p-cad® 2002

PROFESSIONAL TOOLS FOR BOARD LAYOUT SPECIALISTS ™

Library Executive

p-cad®

PCB layout system from Altium™

Copyrights

Altium Limited
www.altium.com

Table of Contents

chapter 4 Libraries

chapter 5 The Pattern Editor

chapter 6 The Symbol Editor

chapter 7 The Source Browser

chapter 8 Querying Libraries

chapter 9 Importing Data from an External Source

chapter 10 Library Verification

chapter 11 Reports

chapter 12 Component Placement

chapter 13 Extended Library Features

chapter 14 File Commands

chapter 15 Library Commands

CHAPTER 1

Introduction to the Library Executive User's Guide

This introduction to the Library Executive User's Guide summarizes the powerful, advanced features of Library Executive, as well as details the contents of the manual to follow.

Library Executive is a utility, with a user-friendly interface, used in conjunction with P-CAD PCB and P-CAD Schematic to efficiently create and maintain component data. These components can then be placed into PCB and Schematic designs.

With Library Executive you can search libraries for a variety of attributes using the Query tool. External data files may be imported, queried, or used to create or update your P-CAD libraries. Also, components in a design can quickly be verified and modified, if desired, when updates are made to a library. Many of these features, including Query, can be used directly from P-CAD PCB and Schematic.

Library Executive is partnered with a Pattern Editor and Symbol Editor that include wizards that accelerate pattern and symbol creation. Once created, the patterns and symbols can be easily attached to components. General knowledge of P-CAD PCB or P-CAD Schematic may be a prerequisite to using certain Library Executive functions. Refer to your *P-CAD PCB User's Guide* or *P-CAD Schematic User's Guide* for information about these products.

About this Manual

This manual is a combined user's guide and reference manual for P-CAD Library Executive.

Chapter 2: Chapter 2 is useful for setting up and learning how to use the basics of Library Executive.

Chapter 3: Chapter 3 consists of a tutorial that takes you through the process of creating a new library with new symbols, patterns and components.

Chapter 4: Chapter 4 describes library concepts, such as how patterns, symbols and components occupy the same library, the use of component, pattern and symbol aliases, and the important distinction between pin designators and pad, or pin, numbers.

Chapter 5 and 6: Chapter 5 and 6 summarize the features available in the Pattern Editor and Symbol Editor. These editors include the Pattern and Symbol Wizard, quick and easy utilities that jump start component creation.

Chapter 7 to 13: Chapters 7 through 13 describes the Source Browser and Query features, as well as how to import component data from a non-P-CAD source, update, create and verify libraries and produce reports.

Chapter 14 to 22: Chapters 14 through 22 contain the Command Reference for Library Executive. They are useful for looking up specific information about the program commands. They list and describe all of the menu commands in the order that they appear on the Library Executive menu. These chapters also provide all related functions and options that work in conjunction with, or are alternatives to, the listed commands.

Installation and Setup

This section lists the required hardware and software settings you need to install the P-CAD Suite.

System Requirements

Make sure that your PC and its software conform to the following P-CAD requirements and recommendations.

Recommended System

- Windows NT 4/2000 Professional
- PC with Pentium III Processor
- 128MB RAM (256MB for high component/net count)
- 400MB Hard Disk Space
- Desktop area 1024x768 pixels
- 32-bit Color Palette
- CD-ROM Drive
- Mouse or compatible pointing device

Minimum System

- Windows 95/98/2000Me
- PC with Pentium 166MHz
- 64MB RAM
- 200MB Hard Disk Space (without ISO libraries)
- Desktop area 800x600 pixels
- 256 Color Palette
- CD-ROM Drive
- Mouse

Installing P-CAD Products

For up-to-date installation information refer to the file Readme.WRI, located on the product CD. This file can also be found in the application program folder `(\Program Files\P-CAD 2002)` after installation. Note that the setup program on the Product CD can also be used to Repair or Remove an existing P-CAD Installation.

CHAPTER 2

Library Basics

This chapter provides important information about the basic functions and features of the P-CAD Library Executive, such as:

- A description of the user interface
- A description of the *Symbol View*, *Pattern View* and *Pins View* dialogs spreadsheet
- A description of the advanced views, including the Source Browser and Query
- Spreadsheet editing
- Library Executive commands
- Keyboard specifics.

User Interface

This section provides an overview of the Library Executive user interface.

Menu Bar

The menu bar provides access to Library Executive commands and functions.

To activate a menu, click the **Menu Title** or press the **ALT** key in combination with the underlined letter of the menu title (e.g. **ALT**+**C** to display the *Component* menu). When the menu appears, click the **Menu Item**, or press the underlined key, to choose a command.

A command followed by three dots (e.g. Open...) opens a dialog when you choose it.

Toolbar

A toolbar provides shortcuts to commonly used commands and functions. It appears when you choose the **View » Toolbar** command. You can use your mouse to drag it to a new location on your screen.

Tool Tips explain each of the toolbar buttons. To activate a Tool Tip, place the mouse over the button. The Tool Tip pops up.

The toolbar buttons are shortcuts for the following commonly used commands.

Use this button	To do this
	Component » **New**: Create a new component.
	Component » **Open**: Open an existing component.
	Component » **Save**: Save the current component.
	Edit » **Cut**: Cut whatever is selected in the spreadsheet to the clipboard.
	Edit » **Copy**: Copy whatever is selected in the spreadsheet to the clipboard.
	Edit » **Paste**: Paste the contents of the clipboard to the current selection.
	Edit » **Slide Selection Up**: Slides the selected information up one row in the spreadsheet.
	Edit » **Slide Selection Down**: Slides the selected information down one row in the spreadsheet.
	Edit » **Component Attr**: Modify the attributes of the current component.
	View » **Component Information**: Opens and closes the *Component Information* dialog.
	View » **Pins View**: Opens and closes the *Pins View* dialog.
	View » **Pattern View**: Opens and closes the *Pattern View* dialog.
	View » **Symbol View**: Opens and closes the *Symbol View* dialog.
	Component » **Validate**: Validates the component for errors.

Use this button	To do this
	Edit » **Undo**: Allows you to undo the last action.
	Source Browser: Opens the *Source Browser* dialog.

You can turn on or off the display of the Toolbar by choosing the **View** » **Toolbar** command.

Custom Toolbar

You can create a Custom Toolbar for quick access to other applications from Library Executive. Programs are added to the Custom Toolbar by choosing the **Utils** » **Customize** command. You can start any of the applications by clicking their **Custom Toolbar** buttons or selecting them from the list of applications that appear in the Utils menu.

Complete instructions on creating and using a Custom Toolbar are found in *Utils Commands (page 263)*.

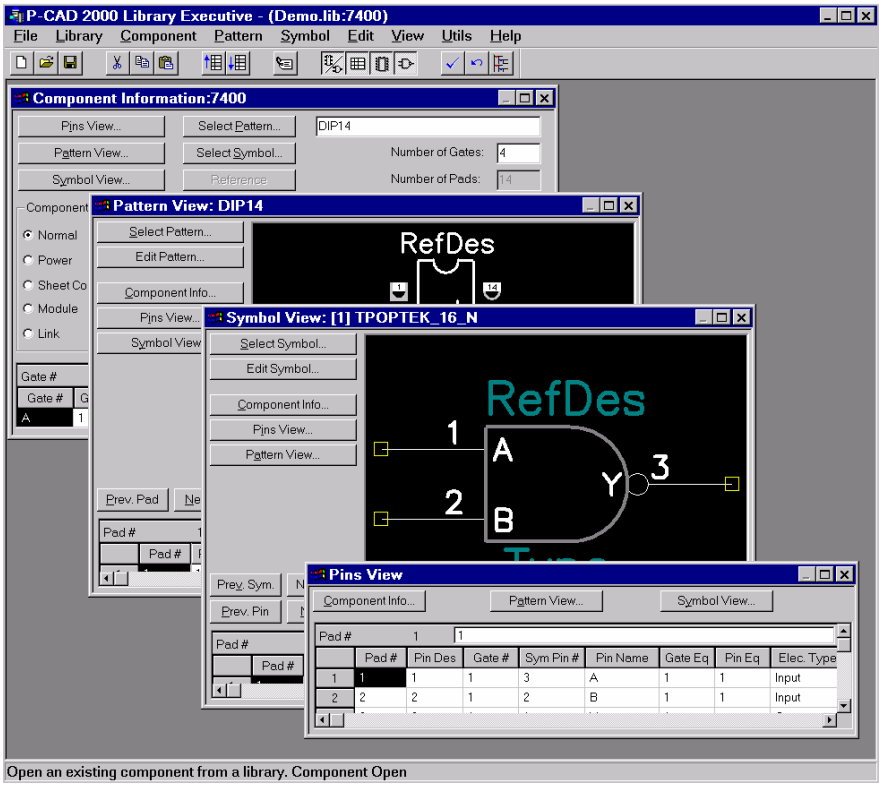
Prompt Line

The prompt line at the bottom of the window displays information prompting you on the appropriate action to take. You can turn on or off the display of the Prompt Line by choosing the **View** » **Prompt Line** command.

Open an existing component from a library. Component Open

Library Executive Layout

When you first access Library Executive, the interface is blank. All toolbar functions except **Component New** and **Component Open** are grayed, indicating they are unavailable. When you open a component (**Component** » **Open**) or create a new component (**Component** » **New**), the *Component Information* dialog opens and tool bar functions are enabled. Three other dialogs provide detailed information about the component and let you create pad to pin associations.



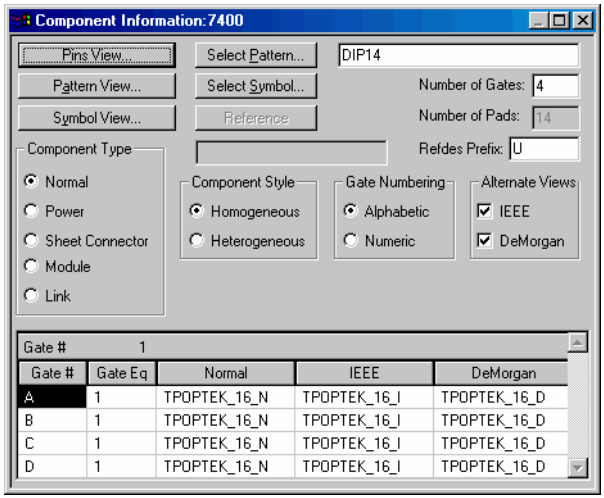
The **Component interface** comprises four main dialogs. The *Component Information* dialog provides information about a component. The three *View* dialogs provide visual feedback for patterns and symbols as the component is being created. This includes highlighting of pins and pads and the automatic selection of a spreadsheet row (in all three views) based on your selection within the pattern or symbol browse window. During component creation as much information as possible is automatically filled in for you. Changes you make to one dialog are reflected in the other three.

Each of the dialogs may be resized to offer more or less viewing area depending on your needs. Each dialog can be accessed from any of the other three.

A brief description of each dialog is presented in this section. For a detailed description of these dialogs, see *View Component Info (page 255)*.

Component Information

The *Component Information* dialog, shown in the following figure, provides general information about a component.

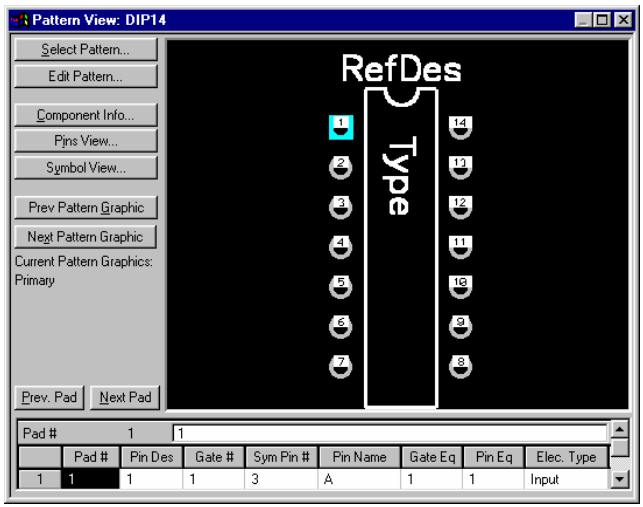


From this dialog, you can set the component type as homogeneous or heterogeneous, select alternate views, set the number of gates and gate numbering style, define the total number of pins, attach patterns and symbols and assign a reference designator prefix.

The maximum number of gates per component is 5,000. You can have as many as 10,000 pins per component.

Pattern View

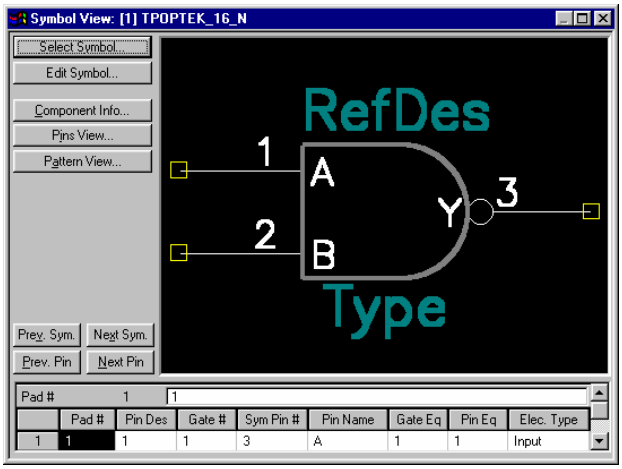
The *Pattern View* dialog lets you select a pattern for the component and displays the information as shown in the following figure.



In the *Pattern View* dialog you can associate pin information to each pattern pad. A browse window displays the current pattern and the spreadsheet displays information about the pin/pad association. Click the **Select Pattern** button to open the *Library Browse* dialog where you can select a new pattern.

Symbol View

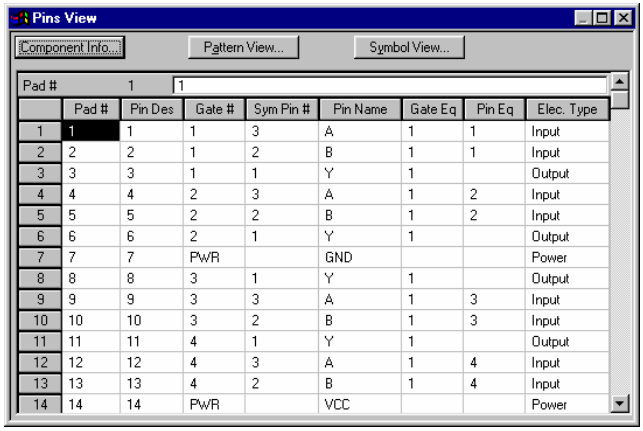
The *Symbol View* dialog lets you define symbols for the component.



Use the *Symbol View* dialog to build an association between symbol pin information and the pattern's pads. A browse window displays the attached current symbol(s) and the spreadsheet displays information about the pin/pad association. Click the **Select Symbol** button to open the *Library Browse* dialog where you can select a new symbol.

Pins View

The *Pins View* dialog is a spreadsheet view of the component, as shown in the following figure.



	Pad #	Pin Des	Gate #	Sym Pin #	Pin Name	Gate Eq	Pin Eq	Elec. Type
1	1	1	1	3	A	1	1	Input
2	2	2	1	2	B	1	1	Input
3	3	3	1	1	Y	1		Output
4	4	4	2	3	A	1	2	Input
5	5	5	2	2	B	1	2	Input
6	6	6	2	1	Y	1		Output
7	7	7	PWR		GND			Power
8	8	8	3	1	Y	1		Output
9	9	9	3	3	A	1	3	Input
10	10	10	3	2	B	1	3	Input
11	11	11	4	1	Y	1		Output
12	12	12	4	3	A	1	4	Input
13	13	13	4	2	B	1	4	Input
14	14	14	PWR		VCC			Power

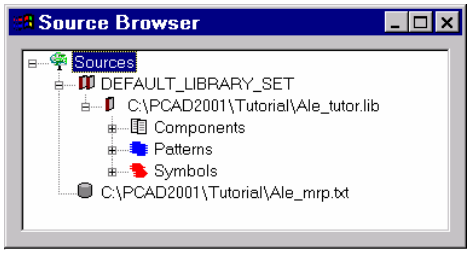
This dialog lets you associate pin information with each pattern pad, where each row corresponds to the pad number of an attached pattern, if a pattern is attached. The number of pins cannot be greater than the number of pads.

The *Pins View* dialog consists of rows and columns identifying component properties such as **Pin Designator**, **Gate #** and **Sym Pin #**, all of which can be edited (e.g. to assign an electrical type). Above these rows and columns is a spreadsheet edit box (in the top-left corner) and buttons to open other views.

Source Browser

The Source Browser provides easy access to all of the component data from a variety of sources and most of the commands in Library Executive.

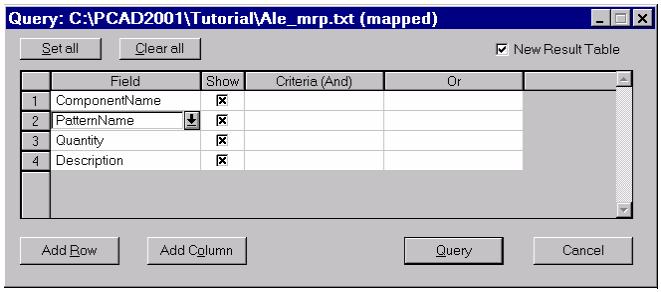
The Source Browser is shown in the following figure.



The Source Browser is a storage place for P-CAD libraries, library sets, an imported source file and query results. Its tree like structure provides easy access to each of these sources and their contents. When a source is selected, the shortcut menu provides access to the commands available for execution on that source. Refer to *The Source Browser (page 91)* for additional information.

Query Dialog

The *Query* dialog is a spreadsheet dialog in which component search criteria may be entered, as shown in the following figure.



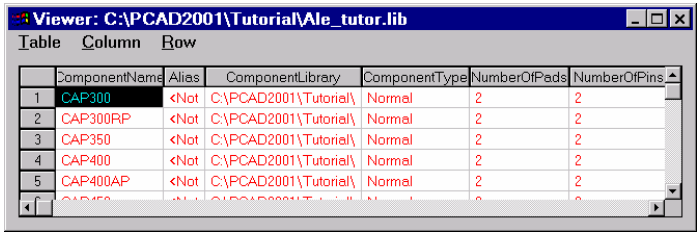
When **Query** is chosen, the selected source is searched for components satisfying the specified criteria. These components are listed in a *Query Result Viewer* dialog. See *Viewer Dialog (page 10)* for more information.

The *Query* dialog is a spreadsheet. Refer to the information in *About the Spreadsheet Views (page 10)* for details on manipulating spreadsheets.

The *Query* dialog can be accessed directly from P-CAD PCB and Schematic. Components in the embedded Query Result can be placed directly into the current design. Refer to *Querying Libraries (page 103),* for additional information.

Viewer Dialog

The *Viewer* dialog displays the components contained within a variety of sources. Query results, imported library source files, cross link results and P-CAD libraries all have their contents displayed in the spreadsheet Viewer.



	ComponentName	Alias	ComponentLibrary	ComponentType	NumberOfPads	NumberOfPins
1	CAP300	<Not	C:\PCAD2001\Tutorial\	Normal	2	2
2	CAP300RP	<Not	C:\PCAD2001\Tutorial\	Normal	2	2
3	CAP350	<Not	C:\PCAD2001\Tutorial\	Normal	2	2
4	CAP400	<Not	C:\PCAD2001\Tutorial\	Normal	2	2
5	CAP400AP	<Not	C:\PCAD2001\Tutorial\	Normal	2	2

From the *Viewer* dialog, you can access many of the Library Executive features and functions, including query, verification and reports. Also, components selected in the Viewer can be placed directly into a P- CAD PCB or Schematic design.

