# 3

# Working with database parts

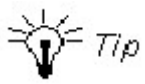
This chapter describes the various procedures you use when working with database parts, including the following:



## 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.

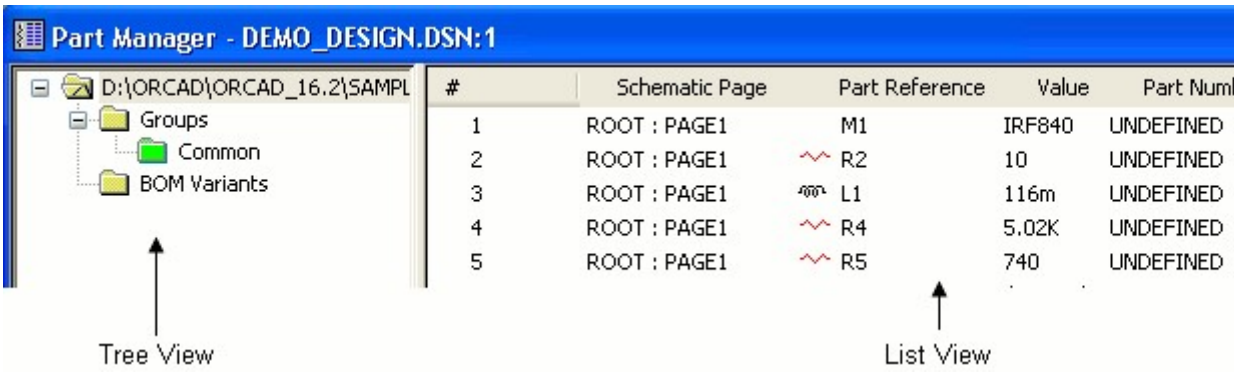
For descriptions of the functions of the windows in the CIS interface, see "The CIS work environment" .




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 part manager

## window




The part manager window is composed of a component property view in the right pane, and a tree view in the left pane. The tree view is used for creating groups and subgroups of components for bill of materials variants. If you are not working with design variants, you can close the tree view. For more information on using the tree view, see "Defining and Using Groups and Subgroups" .

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.

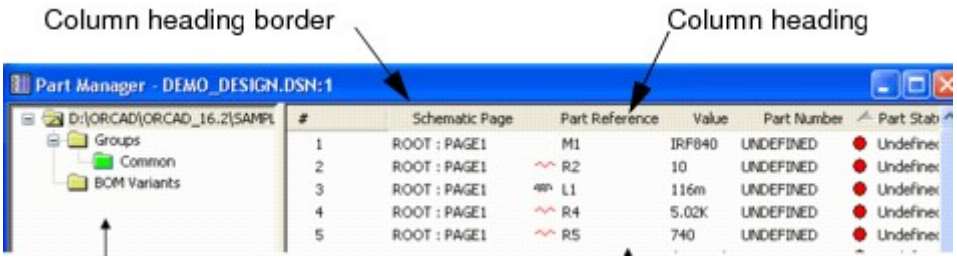
***To open the part manager window, do the following:***

→ 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" .

***To sort information in the part manager***

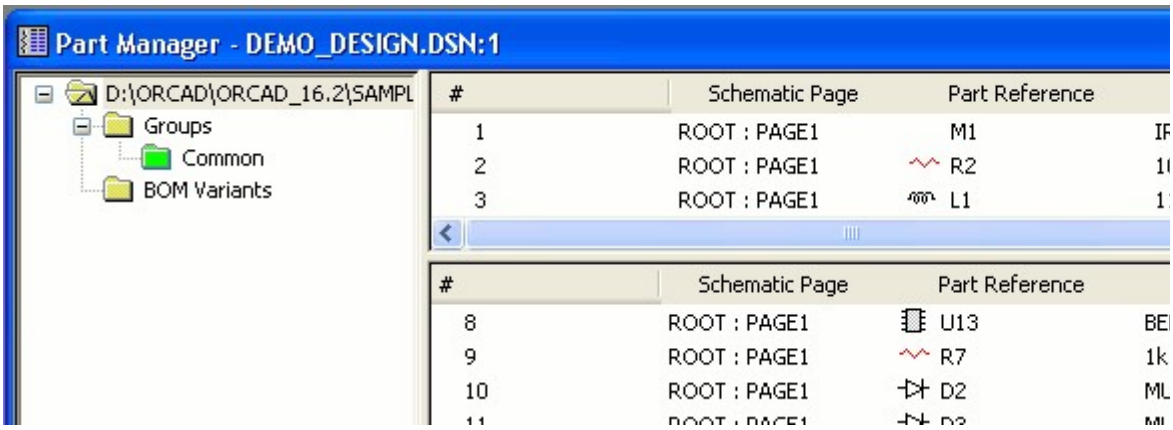
→ Click the heading of the column you want to sort by. To reverse the sort order, click the same column heading again.



.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.

***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.

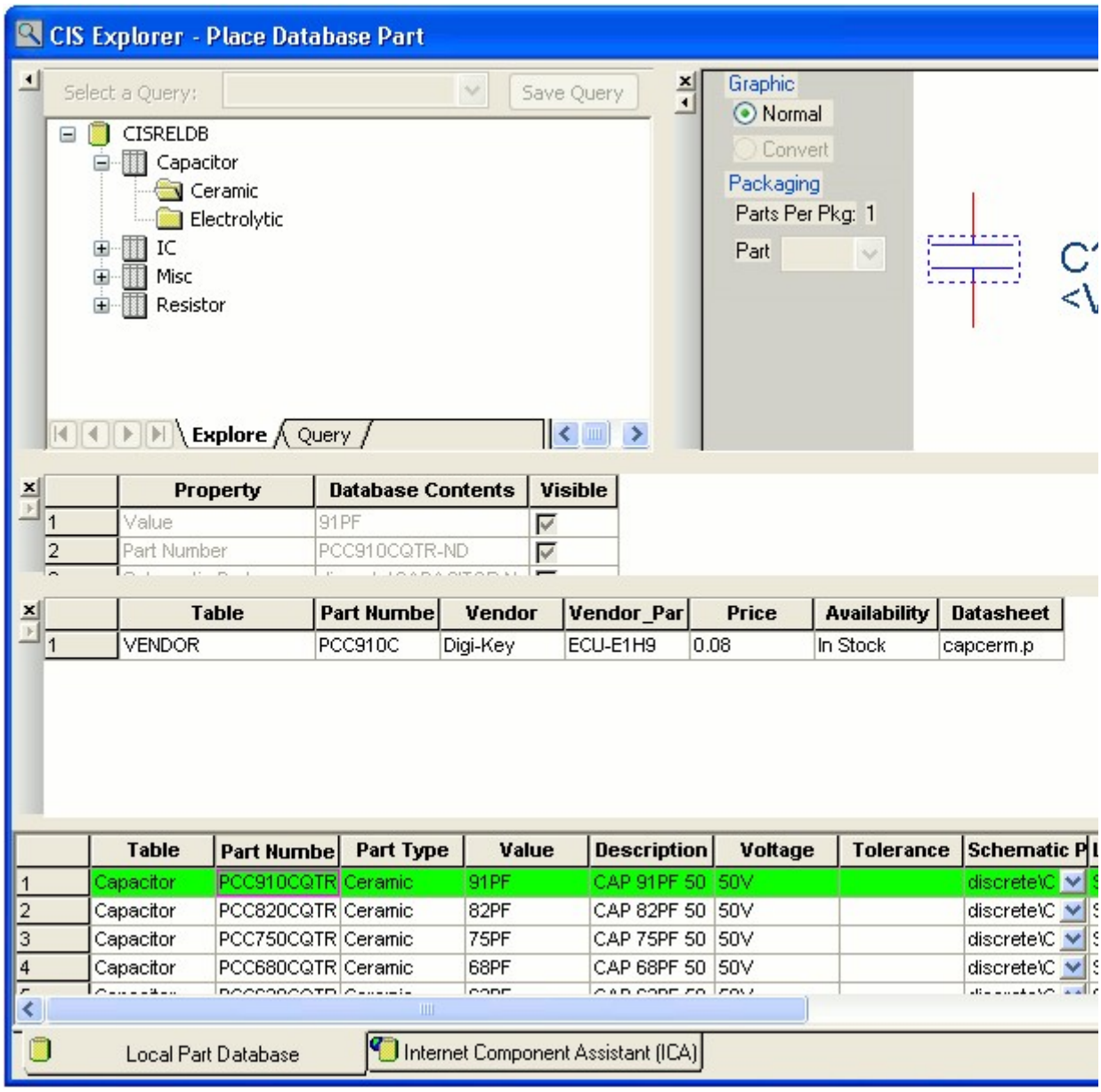



***To display the part manager toolbar***

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.
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:

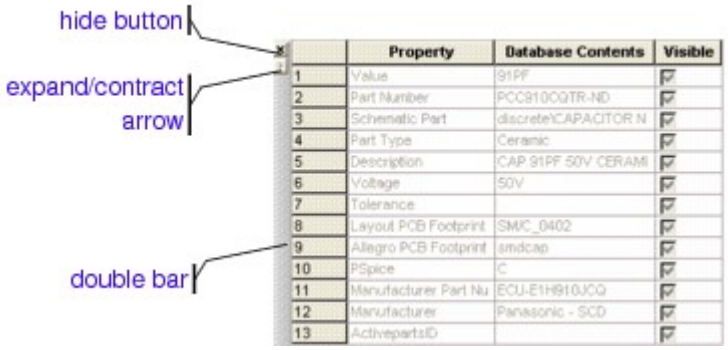
- Docking to the main window
- Expanding or contracting relative to other windows
- Floating over other docking windows

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

windows. 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



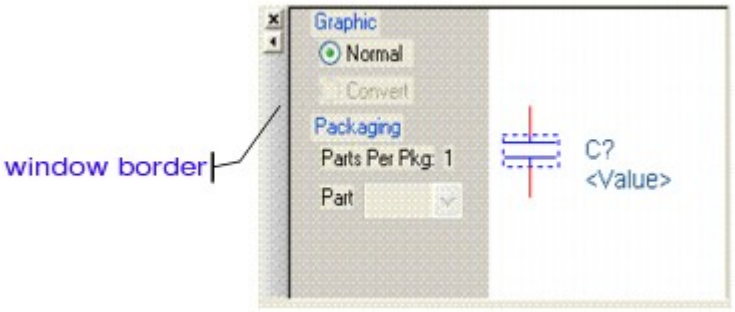

border.

