new database part from a placed part on page 3-77</u>.

- Locate and create a new database part using the internet component assistant (ICA). See <u>Using the ICA to locate and create a new database part on page 3-81</u>.

Note   *You must have write permission for the part and footprint libraries to save a part with the internet component assistant (ICA).*

Deriving a new database part from the local part database

You can create new database parts using parts that are already stored in your database. Once you have selected and altered an existing database part, you can store it as a temporary part in your database and place instances of the new part on schematic pages.

To derive a new part for the part database

1   If a schematic page is not already open and active,
    open a schematic page.

2   From the Place menu, choose Database Part. CIS
    displays the CIS explorer.
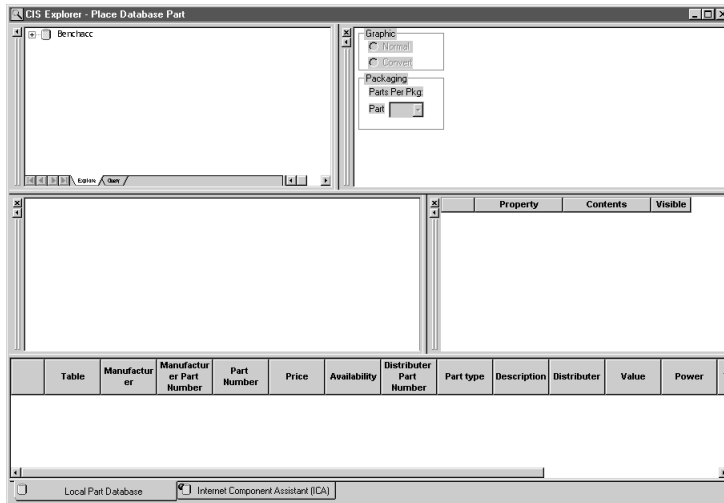
3   Use the explore and query features to locate the part
    you want to use as the basis for creating your new
    database part.

4   Select the desired part in the database parts window.

For more information about locating parts
in your local part database, see Placing
a database part on a schematic
page on page 3-66.

5   From the Edit menu, choose Derive New Database
Part. CIS displays the New Database Part dialog box.
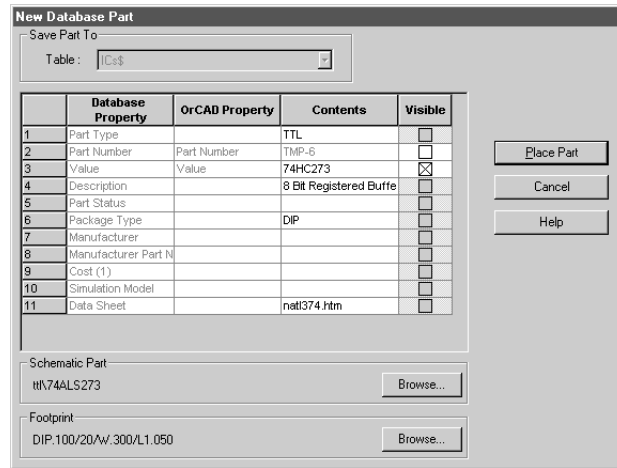
You cannot change the selected table in the
Save Part To area. This is because you are
deriving the new part from a placed part
that is already in your part database, so the
new part is automatically placed in the
same table as the existing database part.

6   Enter the appropriate values for the new part's
database properties in the Contents column.

Note   *If you have selected Assign Temporary Part Numbers
Automatically in the configuration, a temporary part number
is already assigned which you cannot modify.*

7   Modify the visibility settings for each property as
desired.

8   If you want to choose alternate schematic parts or
footprints for the new database part, do the following:

a   Click the Browse button in the Schematic Part area
to choose alternate schematic parts or in the
Footprint area to choose alternate footprints. CIS
displays the Select Schematic Part or Select
Footprint dialog box (see graphic below).

b   If you want to include the library name with the selected schematic parts or footprints, select the Attach the Library Name option. You can include the library name to avoid confusing the schematic part or footprint name with an identically named one in a different library.

c   In the Schematic Part or the Footprint list, select a part or a footprint. The part or footprint displays in the preview window.

Tip *If you know what library your footprints or schematic parts are stored in, highlight only that library in the Configured Libraries list. This will greatly reduce the number of schematic parts you have to scroll through to find part or footprint names.*

Caution *Only select a schematic part from your own custom libraries; do not select a part from the libraries that are supplied with Capture (resource libraries). Resource libraries are often changed by upgrades to new versions of Capture. If the name or path of the schematic part you select is changed during an upgrade, it will become invalid for all your database parts.*

Caution *If you are deriving a new mechanical (non-electrical) part and choosing an alternate schematic part, you must choose a schematic part with no pins. If mechanical parts with pins are placed in a design, they will invalidate netlists generated from that design.*

d   Click the Add button to move the schematic part or footprint to the Selected Schematic Parts or Selected Footprints list. You can remove it from the list by selecting it and clicking the Remove button.

e   Repeat steps **c** and **d** until you have selected all the schematic parts or footprints that you want to be available for use with the database part.

f   If you selected more than one schematic part or footprint, use the up and down arrows to arrange the order of the parts or footprints. The top one will be the default value for the database part. The order of the whole set determines the order in which they will display in the scroll-down list of the Schematic Part or PCB Footprint property value cells of the CIS explorer's database parts window.

Order in the Selected Schematic Parts list

Order in scroll-down list of Schematic Part property value cell in CIS explorer's database parts window

g   Click OK. CIS returns you to the New Database Part dialog box.

Note *Once the new part is added to the database, you must have administrative privileges for the database to modify the properties that you set above.*

9   Click the Place Part button. The dialog box closes, the new part is added to the part database, and you are returned to the schematic page with the pointer in place part mode.

10   Click to place the part. Press [Esc] when you're finished placing instances of the part.

Deriving a new database part from a placed part

You can create new database parts using parts that you have already placed on schematic pages. While you are deriving a new database part, you can change the values and visibility settings of the database part properties.

By default, CIS does not automatically copy all placed part properties to the part database. The properties that are automatically transferred have all three of the following characteristics:

- They are mapped to database part properties in the database configuration.

- They have been set to transfer to designs in the database configuration.

- They have a value for the placed part.

If you want other property values to be added to the database, you can manually add them during this procedure, but only if the property is mapped in the database configuration.

As soon as the new part is stored in the database, you can place instances of the part on schematic pages. The part will remain a temporary part, however, until your database administrator assigns a permanent part number.

To derive a new part from a database part

1   Open the schematic page that contains the placed part you want to use to derive a new database part.

2   Select the placed part.

3   From the Edit menu, choose Derive Database Part. CIS displays the New Database Part dialog box.

New Database Part				
Save Part To				
Table :	Resistor			
	Database Property	**OrCAD Property**	**Contents**	**Visible**
1	Part Number	Part Number	TMP-6	☑
2	Part type			☑
3	Value	Value	330	☑
4	Description			☑
5	Power	Power	1/10W	☑
6	Tolerance	Tolerance	5%	☑
7	Manufacturer Part N			☑
8	Manufacturer			☑
9	Distributor Part Num			☑
10	Distributor			☑
11	Price			☑
12	Availability			☑
13	Datasheet			☑

Derive Part
Cancel
Help

Schematic Part
RESISTOR   Browse...

Footprint
SM/R_0805   Browse...

4   In the Save Part To area, choose the database table in which you want to save the new database part.

Note   *If you are deriving the new part from a placed part that is already in your part database, the new part is automatically placed in the same table as the existing database part. CIS will not allow you to select a different table.*

5   If you want to change the value of a database property, enter the new value in the Contents column.

Note   *If you have selected the Assign Temporary Part Numbers Automatically option in the configuration, a temporary part number is already assigned which you cannot modify.*

6   Modify the visibility settings for each property as desired.

7   If you want to choose alternate schematic parts or footprints for the new database part, do the following:

a   Click the Browse button in the Schematic Part area to choose alternate schematic parts or in the Footprint area to choose alternate footprints. CIS

displays the Select Schematic Part or Select Footprint dialog box.

b If you want to include the library name with the selected schematic parts or footprints, select the Attach the Library Name option. You can include the library name to avoid confusing the schematic part or footprint name with an identically named one in a different library.

c In the Schematic Part or the Footprint list, select a part or a footprint. The part or footprint displays in the preview window.