The file *Viewer* dialog is a spreadsheet. Refer to the information in *About the Spreadsheet Views (page 10)* for details on manipulating spreadsheets.

Each of the Viewer dialogs are identical, independent of the source. However, the commands that may be accessed from the Viewer menu vary. For a summary of the Viewer dialog menu commands, refer to *Viewer Menu Commands (page 15)*. For information on the Imported Source File Viewer, refer to *Importing Data from an External Source (page 121)*. For information on the Query Results Viewer, refer to *Querying Libraries (page 103)*. The commands accessible from a P-CAD library and Cross Link Results Viewers are the same as from a Query Result Viewer.

About the Spreadsheet Views

In Library Executive, each of the three view dialogs (*Pattern View*, *Symbol View* and *Pins View*) contains a spreadsheet displaying information about the pin/pad association of the component. The *Query* dialog and the file Viewer are also spreadsheets. These dialogs display the component attributes and/or their search criteria.

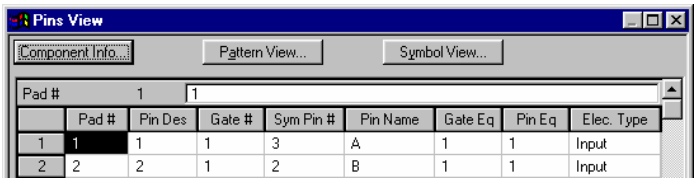
This section describes the spreadsheet views of Library Executive. It includes detailed definitions of the fundamental building blocks of a component.

Information about spreadsheet editing techniques and shortcuts can be found in *Spreadsheet Editing (page 12)*.

Spreadsheet Columns

The columns, which appear in all three common Library view dialogs, are used as follows:

- **Row Number**: Each row number represents a pattern pad if a pattern is attached.



	Pad #	Pin Des	Gate #	Sym Pin #	Pin Name	Gate Eq	Pin Eq	Elec. Type
1	1	1	1	3	A	1	1	Input
2	2	2	1	2	B	1	1	Input

- **Pin Des**: The pin designator is the name assigned to each pad/pin association.
- **Gate #**: The gate number defines the gate on which the pin is located. In multi-gate components, the gates are uniquely numbered from 1 through *n*.
- **Sym Pin #**: The number of the pin on the attached symbol. Symbol pin numbers must be unique and must exist in the attached symbol.
- **Pin Name**: The pin name for the corresponding pad/pin association.
- **Gate Eq**: The gate equivalence column indicates which gates within a component, or between components of the same type, are logically equivalent and may be swapped by choosing the **Utils » Optimize Nets** gate swap command. For two gates to be considered logically equivalent, the Gate Eq values must be identical and non-zero.

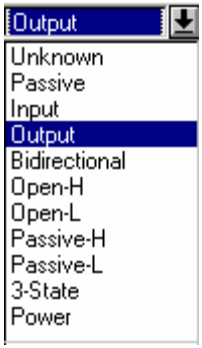
  Equivalent gates must have the same number of pins. A zero value (shown as a blank in the spreadsheet) indicates that the gate isn't swappable. See the *Utils Optimize Nets* command section in the *P-CAD PCB User's Guide* for information on swapping gates.
- **Pin Eq**: Indicates which pins within a gate are logically equivalent and may be swapped by choosing the **Utils Optimize Nets** pin swap command. The pin equivalence values must be identical for a swap to occur between two pins. Non-swappable pins are indicated with a zero value.

  Jumper pins are listed as *JMP-n*, where all pins with the same -n value are considered jumpered together. A *JMP1* jumpered pin is displayed as *JMP*.
- **Elec Type**: The type of pin. Options appear in a shortcut menu, described in *Electrical Pin Types (page 11)*.

If a cell is blank, it has a value of zero (or an electrical type of Unknown).

Electrical Pin Types

The *Elec Type* cells are a combination of edit box and list. Click in the cell to select it and then click the **Down Arrow** next to the box at the top of the window to display a list of electrical pin types. Select the pin type you want from the list and it changes in the cell.



You can also access electrical types by typing the first letter of a given type. Where more than one type begins with the same letter (e.g. Passive-H and Passive-L), pressing the letter scrolls through the types.

Spreadsheet Editing

This section describes some common spreadsheet editing functions and also provides a list of keyboard shortcuts.

To perform this task:	Follow these steps:
Edit a cell	Type directly in a highlighted cell or select the cell and enter information. When you move off the cell (using one of the arrow or mouse keys) or press **ENTER**, the new information appears in the cell. **ESC** cancels the changes. When a list is available, the contents of a cell can be modified either by selecting an option from the list or typing the entry directly.
Select a cell or cells	To select a single cell press the **left mouse button** and drag the cursor to select additional cells. You can also hold down the **SHIFT** key and use the arrow keys to select multiple cells.
Select a column	Pad # | Pin Des | Gate # | Sym Pin # | Pin Name | Gate Eq | Pin Eq | Elec. Type Click the **column heading** button to select all cells in the corresponding column.
Select a row	1 2 Click the **row** button to highlight an entire row.
Cut a cell or cells	Choose the **Edit** » **Cut** command to remove the selection and place it onto the Clipboard. The selection can be a cell or a cell range. **Shortcuts**: Click the **Cut** button Press the **CTRL**+**X** keys

To perform this task:	Follow these steps:
Copy a cell or cells	Choose the **Edit » Copy** command to copy the selection onto the Clipboard. The selection can be a cell or a cell range. **Shortcuts**: Click the **Copy** button Press the **CTRL**+**C** keys
Paste information into a cell or cells	Choose the **Edit » Paste** command to paste the contents of the Clipboard into a cell. If you select multiple cells, the contents of the Clipboard will be pasted into each cell. The paste area can be a single cell or a cell range. If the paste area is a single cell, this cell is the upper-left corner of the paste area; the rest of the copied data is pasted below and to the right of this cell. **Shortcuts**: Click the **Paste** button Press the **CTRL**+**V** keys
Slide selected cells up or down	To slide row 12 up to row 10: 1. Select the columns in row 12 that you want to slide. 2. Press the **CTRL** key and slide the row up. 10 10 10 3 2 11 11 11 4 1 12 12 12 4 3 13 13 13 4 2 Before sliding row 12 data into row 10 3. Notice that row 10 and 11 have been pushed down. 10 10 12 4 3 11 11 10 3 2 12 12 11 4 1 13 13 13 4 2 After sliding row 12 data into row 10 **Shortcuts**: Press the **CTRL**+**UP** keys Press the **CTRL**+**DOWN** keys

To perform this task:	Follow these steps:
Undo a task	Choose the **Edit** » **Undo** command to undo the previous action. **Shortcuts**: Click the **Undo** button Press the **CTRL**+**Z** keys
Move between cells	Press the **TAB** key to move to the next cell; **SHIFT**+**TAB** to move back to the previous cell. **Shortcuts**: Click a cell
Move a Column	Select the column in the *View* dialog. Press the **left mouse button** and the **SHIFT** key. Drag the column to its new location. A black line indicates where the column will be placed when the mouse button is released.
Enable All	In the *Save To Library* and *Query* dialogs, columns containing check boxes can be entirely selected by clicking the **column header**. For example, click **Show** to select all check boxes in the *Show* column. In the *Save To Library* dialog, only one column can be selected at a time; clicking a **column header** button toggles the check box columns.
Place a Queried Component	In the embedded *Query* dialog, double click a **component row** to select it for placement. The workspace appears in **Place Component** mode with the selected component available for placement.

Resizing Spreadsheet Views

You can resize a spreadsheet view when you want to simultaneously display multiple windows, or when you want to add or reduce the number of spreadsheet rows in a window.

To perform this task:	Follow these steps:
Reduce a view to an icon	Click the **Minimize** button
Restore a view to its previous size and location	Click the **Restore** button
Close a view window	Click the **Close** button
Change the size of a view window	Point to the window's border. When the pointer becomes a ↔, click and drag the border to size the window proportionately.

To perform this task:	Follow these steps:
Expand the number of displayed spreadsheet rows	Point to the window's border. When the pointer becomes a ↔ hold the **SHIFT** key and drag the border until the desired number of rows appears.

Menu Commands

Additional menu commands exist in the Viewer and from the shortcut menus.

Viewer Menu Commands

This section describes how to use the menu commands for the *Library Executive Viewer* dialog. These commands are listed in the order that they appear on the dialog menu: **Table**, **Column** and **Row**.

The Viewer is used for Query Results, P-CAD libraries, Cross Link Results and imported external source files. The *Viewer* menus are identical for the four sources with the exception of the **Table** command **Map Fields**, available for the external source file *Viewer* only.

Table Commands

Table commands allow you to access other features of Library Executive, including **Query**, **Cross Link**, **Verify**, **Report** and **Library Save**. They also allow you to rearrange the contents of the table by using cut, copy and paste. Of particular importance for importing files from an external source are the **Map Fields** and **Cross Link** commands, which are also entered from the **Table** menu.

The **Table** menu is used in the external source file *Viewer* dialog. The **Map Fields** command is not available in the Table menu of the *Viewer* dialog for Query Results, Cross Link Results and P-CAD library files.

- **Cut**: Removes any selected component row from the *Viewer* and moves it to the Clipboard.
- **Copy**: Copies any selected item from the Viewer onto the Clipboard.
- **Paste**: Pastes items from the Clipboard onto the Viewer.
- **Map Fields**: Opens the *Map Fields* dialog where you can map the field names imported from an external source file to field names recognized by P-CAD Library Executive.
- **Cross Link**: Opens the *Cross Link* dialog where you can link component attributes in the selected file with their corresponding components in another source. The Cross Link Results Viewer displays a combination of the attributes in the two source files for the linked components.
- **Query**: Opens the *Query* dialog where you can narrow the component list of the imported library by including search criteria.

- **Verify**: Opens the *Verify* dialog where you can compare the components in the *Viewer* with components in a design or library.
- **Report**: Opens the *Report* dialog where you can set up and generate a report on the components included in the Viewer.
- **Save to Library**: Opens the *Save Source* dialog where you can save components in the Viewer to a P-CAD library.
- **Close**: Closes the Viewer.

The **Verify** and **Save to Library** commands are only available if the primary field, *ComponentName*, is present.

Column Commands

The Column commands for a selected column are available from the **Column** menu as well as from the shortcut menu. The commands allow you to add, remove, rename and sort the columns of fields and their values.

- **Add**: Adds a column to the *Viewer* to the left of the currently selected column. Displays a dialog prompting you for the new field name. A pick list of predefined field names is available in the *Add* dialog.
- **Delete**: Removes the currently selected column from the Viewer.
- **Rename**: Displays a dialog that requests a new name for the currently selected column.
- **Sort**: There are two methods available to sort columns: **Allow Duplicates** and **Resolve Duplicates**.

  The **Allow Duplicates** command sorts the field values, grouping duplicate values together.

  If you would like unique field values, use the **Resolve Duplicates** command. The suffixes _1, _2, _3, etc. are added to the end of repeated values. The **Resolve Duplicates** command may be preferred, for example, when sorting the *ComponentName* field, since P-CAD requires a unique name for each component. The **Resolve Duplicates** command is available only for fields with data type String.

> The sorting method for String and Integer data types differs. The default data type, String, is always used when saving attributes to a P-CAD library. Only modifiable attributes (not read-only) are saved to the library.

Row Commands

The Row commands for a selected column are available from the **Row** menu as well as from the shortcut menu. With the Row commands, you can add a row or remove the component in the selected row. You can place a component in a P-CAD PCB or Schematic design. You can also open a component to view its component information.

- **Delete**: Removes the currently selected row from the *Viewer*.

- **Place**: Places the selected component in a running PCB or Schematic design and opens the *Place Part* or *Place Component* dialog open with the component selected.
- **Open**: Displays the *Component Information* dialog. Refer to *View Commands (page 255),* for additional information.

  The **Open** command is only available if the row contains the ComponentName and ComponentLibrary field. The **Place** command requires the row represents a valid component with an attached symbol or pattern.

Source Browser Shortcut Menu Commands

When you select an object in the Library Executive Source Browser and click the **right mouse button**, a shortcut menu opens, providing access to common commands performed on selected objects. The commands on the shortcut menu change depending on the object selected.

The following commands appear on the shortcut menu.

Choose this Command:	To do this:
Add Library	Opens a standard window where you can navigate to and select the library you want to add to a selected library set.
Cross Link	Opens the *Cross Link* dialog where you can link component attributes in the selected file viewer with their corresponding components in another source. The Cross Link Results Viewer displays a combination of the attributes in the two source files for the linked components.
Delete	Deletes the selected library set. You can also press the **DELETE** key.
Delete Attributes	Deletes selected user-defined attributes from a P-CAD library.
Map Fields	Opens the *Map Fields* dialog where you can map the field names imported from an external source file to field names recognized by Library Executive.
New Comma-delimited File	Opens the *Import Comma-delimited File* dialog where you can select a non-P-CAD component data file to import into Library Executive.
New Library Set	Adds a new library set to the Source Browser.
Open	Opens the *Component Information* dialog. See *View Commands (page 255),* for additional information.
Place	Opens the *Place Component* dialog in PCB or the *Place Part* dialog in Schematic with the selected component name selected for placement.

Choose this Command:	To do this:
Query	Opens the *Query* dialog where you can search the selected source. Query can be selected from a library, library set, an external source file, or a Query Result.
Reload	Reloads an external file, a library, or a library set's contents into P-CAD Library Executive. The contents of a library or external source file are read automatically when the library components are viewed, queried, verified, or used in a report. If, for example, the library contents are modified by another designer, you can refresh the library contents in Library Executive by selecting the library and choosing **Reload**.
Remove	Removes the selected source from the Source Browser. You can also press the **DELETE** key.
Rename	Renames the library set.
Report	Opens the *Report* dialog where you can set up and generate a report on the selected source.
Save to Library	Opens the *Save Source* dialog where you can create or update a P-CAD library from the selected source.
Verify	Opens the *Verify* dialog where you can generate a difference report comparing the selected source to a P-CAD library or library set.
View	Opens the selected pattern, symbol, or source Viewer.

Keyboard Shortcuts

The Library Executive spreadsheet editors have specific P-CAD keyboard features in addition to some standard Windows accelerators (shortcut keys). They are listed in the following table.

Press these keys:	To do this:
ALT+F4 (Exit)	Closes the active dialog. If no dialog is active, this works as a shortcut for **Component Exit** or **File Exit**, which exits the Library Executive program. If the current component has been modified since the last save, you will be prompted whether you want to save the changes. The program will write information to the `cmp.ini` file when you exit.
CTRL+C (Edit Copy)	A shortcut for **Edit » Copy**. Copies text from the selected cell(s) in the spreadsheet to the Clipboard. Main window must have focus.

Press these keys:	To do this:
CTRL+Down (Edit Slide Down)	Slides the selected information down in the spreadsheet.
CTRL+N (Component New)	A shortcut for **Component » New**. This command clears the spreadsheet to load the new component. Main window must have focus.
CTRL+O (Component Open)	Opens the *Component Open* dialog where you can choose a component. Main window must have focus.
CTRL+S (Component Save)	Saves changes to the current component without closing it. To save the component to a different name or location, choose **Component » Save As**. To clear the spreadsheet, choose **Component » New**. Main window must have focus.
CTRL+UP (Edit Slide Up)	Slides the selected information up in the spreadsheet.
CTRL+V (Edit Paste)	A shortcut for **Edit » Paste**. You can paste information from the Clipboard to the selected cells in the spreadsheet.
CTRL+X (Edit Cut)	A shortcut for **Edit » Cut**. Removes text from the selected cell(s) in the spreadsheet and places it on the Clipboard.
F1 (Help)	Displays context-sensitive help. If you put focus on a Library Executive command or dialog (by mouse or keyboard) and press **F1**, the Help window appears containing information specific to the focus item.
F2 (Edit)	Moves the cursor to a cell where you can modify its contents.
TAB	Moves forward from left to right one cell at a time. If you made a change to the previous cell, the change takes effect when you press **TAB**.
SHIFT+TAB	Moves backwards through the cells one at a time.
Arrow keys	If you haven't specified any change to a cell (nothing entered in the edit box), then the arrow keys will function normally, moving between cells in any direction. If you have made changes, the arrow keys allow you to commit the change and move to another cell. The **SHIFT+arrow** keys extend the selection.

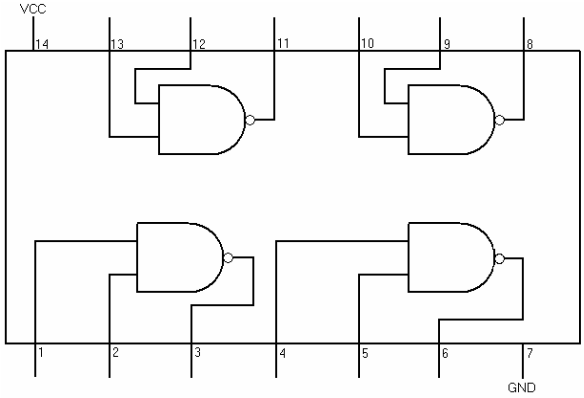
Press these keys:	To do this:
DELETE	Deletes items from the selected cell or cells.
HOME	Moves the cursor positioned in an edit box to the beginning (left side) of the box. When the cursor is positioned in the spreadsheet, it moves to the beginning of the row.
END	Moves the cursor positioned in an edit box to the end (right side) of the box. When the cursor is positioned in the spreadsheet, it moves to the end of the row.
ENTER	Transfers the change you make in the edit box to the cell that is being edited and moves the selection to the next cell below the current one.
PgUp	Scrolls the spreadsheet one page up. If the spreadsheet is only one page, scrolls to the top of the page.
PgDn	Scrolls the spreadsheet one page down. If the spreadsheet is only one page, scrolls to the bottom of the page.

CHAPTER 3

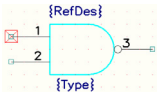
Library Executive Tutorial

Welcome to the P-CAD Library Executive tutorial. In this chapter, we will be investigating how to create libraries and components from scratch, as well as how to use the Symbol and Pattern Editors. After we have created a new symbol, pattern and component, we will have a quick look at the Source Browser.

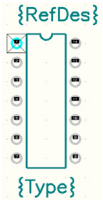
In this tutorial, we will create a symbol and pattern for a component, which is based on the logical diagram of the 7400 below.



When creating our new component, we will follow these general steps:



1. Create a new library, or open an existing one in which to store the component.
2. Create a new symbol, e.g. NAND, or use an existing P-CAD supplied one, which will represent the component parts in a Schematic design.
3. Create a new pattern, or use an existing P-CAD supplied one, e.g. DIP14, which will represent the component parts in a PCB design.



4. Create a new component, complete with Pin View information, which links the symbol, pattern and component information together.
5. Validate and save the component into a library.