***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.

***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**

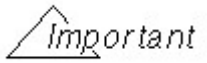
**window**

| | Table | Part Number | Part Type | Value | Description | Voltage | Manufacturer Part Number | Manufacturer | L |
|---|---|---|---|---|---|---|---|---|---|
| 1 | Capacitor | PCC910CQTR | Ceramic | 91PF | CAP 91PF 50 | 50V | ECU-E1H910J | Panasonic - S | S |
| 2 | Capacitor | PCC820CQTR | Ceramic | 82PF | CAP 82PF 50 | 50V | ECU-E1H820J | Panasonic - S | S |
| 3 | Capacitor | PCC750CQTR | Ceramic | 75PF | CAP 75PF 50 | 50V | ECU-E1H750J | Panasonic - S | S |
| 4 | Capacitor | PCC680CQTR | Ceramic | 68PF | CAP 68PF 50 | 50V | ECU-E1H680J | Panasonic - S | S |
| 5 | Capacitor | PCC620CQTR | Ceramic | 62PF | CAP 62PF 50 | 50V | ECU-E1H620J | Panasonic - S | S |

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

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




In the Japanese version of CIS on a Windows 98 SE machine, text in the database parts window appears garbled. To fix this problem, you need to install ddlinkx.dll and define a section in the Capture.ini file as shown below.

```
[CIS Spreadsheet]

Font = {required Japanese font}

Size = {size of the font}
```

**Note:** 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.

**Note:** Roll the mouse wheel up and down to scroll through vertically in the database parts window.

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

**Note:** Click the mouse wheel button and drag it to the right or left in the database parts window to scroll horizontally.

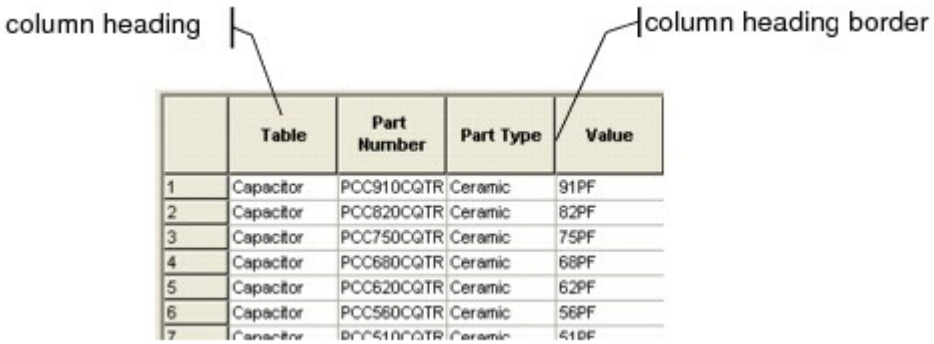
**Note:** Click the mouse wheel button and drag it up or down in the database parts window to scroll vertically.

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

***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.




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 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.

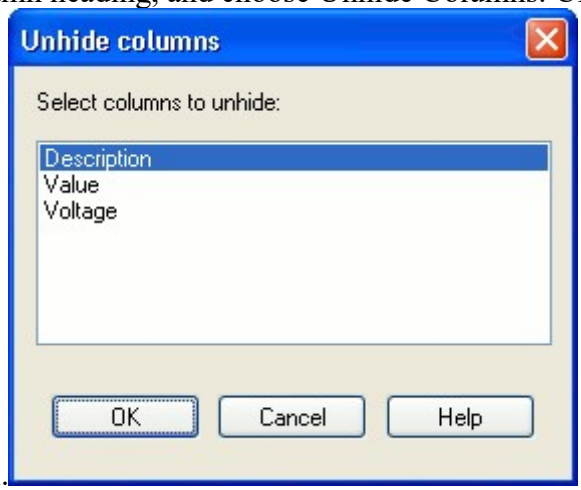
***To hide columns***

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.

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.

For information about locating and placing parts from the internet component assistant (ICA), see "Using the ICA to locate and create a new database part" .

## 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.

   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 abo adjusting the display of part information and property columns in the database parts windov see "Using the database parts window"

| | Table | Part Numbe | Part Type | Value | Description | Voltage | Tolerance | Schemat |
|---|---|---|---|---|---|---|---|---|
| 1 | Capacitor | PCC910CQTR | Ceramic | 91PF | CAP 91PF 50 | 50V | | discrete\( |
| 2 | Capacitor | PCC820CQTR | Ceramic | 82PF | CAP 82PF 50 | 50V | | discrete\( |
| 3 | Capacitor | PCC750CQTR | Ceramic | 75PF | CAP 75PF 50 | 50V | | discrete\( |
| 4 | Capacitor | PCC680CQTR | Ceramic | 68PF | CAP 68PF 50 | 50V | | discrete\( |

**Note:** CIS uses the part type property contents from your part database to determine the folder hierarchy.

**Note:** While placing a database part, if you get the error message "Could not read part information", then make sure that you check the CIS configuration settings and the .INI file.

## Using the query feature to locate database parts

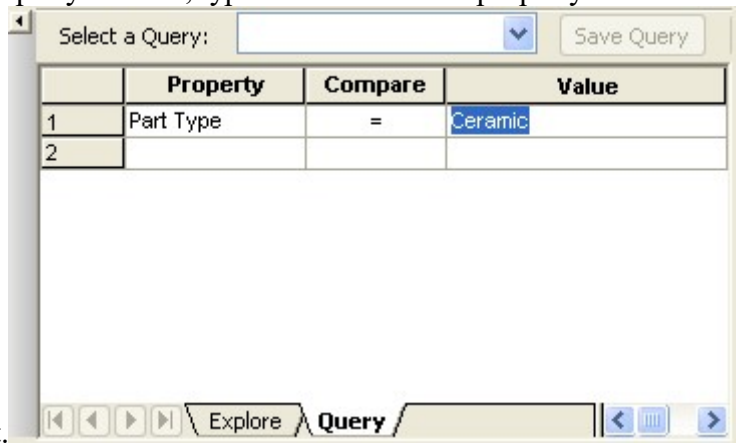
You can also use the query feature in the CIS explorer window to locate all parts in the database that fit a set of criteria. This is useful when you want to view all the available parts that match a set of attributes. To use the query feature, you must first create the query criteria, then execute the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

- Save and retrieve queries, see "Saving and retrieving a non-relational query"
- Modify saved queries, see "Modifying a saved query"
- Delete saved queries, see "Deleting a saved query"

### Creating and executing a non-relational query

Capture CIS allows you to create non-relational queries on a flat structure database.

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.




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

Repeat steps Step 3 through Step 5 as necessary to construct subsequent rows of search criteria. You can delete a query row by selecting the row and then pressing Delete. 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. To perform the query, press Enter or choose Re-search Database from the Update menu. CIS displays the results in the database parts window.

**Note:** If you make changes to column width or hide a column in Query view (Query tab), the same settings will not be retained when you change to the Explore view (Explore tab) and vice-versa.

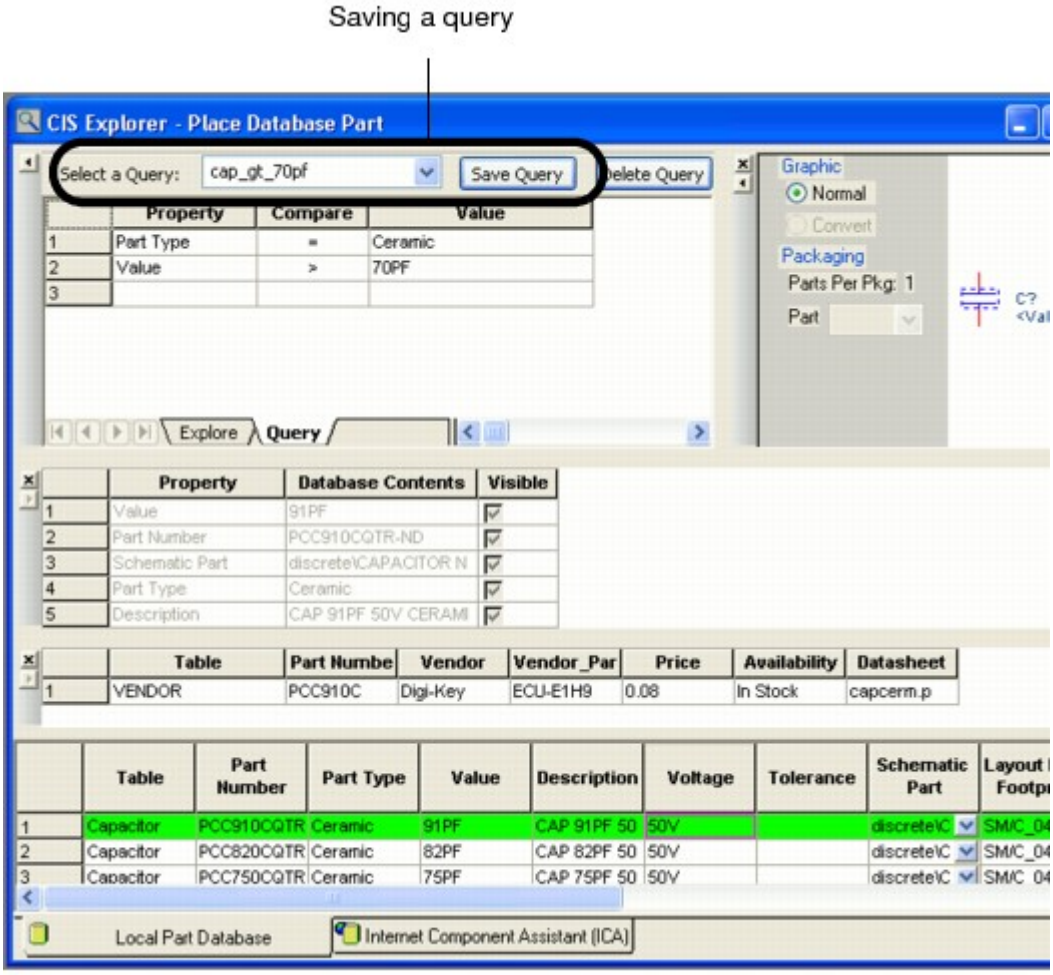
**Note:** You can save your query definitions. See "Saving and retrieving a non-relational query" for more information.