Caution *Only select a schematic part from your own custom libraries; do not select a part from the libraries that are supplied with Capture (resource libraries). Resource libraries are often changed by upgrades to new versions of Capture. If the name or path of the schematic part you select is changed during an upgrade, it will become invalid for all your database parts.*

Caution *If you are deriving a new mechanical (non-electrical) part and choosing an alternate schematic part, you must choose a schematic part with no pins. If mechanical parts with pins are placed in a design, they will invalidate netlists generated from that design.*

Tip *If you know what library your footprints or schematic parts are stored in, highlight only that library in the Schematic Part Libraries or Configured Libraries list. This will greatly reduce the number of schematic parts you have to scroll through to find part or footprint names.*

d   Click the Add button to move the schematic part or footprint to the Selected Schematic Parts or Selected Footprints list. You can remove it from the list by selecting it and clicking the Remove button.

e   Repeat steps **c** and **d** until you have selected all the schematic parts or footprints that you want to be available for use with the database part.

f   If you selected more than one schematic part or footprint, use the up and down arrows to arrange the order of the parts or footprints. The top one will be the default value for the database part. The order of the whole set determines the order in which they will display in the scroll-down list of the Schematic Part or PCB Footprint property value cells of the CIS explorer's database parts window.

Order in the Selected Schematic Parts list

Order in scroll-down list of Schematic Part property value cell in CIS explorer's database parts window

g   Click OK. CIS returns you to the New Database Part dialog box.

8   Click the Derive Part button. The part is added to the part database with the automatically assigned part number.

Using the ICA to locate and create a new database part

You can use the ICA to access the activeparts web site's part database and locate and create parts that are not yet stored in your local part database.

When you need to locate and create parts during a design session, do the following:

1 Locate a part using the ICA. See <u>Using the ICA to locate a part on page 3-81</u>.

2 Create and place a part using the ICA. See <u>Using the ICA to create and place a part on page 3-82</u>.

Using the ICA to locate a part

Through the ICA, you can locate new parts using the part database at the activeparts web site.

Note   *The activeparts database is updated as new part information becomes available without requiring you to update your CIS software installation.*

To use the ICA to locate a part

1 Open a new or existing schematic design in Capture and, if a schematic page is not already open and active, open a schematic page.

2   From the Place menu, choose Database Part. CIS displays the CIS explorer.

Internet Component Assistant (ICA) tab
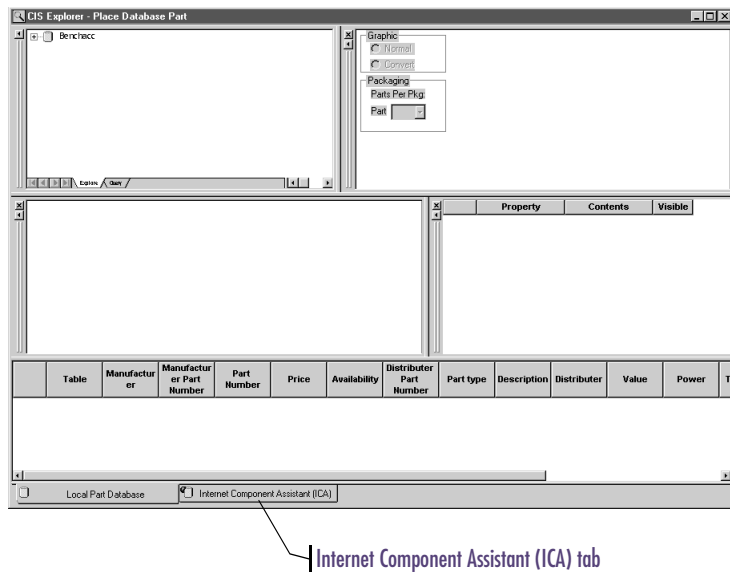
3   Click the Internet Component Assistant (ICA) tab at the bottom of CIS explorer window. CIS displays the activeparts web site.

Note   *The first time you access the activeparts web site, you will have to create a login.*

4   Use the activeparts part search features to locate the part you want to place in your design or add to your preferred parts database.

Note   *For information about using the activeparts search features, refer to the activeparts online help.*

Using the ICA to create and place a part

When you have found the part you want to create, you can add the part to your database and place instances of the new database part on a schematic page.

To use the ICA to create and place a part

1 From the activeparts web site in the ICA window, find the part you want to create and place. If you need instructions for finding a part, see Using the ICA to locate a part on page 3-81.

2 Click the Place in Schematic button. CIS displays Step 1 of the New Database Part Wizard.

The number of steps (1 to 3) in the wizard and the options available (see the table below) in Step 1 of the wizard will vary depending on the following:

- The presence of the correct schematic part for the database part in your local libraries.

- The presence of the correct, configured Layout footprint for the part in your local libraries.
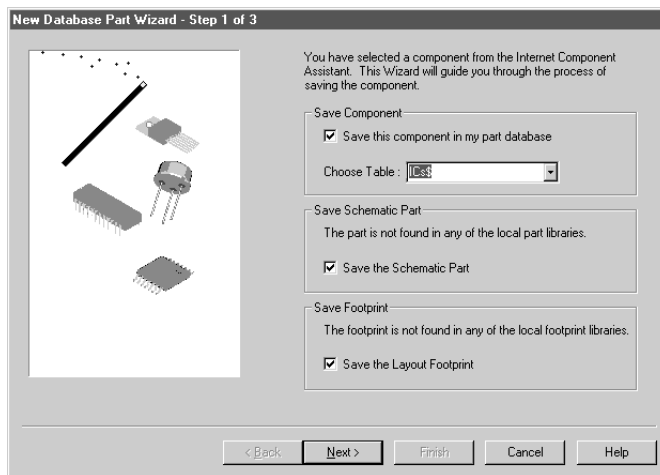
3 Select options in the wizard using the following table as a guide:

If you want to...	Do the following...
Save the part in your part database*	Select the Save This Component In My Part Database option, and choose the database table in which you want the new database part to be stored.
Save the schematic part in a local library**	Select the Save the Schematic Part option.
Save the Layout footprint in a local library**	Select the Save the Layout Footprint option.

* If you choose not to add the part to your part database, CIS will assign a temporary part number to the placed part, but will not add the temporary part number to your database.

** These options will not be available if you already have the correct schematic part and Layout footprint names for the new part stored in your local libraries.

If you accept the CIS default library name and the library has not already been created, Capture will prompt you to allow it to create the library and add the new part to that library. However, before you can place the part from your local part database, you must add the new library to your Capture configuration by doing the following:

1   From the schematic page editor's Place menu, choose Part.

2   In the Place Part dialog box, click the Add Library button.

3   Browse for and select the new library then click Open.

4   Click Cancel.

5   Exit and restart Capture.

You can now place the part from your local part database.

4   If you are saving a schematic part or Layout footprint to a local library, complete the subsequent wizard steps. You will need to either accept the default library name supplied by CIS or browse to find a library in which to store the part or footprint.

Caution   *Only save schematic parts to your own custom libraries; do not save parts to the libraries that are supplied with Capture (resource libraries). Resource libraries are often changed by upgrades to new versions of Capture. If the library in which you save the schematic part is replaced during an upgrade, you will lose the schematic part information associated with the new database part.*

When you finish the New Database Part wizard, CIS displays the New Database Part dialog box.