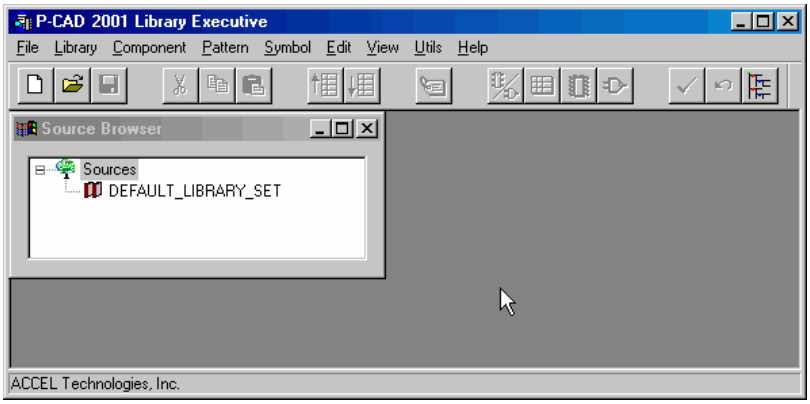
For more information about the topics in this tutorial, press **F1** for online Help.

Creating a New Library

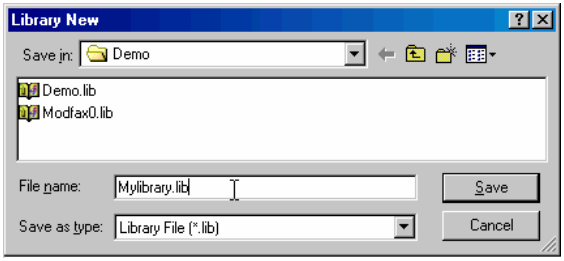
First we will create a new library to house the symbol, pattern and component we will make.

To create a new library:

1. Open P-CAD Library Executive. You can open the Library Executive as a program by itself or by choosing **Utils » Library Executive** from the main menu of P-CAD Schematic or P-CAD PCB. The Library Executive window opens. A Default Library Set appears in the Source Browser window.



2. Choose **Library » New**. The *Library New* dialog displays.

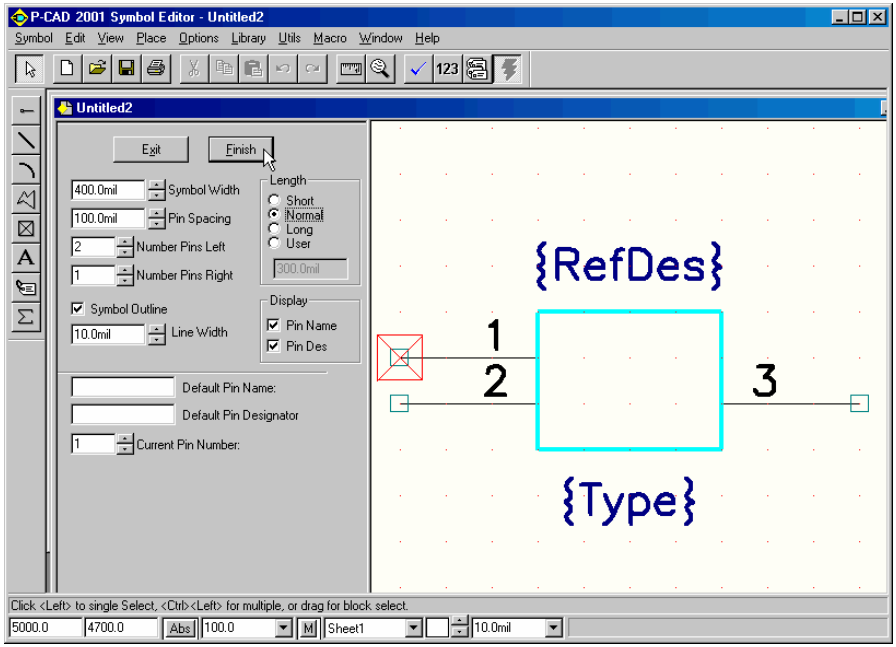


3. Navigate to the folder you want the library saved into, e.g. the `Demo` folder in the P-CAD installation directory.
4. Type in the new library name, e.g. `Mylibrary.lib`, and click **Save**. A new library has been created in the Demo folder.

Creating a New Symbol

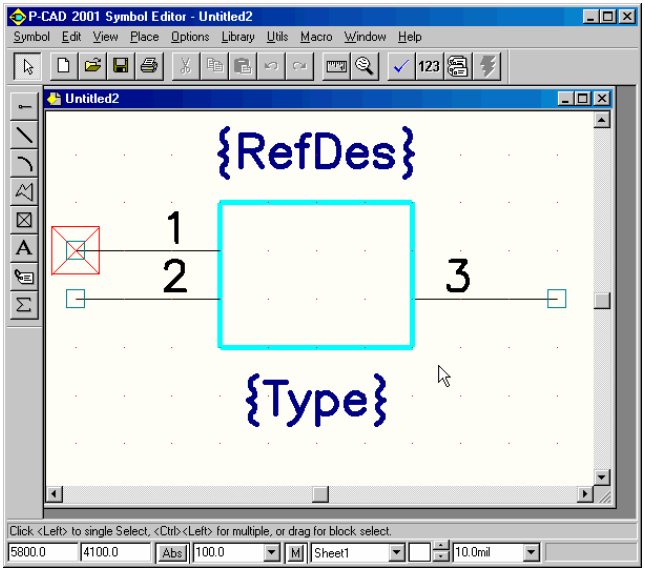
The new symbol we will create will be based on a symbol created by the Symbol Wizard. This will give us the basic shape we require and then we will use the Symbol Editor to make some alterations.

1. Create the new symbol using **Symbol » New** to open the Symbol Wizard.

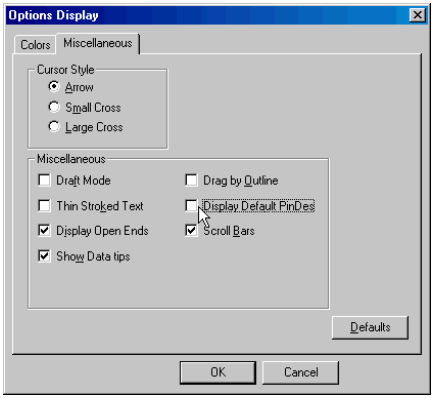


2. Fill in the fields as illustrated above, with a Symbol Width of 400 mil, Pin Spacing of 100 mil, 2 pins left, 1 pin right and **Symbol Outline**, **Pin Name** and **Pin Des** selected to display.

3. Click **Finish** to close the Symbol Wizard. The Symbol Editor is now displayed in which you can modify the new symbol if necessary.



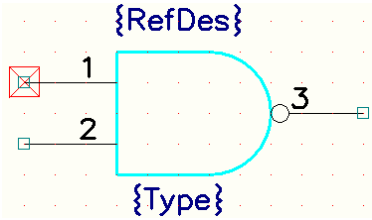
4. Choose **Options » Display** to turn off a pin display option. Click on the **Miscellaneous** tab.



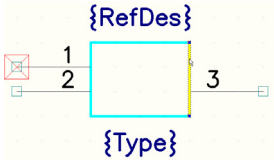
5. Deselect **Display Default Pin Des** and click **OK**.

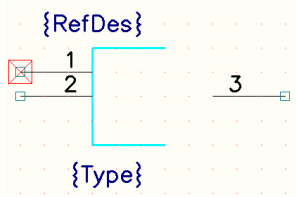
Modifying a Symbol

Now we can modify the symbol created by the Wizard to form a NAND gate symbol as shown in the following figure.

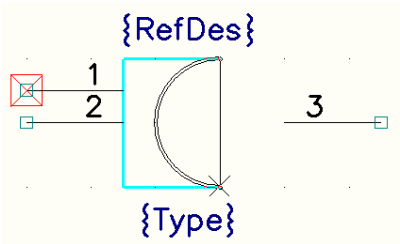


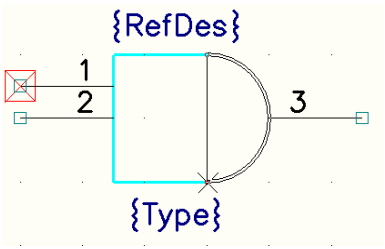
1. First, we will add a new grid spacing of 20mil to allow for easier object placement. To do this, type **20** into the Grid Spacing box located on the status line and press **Enter**.
2. Use the zoom keys **+** (zoom in) and **–** (zoom out) to view the symbol as necessary.
3. Now we will make the symbol bigger by selecting and deleting the right-hand vertical line of the box. Click on the line to select it and press the **Delete** key.
4. Make the remaining body of the symbol 400 mils high by selecting the bottom line and the {Type} field and moving them down 100 mils as shown.





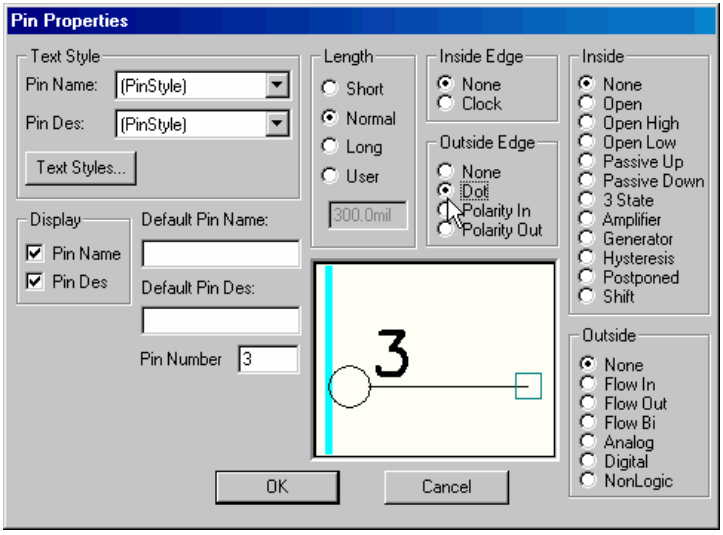
5. Extend the remaining vertical line to meet the bottom line. To do this, select the vertical line, click on the editing handle at the end and drag the line to its new position.
6. Select the pin number 3 and move it out of the way while we create the arc.
7. To create an arc with a 200mil radius, select **Place » Arc** or click the  toolbar button. Place the cursor over the end of the line at the top right of the symbol where you want the arc to start. Click and drag the cursor from the top to the bottom and release. An arc appears. Press **F** to flip the arc over and then click again at the bottom to place it. Right-click, or press **ESC**, to end arc placement.





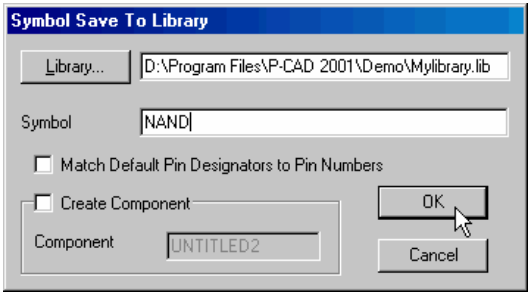
8. Choose the Select tool to reposition the {Refdes} and {Type} fields and move pin number 2 down 100 mils. Simply select the objects and drag them to their new positions.

9. Double-click on pin 3 to open the *Pins Properties* dialog and add a dot to the Outside Edge. Select **Pin Name** and **Pin Des** in the Display frame so these values will display with the pins and click **OK**.



10. Your symbol should now resemble the NAND gate symbol shown at the beginning of this section, so let's save it to the library `Mylibrary.lib`.

11. Choose **Symbol » Save** and the Symbol Save to Library displays.



12. Choose the library we created earlier by clicking on **Library** and navigate to `Mylibrary.lib`. Type `NAND` in the Symbol field and click **OK**.

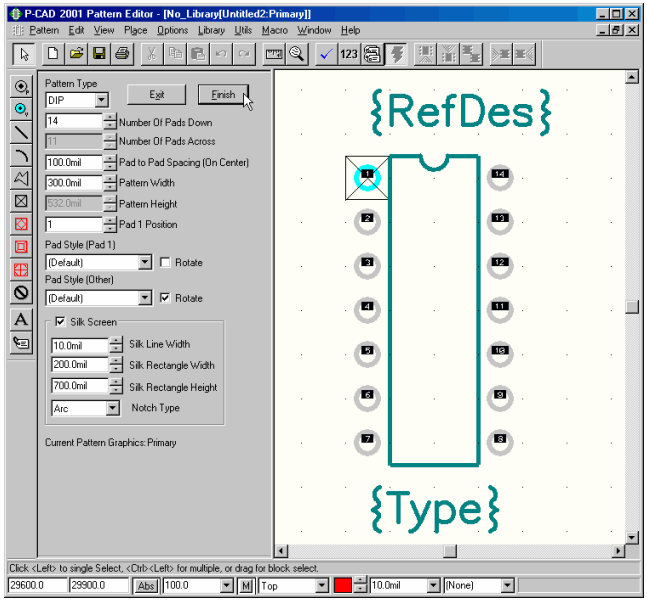
13. Close the Symbol Editor by choosing **Symbol » Exit** or click on the **Close** button .

14. You will be asked to save your symbol again but choose **No** since no changes have been made since you last saved it.

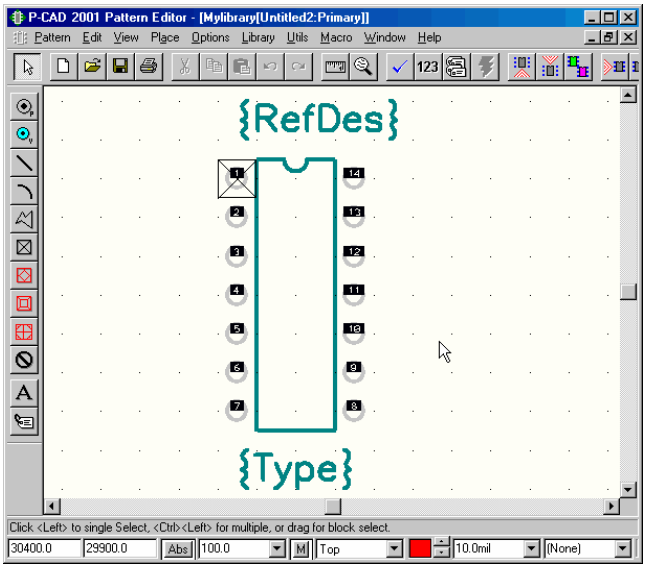
Creating a New Pattern

Although we have a DIP14 already created and available through a number of P-CAD supplied libraries, we will take a quick detour to look at the Pattern Editor and how the Pattern Wizard would create a pattern to your specifications.

1. Choose **Pattern** » **New** from the Library Executive main menu to open the Pattern Wizard.
2. Select **DIP** from the Pattern Type list and fill in the other fields as per the illustration below, i.e. 14 pads down, 100mil Pad to Pad Spacing, 300mil Pattern Width, Pad 1 Position of 1. Make sure the Silk Screen checkbox is selected and the Silk Rectangle Width is 200mil and Height is 700.0mil. Leave the Pad Styles as defaults.

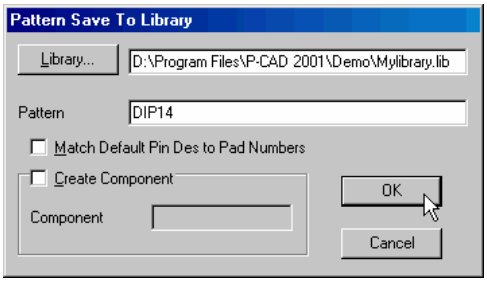


3. Click **Finish** to close the Pattern Wizard. The Pattern Editor is now displayed in which you can modify the new pattern if necessary by using the standard P-CAD PCB drawing tools.



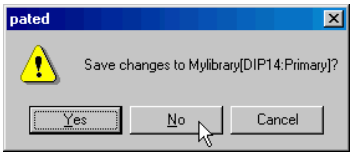
4. For this tutorial we will not need to alter the pattern that was created by the Pattern Wizard, so we will save the pattern into our library by choosing **Pattern » Save**. Navigate to the library `Mylibrary.lib` by clicking on the **Library** button, enter the library pathname and click **Open**.

5. Type `DIP14` in the Pattern field and click **OK**.



6. Close the Pattern Editor by choosing **Pattern » Exit** or clicking on the **Close** button ☒.

7. You will be asked to save your pattern again but choose **No** as no changes have been made.

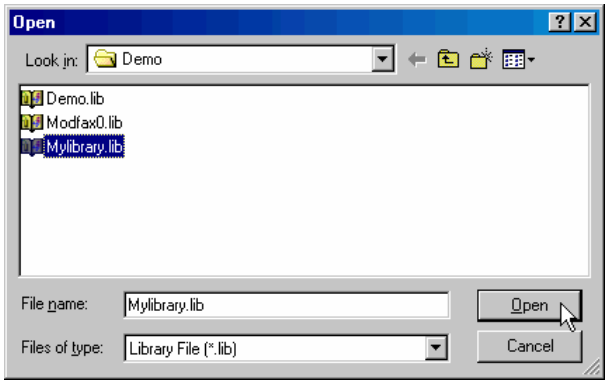


8. You are returned to the Library Executive window. Now we will create a component that uses the symbol and pattern we have created.

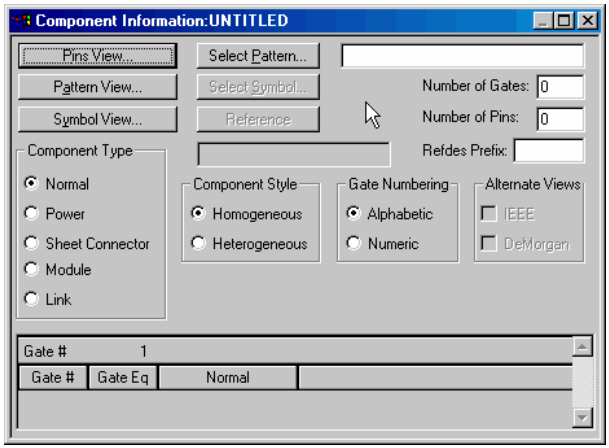
Creating a New Component

Creating a new component will link a symbol and pattern together with necessary pin information.

1. Choose **Component » New** in the Library Executive. You are prompted to open the library into which this component will be stored.



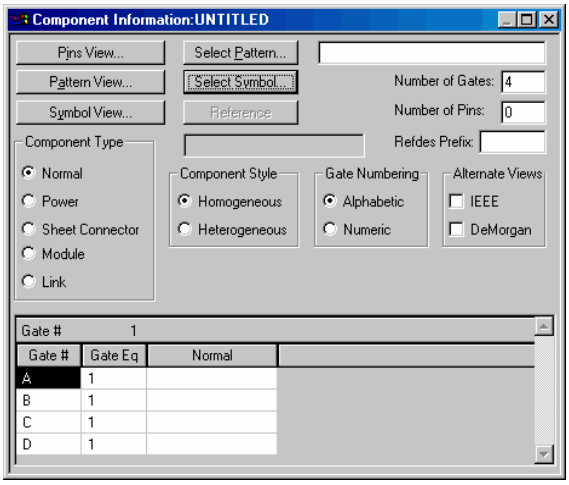
2. Select `Mylibrary.lib` from the `Demo` folder and click **Open**. The *Component Information* dialog displays.



3. Enter the number of gates (`4`) and press **Enter**. The 'gates' referred to when creating components in this section are referred to as 'parts' when they are being placed in a Schematic design.

4. Notice that the Gate # spreadsheet appears at the bottom of the *Component Information* dialog. Gate Equivalence (Gate Eq) values indicate which gates (parts) within a component are logically equivalent and may be swapped in a PCB design. For two gates to be considered logically equivalent, the Gate Eq values must be identical and non-zero. Equivalent gates must also have the same number of pins. A zero value (or a blank cell in the spreadsheet) indicates that the gate is not swappable.

> See the *Utils Optimize Nets* command section in the *P-CAD PCB User's Guide* for information on swapping gates.