### Saving and retrieving a non-relational query

OrCAD Capture CIS enables you to save your queries and retrieve them in subsequent Capture CIS sessions.

***To save a query***

1. Create a query, see "Creating and executing a non-relational query"




.

2. Specify a name for the query in the Select a Query combo box.
3. Click Save Query. The query is saved and the query name appears in the Select a Query combo box.

**Note:** Capture CIS saves the query name along with the query definitions in a file called CISQuery.txt file. This file is created only when you save your first query. The file by default is saved in your local Windows installation directory, for example, WINNT (Windows 2000) and WINDOWS (Windows XP).

Caution ***Do not change the default location of the CISQuery.txt file. Otherwise, you will not be able to retrieve your saved queries. To know how to retrieve a saved query, see "To retrieve a saved query" .***

Caution ***Do not modify a query manually in the CISQuery.txt file. This may result in problems while retrieving the modified query later.***

**To retrieve a saved query**

In the Query tab of the Explorer window, select the query you want to retrieve from the Select a Query combo box. The query definitions for the selected query appear under the respective column headers in the Explorer window.

**Note:** You can also retrieve the query definitions of a saved query by entering the query name

in the Select a Query combo box and pressing Enter.

### Modifying a saved query

You can change the existing query definitions in a saved query and use it later.


***It is recommended that you modify the queries from the Save Query UI and not manually in the CISQuery.txt file.***

#### *To modify an existing saved query*

In the Query tab of the Explorer window, select the query you want to modify from the Select a Query combo box. The query definitions appear under the respective column headers in the Explorer window.

1. Change the query definitions as desired.
2. Click Save Query. The query is updated.

### Deleting a saved query

You can delete an existing saved query.

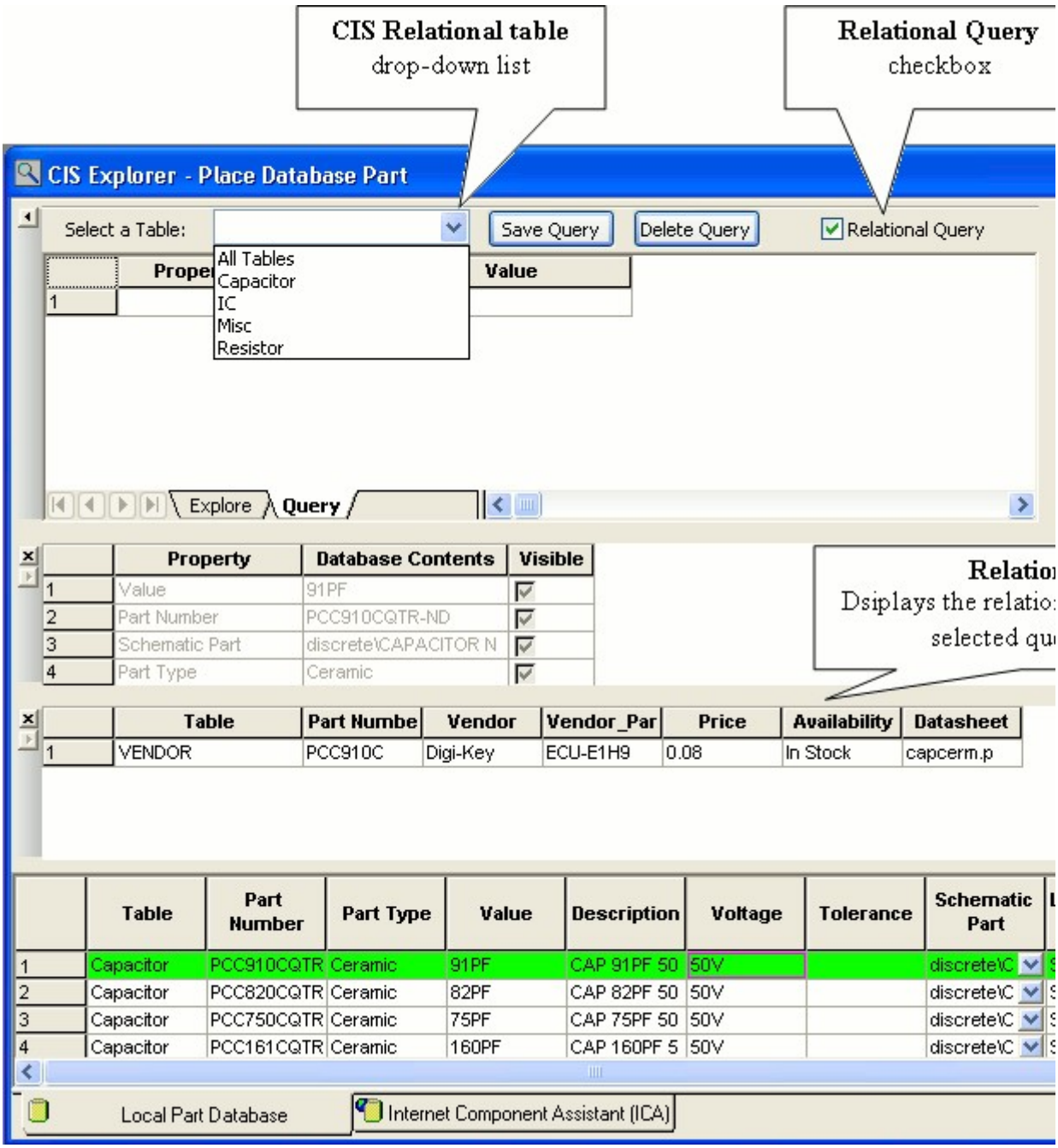
#### *To delete an existing saved query*

1. In the Query tab of the Explorer window, select the query you want to delete from the Select a Query combo box. The query definitions appear under the respective column headers in the Explorer window.
2. Click Delete Query. The query along with the query definitions are deleted from the CISQuery.txt file and the query name does not appear in the Select a Query combo box.

### Creating and executing a relational query

Capture CIS also allows you to create non-relational queries on a flat structure database. However, the relational view will be enabled in CIS explorer only if at least one relation is defined in CIS

configuration.




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. Select the Relational Query checkbox.
4. In the Select a Table drop-down list, choose a primary table to query.

**Note:** The drop-down list will show only the tables for which relational query has been defined.

**Note:** When you select particular table for running a query, all the properties of that table are available in the property tab to formulate a query. If you select "All Tables" the union of the properties of all the relational tables is available for forming the query.

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

6. In the Compare column, type in a logical operator or choose one from the cell's drop-down list.
7. 6. 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.

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

8. To execute the query, press Enter or choose Re-search Database from the Update menu. CIS displays the results in the database parts window.

## 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" .

**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.

**Note:** 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 Capture User's Guide for more information.

### Setting path for custom libraries

Before placing database components, you should make sure that path for all the libraries including custom libraries, is initialized. There are three ways to initialize a library path:

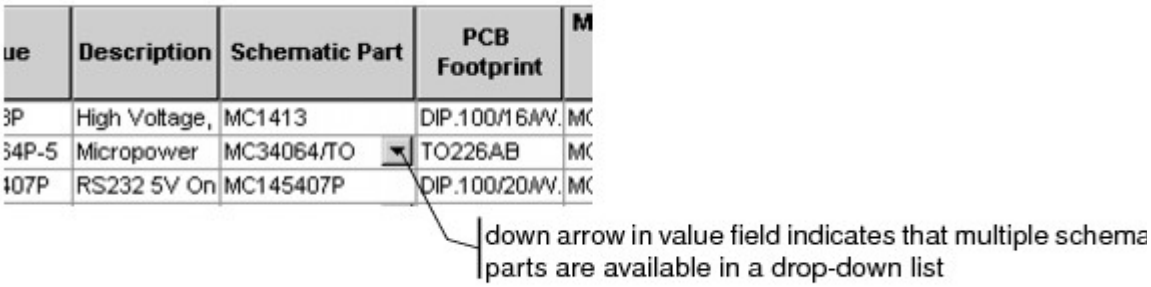
- Add a library from the custom library path in the Place Part dialog box and restart Capture. When you restart Capture, the newly modified Capture.ini file will be read.
- Manually add path of the custom libraries to the Capture.ini file before opening Capture.
- Use the OrCAD INI File Administrator utility to define the paths for custom libraries.

For more information, see "OrCAD Capture CIS Starter Database Kit" .

***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.




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.
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.

For more information about placing and linking database parts, see "Placing a database part on a schematic page" and "Linking a placed part to a database part" .

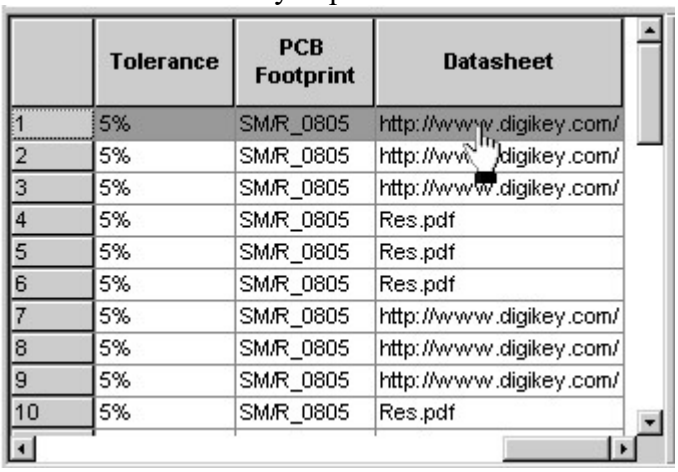
# Browsing part properties

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.

For information about configuring CIS, see "Creating a configuration file" .

***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.



| | Tolerance | PCB Footprint | Datasheet |
|---|---|---|---|
| 1 | 5% | SMR_0805 | http://www.digikey.com/ |
| 2 | 5% | SMR_0805 | http://www.digikey.com/ |
| 3 | 5% | SMR_0805 | http://www.digikey.com/ |
| 4 | 5% | SMR_0805 | Res.pdf |
| 5 | 5% | SMR_0805 | Res.pdf |
| 6 | 5% | SMR_0805 | Res.pdf |
| 7 | 5% | SMR_0805 | http://www.digikey.com/ |
| 8 | 5% | SMR_0805 | http://www.digikey.com/ |
| 9 | 5% | SMR_0805 | http://www.digikey.com/ |
| 10 | 5% | SMR_0805 | Res.pdf |

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.