5   Enter the appropriate values for the new part's properties in the Contents cells.

Note   *If you have selected Assign Temporary Part Numbers Automatically in the configuration, a temporary part number is already assigned which you cannot modify.*

6   Modify the visibility settings for each property as desired.

7   If you want to choose alternate schematic parts or footprints for the new database part, do the following:

a   Click the Browse button in the Schematic Part area to choose alternate schematic parts or in the Footprint area to choose alternate footprints. CIS displays the Select Schematic Part or Select Footprint dialog box.

b   If you want to include the library name with the selected schematic parts or footprints, select the Attach the Library Name option. You can include the library name to avoid confusing the schematic part or footprint name with an identically named one in a different library.

c   In the Schematic Part or the Footprint list, select a part or a footprint. The part or footprint displays in the preview window.

Caution   *Only select a schematic part from your own custom libraries; do not select a part from the libraries that are supplied with Capture (resource libraries). Resource libraries are often changed by upgrades to new versions of Capture. If the name or path of the schematic part you select is changed during an upgrade, it will become invalid for all your database parts.*

Caution   *If you are creating a new mechanical (non-electrical) part, the associated schematic part must not have pins. If mechanical parts with pins are placed in a design, they will invalidate netlists generated from that design.*

d   Click the Add button to move the schematic part or footprint to the Selected Schematic Parts or Selected Footprints list. You can remove it from the list by selecting it and clicking the Remove button.

e   Repeat steps **c** and **d** until you have selected all the schematic parts or footprints that you want to be available for use with the database part.

f   If you selected more than one schematic part or footprint, use the up and down arrows to arrange the order of the parts or footprints. The top one will be the default value for the database part. The order of the whole set determines the order in which they will display in the scroll-down list of the Schematic Part or PCB Footprint property value cells of the CIS explorer's database parts window (see graphic below).
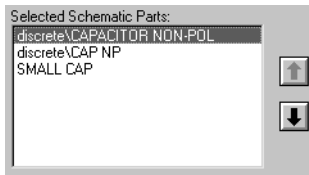
Order in the Selected Schematic Parts list

Order in scroll-down list of Schematic Part property value cell in CIS explorer's database parts window

Note   *Make sure that you have a schematic part selected before continuing. If you do not, CIS will return an error when you attempt to place the part and you will have to repeat this procedure to begin placing the part.*

selected schematic part

g   Click OK. CIS returns you to the New Database Part dialog box.

8   Click the Place Part button. The dialog box closes and you are returned to the schematic page with the pointer in place part mode.

9   Click to place the part. Press [Esc] when you're finished placing instances of this part.
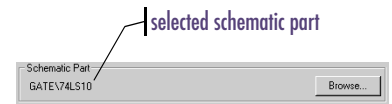
Note *If you chose to add the new part to the database in step 3, a new record is added for the part at this point. Once this new part is added to the database, you must have administrative privileges for the database to modify the properties that you set above.*

Linking a placed part to a database part

Using the visibility window, you can override the default visibility settings of one or more part properties.

⊠ CIS displays the property with the part on the schematic page.

☐ CIS does not display the property with the part on the schematic page.

■ CIS does not modify the property visibility. If the property does not exist, it is set to invisible.

You cannot change the visibility setting of properties that have not been configured as transferable in the database. The row of an unchangeable property has a light gray background in the Visible column.

You can group similar parts in the part manager by sorting on one of the columns (for example, the Value column). Choose a column heading to sort the list by the information in that column; choose the same heading again to reverse the sort order.

You can use the part manager or the Link Database Part command to assign a database part (and the associated transferable properties) to one or more placed parts. By linking a placed part to a database part, you replace the part and its properties with the database part and its transferable properties. If you are linking a part occurrence to a database part, both the occurrence and instance properties will be set to the database properties.

Note *CIS warns you if you select a database part to link to a group of placed parts whose part reference prefixes or values differ. CIS will, however, allow you to link a placed part to a database part with a different schematic part name.*

To link using the part manager

1 From the project manager's Tools menu, point to Part Manager and choose Open. CIS displays the part manager window.

2 In the part manager, select a placed part or a group of placed parts. To select a group of parts, do one of the following:

• For a group of nonadjacent parts, hold down Ctrl and click the parts.

• For a group of adjacent parts, hold down ⇧ Shift and click the first then the last part in the range.

3 From the Tools menu, choose Link Database Part.

      or

Click the Link Database Part toolbar button.

4 In the part database explorer:

Double-click the database part you want.

      or

Select the database part you want and, from the Update menu, choose Link Database Part.

If the schematic part property for the placed part is different from that of the selected database part, CIS will highlight the database part's row yellow when you select it in the database parts window. Then, if you double-click or choose Link Database Part, CIS will allow you to change the schematic part property of the placed part to match the database part.

Note   *In the visibility window, CIS displays the values of the selected database part's properties in the Database Contents column and the values of the placed part's properties in the Schematic Contents column. If the value of placed part's property is green, then it matches the value of the property for the database part. If the value of placed part's property is red, then the value does not match that of the database part.*

Caution   *Because the schematic part used for the placed part will change, you must check the schematic page to make sure that you have correct connectivity when you are finished linking.*

CIS links it to the placed part, and the appropriate properties are transferred.

To link using the schematic page editor

1   Select one or more parts on a schematic page.

2   From the Edit menu, choose Link Database Part. CIS performs a database query, using the keyed properties

for the database, then displays the results of the query in the database parts window.
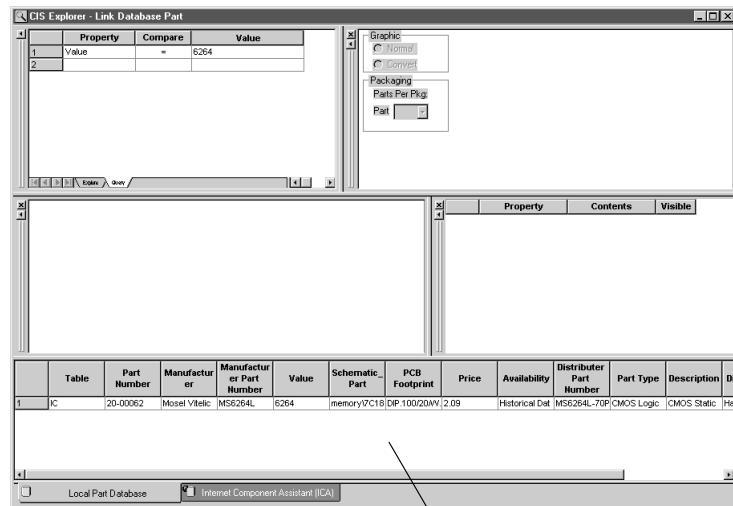
database parts window

If you select several parts, CIS uses the keyed properties and part reference prefix of the first part in the group (from top left to bottom right) for the initial query.

Keyed properties are defined in the database configuration. Generally, though, the Value property is the primary keyed property. The initial query searches for parts with a value matching that of the placed part.

CIS also uses the configuration settings for Allowed Part Reference Prefixes and Part Reference Associations to filter the database for only those parts that apply to the reference prefix of the selected parts.

3 Select the database part you want to apply to the selected placed parts. You can also create a new query if the initial query did not locate the part of interest.

4 In the part database explorer, do one of the following:

Double-click the database part you want.

    or

Select the database part you want and, from the Update menu, choose Link Database Part.

If the schematic part property for the placed part is different from that of the selected database part, CIS will highlight the database part's row yellow when you select it in the database parts window. Then, if you double-click or choose Link Database Part, CIS will allow you to change the schematic part property of the placed part to match the database part.

Note   *In the visibility window, CIS displays the values of the selected database part's properties in the Database Contents column and the values of the placed part's properties in the Schematic Contents column. If the value of placed part's property is green, then it matches the value of the property for the database part. If the value of placed part's property is red, then the value does not match that of the database part.*

Caution   *Because the schematic part used for the placed part will change, you must check the schematic page to make sure that you have correct connectivity when you are finished linking.*

The database part is linked to the placed parts, which also receive the transferable properties, including the part value.