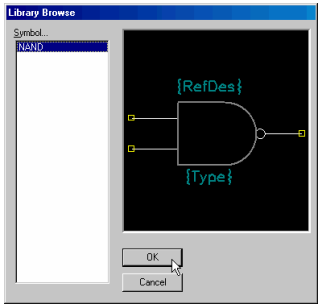
5. Leave the Component Type information as defaults since we are creating a normal component type that is homogeneous (all gates use the same symbol) with alphabetic gate numbering. Since you set the component style to Homogeneous, then all Gate Eq values are set to the first Gate Eq value of 1.

> There are five types of components: Normal, Power, Sheet Connector, Module, and Link. Naming a component type allows that component to properly associate with net connections. The component type also determines whether the component will appear in the netlist or bill of materials.

Adding the Symbol

First we will add the symbol NAND to the component. This symbol is a representation of the part of a component when placed in a schematic design using P-CAD Schematic.

1. Click on **Select Symbol** to choose which symbol you are attaching to the component. The Library Browse window displays the symbols available in the selected library.



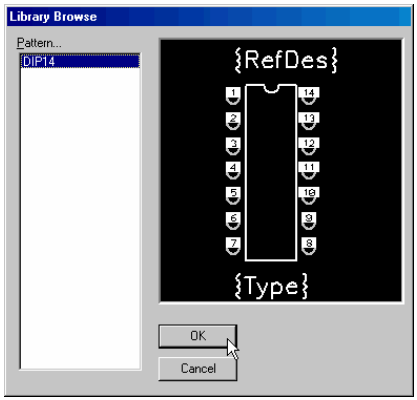
2. Our NAND symbol is already selected, so click **OK** to add it to the component. You are returned to the *Component Information* dialog.

3. Notice that the NAND symbol name has been added into the Gate # spreadsheet at the bottom of the *Component Information* dialog.

Adding the Pattern

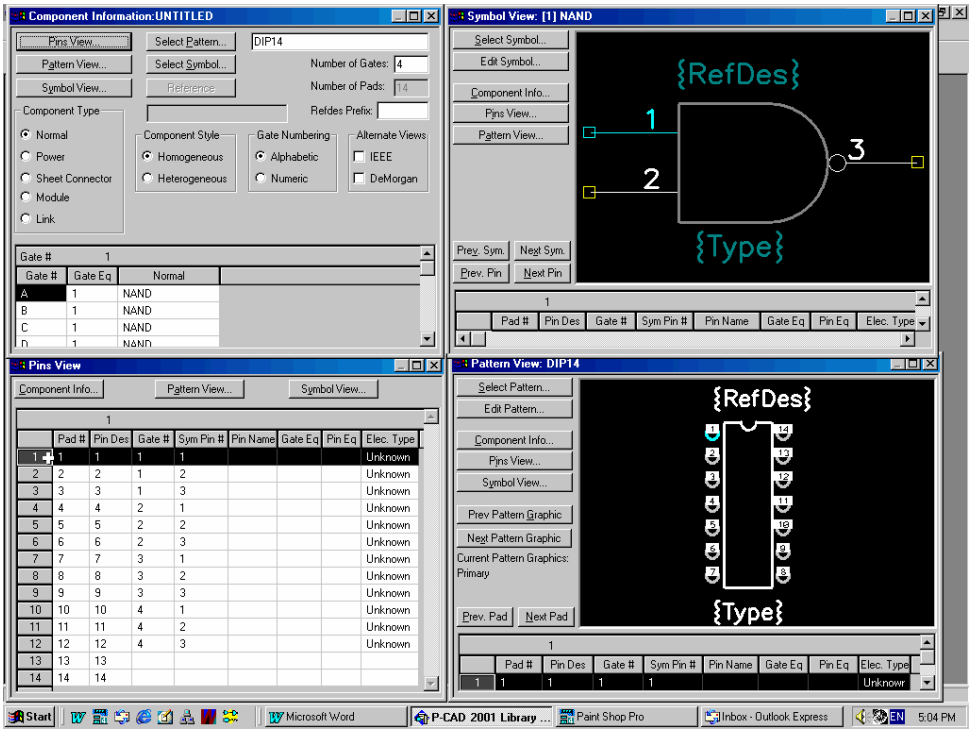
Next a pattern is added to the component to represent the component's footprint when placed in a PCB design using P-CAD PCB. Pad information from the pattern is used for matching pin to pad numbers when editing the Pins View later in this tutorial.

1. Click on **Select Pattern** to choose the pattern this component will use when placed in a PCB design. The Library Browse window displays the patterns available in the selected library.



2. Our DIP14 pattern is already selected, so click **OK** to add it to the component. You are returned to the *Component Information* dialog. Notice that the pattern name, DIP14, has been added to the top right of the dialog and the number of pads has been set to 14.

3. To view all the different parts of our component, click on **Symbol View**, **Pattern View** and **Pins View** and arrange the windows on your screen so that you can see what is happening as you change the pin information.



The final stage of setting up our new component is editing the Pins View spreadsheet so that the logical pins and the physical pins are correctly matched.

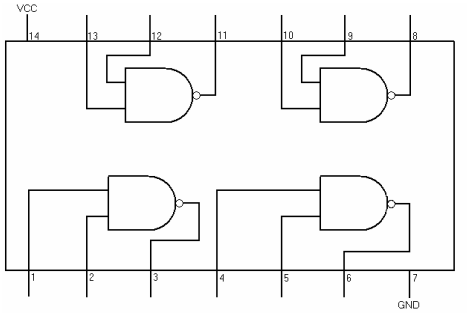
Editing the Pins View Spreadsheet

You define the electrical properties of your component in the Pins View spreadsheet. When you attach a pattern to a component, pad numbers and pin designators (Pin Des) are filled in. When you select a symbol, gate numbers (Gate #) and pin numbers for that symbol (Sym Pin #) are automatically added. Electrical types default to Unknown.

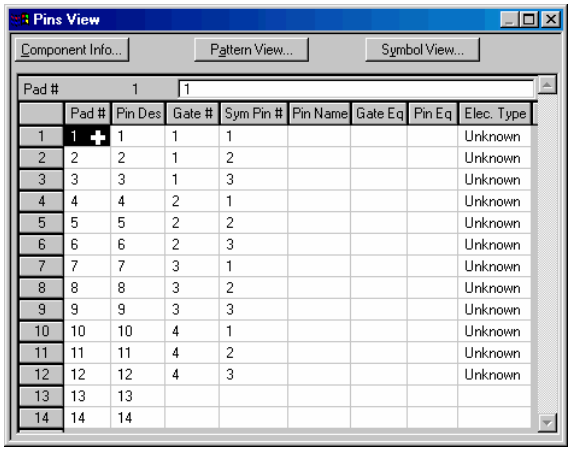
For information about spreadsheet editing techniques, refer to *Library Basics (page 3)*.

1. Click on the Pins View spreadsheet that you opened by clicking on **Pins View**. Notice that the program has automatically filled in some of the columns from information already supplied. We will be checking this information and adding new data.

2. Make sure that the pin information corresponds to the pads on the pattern in a way that matches the 7400 component specifications by checking the pin designators.



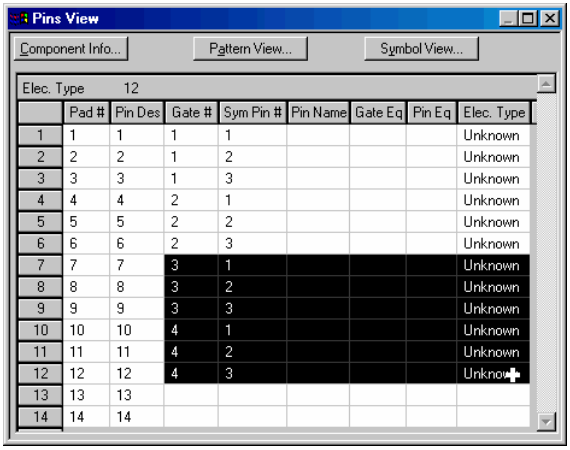
3. In this example, the pin designators coincide with the pad numbers and so can be left as entered.



	Pad #	Pin Des	Gate #	Sym Pin #	Pin Name	Gate Eq	Pin Eq	Elec. Type
1	1	1	1	1				Unknown
2	2	2	1	2				Unknown
3	3	3	1	3				Unknown
4	4	4	2	1				Unknown
5	5	5	2	2				Unknown
6	6	6	2	3				Unknown
7	7	7	3	1				Unknown
8	8	8	3	2				Unknown
9	9	9	3	3				Unknown
10	10	10	4	1				Unknown
11	11	11	4	2				Unknown
12	12	12	4	3				Unknown
13	13	13						
14	14	14						

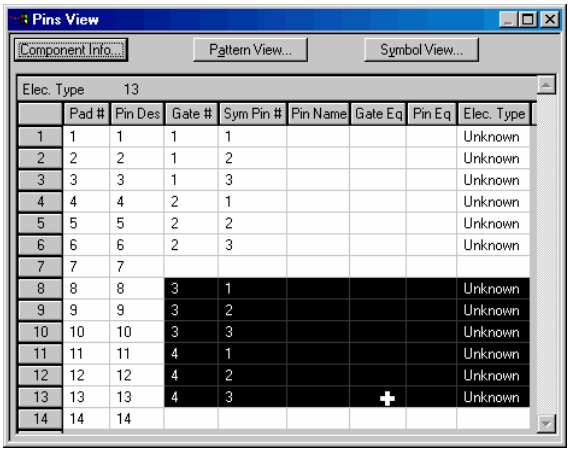
4. Notice that rows 13 and 14 do not have Gate # and Sym Pin # entries. This is because there are only 12 pins on the 4 NAND symbols associated with the pattern, but there are 14 pads on the DIP14. These two extra rows will be used to specify the hidden power and ground pins that are associated with pads 7 and 14 respectively.

5. Let's shift the pin information in rows 7 through 12 down one row to make room for the hidden power pin data to be filled in later. Select the information associated with gates 3 and 4 as shown below.



	Pad #	Pin Des	Gate #	Sym Pin #	Pin Name	Gate Eq	Pin Eq	Elec. Type
1	1	1	1	1				Unknown
2	2	2	1	2				Unknown
3	3	3	1	3				Unknown
4	4	4	2	1				Unknown
5	5	5	2	2				Unknown
6	6	6	2	3				Unknown
7	7	7	3	1				Unknown
8	8	8	3	2				Unknown
9	9	9	3	3				Unknown
10	10	10	4	1				Unknown
11	11	11	4	2				Unknown
12	12	12	4	3				Unknown
13	13	13						
14	14	14						

6. To quickly move the selected cells down to allow room to add power pin information for pin number 7, click the Shift Down button on the Library Executive main toolbar to move the selected data down one row.



	Pad #	Pin Des	Gate #	Sym Pin #	Pin Name	Gate Eq	Pin Eq	Elec. Type
1	1	1	1	1				Unknown
2	2	2	1	2				Unknown
3	3	3	1	3				Unknown
4	4	4	2	1				Unknown
5	5	5	2	2				Unknown
6	6	6	2	3				Unknown
7	7	7						
8	8	8	3	1				Unknown
9	9	9	3	2				Unknown
10	10	10	3	3				Unknown
11	11	11	4	1				Unknown
12	12	12	4	2				Unknown
13	13	13	4	3				Unknown
14	14	14						

7. The Sym Pin # (Symbol Pin Number) is the number of the corresponding pin on the attached symbol. Now we will map the Symbol Pin # to the physical Pin Designators based on specifications found in the manufacturer's data book.

   The Sym Pin # information added automatically is only half correct. Delete the Sym Pin # data for pins 8 to 13 by selecting them and pressing the **Delete** key. Type in the new values as shown right. You can type directly in a highlighted cell or highlight the edit box at the top of the window and type in information. To enter the data, press **Enter** or move the cursor to another cell using the arrow keys or the mouse.

Gate #	Sym Pin #
1	1
1	2
1	3
2	1
2	2
2	3
3	3
3	2
3	1
4	3
4	2
4	1

8. We will only be allocating Pin Names to the power pins in this exercise, so type in `GND`

for pin 7 and `VCC` as the Pin Name for pin 14. Hidden power pins are automatically connected to the net specified by the Pin Name.

9. Now we can add Gate Eq (Gate Equivalence) of 1 to the gates. All gates are equivalent in a homogenous component and so all the gates will be using the same symbol. To quickly fill this column, type `1` in the first cell of the Gate Eq column and press **Enter**.

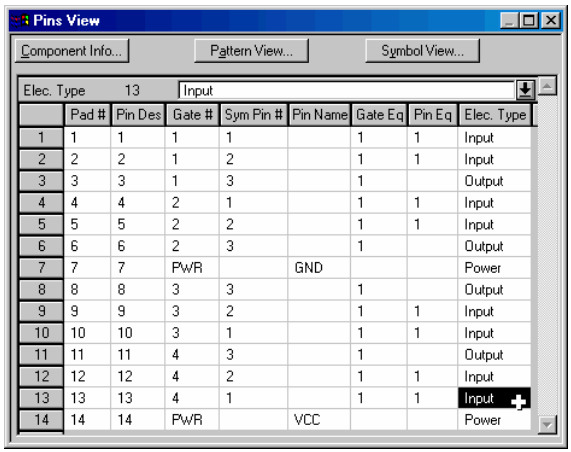
10. Assign Pin Eq (Pin Equivalence) values for the component if pin swapping is required when using the component in both P-CAD PCB and Schematic. The Pin Eq indicates which pins within a gate are logically equivalent and may be swapped. These values must be non-zero and identical for a swap to occur between two pins. Non-swappable pins are indicated with a zero or blank.

    Both input pins (Symbol Pin # 1 and 2) can be swapped in this example, so enter the same code for these pins, e.g. 1.

11. Finally, we will assign the Electrical Types to each pin, e.g. Input, Output and Power. The electrical types may be entered quickly by selecting the appropriate cell and typing the first letter of the electrical type name, for example, `I` for input and `O` for output. If there is more than one electrical type that starts with the same letter, type the letter again to select the next one, e.g. press `P` until the type Power appears.

> Alternatively, select a cell in the Electrical Type column and right-click to choose the **Electrical Types** command from the shortcut menu.

12. When entering electrical types by typing in the first letter, notice that the power and ground pins were automatically filled in with a value of "PWR" for the gate number. If you used the shortcut menu to enter the electrical types, PWR is not automatically added, so type in `PWR` to complete the spreadsheet as below.

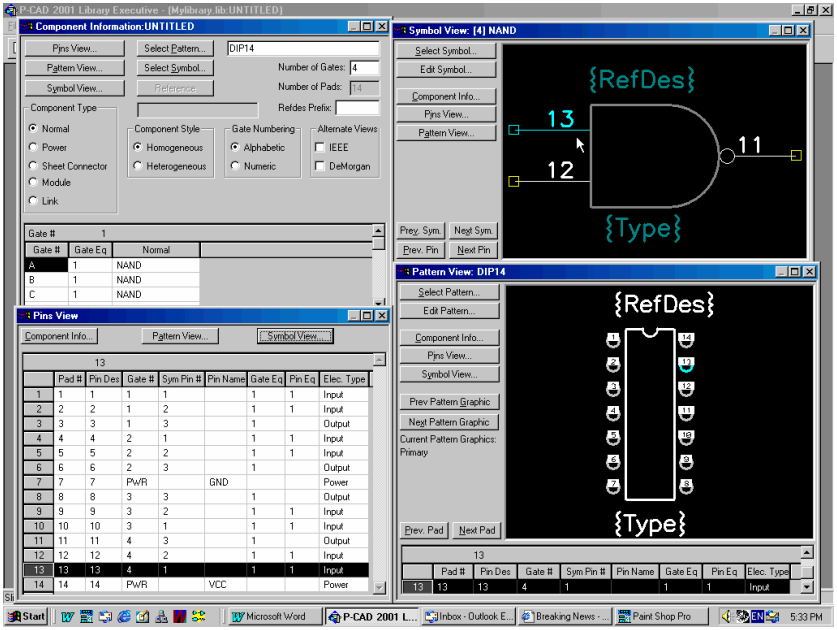


	Pad #	Pin Des	Gate #	Sym Pin #	Pin Name	Gate Eq	Pin Eq	Elec. Type
1	1	1	1	1		1	1	Input
2	2	2	1	2		1	1	Input
3	3	3	1	3		1		Output
4	4	4	2	1		1	1	Input
5	5	5	2	2		1	1	Input
6	6	6	2	3		1		Output
7	7	7	PWR		GND			Power
8	8	8	3	3		1		Output
9	9	9	3	2		1	1	Input
10	10	10	3	1		1	1	Input
11	11	11	4	3		1		Output
12	12	12	4	2		1	1	Input
13	13	13	4	1		1	1	Input
14	14	14	PWR		VCC			Power

Now we can check and validate our new component before we save it to a library.

Visually checking the pin information

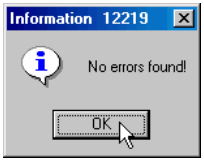
Before we leave the component views, we can have a quick visual check that the pins have been set correctly. As you select a Pin Number's row, the corresponding pin is highlighted in the Symbol View and its corresponding pad is highlighted in the Pattern View. In the example below, pin number 13 has been selected and highlighted in all views.



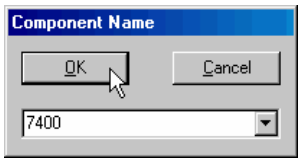
Validating and Saving a Component

As a final check that the component is correct, we can validate it.

1. Choose **Component » Validate** or click on the button on the Library Executive main toolbar. If the component and its pins have been set up properly, there should be no errors. If you have errors, the error description will indicate which part needs to be altered. Fix the error(s) and revalidate the component until the following dialog displays and click **OK**.



2. Choose **Component » Save** to complete the creation of the new component. The *Component Name* dialog displays.

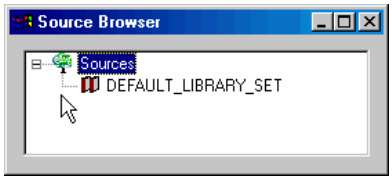


3. Name the new component `7400` and click **OK**.
4. Close the *Component Information* dialog by clicking on the **Close** button .

Using the Source Browser

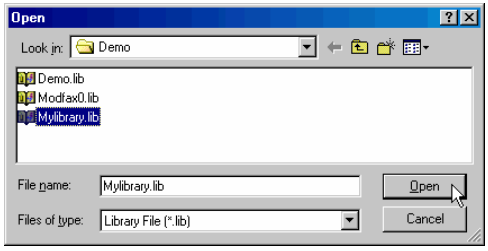
The Source Browser can be used to display all of the component data currently available in the P-CAD Library Executive. We will add our new library to the Default Library Set that will load every time you access the Library Executive and then check the contents of the new library using the Source Browser.

1. To access the Source Browser, click the Source Browser button on the Library Executive toolbar or choose **View » Source Browser** from the menu. The Source Browser window displays.



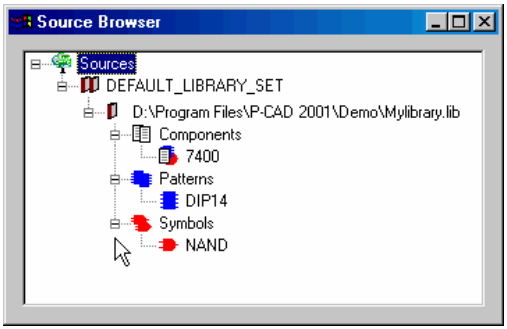
The Source Browser is organized in a tree-like structure that shows the relationship between the component data sources. The tree structure allows you to view all Library Executive library sets and their contents at various levels of detail by expanding or collapsing the branches of the tree. To expand a branch, simply click the **+** to the right of its name. Expanded branches are shown with a - sign. To collapse the branch, click the **-** sign.