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 "Viewing and updating part status" .

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

- Derive a new database part from your local part database. See Deriving a new database part from the local part database below.
- Derive a new database part from a part placed on a schematic page. See "Deriving a new database part from a placed part" .

- Locate and create a new database part using the internet component assistant (ICA). See "Using the ICA to locate and create a new database part" .

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

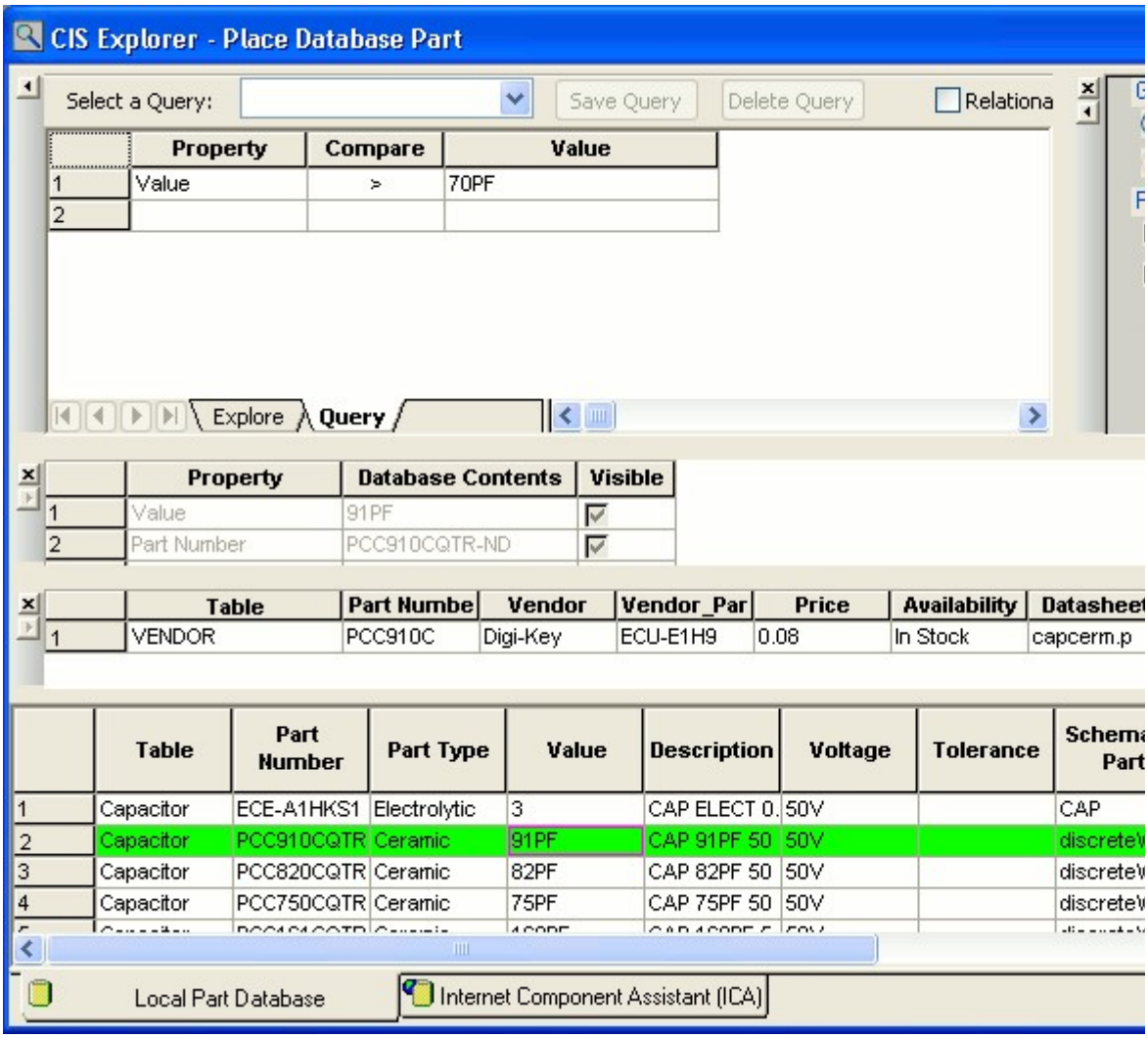
## Deriving a new database part from the local part database

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***

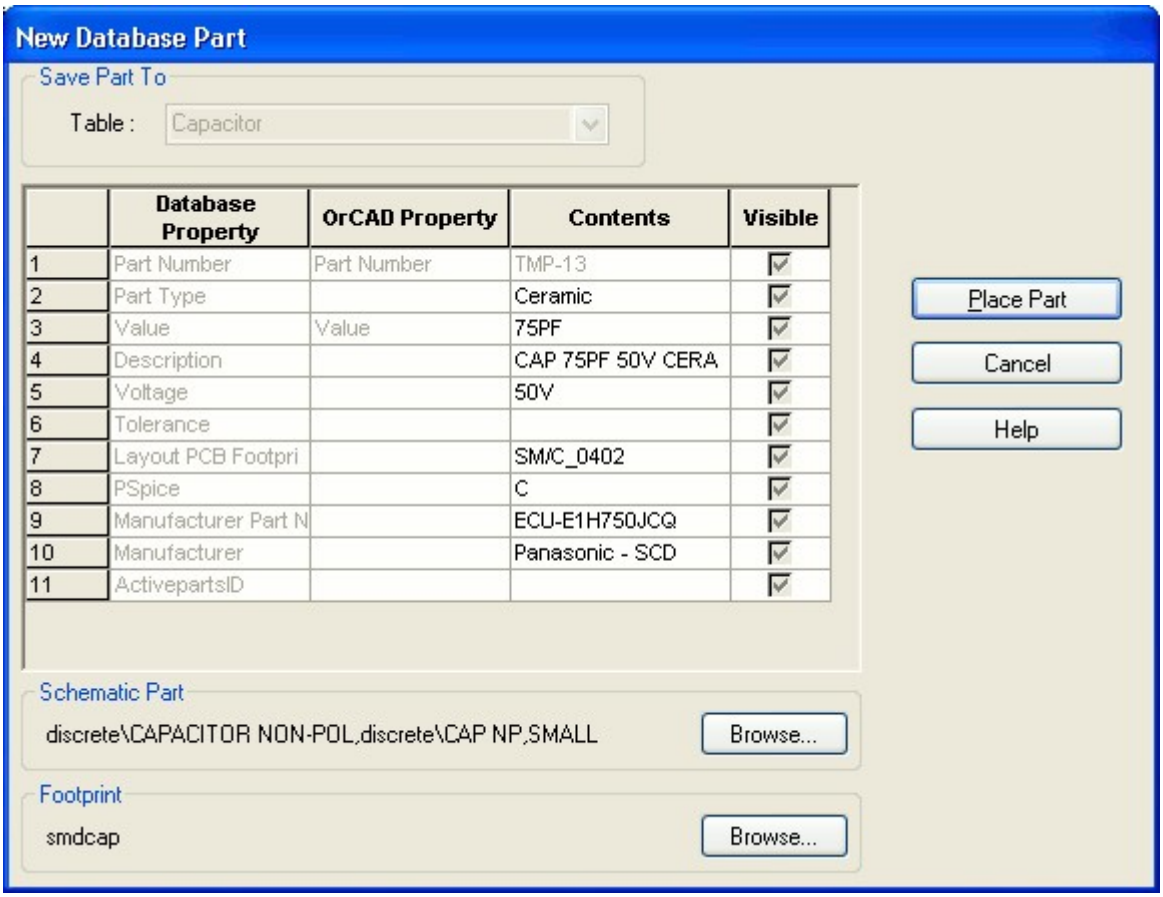
1. If a schematic page is not already open and active, open a schematic page.
2. From the Place menu, choose Database Part. CIS displays the CIS

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. For more information about locating parts in your local part database, see "Placing a database part on a schematic page" .
4. Select the desired part in the database parts window.
5. From the Edit menu, choose Derive New Database Part. CIS displays the New Database Part dialog

box.



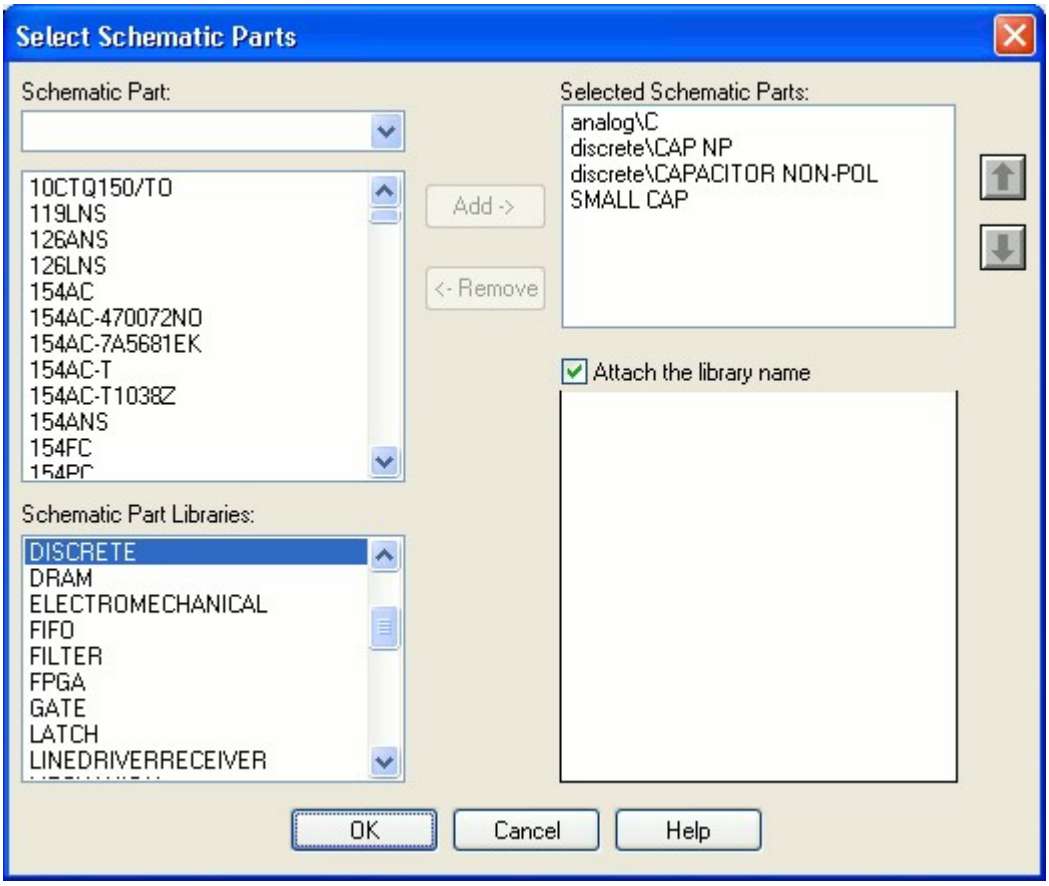

**6.** Enter the appropriate values for the new part's database properties in the Contents column. You cannot change the selected table in the Save Part To area. This is because you are

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.