If any of the selected placed parts are multiple-part packages and have defined part references, CIS updates all other parts in the package at the same time. For example, if the selected placed part is U33B, CIS will update U33A, U33C, and any other parts in the package as well. If the selected placed part reference is undefined, only the selected part is updated.

Viewing a placed part's database properties

You can view a placed part's local and Internet database properties right from the schematic page or the part manager, if the part has been added to the local part database. When you choose to view the part properties, CIS displays a read-only version of the CIS explorer with a comparison of the placed part's properties to those of its linked database part. You can also use this part viewing capability to check the latest data sheet, stocking, and pricing information from the Internet.

To view a placed part's database properties

1   From the schematic page or the part manager, select the part for which you want to view database properties.

Note   *On the schematic page, the selected part must be in your local part database for CIS to display part properties. If you attempt to view the database properties of a part not stored in your local part database, CIS will display the Local Part Database tab of the CIS explorer window with no part information. If you then click the Internet Component Assistant (ICA) tab, CIS will display an error message page.*

2   From the View menu, choose Database Part. CIS displays the CIS explorer window in View Database Part mode.

Note   *The View Database Part mode is a read-only mode. For this reason, you cannot use the search or query functions of the explorer window to search for, place, or link a database part.*

CIS displays the database part property contents in the database parts window. In the visibility window, CIS displays both the contents of the database parts properties and of the properties that have been

transferred to the schematic. If the content of a schematic property is green, then it matches the content of the part database. If the content of a schematic property is red, then the contents do not match the part database.

3   If you want to check the latest data sheet, stocking, or pricing information from the Internet, click the Internet Component Assistant (ICA) tab at the bottom of the CIS explorer. The Internet part property information is displayed for the part, including hyperlinked data sheet URLs for parts that have data sheets associated with them.
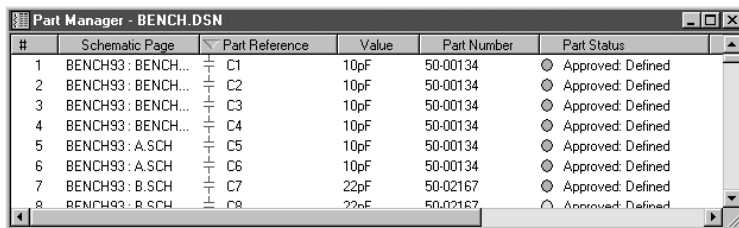
Copying part properties using the part manager

For information about transferring database part properties to placed parts, see .

You can also use the part manager to copy properties from one part to another (or to several others) in the design. CIS copies only those properties defined in the database configuration file as transferable.

To copy properties from one part to another

1   Open the schematic design that contains the part properties you want to copy.

2   From the project manager's Tools menu, point to Part Manager and choose Open. CIS displays the part manager, showing all parts in the design, sorted by part reference.

You can use the Split command (from the part manager's Window menu) to display two areas of the part manager. This can make copying properties from one part to another much easier (by dragging the part across split panes) if the two parts are widely separated in the part manager.

3   Select the part with the properties to be copied.

4   Drag the selected part onto the part on the schematic page that is to receive the copied properties.

To copy properties from one part to a group of parts

1   Open the schematic design that contains the part
    properties you want to copy.

2   From the project manager's Tools menu, point to Part
    Manager and choose Open. CIS displays the part
    manager, showing all parts in the design, sorted by
    part reference.

#	Schematic Page	Part Reference	Value	Part Number	Part Status
1	BENCH93 : BENCH...	C1	10pF	50-00134	Approved: Defined
2	BENCH93 : BENCH...	C2	10pF	50-00134	Approved: Defined
3	BENCH93 : BENCH...	C3	10pF	50-00134	Approved: Defined
4	BENCH93 : BENCH...	C4	10pF	50-00134	Approved: Defined
5	BENCH93 : A.SCH	C5	10pF	50-00134	Approved: Defined
6	BENCH93 : A.SCH	C6	10pF	50-00134	Approved: Defined
7	BENCH93 : B.SCH	C7	22pF	50-02167	Approved: Defined
8	BENCH93 : B.SCH	C8	22pF	50-02167	Approved: Defined

3   In the part manager, select the part with the properties
    you want to copy.

4   From the Edit menu, choose Copy.

5   Select the destination parts by holding down [Ctrl]
    while you select individual parts to add to the selected
    group, or by holding down [⇧ Shift] and selecting a
    contiguous group of parts.

6   From the Edit menu, choose Paste. CIS copies the
    properties from the first part to all the selected
    destination parts.

Finalizing and documenting designs

4

Overview

This chapter describes the various procedures you use when working with the Orcad Component Information System (CIS), including the followinpg:

Viewing and updating part status

You can also set up how CIS sorts and displays the information in the part manager. For more information, see Using the part manager window on page 3-58.

Use the CIS part manager to view and update the database status of placed parts in your design.

Note   *You should always update part status prior to creating a report. By making sure that all part status indicators are green before creating your bill of materials and netlist, you ensure that your report is complete.*
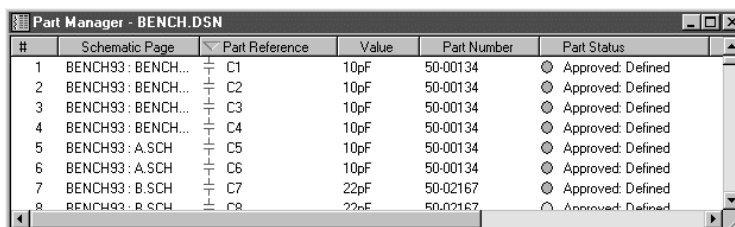
Opening the part manager

When you open the part manager, CIS generates a report describing the status of your design's placed parts the last time they were checked against the database parts. If you have changed part properties on any placed parts or placed any non-database parts since the last time you updated part status, the report generated when you open the part manager will not reflect the current status of your design with respect to the database.

To open the part manager

1   Open a new or existing schematic design.

2   From the project manager's Tools menu, point to Part Manager and choose Open. CIS displays the part manager, showing all parts in the design, sorted by part reference.

If no configuration (.DBC) file has been specified for use with the part manager, no information will be displayed in these columns when you open the part manager for the first time. For information on setting up a part database and configuration file, see **Chapter 2,** Setting up Orcad CIS.

#	Schematic Page	Part Reference	Value	Part Number	Part Status
1	BENCH93 : BENCH...	C1	10pF	50-00134	Approved: Defined
2	BENCH93 : BENCH...	C2	10pF	50-00134	Approved: Defined
3	BENCH93 : BENCH...	C3	10pF	50-00134	Approved: Defined
4	BENCH93 : BENCH...	C4	10pF	50-00134	Approved: Defined
5	BENCH93 : A.SCH	C5	10pF	50-00134	Approved: Defined
6	BENCH93 : A.SCH	C6	10pF	50-00134	Approved: Defined
7	BENCH93 : B.SCH	C7	22pF	50-02167	Approved: Defined
8	BENCH93 : B.SCH	C8	22pF	50-02167	Approved: Defined

Updating the part status for your design

You can update the part status for your design using the part manager. When you update the part manager, CIS checks all the placed parts in your design against the part database, and updates part properties where necessary.

Any properties that you have assigned to part instances are retained when you update part status for your design, even if they differ from the values in the database.

Note   *You can also view an individual part's local and Internet database properties. For instructions, see* <u>Viewing a placed part's database properties on page 92</u>*.)*

To update part status for your design

1   Do one of the following:

   • From the project manager's Tools menu, point to Part Manager and choose Update.

   • From the part manager's Tools menu, choose Update Part Status.

   CIS checks each placed part against the database part to which it is linked. The part database is searched for the Part Number property that matches the placed part, then transferred properties that are configured to be updated are compared.