2. To add our new library to the Default Library Set, right-click on **DEFAULT_LIBRARY_SET** and choose **Add Library** from the shortcut menu. The *Open* dialog displays.



3. Select `Mylibrary.lib` from the list of libraries available in the `Demo` folder of the P-CAD installation directory and click **Open**.

4. Expand the Default Library Set to display the names of libraries already added. Then expand `Mylibrary.lib` to view the components, patterns and symbols stored in it.



5. Double-click on a component name, e.g. 7400, to display its *Component Information* dialog.

6. Double-click on a pattern name, e.g. DIP14, and the pattern will display in the Pattern Viewer. You can then edit the pattern by clicking **Edit** in the Pattern Viewer, if required. Click **Close** to return to the Source Browser.

7. Double-click on a symbol name, e.g. NAND, to view or edit the symbol via the Symbol Viewer. Click **Close** to return to the Source Browser.

For more information about the other features available in the Source Browser, see *The Source Browser (page 91)*.

CHAPTER 4

Libraries

This chapter provides important information about the structure and characteristics of P-CAD libraries, including:

- Integrated, pattern and custom libraries
- Using the Library Index
- Creating, updating and saving libraries
- The relationship between and usage of components, patterns and symbols
- Pin numbers, pad numbers and pin designators
- Pattern library naming conventions
- Updating libraries with swap information
- Errors preventing saving
- Copying, renaming and using aliases
- Compacting libraries

Integrated, Pattern and Custom Libraries

A library with components having both pattern and symbol information is an integrated library. Both schematics and printed circuit boards can be built with the same components from an integrated library.

P-CAD includes 334 new component libraries, containing over 27,000 components. These libraries have been developed by the P-CAD Library Development Center in accordance with the ISO 9001 certifed component development process that they have established.

P-CAD supports two types of libraries: integrated and pattern. In P-CAD, a library can contain three types of items: logical components, pattern graphics and symbol graphics. Components contain only logical information – the associated graphics for Schematic symbols and PCB patterns are stored separately. This is the key to understanding how P-CAD libraries are structured.

Integrated Libraries

Integrated libraries contain intelligent components suitable for use with both P-CAD Schematic and PCB. The components in these libraries have logical pin designators and pin data specific to the type of the component (e.g. SN7400N). These components reference the appropriate Schematic symbol graphics and PCB pattern graphics (also known as footprints or decals).

In a typical integrated library, many components reference the same pattern graphic since those components have the same PCB pattern. Note that the component gives the pattern and symbol graphics all their "intelligence"; pattern graphics and symbol graphics do not contain any logical information such as pin designator values.

Pattern Libraries

Pattern libraries contain generic components that have generic pin designators and pin data and a reference to a PCB pattern graphic. These libraries do not contain Schematic symbol graphics and are unusable with P-CAD Schematic.

If you use P-CAD Schematic, you must use the integrated libraries since the PCB-specific libraries contain no symbol graphics. If you import a netlist from P-CAD Schematic, it is recommend you use the integrated libraries in PCB. This allows you to maintain synchronized PCB and schematic designs between P-CAD Schematic and P-CAD PCB in such areas as:

- Hotlink support
- ECO support
- Pin and gate swapping

If you use a third party schematic editor to create netlists for PCB, it is recommended you use the generic PCB pattern libraries in PCB. These generic pattern libraries are identified by names starting with `pcb`, e.g. `pcbmain.lib`.

Custom Libraries

The component libraries supplied with the P-CAD family of applications may be updated in future releases, so it is **strongly suggested that you do not modify the P-CAD-supplied libraries**. Save your custom components to a separate library, for example `Custom.lib`. Then, you can install future releases of P-CAD libraries and maintain your custom components.

Viewing Contents of the Libraries

To view the contents of any library, use the browse function of the *Place Component* dialog in P-CAD Schematic and PCB, or use the capabilities of the Source Browser and the file Viewer.

You can print a complete, formatted listing of the components within each library with the report feature. For complete information on using the report features, see *Reports (page 145)*.

Using the Library Index

Each new P-CAD supplied library is based on a component range available from a device manufacturer. A complete library index, `Library Index.XLS`, can be found in the `Lib` folder of the P-CAD installation directory. Use this file to review the complete list of new components.

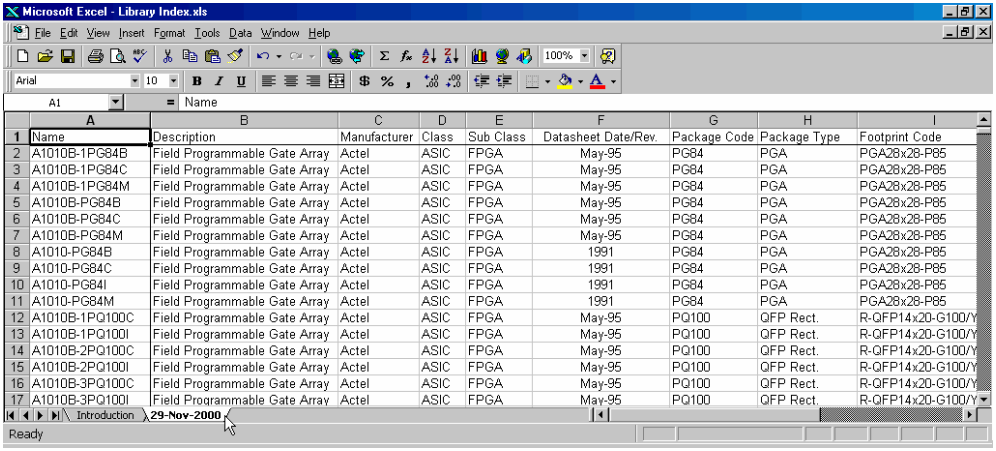
Each component entry has the following information:

Column	Description
Name	This is the Component Type.
Description	This is a description of what the component actually is.
Manufacturer	This is the company that actually makes the component.
Class	This is the general class that the component falls into, e.g. Communication, Logic.
Sub Class	This is the more specific sub class that the component falls into, e.g. for the Logic class, the component might fall into the sub class of Gate, Register, Switch, etc.
Datasheet Date/Rev.	This is the date/revision number of the datasheet associated with the component.
Package Code	This is the manufacturer's code for the component package.
Package Type	This is the type of package that the component comes in, e.g. DIP, CAN.
Footprint Code	This is the name of the associated pattern that is used for the component in the PCB Editor.
SIM Model File	This is the associated model file for the component. If the component has an entry in this field, then the component can be simulated.
Revision History	This is an entry showing when the component was first available in P-CAD and any subsequent modifications that have been made.

Each library name is based on the manufacturer's name and the library sub-class. The library class is also included when necessary to distinguish between sub-classes. For example, the NEC 8-bit Microcontrollers are in a library called "NEC Microcontroller 8-Bit.lib, and the Analog Devices operational amplifiers are in a library called "AD Operational Amplifier.lib".

To use the spreadsheet to locate a component and therefore identify which library you should add to the current open libraries list, follow the steps below:

1. Using Windows Explorer, browse to the Library Index (`Library Index.xls` in the `Lib` folder of the P-CAD installation directory) and double-click on the file;'scon. The Library Index displays as a spreadsheet.



	A	B	C	D	E	F	G	H	I
1	Name	Description	Manufacturer	Class	Sub Class	Datasheet Date/Rev.	Package Code	Package Type	Footprint Code
2	A1010B-1PG84B	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PG84	PGA	PGA28x28-P85
3	A1010B-1PG84C	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PG84	PGA	PGA28x28-P85
4	A1010B-1PG84M	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PG84	PGA	PGA28x28-P85
5	A1010B-PG84B	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PG84	PGA	PGA28x28-P85
6	A1010B-PG84C	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PG84	PGA	PGA28x28-P85
7	A1010B-PG84M	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PG84	PGA	PGA28x28-P85
8	A1010-PG84B	Field Programmable Gate Array	Actel	ASIC	FPGA	1991	PG84	PGA	PGA28x28-P85
9	A1010-PG84C	Field Programmable Gate Array	Actel	ASIC	FPGA	1991	PG84	PGA	PGA28x28-P85
10	A1010-PG84I	Field Programmable Gate Array	Actel	ASIC	FPGA	1991	PG84	PGA	PGA28x28-P85
11	A1010-PG84M	Field Programmable Gate Array	Actel	ASIC	FPGA	1991	PG84	PGA	PGA28x28-P85
12	A1010B-1PQ100C	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PQ100	QFP Rect.	R-QFP14x20-G100/Y
13	A1010B-1PQ100I	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PQ100	QFP Rect.	R-QFP14x20-G100/Y
14	A1010B-2PQ100C	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PQ100	QFP Rect.	R-QFP14x20-G100/Y
15	A1010B-2PQ100I	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PQ100	QFP Rect.	R-QFP14x20-G100/Y
16	A1010B-3PQ100C	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PQ100	QFP Rect.	R-QFP14x20-G100/Y
17	A1010B-3PQ100I	Field Programmable Gate Array	Actel	ASIC	FPGA	May-95	PQ100	QFP Rect.	R-QFP14x20-G100/Y

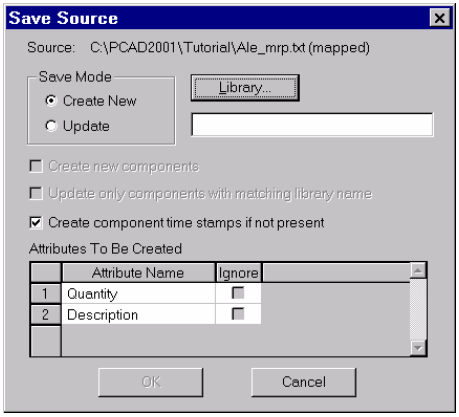
2. Click on the **<Date>** tab in the bottom left-hand corner to make the component sheet active.
3. Search the spreadsheet to find the component you require using the **Edit » Find** command.
4. Determine the name of the library in which the required component resides and add it to the open libraries list.

Creating and Updating Libraries

Creating a New Library

To create a new library from the selected source, follow these steps:

1. Choose the **Save to Library** command. Refer to *Accessing the Save To Library Command (page 46)* for additional information. The *Save Source* dialog appears.

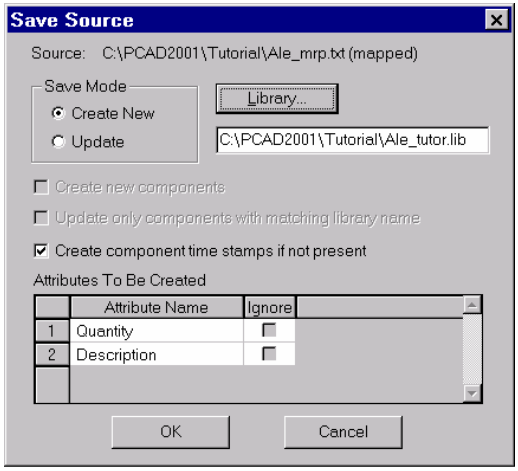


2. Select the **Create New** option button.
3. Enter in the desired library name in the **File Name** box, or click the **Library** button to navigate to and select the desired directory.
4. Click **Save**. The *Save Source* dialog now displays with the desired library name.
5. If the **Create component time stamps if not present** check box is selected, the attributes CreateDate and ModifyDate are added to the component with the current date and time.
6. The **Attributes To Be Created** box lists the attributes in the selected source. All attributes in this list are saved to the specified library unless the corresponding check box in the **Ignore** column is selected.
7. Click **OK**. A new P-CAD library has been created which contains the selected component attributes.

Updating an Existing Library

To update an existing P-CAD library, follow these steps:

1. Choose the **Save to Library** command. Refer to *Accessing the Save To Library Command (page 46)* for additional information. The *Save Source* dialog appears.



2. Select the **Update** option button.
3. Enter the desired library name in the box or click the **Library** button to navigate to and select the desired library. The selected library name is now displayed.

   If the **Create new components** box is selected, components in the source that are not already in the selected library are added to the library.

If the **Update only components with matching library name box** is selected, Library Executive searches the source for components whose ComponentLibrary attribute matches the selected library. Only matched components are updated or added to that library.

When updating a library, the component's time stamp attribute ModifyDate, if present, is automatically updated with the current date and time. If the **Create component time stamps if not present** check box is selected, and the ModifyDate attribute does not exist, the ModifyDate attribute is added to the component library.

All attributes that will be updated in the library are listed in the Attribute Conflict Resolution section.

Read-only attributes cannot be modified and cannot be saved to a P-CAD library. These attributes are displayed in red on the library Viewer and Query Result Table.

In this section, you can specify how to handle discrepancies between the source and the target library. Difference handling can be customized for each attribute by the following simple rules:

- If the **Ignore** check box is selected, the source attribute is ignored. It is neither created nor updated in the specified library.
- If the attribute exists in both the source and the library, and the **Source** check box is selected, the source is favored. The source attribute value replaces the present library attribute value.
- If the attribute exists in both the source and the library, and the **Library** check box is selected, the library is favored. The library attribute value remains unchanged.
- If the attribute in the source does not exist in the library component and the **Ignore** check box is cleared, it is added.

4. Click **OK**. The library has been updated to reflect the new attribute values.

   Any new components have been added if the Create new components check box was selected. Refer to *Creating a New Library (page 42)* for details about the completeness of components created by the Save To Library command.

Saving a Library

You can create or update a P-CAD library from a variety of sources:

- a modified P-CAD library
- an imported library from an external source
- a query result
- a cross link result

The only requirement is that you have the primary field, ComponentName, uniquely defined.

To save a source to a P-CAD library, choose the **Save to Library** command. Refer to *Setting up a Library Set (page 45)* for more information.

What is Saved?

You can customize which component attributes are stored in a P-CAD library. For example, if pricing is important for component selection, you can choose to save the Cost attribute. Later, you can query on this attribute for component selection.

Many attributes can be saved to a P-CAD library, including user-defined attributes that can be tailored to enhance your particular PCB design projects. The default data type, String, is always used when these attributes are saved.

Read-only attributes are unmodifiable and cannot be saved to a P-CAD library. There is one exception to this rule: when a new library is created, the read-only ComponentName field can be saved. Complete, integrated components can be created in this new library.

Read-only attributes are displayed in red on the library Viewer and Query Result Table.

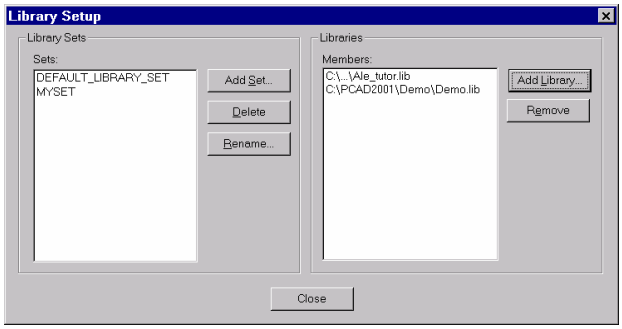
Setting up a Library Set

This section illustrates the process of setting up a library set. `MYSET` will contain the following libraries: `Demo.lib` and `Ale_tutor.lib`. These libraries can be found in the `Demo` and `Tutorial` folders in the P-CAD installation directory, respectively.

To make the `Demo.lib` and `Ale_tutor.lib` libraries available for Library Executive, you can define a library set containing the two libraries. You can define the library set from the Source Browser or from the *Library Setup* dialog.

To use the *Library Setup* dialog, follow these steps:

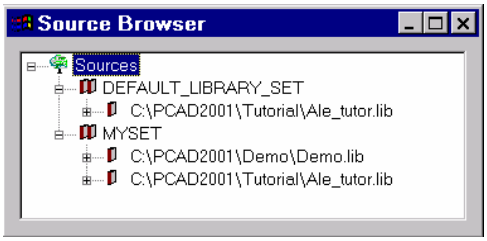
1. Choose **Library Setup** to open the *Library Setup* dialog.



2. Click the **Add Set** button. An edit box appears in the Library Sets box.
3. Type `MYSET` into the edit box to define your new library set. Press **ENTER**.
4. While `MYSET` is selected, click the **Add Library** button to navigate to and select the libraries you want to add to the set.

5. Navigate to the `Demo` folder in the P-CAD installation directory and choose `Demo.lib`. Click **Open**. `Demo.lib` appears in the Members box.
6. Repeat steps 4 and 5 to add the library `Ale_tutor.lib` in the `Tutorial` folder in the P-CAD installation directory to the library set `MYSET`.
7. Click **Close**.

   To view the contents of the library set, click the **Source Browser** button on the toolbar to access the Source Browser tree.
8. Expand both the root of the tree and library set `MYSET` to view its contents.



To add another library to the set, you can simply select the set in the Source Browser tree and choose **Add Library** from the shortcut menu.

Accessing the Save To Library Command

Choose the Save To Library command to update or create a P-CAD component library. The Save To Library command can be accessed from the Source Browser, the File Menu, and the Table Menu. The location from which you choose the Save To Library command defines the components you intend to store in the library. For example, choosing Save To Library from a Query Result table stores the components listed in that Query Result.

This section describes how and where to choose the Save To Library command to get the results you desire.

From the Source Browser

1. Select any library, query result, or external source file from the Source Browser tree.
2. Choose **Save to Library** from the shortcut menu. One of the following results occurs:
   - If you select a library, the components listed in that library are stored in the specified new P-CAD library. If modifications to that library were made while in Library Executive, that same library can be updated with this command to reflect those changes.
   - If you select a query result, the components listed in that Query Result table are stored or updated in the specified library.
   - If you select a cross link result, the components listed in that Cross Link Result table are stored or updated in the specified library.

- If you select an external source file, the components listed in the file Viewer are stored in the specified P-CAD library. Updates can be easily made to P-CAD libraries from an external source file using this command, making changes to your corporate component database easy to incorporate into your PCB designs.

From the Table Menu

1. Choose **Save to Library** from the **Table** menu in the Viewer for imported source files, query results, cross link results and P-CAD libraries.
2. The contents of the current table are saved to a library.

From the File Menu

1. Choose **Save to Library** from the **File** Menu.
2. The mapped, imported source file is saved to a library.

Time Stamps

In Library Executive, you can automatically mark the time when a component in a P-CAD library was created, modified or verified. These times are saved in the user-defined attributes CreateDate, ModifyDate, and VerifyDate fields. Time Stamp makes keeping track of libraries effortless.

The Time Stamp feature of Library Executive can save you both time and headaches, especially if more than one designer is using and modifying the same component libraries. For example, you may want to find all components in a P-CAD library that have been modified after a certain date. With this feature, you can quickly find what changes have been made since you last used the library.

Time stamps are available from these Library Executive commands and features:

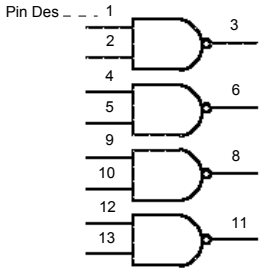
- **Save To Library:** The Save To Library command controls both the CreateDate and ModifyDate fields. The Save To Library command is discussed in to *Creating a New Library (page 42)*.
- **Verify:** The Verify command controls the VerifyDate field. For additional information on the Verify command, see Chapter 10, *Library Verification (page 137)*
- **Query:** Time Stamps can also be used in the Query utility. For information on how to search for a CreateDate, ModifyDate, or VerifyDate field, refer to *Querying Libraries (page 103)*.