**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 figure

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.


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.


***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***

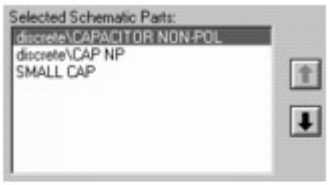
***changed by upgrades to new versions of Capture. If the name or path of the schematic part you select is changed during an upgrade, it will become invalid for all your database parts.***


***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.***

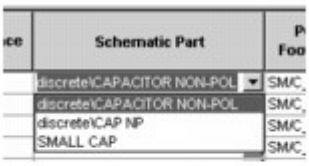
**d.** Click the Add button to move the schematic part or footprint to the Selected Schematic Parts or Selected Footprints list. You can remove it from the list by selecting it and clicking the Remove button.

**e.** Repeat steps Step c and Step 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.

**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.

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

**10.** Click to place the part. Press Esc when you are 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.

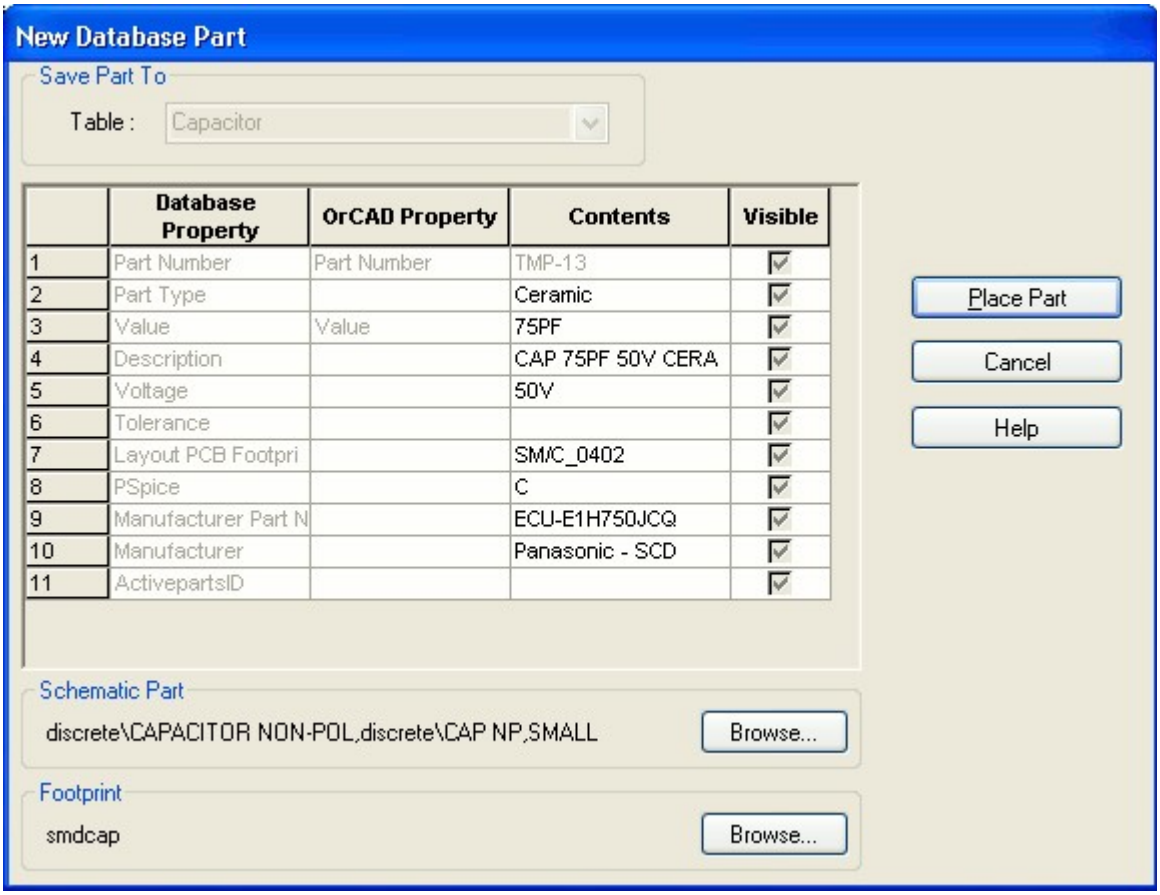
If you want other property values to be added to the database, you can manually add them during this procedure, but only if the property is mapped in the database configuration.

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.




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.

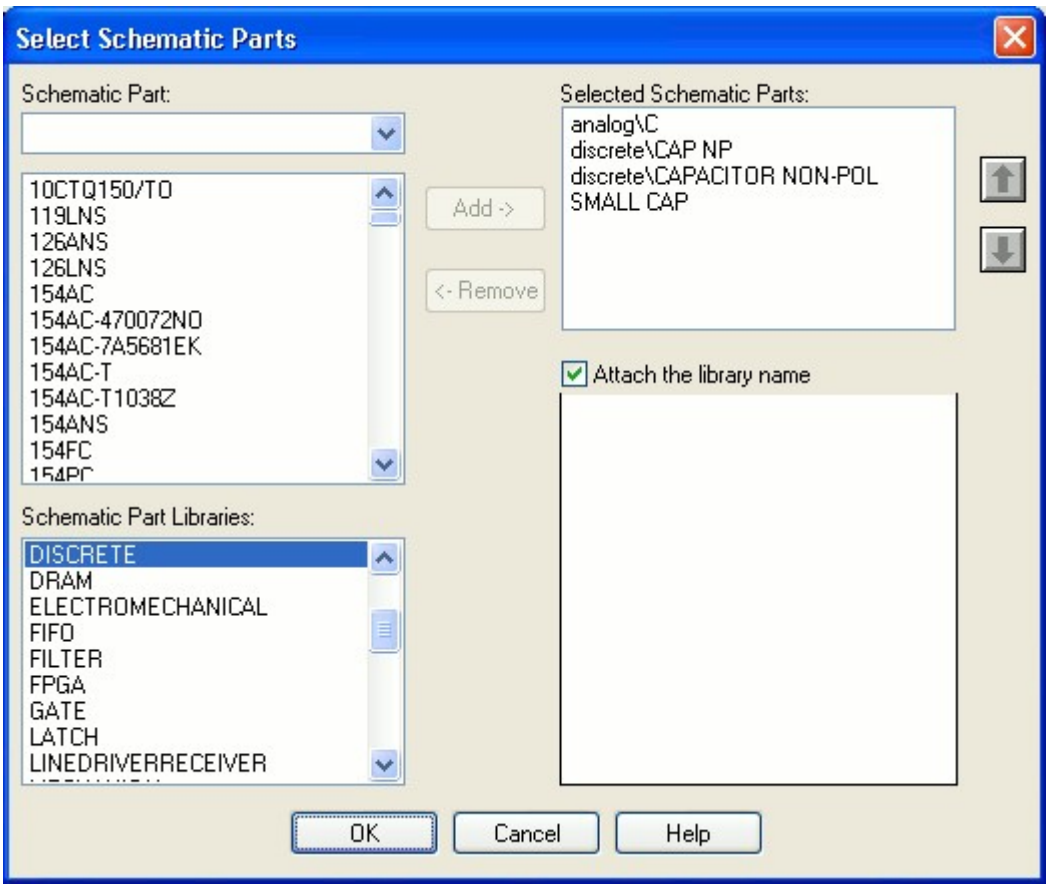
   **Note:** If you have selected the Assign Temporary Part Numbers Automatically option in the configuration, a temporary part number is already assigned which you cannot modify.

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

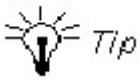
box.




**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.




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.


***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.***

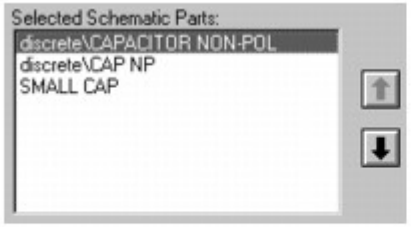

***If you are deriving a new mechanical (non-electrical) part and choosing an alternate schematic part, you must choose a schematic part with no pins. If mechanical parts with pins are placed in a design, they will invalidate netlists generated from that design.***

**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.

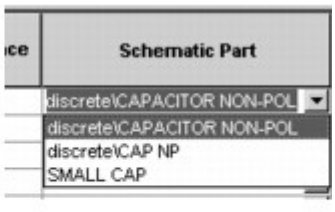
**e.** Repeat steps Step c and Step d until you have selected all the schematic parts or footprints that you want to be available for use with the database part.

**f.** If you selected more than one schematic part or footprint, use the up and down arrow order of the parts or footprints. The top one will be the default value for the database the whole set determines the order in which they will display in the scroll-down list c 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 Schema property value cell in CIS explorer's 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 use the ICA to access the ActiveParts or supplyframe portal 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 Using the ICA to locate a part below.
2. Create and place a part using the ICA. See "Using the ICA to locate and create a new database part" .

### Using the ICA to locate a part

Through the ICA, you can locate new parts using the part database at the Activeparts or supplyframe portals.