   For each placed part that is not current, you are prompted with the Update Part dialog box. This dialog box lets you decide whether or not you want to update the placed part properties with the transferred properties from the database part.
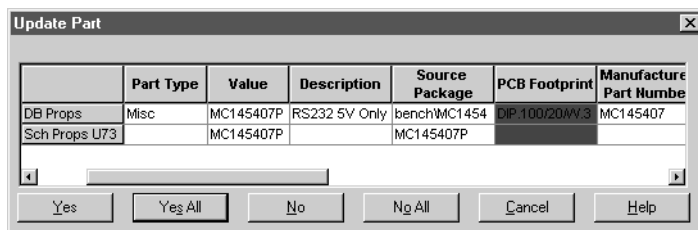
Example: *Assume that an AND gate part in your database has a Part Number value of* `456`*. Suppose that you assign a Part Number value of* `123` *to an instance of an AND gate in your design. That Part Number value applies only to that particular instance of the AND gate. When you update part status, CIS assigns the Part Number value* `456` *to all instances of the AND gate except that instance, which retains the value* `123`*.*

If you want to make sure that none of your parts have been modified since they were placed from their original libraries, from the Options menu in part manager, point to Update Part Status and choose Verify Parts Against Libraries (.OLB). Then, when you update part status, any placed part with a different timestamp than its library part will be flagged yellow.

Note   *If you turn on the Verify Parts Against Libraries (.OLB) option when you have placed parts in the design using a previous version of CIS (Release 9.0 or earlier), all parts placed from the previous Capture libraries will have mismatched timestamps and be flagged yellow. If you get an undesirable result, turn off the Verify Parts Against Libraries (.OLB) option and update part status again.*

The first row lists the database part and its properties, and the second row lists the placed part and its properties. The differences between the database part and placed part are highlighted in red.

2   Click one of the following buttons:

Click this button...	To do this...
Yes	Update the placed part.
Yes All	Update all placed parts whose properties don't match the database part properties.
No	Not update the placed part.
No all	Not update any of the placed parts whose properties don't match the database part properties.

Note   *In some cases, CIS cannot automatically determine the correct database part with which to refresh the placed part. In these cases, you must wait until the update is complete and link the placed part to a database part. (See* Linking a placed part to a database part on page 88*.)*

When complete, a list of updated parts and discrepancies with the database is written to the session log. Then, the part manager window displays the updated part status. The Part Status column contains both text and color-coded icons that indicate whether your placed part is linked to a part in a part database. The following table lists all the possible status conditions and what the each status condition means.

This status...	Means the part...
Green icon	Is approved and current
Yellow icon	Is in the approval process
Red icon	Would be incorrect if you generated a BOM
Approved	Has a defined part number property type
Temporary	Has a temporary part number
Undefined	Has no part number property
Current	Transferable properties match those of the database part
Defined	Has a part number property
Undefined part reference	Has an undefined part reference
Not current	Has at least one transferable property that does not match the database part properties
Duplicate	Has a part number that occurs more than once in the part database
Not found	Has a part number that does not exist in the part database

If your configuration specifies that CIS assign temporary part numbers automatically, each status is preceded by either Temporary Part (if the placed part number prefix is the temporary part number prefix) or Approved Part.

When the status of all the parts in your design is Current, you are ready to generate a report such as a bill of materials.

Once you have updated part status, the part manager also includes tips for parts with a red icon status. The tips give more detail about the part's status and about placed part properties that do not match the database part properties. To display a tip, point briefly over the status column.

Note   *The part status is based only on the part properties you have specified to be transferred from the part database. Other properties that may reside in the part are not checked.*

Saving the status report

You can save the contents of the part manager window to a text (.PRP) file. The report file is saved (in the current sort order) in tab-delimited format which can be edited with a spreadsheet program or word processor.

To save a status report

1   Choose one of the following options:

Example: If your design's filename is MYDESIGN.DSN, CIS automatically names your report MYDESIGN.PRP when you choose the Save command.

To do this...	Select this option...
Save the report file using the name of your design file	From the File menu, select Save.
Save the report file using the name of your choice	From the File menu, select Save As.

Creating design variants

Note   *For more information about design variants and the procedures for creating and documenting them, see the CIS online help.*

Design variants are different assembly configurations of your basic or core design. When manufacturing design variants, the fabricated board will match the core design while the variants define how the board should be populated with parts. You can create and maintain all of your design variants within a single Capture project.

Because your design variations are captured within a single project, you can print variant schematic sheets and create variant bills of materials without having to maintain separate designs. Design variations include different property values for common components and different or not present components for identical footprints.

Examples of product variation requirements include:

- Use in geographical regions with different performance requirements for component attributes such as power level, emissions standards, etc.

- Minor differences between product models such as the number and type of resistors.

Creating reports

A report contains relevant information about the parts in your design. You can customize the report to include both database and placed part properties. CIS supports the following two types of reports:

Note   *If you want mechanical (non-electrical) parts to be included in CIS reports, you must place these parts on the root schematic of the design or on a schematic that is referenced by a hierarchical block on the root schematic. For information about root schematics and hierarchical blocks, see the Capture User's Guide or Capture online help.*

- Standard bills of materials that you create completely within CIS. See Creating a standard CIS bill of materials below.

- Advanced reports that you generate from within CIS using templates created in Crystal Reports. See Creating a report using a Crystal Reports template on page 4-109.

Crystal Reports is a widely-used report design software produced by Seagate Technology, Inc.

You can use Crystal Reports in conjunction with CIS to make customized report templates with more advanced features than the standard CIS bill of materials, including precision formatting and formulas that total, filter, and analyze data for highly specific results.

You must have Crystal Reports installed on your system with CIS to create Crystal Reports templates. If you do not own a copy of Crystal Reports software, visit Orcad's web site at www.orcad.com for reseller information.

If you want to make a Crystal Reports template, see Creating Crystal Reports templates on page 2-54.

Creating a standard CIS bill of materials

A bill of materials lists all the parts in your design. You can use the standard CIS bill of materials feature to create multiple, named report templates so that you can generate separate bills of materials for the different requirements of engineering, purchasing, and manufacturing.

To create a standard CIS bill of materials

Note   *Do not use the Bill of Materials command from the project manager's Tools menu to create your reports. This command does not use your part database information to produce the bill of materials. Only the CIS Bill of Materials commands from the Reports menu utilize your part database information.*

1   From the part manager's Reports menu, point to CIS Bill of Materials and choose Standard. CIS displays the Standard Bill of Materials dialog box.

You can also access the CIS Bill of Materials commands using the project manager's Reports menu.

2   In the Template Name text box, type in a name for the template, or select one from the drop-down list.

3   Select a part property you want in the bill of materials and click the Add button. The part property moves to the Output Format list. You can also double-click a property to move it from one side to the other.

You can create multiple, named templates and define a different format for each. However, since a bill of materials file is, by default, saved using the design name and a .BOM extension, you'll have to save each report under a unique filename using the part manager's Save As command (from the File menu).

Tip   *The Select Properties list only includes schematic properties that you have configured to be transferred from the database. If you want to include a schematic property in your bill of materials that is not in your part database, you must type in its exact name and then choose the Add button. If you want to delete an added schematic property from the Select Properties list, select the property and click the Delete User Property button.*

Note   *The property types that appear in the Select Properties list box include the following:*

✳   Default CIS property names

⬛   Properties transferred from placed parts on schematics

⬜   Properties transferred from database parts

⬛   Properties transferred over the Internet through the ICA

You specify the properties that are transferred from the database to your placed parts when you configure CIS.

The order of the items in the Output Format list box determines the order in which they appear in the report. To change the order of an item, select it and move it using the up or down arrow buttons to the right of the Output Format list box.