Components, Patterns and Symbols

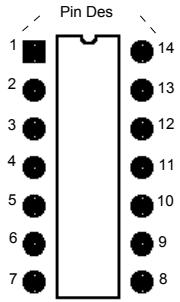
P-CAD integrated libraries contain components, patterns and symbols.

A component contains the logical and electrical data for a device and two graphical representations:

- One or more symbols for a Schematic representation.
- A pattern for a PCB representation.

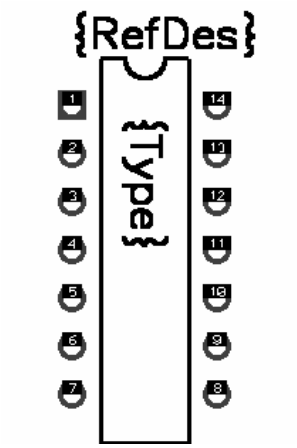


Schematic View of a component

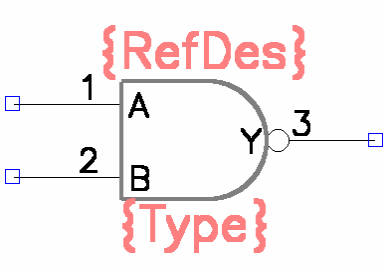


PCB View of a component

Each component consists of one or more logical gates packaged into a physical component. A single pattern is the basic graphical structure used in the creation and display of an entire component in PCB.



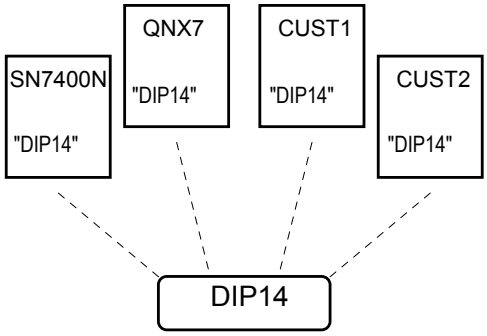
A symbol is the basic graphical structure used in the creation and display of a single gate in Schematic, as shown in the following figure.



The integrated libraries supplied with P-CAD can have symbols of different names that are graphically identical.

PCB cannot use a component or pattern by itself, it requires a component/pattern combination. Schematic needs a component/symbol combination. When a component is placed, it references the pattern and/or symbols attached to it for graphical structure. Numerous components can reference the same pattern or symbol. The component and the pattern and symbols it references must reside in the same library.

The following is an illustration of four components within a P-CAD library, all of which reference a DIP14 pattern. Although not shown, the same would hold true for components sharing symbols in Schematic.



Renaming or deleting a pattern could have a profound impact on a library. If you change the name of a DIP14 pattern to D14, for example, the four components shown above would reference a nonexistent pattern.

The graphical pattern DIP14 only needs to be stored as one entity. When you place a component, the component locates the named pattern structure and imports it into the design along with the component information. The same holds true for symbols.

A P-CAD library could contain many components, but only one pattern and one symbol, and it would be complete as long as all the patterns and symbols that the components reference exist in the library.

It is likely that a library will have multiple components using the same patterns and symbols. This saves space and makes global edits of a pattern or symbol efficient. It is also potentially dangerous, as any changes to a pattern or symbol affect all components referencing that pattern or symbol.

Component Types

There are five component types: Normal, Power, Sheet Connector, Module and Link. By specifying a component type, you can properly associate the net connections for that component. The component type is specified in the *Component Information* dialog.

- **Normal**: The most common type of component, for example diodes, DIPs and resistors. On a Normal component, the pins are associated with the net to which they are connected. Normal components appear in the netlist and the Bill of Materials.

- **Power**: A special component, used only in Schematic designs, representing a connection to a power source. This component type does not appear in the netlist or Bill of Materials. However, any pin connected to this component will be associated with the power net.
- **Sheet Connector**: A component with no net intelligence. This component type is commonly used to import non-intelligent graphics from a library, such as company logos or a chart. It does not appear in a netlist or Bill of Materials.
- **Module**: Used with hierarchical designs, a module component represents another sheet of circuitry in a design. Module components are used on the parent sheet to maintain connections to the link components.
- **Link**: Used with hierarchical designs to represent another sheet of circuitry in a design. Links are used on the child sheet to maintain connections to the module components.

Pattern Graphics in the Library Executive

Pattern graphics are the graphical representation of the pattern that is used when placing components in P-CAD PCB. Every pattern has at least one pattern graphic with the default name of Primary that is created when you use the Pattern Editor.

You can set up multiple and varying pattern graphics for the patterns associated with components. This allows differing pattern graphics to be automatically applied when placing a component, depending on whether the component is placed on the top or bottom of the board, how it is rotated and the chosen direction of solder flow. Pattern graphics may be a different orientation or a different size from the base pattern.

The pattern graphics are stored in the pattern file within a library. You can attach (copy) pattern graphics from other patterns to a target pattern as long as they are all in the same library. You can create new pattern graphics or modify existing pattern graphics using the Pattern Editor. Refer to *Pattern Graphics in the Pattern Editor (page 68)*.

Adding existing pattern graphics to a target pattern

1. Choose **Library » Pattern Graphics**. The *Library Pattern Graphics* dialog displays.
2. Open a source library where the target pattern and the already created pattern graphics reside. Click on **Library** to open the *Library Select* dialog and navigate to the required library and click **Open**.
3. Choose a target pattern to which you want to add additional pattern graphics by selecting the required pattern name from the Target Pattern drop down list. If only one pattern graphic exists for this target pattern, the word ‘Primary’ displays and the primary pattern graphic is shown in the view window. If there are other pattern graphics already attached, their names will be listed as well.
4. Click on the **Alternates** tab and search through a list of all source patterns in the library and select the new pattern graphic you want to copy to the target pattern. Type in a new pattern graphic name and click **Copy to Target**. The new name is added to the Target Pattern Graphics list. Click on names in this list to see the pattern graphics associated with the target pattern in the view window.

5. Click on the **Orientations** tab to specify which pattern graphics are to be used for a specific orientation. Up to eight different pattern graphics can be used to represent various component orientations, depending on whether the component is flipped or rotated by 0°, 90°, 180° or 270°.
6. Click on **Apply Pattern Changes to Library** and click **OK** to close the dialog.

For more details, refer to *Library Pattern Graphics (page 214)* in the command reference section.

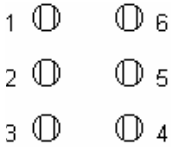
Pin Numbers vs. Pad Numbers vs. Pin Designators

An important fact to understand about components and patterns or symbols is that **pad numbers do not equal pin designator values** and that **symbol pin numbers do not equal pin designator values**, except by a coincidence.

Pin designators are the critical piece of data that map physical pads (in PCB) to logical pins (in Schematic). Pad numbers are just identifiers for pads, used to cross-reference with the pin designators when assigning the pin designator values. Similarly, symbol pin numbers are just pin identifiers. Using an ordered, linear, pad or pin numbering system makes identifying pad numbers and symbol pin numbers easier, but you could just as easily number them randomly and still assign pin designator values to them.

In the component/pattern/symbol combination, pattern and symbols contain only pad and symbol pin numbers, whereas components contain pin designators. Each pin designator references one pad number and one symbol pin number.

For example, suppose you are creating a 6-pin component with pin designators **A** through **F**. You could number your pads and assign pin designators as follows.

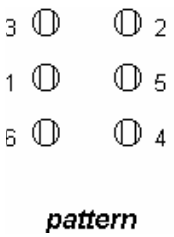


pattern

pin des	pad
A	1
B	2
C	3
D	4
E	5
F	6

component

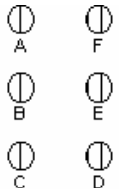
If your pads were numbered differently, the mapping would change, but the pin assignments would remain the same.



pin des	pad
A	3
B	1
C	6
D	4
E	5
F	2

component

Therefore, the pad numbers are irrelevant, except for the purpose of assigning pin designators. Regardless of the numbering, the pin designator assignments in this example are as follows:



This is analogous for Schematic symbol pin numbers.

Pattern Library Naming Conventions

When looking through the pattern libraries, most pattern names will be familiar. However, the following naming conventions may help to clarify the more obscure pattern names.

Patterns for surface mount packages are built to the current standards developed by IPC - "Association Connecting Electronics Industries". IPC claims that these land patterns are transparent to the manufacturing process but recommends that they should be optimized to suit the soldering type (wave, reflow) and assembly (components mounted on one or both sides of the board, etc).

Land patterns for BGA devices follow the assumptions found in standard, IPC-SM-782A, Amendment 2 (April 1999). The pads for this standard are defined by the etched copper, rather than by the solder mask. Land patterns for other surface mount components follow the assumptions found in standard, IPC-SM-782A, Amendment 1 (October 1996).

All pattern dimensions are calculated in metric units. Hard metric dimensions are employed in accordance with the JEDEC JC-11 "Metrication Policy", SPP-003B (February 1998). Departures to this policy are made for some silkscreen dimensions and critical dimensions such as pitch and row spacing.

Pattern Acronyms

The pattern naming convention is in sympathy with the IPC component names and the JEDEC Standard, JESD30-B, "Descriptive Designation System for Semiconductor-Device Packages" (April

1995). The system is based upon a minimum, compulsory 2 to 4-letter code that describes the Package Outline Style. Further optional characters are used to provide additional information such as the number of terminals and their position. Seven fields are available for the assignment of a name to each pattern. Pattern names of Chip Resistors, Chip Capacitors and certain other packages do not follow this system.

The following fields make up the acronym for a pattern name:

1. Features — Package Specific Features
2. Position — Terminal Position
3. Package — Package Outline Style
4. Dimensions — Major dimensions of package
5. -Form — - Lead Form
6. Count — Terminal Count
7. Supplementary — /Supplementary Information

Examples:

TSSO6x14-G16 Thin Shrink (TS) Small Outline (SO) with package dimensions of 6 & 14mm for the respective parameters of D & H and 16 Gull (G) wing leads.

DIP-24/D31 24 pin, Dual In-Line Package with a silk screen length of 31mm for the dimension, D.

1. Package Specific Features (nominal dimension)

Designator	Description
B	Bumpered (Corner Bumpers present) Employed with QFP family
C	Ceramic
E	Enlarged pitch (>1.27 mm)
F	Fine pitch (<=0.50 mm) Restrict to QFP family
L	Low profile (1.4 mm body thickness)
M	Metal
R	Rectangular
S	Shrink pitch (<=0.65 mm). All families except QFP
T	Thin profile (1.0 mm body thickness)
TS	Thin shrink (1.0 mm body thickness, <=0.65 mm pitch)
U	Ultra-thin profile (<1.0 mm body thickness)

2. Terminal/Pin Position (Seating plane is the bottom of the package)

Designator	Description
A	Axial - Terminals extend from both ends in the direction of the major axis of a cylindrical or elliptical package.
B	Bottom - Terminals extend from the bottom of the package.
D	Dual - Terminals are on opposite sides of a square or rectangular package or located in two parallel rows.
P	Perpendicular - Terminals are perpendicular to seating plane on a square or rectangular package. Restrict to PGA family.
Q	Quad - Terminals are on the four sides of a square or rectangular package or located in four parallel rows.
S	Single - Terminals are on one surface of a square or rectangular package in a single row.
Z	Zig-zag - Terminals are on one surface of a square or rectangular package arranged in a staggered configuration.

3. Package Outline Style

Designator	Description
AB	Axial Bipolar Capacitor
CAN	Can
CC	Chip Carrier
CY	Cylinder or Can
DO	Diode Outline
FM	Flange Mount
FP	Flatpack
GA	Grid Array
IL	In-Line Package
IP	In-Line Package (Restrict to DIP/SIP/ZIP)
LF	Long Form Horizontal Package
MELF	Metal Electrode Face
PM	Post/Stud Mount
RB	Radial Bipolar Capacitor
SO	Small Outline

SOT	Small Outline Transistor
TO	Transistor Outline
TP	Tape Pack

4. Major dimensions of package (mm)

Designator	Description
##x##	BGA and PGA - Silk Screen dimensions, DxE
/##x##	Diode Outline - Length by width, mmx10
##x##	QFP (except Bumpered style) - Body of the package dimensions, E1xD1
##x##	Thin Shrink Small Outline - Short package length by largest overall length including terminals, DxH

5. -Lead Form (or Terminal Shape)

Designator	Description
B	Butt or Ball - A short lead or solder ball intended for attachment perpendicular to the land structure.
C	"C" bend - A "C" shaped noncompliant lead bent down and under the body of the package.
F	Flat - A compliant or noncompliant, nonformed flat lead that extends away from the body of the package.
G	Gull wing - A compliant lead bent down from the body of the package with a foot at the end pointing away from the package.
J	"J" bend - A "J" shaped compliant or noncompliant lead bent down and back under the body of the package.
N	No lead - Metallized terminal pads located on the body of the package.
P	Pin or Peg - A tempered lead extending from the bottom of the package and intended for attachment to a plated through-hole in the land structure.
R	Wraparound - A metallized noncompliant terminal wrapped around the package body.
T	Through-hole - A terminal with flat or V-shaped cross-section, extending from the side of the body and intended for attachment to a through-hole.

W	Wire - An untempered wire lead extending from the body of the package.
Y	Screw - A threaded hole.

6. Terminal/Pin Count

Designator	Description
#	Number of terminals. Include those of the type described by the 'lead form' designator
(#)	Number of terminal positions. Include only if this figure differs from the number of terminals given above. Do not apply to Grid Arrays.

7. /Supplementary Information

Designator	Description
F	Flange, Tab or mounting on PCB board
H	Drill Diameter
M	Modified
N	Terminals numbered from a corner of the pattern
S	Stagger lead form
T	Terminals numbered from a center of one side of the pattern
V	Mounted vertically
W	Spacing lead form
#	This pin, at least, is absent

Letters may refer to the dimensions of a pattern parameter. Where this is the case, the dimension is given in millimeters and follows the letter. Example: CAN-12/D9.4 12 pin can with silk screen length of 9.4mm for the dimension, D.

Entering Equivalence Information

When adding Schematic information to existing pattern libraries, there are two equivalence columns in the *Pins View* dialog that should be correctly completed: the Gate Eq column and the Pin Eq column. These columns and their contents are detailed in *Library Basics (page 3)*.

The Gate Eq should be set to reflect actual gate equivalences. If this information is not set correctly, the Schematic **Utils » Renumber** command, the automatic RefDes increment feature in the **Place »**

Part command and the gate swapping functionality of the PCB **Utils » Optimize Nets** command may not behave as expected.

The Pin Eq should be set to reflect actual pin equivalences. If this information is not set correctly, the pin swapping functionality of the PCB **Utils » Optimize Nets** command may not behave as expected.

When the gate equivalence of a pin is filled in, all other pins in that gate automatically have their gate equivalence changed to match.

Errors that Prevent Saving a Component

You cannot save a component using the **Save** or **Save As** command if an error exists. For a list of error messages, see *System Messages (page 275)*.

Copying, Renaming and Using Aliases

When you copy a component to another library, you generally need to copy its pattern and symbol too. If patterns and symbols referenced by the component are not in the same library as the component, the component cannot be placed since its graphics are missing.

You can rename components, pattern and symbols, but this is an action that must be undertaken with care. If you prefer to use a different naming convention for your components, patterns and symbols, you can create aliases for them by choosing the **Library » Alias** command without having to alter the original name.

Aliases are preferable for a number of reasons: it's a safe way to use a variety of naming conventions; there is no danger of making global mistakes (which is possible when you rename) and the flexibility of using aliases allows the component, pattern, or symbol to be referenced by any of its aliases.

Compact Program

The COMPACT program is used to compress P-CAD component libraries that were translated from P-CAD PCB (6/400) pattern libraries or Tango-Schematic part libraries. Compacted libraries can be used directly, without uncompressing. The **Library » Translate** command does not automatically run the COMPACT routine. Uncompacted library files work; they just take up more disk space than compacted library files. The component libraries supplied with P-CAD have already been compacted.

The COMPACT program must be run from a DOS prompt. For example, if you choose **Library » Translate** to create a new P-CAD library called `Mycustom.lib`, then you can create an equivalent, compressed version of this called `Custom.lib` as follows:

```
C> COMPACT MYCUSTOM.lib CUSTOM.lib CUSTOM.ERR
```

The `Custom.err` file contains a list of any errors encountered during the compaction process. Once compaction is complete, you can delete the original `Mycustom.lib` file and the `Custom.err` file.

CHAPTER 5

The Pattern Editor

Introducing Pattern Editor

Pattern Editor allows you to create and modify patterns quickly. Once created, a pattern can be saved to a library or saved as a pattern file. This file can be modified, allowing you to create new patterns. Pattern Editor includes the Pattern Wizard which automates pattern generation. Pattern Editor contains a user-friendly interface which is very similar to P-CAD PCB.

This chapter lists the features which are unique to Pattern Editor. It then describes the Pattern Editor interface, detailing those features which are distinct from P-CAD PCB.

Pattern Editor Features

This section highlights some of the important Pattern Editor features:

- The ability to load and save patterns, or partially created patterns, from a P-CAD library.
- Pattern Wizard automates pattern creation.
- **Utils » Validate** command allows you to validate a pattern before saving it to a library.
- Enhanced **Place » Pad** command to support automatic incrementing of pad numbers.
- Enhanced manual pad renumbering supports single select pad renumbering and drag select for multiple pad renumbering.
- The ability to automatically match a pin's default pin designator to the pin number.
- All pattern attributes, including RefDes and Type, available from a single *Attributes* dialog.
- Separate commands for placing reference points (**Place » Ref Point**), glue points (**Place » Glue Point**) and pick points (**Place » Pick Point**).
- Pattern graphics can be added or removed
- Unique keyboard shortcuts and access from other programs in the P-CAD suite of products.

The Pattern Editor Interface

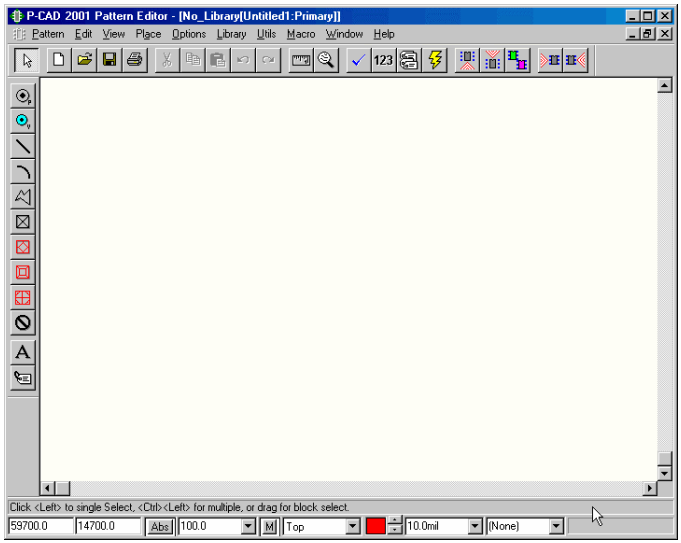
Pattern Editor is very similar to P-CAD PCB so much of it will be familiar. This section details those functions which are unique to Pattern Editor. If you need information about common features, consult your P-CAD user documentation.