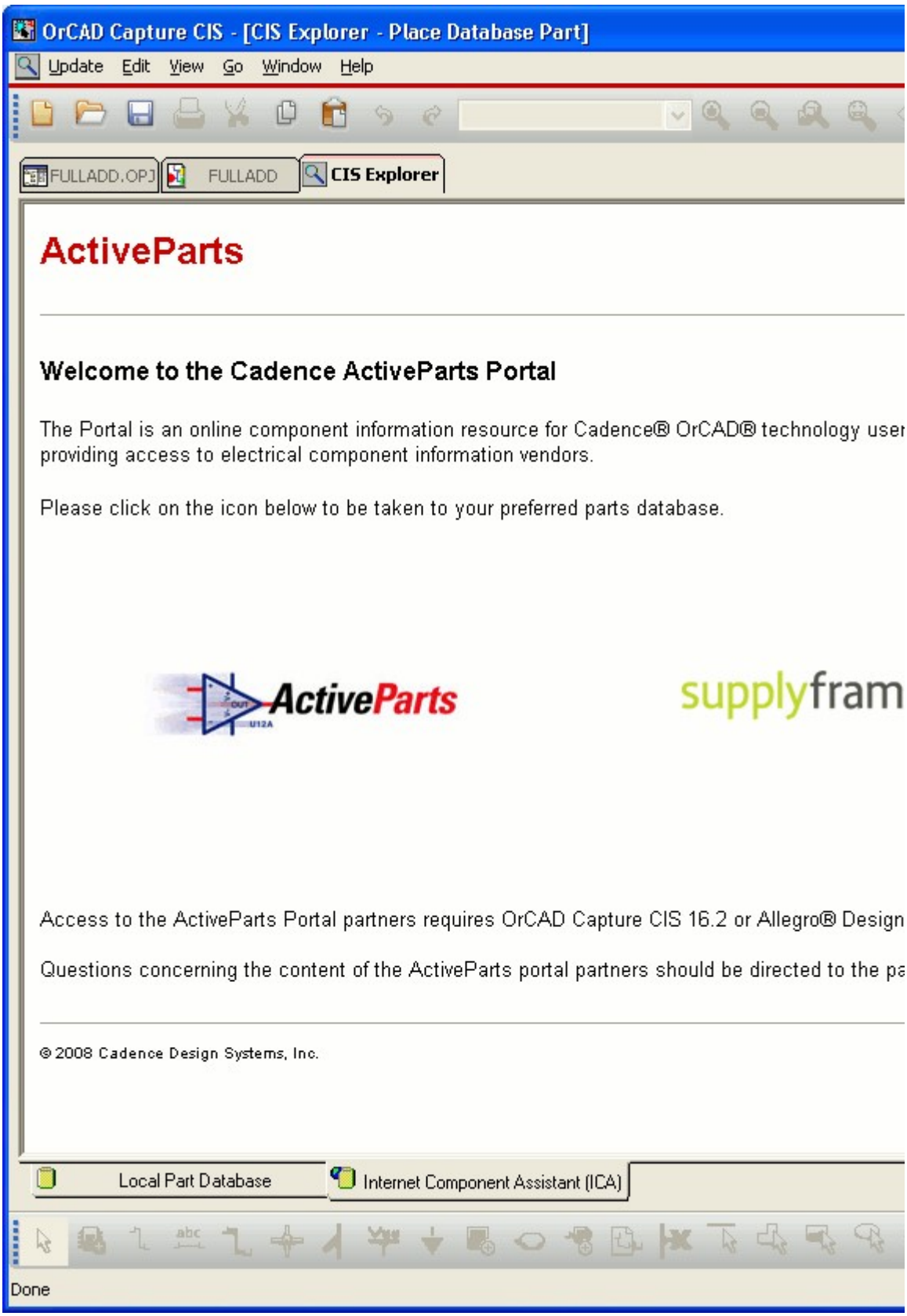
**Note:** These databases are updated as new part information becomes available without requiring you to update your CIS software installation.

***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.
3. Click the Internet Component Assistant (ICA) tab at the bottom of CIS explorer window. CIS displays ICA home page.

This page displays the two ICA portals (ActiveParts & supplyframe) that are

available.




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

1. Click either of these part's portal to search for the part of choice.Then use the 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 search features, refer to the respective online help of these two portals.

### 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 or supplyframe portals 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 and create a new database part" .
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:
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.

   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:

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

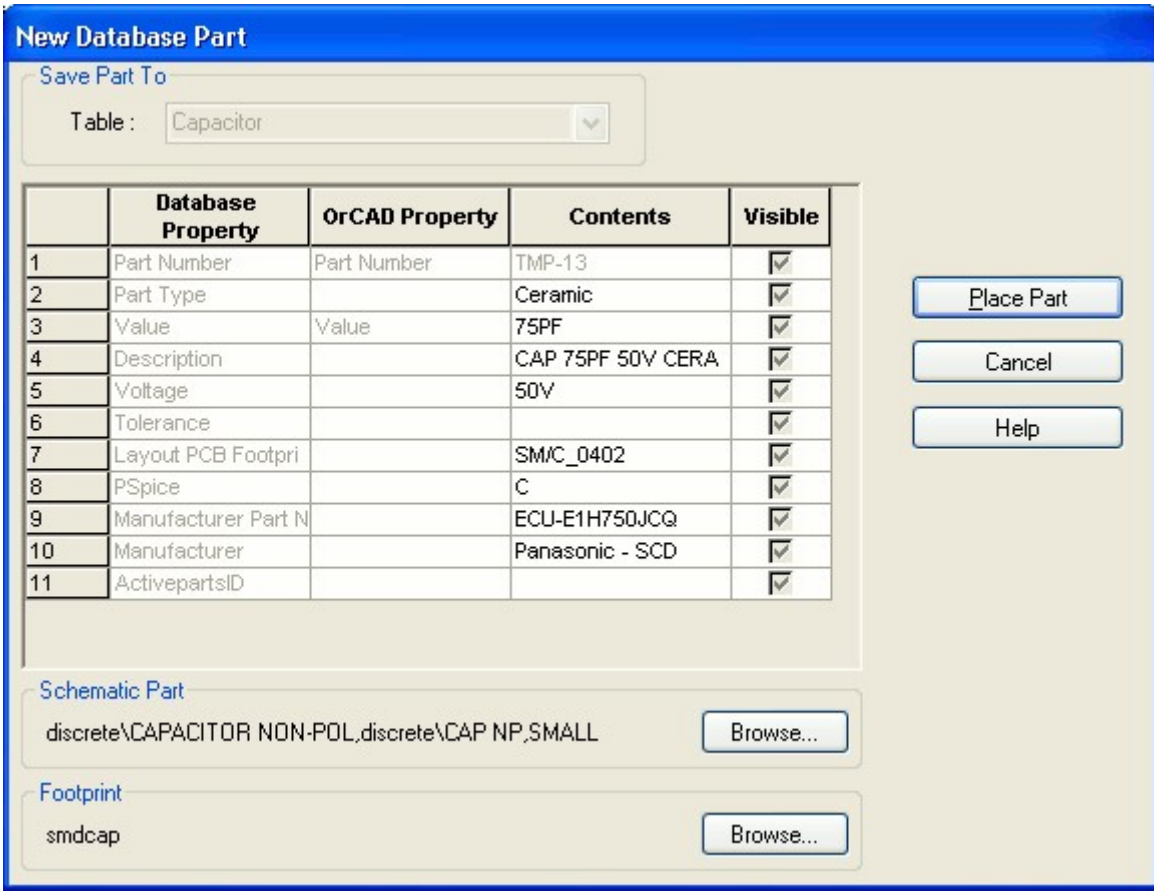
   **b.** In the Place Part dialog box, click the Add Library button.

   **c.** Browse for and select the new library then click Open.

   **d.** Click Cancel.


***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.

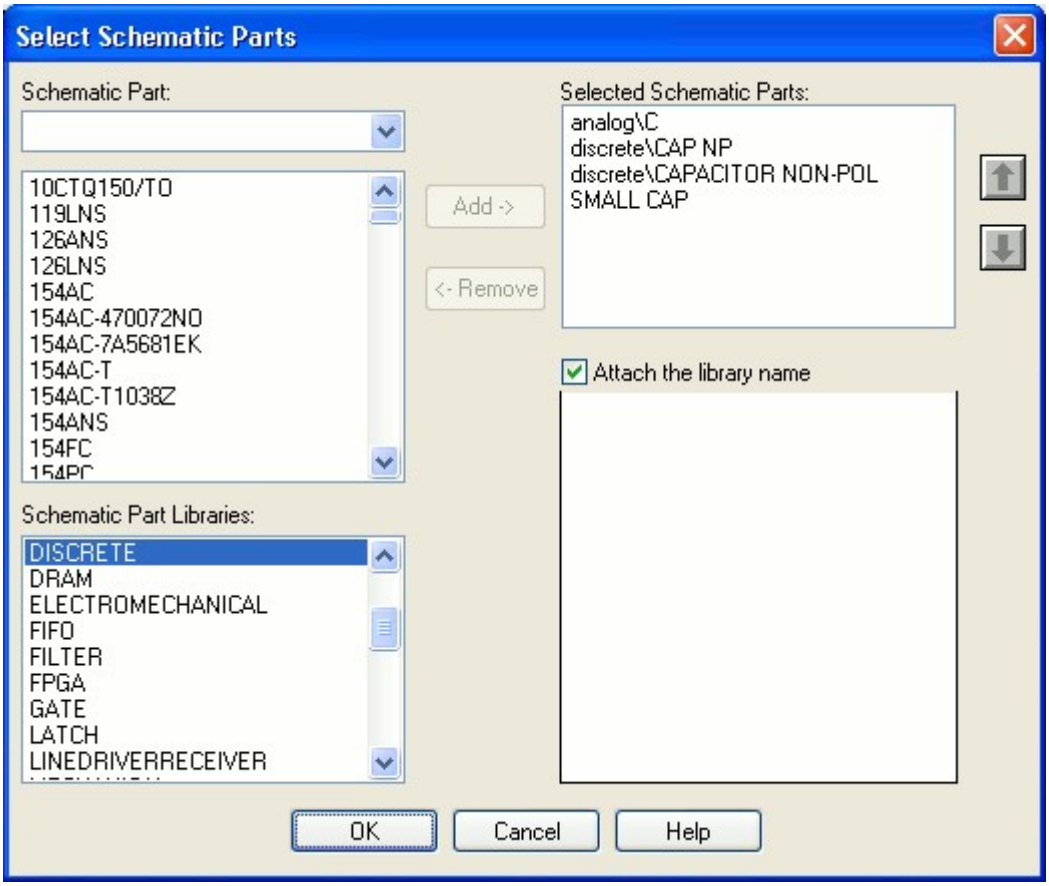



5. Enter the appropriate values for the new part's properties in the Contents cells.

   **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.


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.