4   Repeat step **3** until all the properties you want in the bill of materials appear in the Output Format list box.

5   Select at least one property in the Output Format list box then select the Keyed option. You must do this to specify how CIS matches and groups records in the report. Normally, you should set Part Number to Keyed, so that parts with the same part number will be grouped in the report.

Note   *Keyed properties that are missing from the placed part or have blank contents are not grouped, but remain as distinct items in the report. This is so that undefined parts display as separate line items.*

6   Select one of the Part Reference Options from the
following table:

To do this...	Select this option...
Group parts with matching keyed properties on a single line	Standard
Put each part on a separate line*	Standard – Separate Line Per Part
Allow part ranges (such as R1–R14)	Compressed

* Item Number and Quantity are only listed in the first line of each grouping.

7   In the Exclude Prefixes field, specify part reference
prefixes for parts you do not want to include in the
report. Typically, you use this to omit parts that are
not purchased for manufacture.

8   If you want to export your report to spreadsheet
format and open it automatically in MS Excel, select
the Export BOM Report to Excel option

Note   *If your design has design variants, you will have the option of
selecting the variants whose part information CIS will use to
generate the bill of materials. See the CIS online help for more
information about creating bills of materials for design
variants.*

9   Click the OK button. CIS automatically adds the
template name for the bill of materials you have
designed to the Template Name list so that you can
use the template again the next time you want to
generate a bill of materials.

CIS checks for parts that are grouped (by
having the same part number, for
example), but that have different
properties or packaging or have duplicate
part references between instances of the
parts.

If any design errors are encountered during report
creation, a dialog box prompts you to view the errors
in the session log.

CIS creates a report of the entire physical
design regardless of the schematics and
pages selected in the project manager.

If no design errors are encountered during report creation, the bill of materials report appears in a window.

You can choose a column heading to sort the rows by the values in that column, and choose the same column heading again to reverse the sort order. You can adjust the column widths by dragging the vertical lines between the column heading names to the left or right. If you save the bill of materials, it is saved with the current sort order.

Item Number	Quantity	Value	Description	Part Number	Part Reference
1	6	10PF	CAP 10PF 50V C...	PCC100CQTR-ND	C1 C2 C3 C4 C5 C6
2	9	22PF	22PF 50V CERA...	PCC220CNTR-ND	C7 C8 C24 C25 C26 C2...
3	1	150pF		50-00081	C9
4	3	390PF	390PF 50V CER...	PCC391BNTR-ND	C10 C13 C16
5	3	820PF	820PF 50V CER...	PCC821CGTR-ND	C11 C14 C17
6	2	150PF	150PF 50V CER...	PCC151CGTR-ND	C12 C15
7	1	0.1UF			C18
8	13	0.1UF	CAP 50V .1UF ...	ECE-B1HGE0R1	C19 C39 C43 C52 C63 ...
9	1	0.1UF	.1UFD @ 50VDC...	ECE-A50ZR1	C20
10	5	10UF	CAP 10UF 50V V...	ECE-V1HA100SP	C21 C22 C23 C129 C134
11	11	0.1UF	CAP ELECT .1U...	ECE-A1HKK0R1	C29 C41 C47 C53 C78 ...
12	1	0.1UF	CAP 50V .1UF A...	A104Z15Z5UFW...	C30
13	4	10UF	NHE RADIAL EL...	ECE-A2AGE100	C31 C33 C124 C125
14	1	10UF	CAP 10UF 25V V...	ECE-V1EA100SR	C32
15	7	10UF	10UF/50V SMD ...	ECE-V1HV100SP	C34 C122 C123 C126 ...
16	12	0.1UF	CAP ELECT .1U...	ECE-A1HKS0R1	C37 C40 C57 C62 C68 ...
17	16	0.1UF	0.1UF 50V MINI ...	ECE-A1HKA0R1	C38 C42 C45 C54 C60 ...
18	11	0.1UF	0.1UF/50V SMD...	ECE-V1HA0R1R	C44 C51 C55 C56 C58 ...
19	11	0.1UF	CAP .1UF 50V V...	ECE-V1HS0R1SR	C46 C48 C61 C87 C10...
20	10	0.1UF	CAP .1UF 50V V...	ECE-V1HA0R1SR	C49 C50 C64 C66 C71 ...
21	3	0.1UF	NHE RADIAL EL...	ECE-A1HGE0R1	C59 C81 C99

Note   *If you have included properties in your report that are configured as browsable, CIS will hyperlink (blue underline) the part values for those properties. You can click on the hyperlinks to display the latest version of the source file.*

You can use a spreadsheet application to sort the exported output by multiple keys.

10   To save the report, select one of the save options from the following table:

To do this...	Select this option...
Save the report into the design folder using the default file name*. Note that you will overwrite any report previously saved using this default file name.	From the File menu, choose Save.
Save the report using a new file name into the directory of your choice.	From the File menu, choose Save As.

* CIS creates the default file name by concatenating the design name and the name of the bill of materials template used to generate the report. For example, if the design is named BENCH.DSN and the template is named ENG BILL OF MATERIALS, the bill of materials file will be named BENCH_ENG BILL OF MATERIALS.BOM by default.

11 Specify a filename, save location, and format type. By
    default, CIS saves the report in tab-delimited format,
    but you can choose any of the following formats:

    • TAB - Tab Separated Format

    • (Excel) CSV - Comma Separated Format

    • Access - Compatible with Microsoft Access as well
      as Seagate Crystal Reports.

Creating a report using a Crystal Reports template

In addition to the standard CIS bill of materials, the print
preview, print, and export functions of Seagate's Crystal
Reports software have been integrated into CIS to allow
you to use Crystal Reports templates without having to
install the software on your system. You only need access
to the template files on your local drive or a network to
generate your custom reports.

You can distribute reports you create in Crystal Reports in
many ways, including the following:

• Print and distribute by hand

• Export in Crystal Reports Designer format (.RPT)

• Attach to an e-mail message in MS Word (.DOC) or
  Excel (.XLS) format

• Post in an MS Exchange folder in MS Word (.DOC) or
  Excel (.XLS) format

• Export to HTML and publish on the Internet

When you export a report to a different file
format, you may lose some or all of the
formatting. The export function can only
preserve as much formatting as the export
format allows.

To select and setup a predefined Crystal Reports template

You can also access the CIS Bill of Materials commands using the project manager's Reports menu.

1   From the part manager's Reports menu, point to CIS Bill of Materials and choose Crystal Reports. CIS displays the Crystal Reports Bill of Materials dialog box.

Crystal Reports Bill of Materials

Crystal Reports Template
C:\ORCAD\CAPTURE\SAMPLES\QTY_COST.RPT

QTY_COST   Browse ...

BENCH
ENG_BOM   Delete
PARTNUM
QTY_COST

Part Reference Options
◉ Standard
○ Standard- separate line per part
○ Compressed
Exclude Prefixes:

Print Options

Print ...   Print Preview   Export ...

Variant: United Kingdom

Close   Help
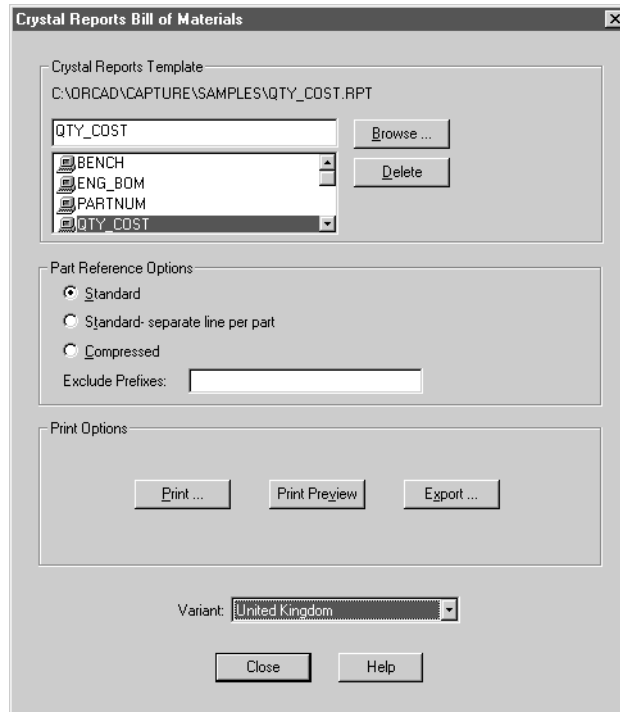
Sample Crystal Reports templates (.RPT files) are provided in ORCAD\CAPTURE\SAMPLES. These templates are designed to be used only with the BENCH.XLS and BENCH.MDB sample part databases. You must create your own Crystal Reports templates to work with your preferred parts database.