Starting Pattern Editor

In addition to starting Pattern Editor from Windows 95, Windows 98, Windows 2000 and Windows NT, you can start it from other P-CAD programs by choosing the **Utils » Pattern Editor** command. You can also access Pattern Editor from Library Executive by choosing the **Pattern » New** or **Pattern » Open** commands.

Pattern Editor is also available from the Source Browser. By choosing the **View** command in the shortcut menu for a selected item in the Source Browser tree, you can view a pattern in the *Pattern Viewer* dialog. The **Edit** button on the *Pattern Viewer* dialog also opens the Pattern Editor.

When you start Pattern Editor, the interface appears as shown in the following figure.



The following Pattern Editor commands and functions are distinctly different in their usage or access than those of P-CAD PCB:

- **Pattern » Open**: Opens pattern files (`.pat`) that can be modified and saved to a library. See *Pattern Editor* online help for details.
- **Pattern » Save to File** and **Save to File As**: Saves a pattern file (`.pat`). This is a unique file type that allows you to keep patterns for later use, without having to save them to a library. Invalidated or incomplete patterns can be saved in this way. See *Working with Pattern Files (page 65)* for details.

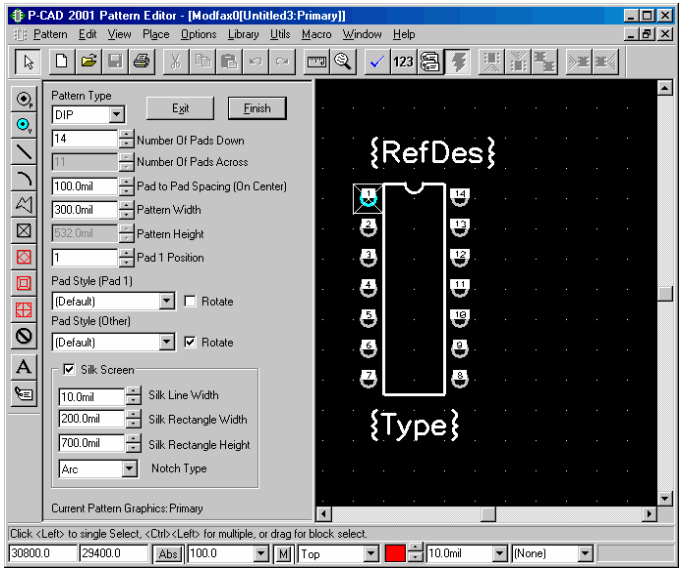
- **Pattern » Save** and **Save As**: Saves the current pattern to a library. See *Working with Pattern Files (page 65)* for details.
- **Pattern » Pattern Wizard**: Opens the *Pattern Wizard* which simplifies pattern creation. See *Pattern Wizard (page 61)* for more details.
- **Pattern » Attributes**: Allows you to create and modify pattern attributes for the current pattern file, including invisible attributes. This command is similar to the *Attributes* function of PCB's **File Design Info** command which presents design-level attributes. See *Pattern Attributes (page 67)* for details.
- **Place » Pad**: Opens a dialog where you can set a starting pad number and increment value before placing pads. See *Pad Placement (page 69)* for details.
- **Place » Ref Point**, **Place » Glue Point**, **Place » Pick Point** and **Place » Test Point**: Separate place commands replace the PCB **Place » Point** command. Toolbar buttons provide easy access to these commands. See *Placing Points (page 74)* for details.
- **Utils » Renumber**: Unlike PCB, pad renumbering in *Pattern Editor* can be performed automatically by dragging the mouse over a series of pads. Pads can also be renumbered manually by selecting one pad at a time. See *Renumbering Pads (page 70)* for details.
- **Utils » Validate**: Allows you to validate a pattern before saving it to a library. See *Working with Pattern Files (page 65)* for details.

Pattern Wizard

Pattern Wizard automates pattern generation, providing a quick way to create a pattern. A pattern created with Pattern Wizard is like any other pattern; you can modify it, save it as a pattern file and save it to a library.

To open Pattern Wizard:

1. Make sure no pads have been placed in the active window.
2. Choose **Pattern » Pattern Wizard** to open the *Pattern Wizard* dialog, shown in the following figure.



Pattern Wizard displays the last pattern you created. The left side of the wizard contains the fields you need to complete to create a pattern; the right side displays the pattern you are creating in a window.

Notice this window is like any other Pattern Editor window. You can resize this window, use zoom in and out and place objects into it.

The Pattern Wizard contains the following items:

- **Pattern Type**: The pattern type may be set to DIP, QUAD or ARRAY to indicate the type of pattern you want to create. The remaining fields change depending on your selection.
- **Number of Pads Down**: A DIP pattern, specifies the number of total pads. For example, 14 pads indicate 7 pads on each side of the DIP pattern. For QUAD or ARRAY pattern types, specifies the number of pads along each side of the pattern.
- **Number of Pads Across**: Specifies the number of pads along the top and bottom of a QUAD or ARRAY pattern. Not used for DIP patterns.
- **Pad to Pad Spacing (On Center)**: The vertical distance between the pads of a DIP pattern or the distance between any two pads for a QUAD or ARRAY pattern. The distance is measured from pad center to pad center.

  The value may be typed in the box using the appropriate mil or mm number. Use the slide arrows to change the value up or down by 10.0mil or 0.50mm. Use the **SHIFT** key with the slide arrows to increment the value by 0.1mil or 0.01mm.
- **Pattern Width**: The horizontal distance between the Pad columns of a DIP pattern measured from pad center to pad center. The distance between the pad columns of a QUAD pattern

measured from the outside edges of the pads. The *Pattern Width* value is not available for ARRAY patterns.

The value may be typed in the box using the appropriate mil or mm number. Use the slide arrows to change the value up or down by 10.0mil or 0.50mm. Use the **SHIFT** key with the slide arrows to increment the value by 0.1mil or 0.01mm.

- **Pattern Height**: The vertical distance between the pad rows of a QUAD pattern measured from the outside edges of the pads. The Pattern Width value is not available for DIP and ARRAY patterns.

  The value may be typed in the box using the appropriate mil or mm number. Use the slide arrows to change the value up or down by 10.0mil or 0.50mm. Use the **SHIFT** key with the slide arrows to increment the value by 0.1mil or 0.01mm.

- **Pad 1 Position**: Indicates which pad will become pad number 1 when the pattern is created on the PCB design. The value is only used for DIP and QUAD patterns. The remaining pads will be numbered in a counterclockwise order starting from pad 1.

  The Pad 1 Position value is not available for ARRAY patterns. The first pad of an ARRAY pattern is always the upper left corner pad (typically designated A1). If that pad has been removed using the Corner Pads options, then the next pad (A2) will be used for pad 1.

- **Pad Style (Pad 1-For DIP patterns only)**: Choose the pad style from those in the open PCB design to be used for pad 1. Use the Rotate box to rotate the pad style 90 degrees separately from the other pads of the DIP pattern.

- **Pad Style (Others-For DIP patterns only)**: Choose the pad style from those in the *Options Pad Style* dialog in PCB to be used for all other pads. Use the Rotate box to rotate the pad style 90 degrees separately from pad 1's style.

- **Pad Style (Top & Bottom-For QUAD patterns only)**: Choose the pad style from those in the open PCB design to be used for the top and bottom pads. Use the Rotate box to rotate the pad style 90 degrees.

- **Pad Style (Left & Right-For QUAD patterns only)**: Choose the pad style from those in the open PCB design to be used for the left and right side pads. Use the Rotate box to rotate the pad style 90 degrees.

- **Pad Style (For ARRAY patterns only)**: Choose the pad style from those in the open PCB design to be used for all pads. Use the Rotate box to rotate the pad style 90 degrees.

- **Corner Pads (For ARRAY patterns only)**: The outside four corner pads and the inside four corner pads (if a cutout section is present in the array pattern) may be added or removed individually by selecting the check boxes in the Corner Pads group.

- **Cutout Pads Down (For ARRAY patterns only)**: The number of pads to be removed from the center of the ARRAY pattern. The number of Cutout Pads Down must be greater than or equal to 0 and less than the Number of Pads Down. Also, the number of Cutout Pads Down must be an odd value if the Number of Pads Down is an odd value and even when the Number of Pads Down is even. When the Cutout Pads Across value is 0, the Cutout Pads Down value will be ignored.

- **Cutout Pads Across (For ARRAY patterns only)**: The number of pads to be removed from the center of the ARRAY pattern. The number of Cutout Pads Across must be greater than or equal to 0 and less than the Number of Pads Across. Also, the number of Cutout Pads Across must be an odd value if the Number of Pads Across is an odd value and even when the Number of Pads Across is even. When the Cutout Pads Down value is 0, the Cutout Pads Across value will be ignored.
- **Total Pads (For ARRAY patterns only)**: The number of pads in the ARRAY pattern.
- **Default Pin Designators (For ARRAY patterns only)**: Accesses the Default Pin Designator Assistant where you can designate the letters or numbers used as the beginning values for the Default Pin Des to name the rows and columns. Specify the starting corner and choose the characters you want to skip when numbering. Skipped characters can only be letters; numbers cannot be skipped. The characters chosen as the beginning of a row or column cannot be used as skipped characters and vice versa.

  For DIP and QUAD patterns, the Default Pin Designators are automatically numbered starting at the pad position indicated by the Pad 1 Position entry.

- **Silk Screen**: The silk screen may be turned on or off by selecting this check box.
- **Silk Line Width**: The line width used to create and display all silkscreen lines.

  The value may be typed in the box using the appropriate mil or mm number. Use the slide arrows to change the value up or down by 1.0mil or 0.10mm. Use the **SHIFT** key with the slide arrows to increment the value by 0.1mil or 0.01mm.

- **Silk Rectangle Width**: The width of the silk screen for the pattern. The value is measured from centerline to centerline.

  The value may be typed in the box using the appropriate mil or mm number. Use the slide arrows to change the value up or down by 25.0mil or 1.00mm. Use the **SHIFT** key with the slide arrows to increment the value by 0.1mil or 0.01mm.

- **Silk Rectangle Height**: The height of the silk screen for the pattern. The value is measured from centerline to centerline.

  The value may be typed in the box using the appropriate mil or mm number. Use the slide arrows to change the value up or down by 25.0mil or 1.00mm. Use the **SHIFT** key with the slide arrows to increment the value by 0.1mil or 0.01mm.

- **Notch Type**: For DIP patterns, the Notch Type may be either None, Arc, Square, or Triangle.

  For QUAD and ARRAY patterns, the Notch Type may be either None, Upper Left, Upper Right, Lower Left, or Lower Left.

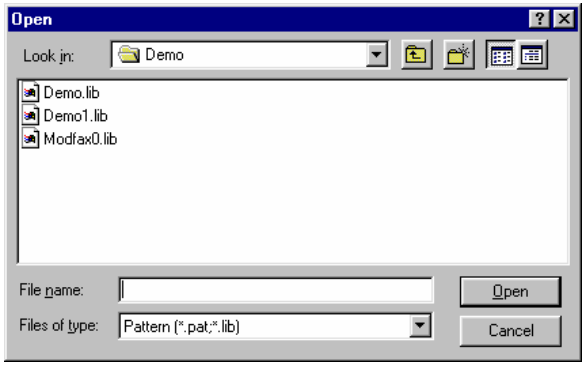
- **Current Pattern Graphics**: Displays the pattern graphic name linked to the component.

Working with Pattern Files

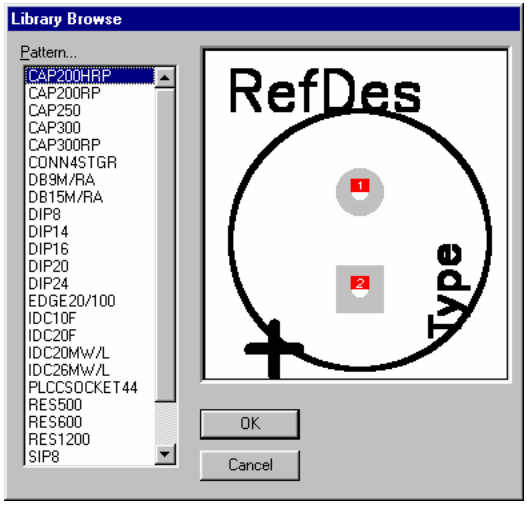
In Pattern Editor, patterns can be saved as pattern files (`.pat`). This allows you to save partially created patterns and to create pattern templates that you can use to create new patterns. Before a pattern can be copied to a library, it must pass a validation test. If it fails, you can save it as a `.pat` file and return to it at a later time.

Loading a Pattern from a Library

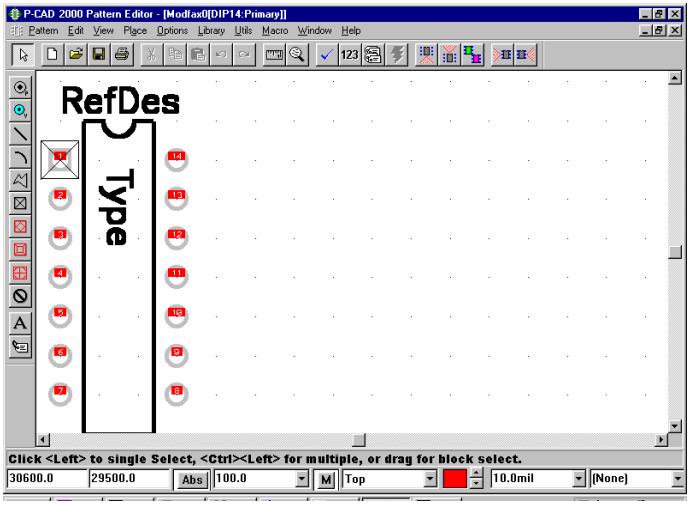
1. You can load an existing pattern from a library and modify it to create a new pattern. When you choose **Pattern » Open**, the *Open* dialog appears.



2. Notice that this dialog allows you to open pattern files (`.pat`) and library files (`.lib`). When you choose a library file and click **Open**, the *Library Browse* dialog opens, as shown in the following figure.



3. Select a pattern from the list (e.g. DIP14) and click **OK**. The pattern is loaded into *Pattern Editor*. When a pattern is loaded from a library, the library and pattern name appear in the title bar.



Notice that the pattern name and source library appear in the title bar even though this pattern has not been saved as a `.pat` file.

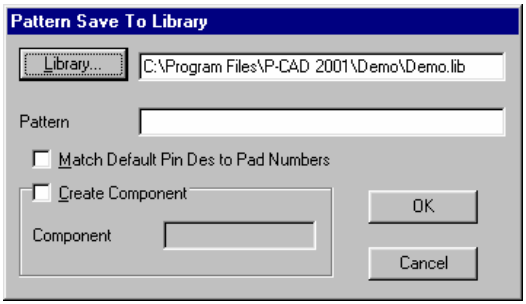
Saving a Pattern to a Library

Once you have created a pattern, you can save it to a library after it has been validated.

1. Choose **Utils** » **Validate**.

> If your pattern fails validation, an error message appears. For a list of error messages, see *Appendix B, System Messages (page 275)*.

2. Choose **Pattern** » **Save** or **Pattern** » **Save As**. The *Pattern Save To Library* dialog opens.



3. Select the library you want to save the pattern to, then specify a pattern name. If the pattern was loaded from a library, that library appears in the Library box and the pattern name appears in the Pattern box.

4. If you want to just create a pattern (not a component), click **OK**.

   To automatically create a component that corresponds to the pattern, select the **Create Component** check box and give the component a new name in the **Component** box before you click **OK**. If the existing default pin designators are correct for the new pattern, you do not have to change them. You can set the default pin designators to **Match Default Pin Des to Pin Numbers** by selecting that check box.

   If the **Create Component** box is left cleared, an electrically complete component can be generated with this pattern.

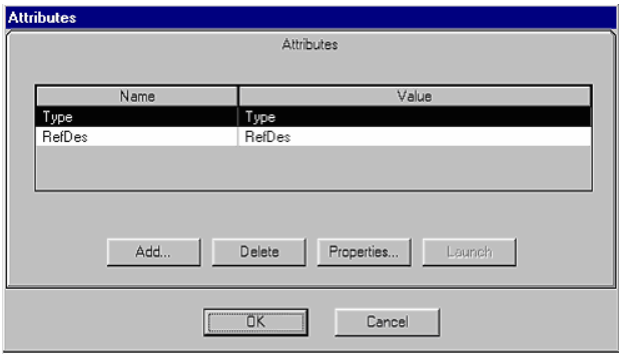
Saving a Pattern as a Pattern File

The **Pattern » Save to File** and **Pattern » Save to File As** commands save a pattern as a pattern file (`.pat`) without having to save them to a library, allowing you to keep patterns for later use.

Pattern Attributes

All pattern-level attributes, including invisible attributes like RefDes and Type, can be viewed and modified by choosing the **Pattern » Attributes** command.

When you choose **Attributes**, the *Attributes* dialog opens.



In the *Attributes* dialog you can view, add, modify, or delete attributes or access a web site. The dialog contains a two-column table showing the name of the attribute in the left column and corresponding value in the right column.

- **Adding an Attribute**: Click the **Add** button to open the *Attribute Property* dialog. Enter a name and value for the attribute and set attribute properties. Click **OK** and the attribute is added to the table.

- **Viewing or Changing Attribute Properties**: Select an attribute from the table and click the **Properties** button (or double-click the attribute) to open the *Attribute Property* dialog to view or change an attribute's properties.
- **Deleting an Attribute**: Select an attribute in the table and click **Delete**, or press the **DELETE** key.
- **Launching a reference link**: When the special attribute *Reference*, whose value is a reference link, is added to the pattern, you can select the Reference attribute and click the **Launch** button to start an application, access a web address, or open a document.

For details about this function, refer to your *P-CAD PCB User's Guide*.

Pattern Graphics in the Pattern Editor

This feature allows you to set up multiple and varying pattern graphics for the patterns associated with components. This allows differing pattern graphics to be automatically applied for a component, depending on whether the component is placed on the top or bottom of the board, how it is rotated and the chosen direction of solder flow. Pattern graphics can be added to a primary base pattern through the Pattern Editor, especially handy when you need to modify a pattern graphic, e.g. change its size, before adding it. The Library Executive, however, includes a more comprehensive way of handling pattern graphics, so refer to *Pattern Graphics in the Library Executive (page 50)*.

Add, Remove and Rename Pattern Graphics

Load the base pattern that you want to add the pattern graphics to. From the **Pattern** menu, select one of the following:

- **Add Pattern Graphics:** This command allows you to create new, empty pattern graphics and add them to the base pattern. Also available by clicking on the button on the main toolbar.
- **Remove Pattern Graphics:** This command allows you to remove pattern graphics from the base pattern. Also available by clicking on the button on the main toolbar.
- **Rename Pattern Graphics:** This command allows you to change the name of any of the pattern graphics already attached to the base pattern. Also available by clicking on the button on the main toolbar.

Next and Previous Pattern Graphics

- Choose **View » Next Pattern Graphic** or **View » Previous Pattern Graphic** to switch between the pattern graphics of a pattern as you can only work on one pattern graphic at a time. Also available by clicking on the (next) and (previous) buttons on the main toolbar.

Orientations

Choose **Utils » Orientations** to specify which pattern graphics are to be used for a specific orientation. Up to eight different pattern graphics can be used to represent various component orientations, depending on whether the component is flipped or unflipped and whether it is rotated by 0°, 90°, 180° or 270°. To use the orientations when placing components in P-CAD

PCB, make sure Autoswap is turned on. Refer to your *P-CAD PCB User's Guide* for more information about pattern graphics orientations.