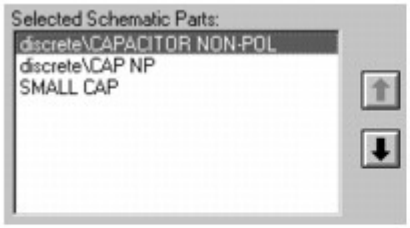

***Only select a schematic part from your own custom libraries; do not select a part from the libraries that are supplied with Capture (resource libraries). Resource libraries are often changed by upgrades to new versions of Capture. If the name or path of the schematic part you select is changed during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


***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.***

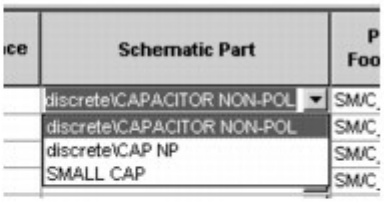
**d.** Click the Add button to move the schematic part or footprint to the Selected Schematic Parts or Selected Footprints list. You can remove it from the list by selecting it and clicking the Remove button.

**e.** Repeat steps Step c and Step 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 figure




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

below).

**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.

**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.

**Note:** If you chose to add the new part to the database in step 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.

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

# Linking a placed part to a database part

You can use the Link Database Part command from the part manager or the schematic page editor to assign a database part (and the associated transferable properties) to one or more placed parts. You can also specify whether you want to:

- replace the symbol, reference designator, and all the properties of the placed part with that of the database part.
- replace the symbol and all the other properties of the placed part with that of the database part, but retain the placed part reference designator.
- replace all the properties of the placed part with that of the database part, but retain the placed part symbol and the reference designator. For information on how to specify these options, see "Setting linking preferences" .

OrCAD Capture CIS allows you to link selected part occurrence(s) or a part with multiple occurrences to a database part. See "Linking part occurrence(s) to a database part" for details.

## Linking External Parts

You can link external parts using the Link Database Part command only from the Parts Manager.

## Setting linking preferences

You can set preferences for linking a placed part to a database part by using the Extended Linking command.

***To set link preferences***

1. From the Capture CIS window, select the Options menu, point to CIS Preferences and choose the Extended CIS Linking command or press SHIFT+E shortcut keys. The Extended CIS Linking dialog box appears.


**Note:** The Extended CIS Linking command is also accessible from the project manager and the schematic page editor.

2. Use the CIS Linking dialog box to specify the following options:

| To... | Select... |
|---|---|
| Retain the symbol and reference designator of the placed part and replace all the properties of the placed part with that of the database part | Preserve Reference Designator and Preserve Symbol check boxes. |
| Retain the placed part symbol, but replace all the other properties of the placed part with that of the database part. | Preserve Symbol check box only. |
| **Note:** When you select the Preserve Symbol check box, the Preserve Reference Designator ch box is selected automatically. | |
| Replace the symbol and all the other properties of the placed part with that of the database part, but retain the placed part reference designator. | Preserve Reference Designator check box on |

**Note:** The default behavior of OrCAD Capture CIS is to replace the symbol, reference designator, and all the properties of the selected placed part with that of the database part.

3. Click OK to save the settings. The settings are saved in the CAPTURE.INI file. The next time you link a placed part to a database part these settings will be used.

**Note:** The link preferences can be changed any number of times during a Capture CIS session using the Extended CIS Linking command.


***Do not select the Preserve Reference Designator check box, if you are linking a placed part to a database part that differ in the number of components. Otherwise, the reference designators for the placed part will not be preserved after the linking is done.***


It is recommended that you set the linking preferences before you use the Link Database Part command to link a placed part to a database part.

***To link a placed part to a database part 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 Control and click the parts.
   - For a group of adjacent parts, hold down Shift and click the first then the last part in the range.


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.

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.

   **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.

   **Note:** CIS displays a warning message, 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.

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

   ***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.***

   **Note:** If you have defined part reference prefixes in the Allowed Part Reference Prefix text box and you link a placed part in your design to a database part with the Preserve Reference Designator check box checked in the CIS Extended Linking dialog box, the reference designator of the placed part is retained and all the transferable properties of the database part are transferred to the placed part. Now, when you update the part status from the part manager (using Update All Part Status command), the part status column displays: Approved: Part not found. This is because CIS is unable to find the part (with matching part reference prefix and properties) in the database table. You may avoid this situation by changing the reference designator of the placed part to the one you defined in the Allowed Part Reference Prefix text box or delete the part reference prefix from the Allowed Part Reference Prefix text box.

   **Note:** CIS, by default, checks only for the symbol name while linking a placed part to a database part. If you want CIS to check for the symbol name, the complete path of the library, and the date/time stamp of the symbol during the linking process, add the following entry under [Symbol Libraries] section in CAPTURE.INI file:

   ```
   [Symbol Libraries]
   ```

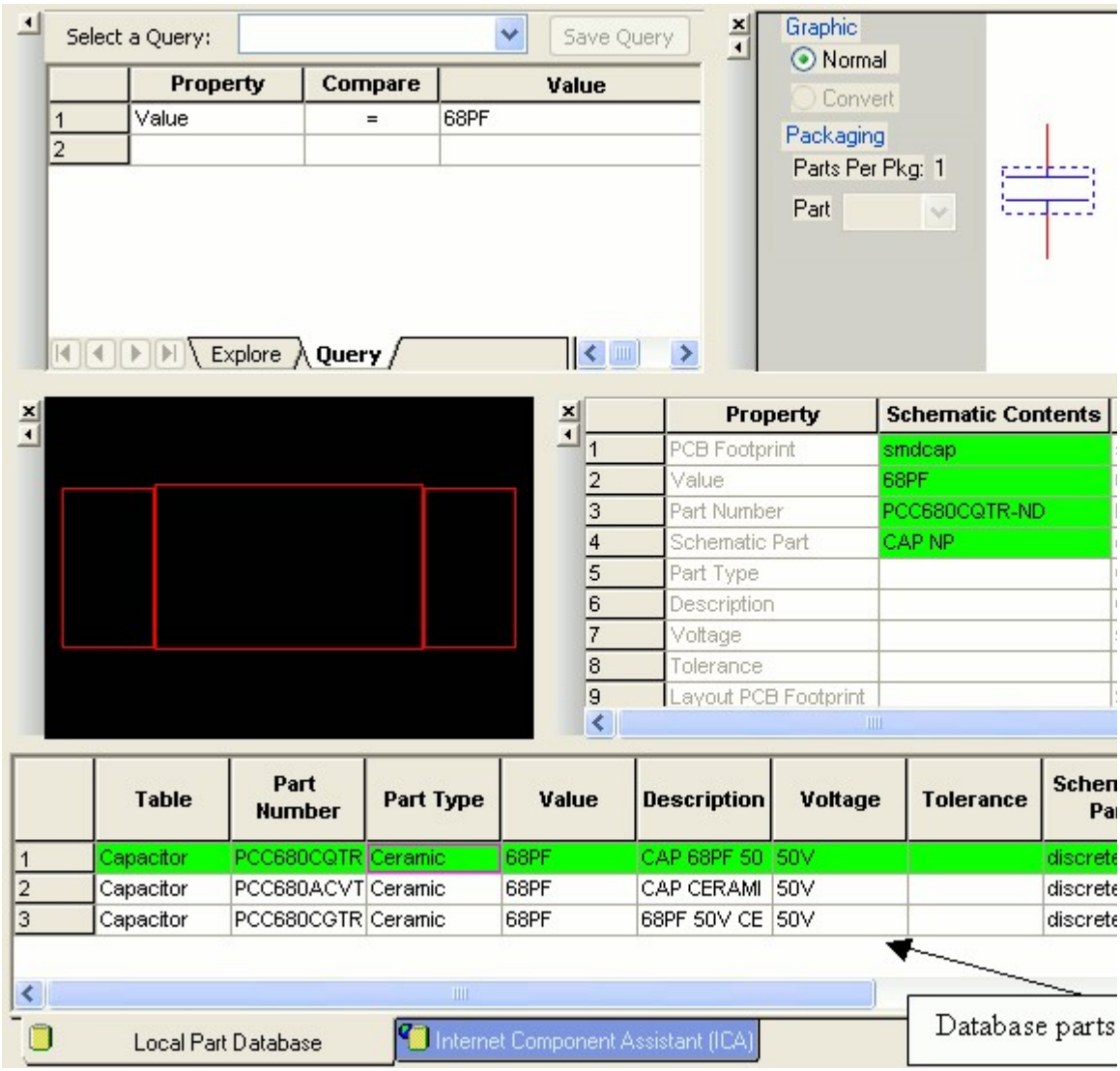
```
Strict Cache Check=1
```

**Note:** While linking, if a database part is a package alias of placed part, then the schematic part property of the database part will not transferred to the placed part on schematic. This is because the tool does not differentiate between a package and its alias.

***To link a placed part to a database part 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 for the database, then displays the results of the query in the database parts

window.




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. 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

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.

**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.

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.


***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.***

**Note:** CIS, by default, checks only for the symbol name while linking a placed part to a database part. If you want CIS to check for the symbol name, the complete path of the library, and the date/time stamp of the symbol during the linking process, add the following entry under [Symbol Libraries] section in CAPTURE.INI file:

```
[Symbol Libraries]

Strict Cache Check=1
```

**Note:** While linking, if a database part is a package alias of placed part, then the schematic part property of the database part will not transferred to the placed part on schematic. This is because the tool does not differentiate between a package and its alias.

## Linking part occurrence(s) to a database part

You can select whether you want to link all the occurrences of a selected part (s) or only the selected occurrence (s) of a part to a database part.

***To link part occurrence(s) to a database part***

1. Select one or more parts on a schematic page or the part manager.
2. Right-click on the selection and choose Link Database Part from the pop-up menu.
3. If the selected part (s) has:
   - only a single occurrence in the design, then the part database explorer appears where you can select the database part to be linked to the selected part (s).
   - more than one occurrence in the design, then the Occurrence Level Settings dialog box appears.