If you want to create more templates, see Creating Crystal Reports templates on page 2-54. You must have a copy of Crystal Reports software to create your own templates. Visit Orcad's web site at www.orcad.com for reseller information.

2   In the Crystal Report Template list, select a report name, or click the Browse button and locate another template.

3   Select one of the Part Reference Options from the following table:

To do this...	Select this option...
Group parts with matching keyed properties on a single line	Standard
Put each part on a separate line	Standard – Separate Line Per Part
Allow part ranges (such as R1–R14)	Compressed

4   In the Exclude Part Reference Prefixes field, specify part reference prefixes for parts you do not want to include in the report. Typically, you use this to omit parts that are not purchased for manufacture.

Note   *If your design has design variants, you will have the option of selecting the variants whose part information CIS will use to generate the report. See the CIS online help for more information about creating reports for design variants.*

5   Select an option from the following table:

To do this...	Click this button...	Then proceed to...
Print your report	Print	To print a Crystal Reports report below
Preview your report before printing or exporting	Print Preview	To preview a Crystal Reports report below
Export your report to a file, an MS Exchange folder, a Lotus Notes database, or MS Mail.	Export	To export a Crystal Reports report on page 4-113

111

CIS creates a report of the entire physical design regardless of the schematics and pages selected in the project manager.

Note *After you select one of the above options, CIS generates the report. This can take several minutes, depending on the number of parts in your design and the complexity of the Crystal Reports template. CIS checks for parts that are grouped (by having the same part number, for example), but that have different properties or packaging or have duplicate part references between instances of the parts.*

To print a Crystal Reports report

1   If you have not done so already, complete To select and setup a predefined Crystal Reports template on page 4-110.

2   When CIS displays Crystal Report's Print dialog box, select the print options you want and click OK.

Depending on the design of the Crystal Reports template, you might be required to input some information about your design (for example, part number or revision level) before you print.

To preview a Crystal Reports report

1   If you have not done so already, complete To select and setup a predefined Crystal Reports template on page 4-110.

2   When CIS displays Crystal Report's print preview window, you can browse through the report using the navigation buttons on the toolbar.

The toolbar also displays status information about the records in your report.

total to be
processed

Total:80   100%   80 of 80

percentage of
total processed

number
read

current page

1 of 7   64%

zoom setting

page backward
controls

page forward
controls
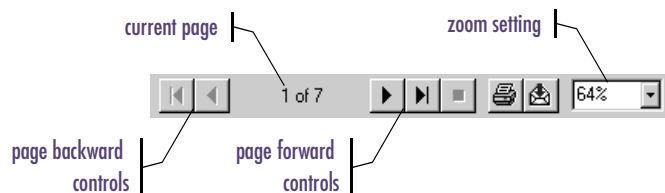
3   When you are ready to print or export your report, select one of the toolbar options from the following table:

To do this...	Click this button...	Then proceed to...
Print your report	🖨	To print a Crystal Reports report on page 4-112
Export your report to a file, an MS Exchange folder, a Lotus Notes database, or MS Mail.	📩	To export a Crystal Reports report below

To export a Crystal Reports report

1   If you have not done so already, complete To select and setup a predefined Crystal Reports template on page 4-110.

2   When CIS displays Crystal Report's Export dialog box, select the format in which you want to export the report. For example, if you want to convert the report to Microsoft Excel 4.0 format, choose Excel 4.0 (.XLS) from the list.

Export

Format:
Character-separated values

Destination:
Disk file

OK
Cancel

3   From the Destination list, select one of the following options:

To do this...	Select this...
Save the report to a disk file	Disk file
Attach the report to a MS Mail e-mail document	Microsoft Mail (MAPI)
Store the report in an MS Exchange folder	Exchange Folder
Attach the report to a Lotus cc:Mail e-mail document	Lotus cc:Mail (VIM)
Send the report to a Lotus Notes database	Lotus Notes Database

When you export a report to a disk file, CIS displays the Choose Export File dialog box. Do the following:

1   Select a path and file name for your file.

2   Click OK. The program exports the report to a disk file in the format you specify.

4   Click OK. The export process begins.

While the program assigns the native extension to all files you export in a specific word processor, database, or spreadsheet format, it automatically assigns the .TXT extension for all files you export in one of the common data interchange formats. However, the program you want to use the data in may require an extension other than .TXT. Consult the manual for that program to determine the correct file extension, and change the extension accordingly in the File Name text box.

Glossary

administrative preferences	Options that you can set to customize database configuration features (for example, allowing duplicate part numbers or assigning temporary part numbers automatically).
allowed part reference prefixes	The part reference prefixes that you assign to the data source tables of a part database during configuration. These assignments decrease search time when you are linking database parts by limiting the data source tables that CIS searches.
applicable part reference prefix	The part reference prefixes that you assign to the part types of a part database during configuration. These assignments decrease search time when you are linking database parts by limiting the part types that CIS searches.
bill of materials	See [BOM](#).
blank property	A database part property that does not have a value.
BOM	Bill of materials. A report that includes a set of properties for all the physical parts required to build a design.
browsable property	A property that is set in the configuration (.DBC) file to allow references to datasheets, drawings, and documents in your part database. Referenced file locations can be local or Internet URLs.
CIS	Component Information System. A part management system that helps you manage part properties within your schematic designs, including part information required at each step in the printed circuit board design process.
CIS explorer	The environment where you search for and retrieve a variety of part information for use in your schematic designs.

core design	A schematic design from which design variants can be created.
Crystal Reports	A widely-used report design software produced by Seagate Technology, Inc. The Crystal Reports Print, Print Preview, and Export options have been integrated into Orcad CIS to create customized bill of materials reports for printed circuit board designs.
data sheets	Documents published by component manufacturers that contain the specifications engineers need to select the correct parts for their printed circuit board designs.
data source	A data source consists of a database filename or server name and an associated ODBC driver with which to access the database.
database	A collection of information related to a particular subject or purpose, such as maintenance of a list of approved parts.
database configuration (.DBC) file	The file that enables CIS to access and use your part database. When you configure CIS, you can specify the database tables that CIS uses, the part properties that will be transferred to your designs, the visibility of transferred part properties, and the part type associations. This information is stored in the .DBC file.
database field name	See table property name.
database part	A part whose identifiers and properties are collected in a database.
database table	A subset of database information with a common identifier or property, such as capacitors. Tables organize data into columns (called fields) and rows (called records).
DBC	See database configuration (.DBC) file.
design errors	Errors in the electrical connectivity of your design. If CIS finds design errors while generating a bill of materials, a dialog box will prompt you to view the errors in the session log so that you can fix them and generate a valid bill of materials.

design variant   A variation of the core design of a project that you can use to create a bill of materials for a different version of the project. All variations are kept within a single design. For more information about design variants, see the CIS online help.

EE   Enterprise Edition. A previous release of the Orcad Enterprise CIS product offering. Enterprise Edition evolved from the DDL product originally developed by an Orcad Industry Partner called Q-Point. The software was packaged and shipped under the product names Orcad Capture EE and Orcad Express EE. The current release is Orcad Capture CIS with the option to include Express and PSpice.

Enterprise Edition   See EE.

ERP   Enterprise resource planning. See MRP.

footprint   See PCB footprint.

ICA   Internet component assistant. A tabbed window of the CIS Explorer that makes new components accessible to you over the Internet by providing access to the activeparts web site. Using the ICA, you can download component information to create temporary parts in your local preferred part database for immediate use in your schematic designs.

Activeparts web site   A web site that allows access to a part database and is itself accessible through the internet component assistant (ICA). When you locate a part at activeparts through the ICA, you can place the part directly onto a schematic page and add the part to your local preferred parts database (PPD).

ICA property name   A property that you select during database configuration that maps property names on new parts created with the internet component assistant (ICA) to corresponding database property names in your preferred part database.

ICA supplier preferences   Preferences that you set when configuring your database to specify the component suppliers you wish CIS to search for part pricing and availability and the order in which you want them searched.

instance   A part or a symbol that you have placed on a schematic page.

intelligent unit conversion	The method that the database query feature uses to interpret common magnitude identifiers in part definitions. For example, intelligent unit conversion recognizes 2.7K, 2,700, 2.700K, and 0.0027M to be equal.
internet component assistant	See ICA.
Internet part	A part stored in a web-based part database that is searchable over the Internet. Parts stored at the activeparts web site are Internet parts.
keyed property	A part property that CIS searches your preferred part database for when you are trying to link a placed part to a database part. You set which properties are keyed when you are configuring your database. Normally, you set only the Value property as a key so that CIS searches the database for parts with a specific value.
Link Database Part	The command you use to link a part you've placed on a schematic page with a part in your preferred parts database. Once these parts are linked, you can automatically update a placed part when the database part changes.
manufacturing resource planning	See MRP.
MIS	Management information services. The department responsible for creating and maintaining company-wide information systems.
MRP	Manufacturing resource planning. The computerized method for planning the use of a company's resources, including scheduling raw materials, vendors, and production equipment processes.
not present	See part not present.
occurrence	An instance placed on a schematic page with one or more property values that have been changed from the normal instance values.
ODBC	Open Database Connectivity. Microsoft technology that provides a common interface for accessing heterogeneous databases. Using ODBC, CIS interfaces directly with whatever database or spreadsheet you want to use to create and maintain your part database.
Open Database Connectivity	See ODBC.

package   A physical part that contains one or more logical parts. 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 whose part references are U15A, U15B, U15C, and U15D.

part   A part is a basic building block of a design. A part may represent a physical component, or it may represent a function, a simulation model, or a text description for use by an external application. A part's behavior is described by a SPICE model, an attached schematic folder, HDL statements, or other means. Parts usually correspond to physical objects–gates, connectors, and so on–that come in packages of one or more parts. Packages with more than one part are sometimes referred to as multiple-part packages. See also package.

part database   A collection of part information. See also database.

part description   A brief textual description of a part.

part instance   A part placed on a schematic page.

part manager   A CIS window that summarizes the status of all the parts in a design. Using the part manager, you can update part status, link database parts to placed parts, and create design variants.

part not present   A variant part setting indicating that a core schematic's part is not installed in the footprint for a design variant. You can set parts as present or not present when you create design variants.

part number   The number that uniquely identifies a part within the database.

part property   A property of a part. See also property.

part reference association   An association between a particular part type and the part reference prefixes used in the part database for that part type.

part reference prefix   A single letter designator for a part type. For example, C is the part reference prefix for the part type capacitor.

part status	The status of a placed part relative to the part database. Part status can be approved and current (green), in the approval process (yellow), or incorrect in the bill of materials (red). For a complete list of possible part statuses and their definitions, see the table on page 101.
part type	A group of parts that have similar electrical functionality, such as capacitors or resistors.
part type delimiter	The character that indicates a hierarchical level within a path in the database. You can choose your part type delimiter when you are configuring your database. By default, the delimiter is a backslash (\) character.
part type property contents	The part type as defined in your database's part type property (such as resistor, capacitor, or IC). When you configure your database, you can associate these part type properties with part reference prefixes.
PATH environment variable	The directory path that a Windows application uses to search for a file if the file is not in the application's installation directory or the current working directory.
PCB footprint	A description of the physical layout dimensions of pads for a component on a printed circuit board.
PDM	Product data management. See MRP.
Place Database Part	The command you use to place a part on a schematic page from your preferred parts database or the internet component assistant (ICA). Once you have placed a part, CIS can automatically update the placed part when the database part changes.
PPD	Preferred parts database. The database in which you store and maintain all the parts and part properties you use in your designs. The database parts you place on or link to your designs from the PPD remain linked to your PPD. This way, when you change a database part, CIS can automatically update the placed parts on your designs.
preferred parts database	See PPD.
present	See present part.
present part	A variant part with a core schematic's part installed in the footprint. You can set parts as present or not present when you create design variants.

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 .1uF and red.

property name   The name of a part property. You can use a different name for the same property in your database than for your placed parts. You define property names when you are configuring your database. See also property.

property type   The property type determines how CIS interprets the database property. Your database must include a property of type Part_Number in every table. Other examples include Normal, Schematic_Part, and Part_Type. See also property.

property visibility   The default visibility setting of a property when it is copied to from your database to a placed part. You set property visibility when you are configuring your database. See also visibility.

query   A question about the data stored in your database tables, or a request to perform an action on the data. Using CIS, you can query your preferred part database and the part database at the activeparts web site.

report   A report is an organized presentation of data. Using CIS, you can create standard bills of materials as well as more advanced reports using Crystal Reports templates.

schematic part   The name of a part as listed in a local Capture library (.OLB files) and the database parts window. The CIS explorer searches these libraries for schematic parts when you place, link, or derive database parts.

schematic part property   The part name property. This property is required to use the Place Database Part command. When entering a name for this property, you may include the library name, a backslash (\), and then the name of the part as listed in the Capture library.

source library   A Capture or Express library from which you can place a part.

source package   The name of a part as it appears in a Capture or Express library.

SQL	Structured query language. A language used in querying, updating, and managing relational databases.
structured query language	See SQL.
tab-delimited file	A file whose discrete elements of data (such as part property values) are separated by tabs. You can save CIS report files in tab-delimited format and then modify the information with a spreadsheet editor or word processor.
table property name	The name of a part property as it is defined in the part database. Also known as the *database field name*.
table property type	The data type for a part property. Most properties are type Text, but there may be other data types.
temporary part number	A part number that CIS automatically generates to track newly created parts. CIS enters the number into the part record in the database as well as in a special table named TMPPRTS.
temporary part number prefix	The prefix that CIS appends to temporary part numbers. You can set this prefix when you configure CIS.
TMPPRTS table	A table in your part database that CIS creates when you enable temporary part number tracking. CIS automatically monitors the TMPPRTS table and notifies you if a temporary part in the design has been promoted to an approved part.
transferable properties	Properties that CIS transfers from database parts to parts you place on your schematic. You can set the properties that CIS transfers to your designs during database configuration.
uniform resource locator	See URL.
Update Part Status	The command you use to update your design's placed part properties to reflect changes in the part database. When you update part status for your design, CIS detects property values that you have changed for part instances and allows you to retain the values for the schematic part or, when possible, choose to automatically update the values to match those in the database. CIS displays the updated part status in the part manager.

URL Uniform resource locator. Text used for identifying and addressing an item in a computer network (for example, `http://orcad.com`). In short, a URL provides location information so that your browser can find and display items from a remote network.

value The part value. Examples are 1.2K, 10.0uF, and 74ALS374. CIS supports the use of common magnitude identifiers (such as K and uF).

variant property A part property with a different set of values than the same part property in the core design. Variant properties are used to create design variants.

visibility The setting that determines whether a part property is visible on your schematic page. You can set the default visibility for part properties during database configuration. You can also override default visibility settings when you place parts and when you select placed parts on the schematic page.

Index