**Note:** The Occurrence Level Settings dialog box is not displayed in the following situations, where the selected part(s) has only a single occurrence in design or the Do not show this dialog again check box is selected.

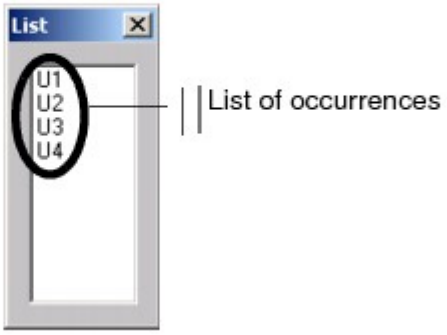
4. Select Link to Selected Occurrences Only check box to link the database part to the part (s) you have selected on the schematic page or the part manager.

   or

   Select Link all Occurrences check box to link the database part to all the occurrences of the selected part (s) on a schematic page.

   **Note:** CIS does not allow you to link a database part to part(s) in a group, which have multiple

   **Note:** If you want to view all the occurrences for the selected part(s) in a design, click the List button. This is a view-only list.

   occurrences .

5. Click OK. The part database explorer appears where you can select the database part to be linked to the selected part (s). The settings are saved in the CAPTURE.INI file.

   If you select the Do not show this dialog again check box, the Occurrence Level Settings dialog box will not be displayed the next time you link a part (s), which has multiple occurrences in a design. To display this dialog box again, you must change the following entry under the [Link Settings] section in the CAPTURE.INI file:

   ```
   [Link Settings]

   Show Occurrence Link Dialog=FALSE
   ```

   to

**Note:** Show Occurrence Link Dialog=TRUECIS, by default, checks only for the symbol name while linking a placed part to a database part. If you want CIS to check for the symbol name, the complete path of the library, and the date/time stamp of the symbol during the linking process, add the following entry under [Symbol Libraries] section in CAPTURE.INI file:

```
[Symbol Libraries]

Strict Cache Check=1
```

**Note:** While linking, if a database part is a package alias of placed part, then the schematic part property of the database part will not transferred to the placed part on schematic. This is because the tool does not differentiate between a package and its alias.


***If you choose to link the database part to the selected occurrence without preserving the symbols, then all the occurrences in the design will be updated with the database part symbol.***

# 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.

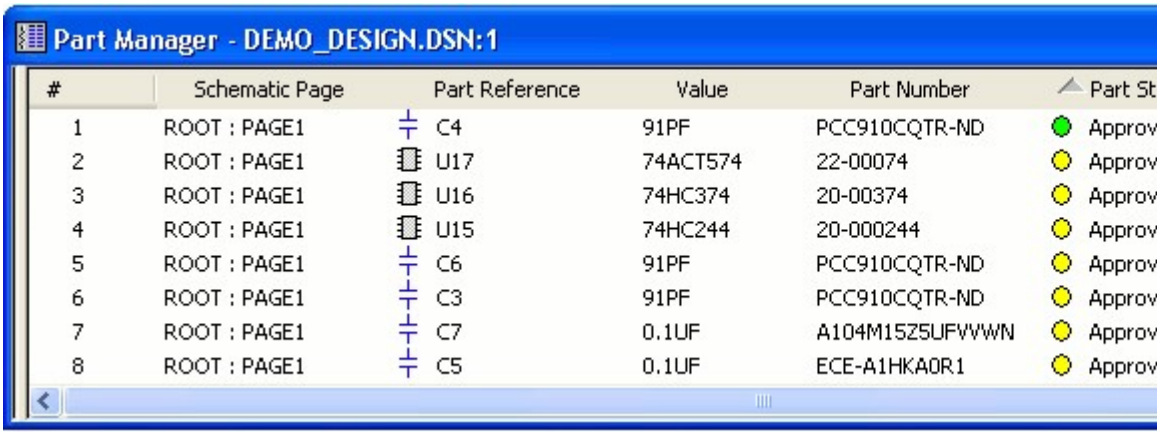
# Copying part properties using the part manager

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 dis design, sorted by part

reference.



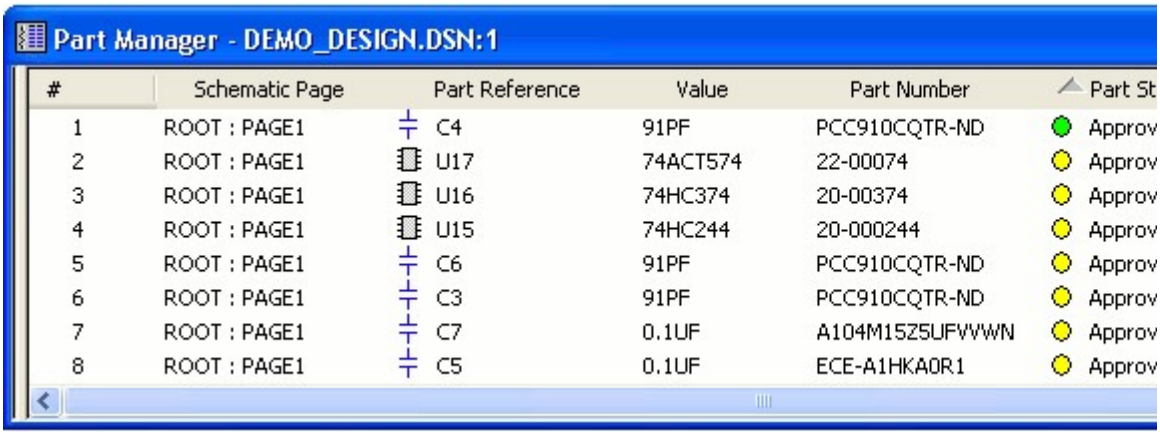

| # | Schematic Page | Part Reference | Value | Part Number | Part St |
|---|---|---|---|---|---|
| 1 | ROOT : PAGE1 | C4 | 91PF | PCC910CQTR-ND | Approv |
| 2 | ROOT : PAGE1 | U17 | 74ACT574 | 22-00074 | Approv |
| 3 | ROOT : PAGE1 | U16 | 74HC374 | 20-00374 | Approv |
| 4 | ROOT : PAGE1 | U15 | 74HC244 | 20-000244 | Approv |
| 5 | ROOT : PAGE1 | C6 | 91PF | PCC910CQTR-ND | Approv |
| 6 | ROOT : PAGE1 | C3 | 91PF | PCC910CQTR-ND | Approv |
| 7 | ROOT : PAGE1 | C7 | 0.1UF | A104M15Z5UFVVWN | Approv |
| 8 | ROOT : PAGE1 | C5 | 0.1UF | ECE-A1HKA0R1 | Approv |

3. Select the part with the properties to be copied. 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.
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 dis design, sorted by part

reference.




| # | Schematic Page | Part Reference | Value | Part Number | Part St |
|---|---|---|---|---|---|
| 1 | ROOT : PAGE1 | C4 | 91PF | PCC910CQTR-ND | Approv |
| 2 | ROOT : PAGE1 | U17 | 74ACT574 | 22-00074 | Approv |
| 3 | ROOT : PAGE1 | U16 | 74HC374 | 20-00374 | Approv |
| 4 | ROOT : PAGE1 | U15 | 74HC244 | 20-000244 | Approv |
| 5 | ROOT : PAGE1 | C6 | 91PF | PCC910CQTR-ND | Approv |
| 6 | ROOT : PAGE1 | C3 | 91PF | PCC910CQTR-ND | Approv |
| 7 | ROOT : PAGE1 | C7 | 0.1UF | A104M15Z5UFVWN | Approv |
| 8 | ROOT : PAGE1 | C5 | 0.1UF | ECE-A1HKA0R1 | Approv |

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 the Ctrl key while you select individual parts to add to the selected group, or by holding down the Shift key 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.

# Exporting Variant List to PCB Editor using Part Manager

In Allegro PCB Editor, you cannot view the assembly drawings for the variants of a design created in Capture CIS. The information about the variants, the part changes for the variants of the design, can be exported to the variants.lst file. Allegro PCB Editor can use this file to import information about the variants. As a result, the variants.lst file serves as a mechanism for passing variant information to Allegro PCB Editor.

Once you are done with creating variants for a design, you can create the variants.lst file in CIS. You can create this file using the Export Variants List dialog box. The variants.lst file can then be imported in the board in PCB Editor.

To create the variants.lst file, CIS needs two inputs:

1. Information about the variants of the design. This is obtained from the CIS database.
2. Information about the mappings between properties in Capture and those in Allegro PCB Editor. This is given by you in the variant.cfg file.

## The Variant.cfg File

The mappings between CIS property names and Allegro PCB Editor injected property names are specified in the variant.cfg file. A sample variant.cfg file is located in the same folder as the Capture.exe file. The variant.cfg file contains entries of the following types:

```
_ <CIS property name> = YES
```

This means that `<CIS property name>` has significance in Allegro PCB Editor. It needs to be written into the variants.lst file. A space in the property name is replaced by an underscore. For example, Part Number = YES is written as Part_Number in the variants.lst file.

```
_ <CIS property name> = NO
```

This means that `<CIS property name>` does not have any significance in Allegro PCB Editor. It does not have to be written into the variants.lst file.

```
_ <CIS property name> = <PCB Editor Property Name>
```

This means that the equivalent of `<CIS property name>` in Allegro PCB Editor is `< PCB Editor Property Name>`. So, the entry in the variants.lst file is made as `<PCB Editor Property Name>`. For example, PCB Footprint = JEDEC_TYPE is written as JEDEC_TYPE in the variants.lst file.

## To create the Variants.lst File

1. From the *Tools* menu of Part Manager, choose *Export Variant List*. The Export Variants List dialog box appears.
2. The *Output File (Variants.lst) Path* field specifies the default location for the Variants.lst file.

   The default location is the Allegro folder in the design directory. You can change this path.
3. The *Config File (Variant.cfg) Path* field specifies the default path for the Variant.cfg file.

   The default path is the same as that of the Capture.exe file. You can change this path also.
4. Click *Export*.

   CIS creates a Variants.lst file containing information about all the variants of the design.

   The properties listed are the ones specified in the Variant.cfg file.

Return to top of page

---