Pattern Graphics in the Pattern Wizard

Whenever the Pattern Wizard is invoked, it operates on the current pattern graphics. The Wizard includes a non-editable field that displays the current pattern graphic. The Title bar also displays which pattern graphic is current, in the format:

```
[Library name [Pattern name: Current Pattern Graphics name]]
```

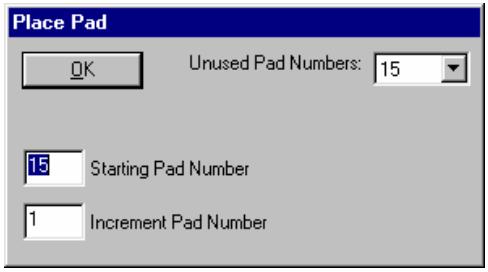
This means that you can add several blank pattern graphics to a pattern, switch to the pattern graphic you want to work with using the Next/Previous Pattern Graphics options and then use the Pattern Wizard to place the particular graphic elements.

Pad Placement

In Pattern Editor, the **Place » Pad** command shows you the next available pad number and lets you increment pad numbers.

To place a pad:

1. Choose **Place » Pad** or click the toolbar **Pad** button to open the *Place Pad* dialog.



2. The Unused Pad Number box defaults to the next unused pad number; click the **Down Arrow** to display the list of unused pads. Choose a number or choose **>#** to begin numbering from that number (#) onwards.
3. The Starting Pad Number box defaults to the next available pad number. To change the default value, type a new number in the box.
4. If you place a pad with a duplicate pad number, you hear a warning beep.
5. Enter the number (positive or negative) by which you want to increment pad numbers in the Increment Pad Number box.

   You're now ready to start placing pads.
6. Click **OK** to begin.

   You can click, drag and release for more accurate placement with a ghosted pad outline.

The Status Line shows the pad number of the pad about to be placed and also indicates if the pad number is a duplicate.

The **N** key increments the pad number about to be placed by one; **SHIFT**+**N** decrements the number. Thus if 6 is the next available pad number, pressing **N** would result in the pad being assigned number 7; pressing **SHIFT**+**N** would result in the pad being assigned number 5.

7. Continue adding a sequence of pads by clicking in the workspace. As you do, each is incremented according to the options you set in the dialog.
8. To stop, **right-click**, press **ESC**, or click the **Select** toolbar button.

   The **Undo** command removes the entire sequence of pads. The **BACKSPACE** key removes the last placed pad.

Rotate or Flip a Pad

In Pattern Editor, pads can be flipped as well as rotated during placement using one of the following methods:

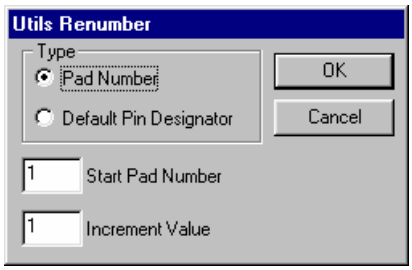
- To flip a pad as you are placing it, press **F** while the pad is ghosted before final placement. When you flip a pad, the hole range does not flip with it.
- To rotate a pad as you are placing it, press the **R** key while the pad is ghosted before final placement. The resulting rotation angle is applied to the next pad you place.
  Refer to your *P-CAD PCB User's Guide* for additional information.
- To rotate or flip a pad after it has been placed, select it and press **R** to rotate or **F** to flip. The **R** key rotates the pad 90 degrees, **SHIFT**+**R** rotates by the rotation increment set in **Options » Configure**.

Renumbering Pads

The renumbering function in Pattern Editor, unlike PCB, supports single select pad renumbering and drag select for multiple pad renumbering. Remember to turn on the display of the **Pad Numbers** in the **Miscellaneous** tab of the *Options Display* dialog.

To number the pads you placed:

1. Choose the **Select** tool.
2. Change to the layer on which the pads are placed.
3. Choose **Utils » Renumber** to open the *Utils Renumber* dialog.



4. Specify **Starting Pad Number** and **Increment Pad Number**, as appropriate.
5. Click **OK**. Notice that the cursor changes to the crosshair shape.

   You are in a temporary mode of assigning numbers, so every time you hold down the left-button and drag it over a pad, you renumber that pad (you can also click a pad to renumber it).

   For example, the first pad you drag the mouse over would be number 1 (if Starting Pad Number was specified as 1), the second pad number 2 (if the Increment Pad Number was specified as 1).

   As you drag the mouse over a pad while in the **Renumber** mode, it shows that a number has been assigned. Once a pad is renumbered, you cannot change its number without choosing the **Renumber** command again.

   The **Status** line information area displays the pad number every time you (re)number a pad.

   The **N** key increments the pad about to be renumbered by one; **SHIFT**+**N** decrements the number. Thus if 6 is the next available pad number, pressing **N** would result in the pad being assigned number 7; pressing **SHIFT**+**N** would result in the pad being assigned number 5.

   You can use the unwind feature to reverse the renumbering process. The **BACKSPACE** key unwinds the renumbering.
6. **Right-click** or press **ESC** to end the Renumber sequence.

   The **Undo** command reverses the entire renumbered sequence.

   You can also select a pad and choose **Properties** to open the *Pad Properties* dialog and assign a pad number.

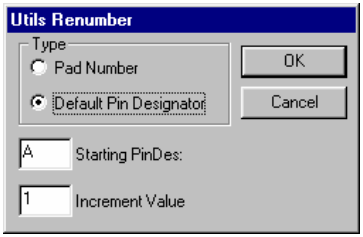
Renumbering Default Pin Designators

The renumbering function in Pattern Editor, unlike in PCB, supports single select pad renumbering and drag select for multiple pad renumbering. Remember to turn on the display of the Pin Des in the **Miscellaneous** tab of the *Options Display* dialog.

To number the pin designators on the pads you placed:

1. Choose the **Select** tool.
2. Change to the layer on which the pads are placed.

3. Choose **Utils » Renumber** to open the *Utils Renumber* dialog.



4. Select **Default Pin Designator** for the Type.
5. Specify **Starting PinDes** and **Increment Value**, as appropriate.
6. Click **OK**. Notice that the cursor changes to the crosshair shape.
7. You are in a temporary mode of assigning numbers, so every time you hold down the left-button and drag it over a pad, you renumber that pad's **Default Pin Designator**. You can also click a pad to renumber it.

   For example, the **Default Pin Designator** of the first pad you drag the mouse over would be number A1 (if **Starting PinDes** was specified as A), the second pad number A2 (if the **Increment Value** was specified as 1).

   As you drag the mouse over a pad while in the **Renumber Pin Des** mode, it shows that a default pin designator has been assigned. Once a pad's DPD is renumbered, you cannot change its DPD without rerunning the **Renumber** command.

   The Status Line information area displays the Default Pin Designator every time you (re)number.

   The **N** key increments the pad about to be renumbered by one; **SHIFT**+**N** decrements the number. Thus if 6 is the next available pad number, pressing **N** would result in the pad being assigned number 7; pressing **SHIFT**+**N** would result in the pad being assigned number 5.

   You can use the unwind feature to reverse the renumbering process. The **BACKSPACE** key unwinds the renumbering.

8. Right mouse-click or press **ESC** to end the Renumber sequence.

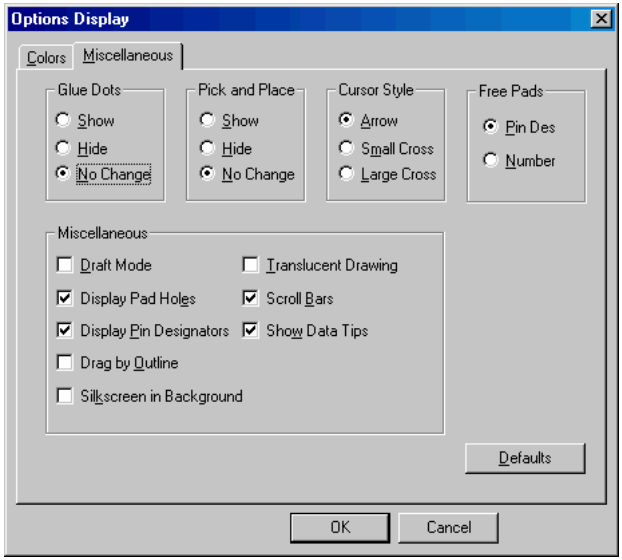
   The **Undo** command reverses the entire renumbered sequence.

   You can also select a pad and choose **Properties** to open the *Pad Properties* dialog and assign a pad number.

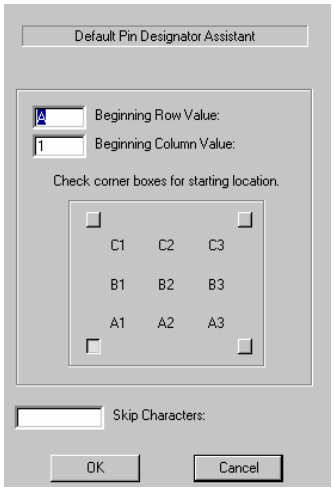
Default Pin Designators in the Wizard

While in the Pattern Wizard, the default pin designators for ARRAY type patterns can be changed using the Default Pin Designator Assistant.

1. In the **Miscellaneous** tab of the *Options Display* dialog, select the **Pin Des** check box in the Free Pads frame.



2. Click **OK** to return to the Pattern Wizard.
3. Click the **Default Pin Designator** button to display the Default Pin Designator Assistant.



The example box in the center of the dialog shows how the different values affect the numbering. Any changes made to the row and column values are immediately reflected here, except for the Skip Characters.

4. Specify the **Beginning Row Value** and **Beginning Column Value** in the appropriate boxes. You can use letters or numbers as a beginning value.
5. Check the corner where the numbering should begin. In the example, the upper left corner is selected.
6. If you want to skip a letter, enter the letter in the Skip Characters box. Only letters can be skipped. A character used as a Beginning Row Value or Beginning Column Value cannot be designated as a Skip Character.
7. Click **OK** to apply the default pin designator renumbering.

Placing Points

Separate commands let you place Reference Points, Glue Points, Pick Points and Test Points on your pattern. Those commands are discussed in this section.

Reference Points

The **Place » Ref Point** command places a Reference Point on a pattern before it is saved to a library.

You need to specify a Reference Point on a pattern object before it is saved to a pattern library. The Ref Point appears in the same color as the 1x grid color in **Options » Display**.

When you place a Reference Point on an object, the pattern will move with the cursor at the reference point. Patterns are flipped and rotated about the Reference Point. For most patterns, select a reference point in the center of pad #1.

Pattern Editor automatically places a Ref Point on a pad or at center of pads if you save a pattern to a library without a Ref Point.

Glue Points

The **Place » Glue Point** command places a glue point on a pattern before it is saved to a library.

Glue Points are used to hold components in place until they are soldered during manufacturing. A glue dot can be placed as part of a pattern before saving the pattern to a library.

Pick Points

The **Place » Pick Point** command places a Pick Point on a pattern before it is saved to a library.

Pick Points provide reference points in directing the pick and place mechanism (or auto insert) in manufacturing (picking up the component and placing it on the board).

Test Points

The **Place » Test Point** command places a Test Point on a pattern before it is saved to a library.

Test Points can be placed on any non-signal layer and can be placed either on the Top or Bottom of the board. The side of the board is independent of the layer name. For example, if you place a test point on the Bottom Silk layer and set the side of board to Top, the test point can be tested from the top. New design rules/attributes have been implemented to allow for checking of test point violations.

Placing a Point

To place a point:

1. Choose the appropriate **Place** command (**Ref Point**, **Glue Point** or **Pick Point**) or click one of the point buttons on the toolbar.
2. Click the cursor where you want to place the point.

   You can move a point by selecting it and dragging it to a new location.

Pick Points and Glue Points can be flipped to the bottom layer. They use the line color from the layer that they are on.

Keyboard Shortcuts

This section contains a list of keyboard shortcuts unique to Pattern Editor.

Press these keys:	To do this:
BACKSPACE (Unwind)	Used as unwind command while placing objects with multiple segments (e.g. lines, polygons, pads). Each **BACKSPACE** stroke unwinds the previously placed item.
N and **SHIFT**+**N** (increment and decrement)	**N** increments a pad by one when placing or renumbering pads. **SHIFT**+**N** decrements a pad by one when placing or renumbering pads.

CHAPTER 6

The Symbol Editor

Introducing Symbol Editor

Symbol Editor allows you to create and modify symbols quickly. Once created, a symbol can be saved to a library or as a symbol file that can be modified, allowing you to create new symbols. Symbol Editor includes the Symbol Wizard that automates symbol generation.

Symbol Editor's interface is similar to P-CAD Schematic.

This chapter lists Symbol Editor features and describes the Symbol Editor interface, detailing those features which are distinct from P-CAD Schematic.

Symbol Editor Features

This section highlights some of the important Symbol Editor features:

- The ability to load and save symbols from a P-CAD library.
- The ability to load and save partially created symbols.
- Symbol Wizard automates symbol creation.
- **Utils » Validate** command validates a symbol before saving it to a library.
- Enhanced **Place » Pin** and **Pin Properties** functions support automatic incrementing of pin numbers, easy to use display characteristics and a display area that shows how the pin will appear.
- **Pin Properties** includes ability to increment pin names.
- Enhanced manual pin renumbering supports single select pin renumbering and drag select for multiple pin renumbering.
- The ability to automatically match a pin's default pin designator to the pin number.

- All symbol attributes, including RefDes and Type, available from a single *Attributes* dialog.
- Unique keyboard shortcuts.
- Access from other programs in the P-CAD product suite.

The Symbol Editor Interface

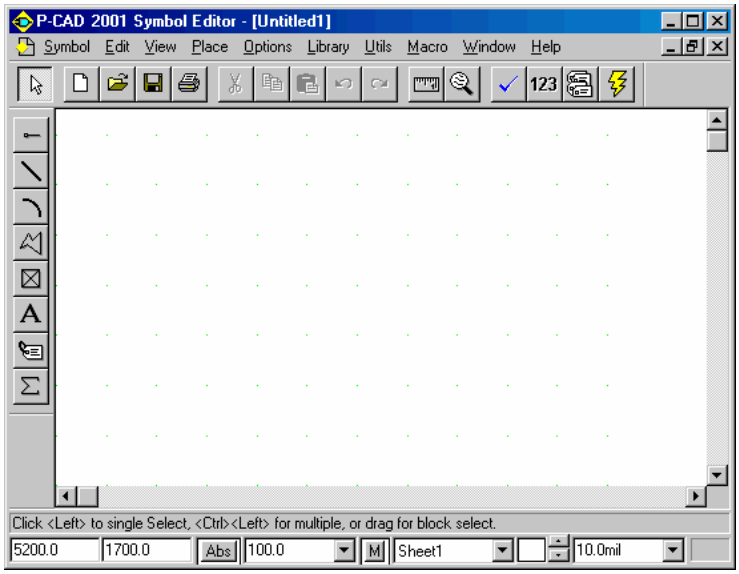
Symbol Editor is very similar to P-CAD Schematic so much of it will be familiar. This section details those functions that are unique to Symbol Editor. If you need information about common features, consult your P-CAD user documentation.

Starting Symbol Editor

In addition to starting Symbol Editor from Windows 95, Windows 98, Windows 2000 and Windows NT, you can start it from other P-CAD programs by choosing the **Utils » Symbol Editor** command. Symbol Editor can also be accessed from Library Executive by choosing the **Symbol » New** or **Symbol » Open** command.

Symbol Editor can be started from the Library Executive Source Browser by choosing **View** in the shortcut menu for a selected item in the Source Browser tree. You can then view a symbol in the *Symbol Viewer* dialog. The **Edit** button on the *Symbol Viewer* dialog also starts Symbol Editor.

When you start Symbol Editor, the interface appears as shown below.



The following Symbol Editor commands and functions are distinctly different in their usage or access than those of P-CAD Schematic:

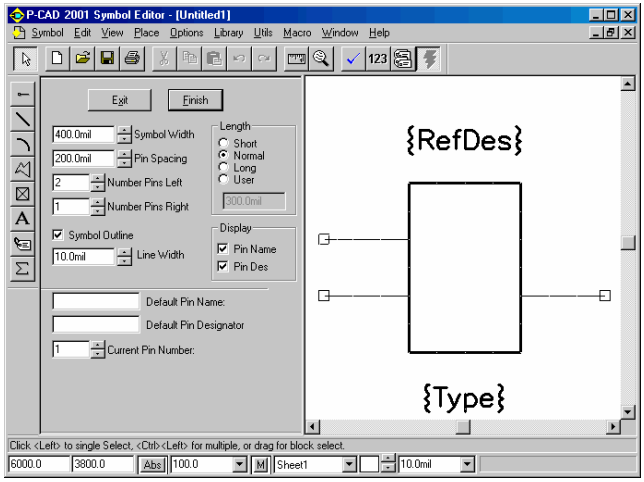
- **Symbol » Open**: Opens symbol files (`.sym`) which can then be modified and saved to a library. See Symbol Editor online help for details.
- **Symbol » Save to File** and **Save to File As**: Saves a symbol to a file (`.sym`) which allows you to keep symbols for later use, without having to save them to a library. Invalidated or incomplete symbols can be saved in this way. See *Working with Symbol Files (page 81)* for details.
- **Symbol » Save** and **Save As**: Saves the current symbol to a library. See *Working with Symbol Files (page 81)* for details.
- **Symbol Wizard**: Opens a wizard used to simplify symbol creation. See *Symbol Wizard (page 79)* for more details.
- **Symbol » Attributes**: Allows you to create and modify symbol attributes for the current symbol file, including invisible attributes. See *Symbol Attributes (page 83)* for details.
- **Place » Pin** and **Pin Properties**: Opens the *Place Pins* dialog where you can set a starting pin number and increment value before placing pins. Option buttons make it easy to set display characteristics, and a display area shows you the results of your selection. See *Pin Placement (page 84)* for details.
- **Utils » Renumber**: Automates pin renumbering in Symbol Editor by dragging the mouse over a series of pins. Pins can also be renumbered manually by selecting one pin at a time. See *Renumbering Pins (page 88)* for details.
- **Utils Validate**: Allows you to validate a symbol before saving it to a library. See *Working with Symbol Files (page 81)* for details.

Symbol Wizard

Symbol Wizard automates symbol generation, providing a quick way to create a symbol. A symbol created with Symbol Wizard is like any other symbol; you can modify it, save it as a symbol file and save it to a library.

To open Symbol Wizard:

1. Make sure no pins have been placed in the active window.
2. Choose **Symbol » Symbol Wizard** to open the *Symbol Wizard* dialog.



Symbol Wizard displays the last symbol you created. The left side of the wizard contains the fields you need to complete to create a symbol; the right side displays the symbol you are creating in a window.

Notice this window is like any other Symbol Editor window. You can resize this window, use zoom in and out and place objects into it.

The Symbol Wizard contains the following items:

- **Symbol Width**: The width of the symbol being created. The slide arrows increase and decrease the width by the amount set in the Pin Spacing field or in 100 mil increments if you hold down the **SHIFT** key.
- **Pin Spacing**: The spacing between pins. The slide arrows increase and decrease the spacing in 25 mil increments or in 1 mil increments if you hold down the **SHIFT** key.
- **Number of Pins Left**: The number of pins on the left side of the symbol.
- **Number of Pins Right**: The number of pins on the right side of the symbol.
- **Symbol Outline**: Turns on or off the graphical representation of the symbol.
- **Line Width**: The line width. The slide arrows scroll through the values defined by choosing the **Options Current Line** command. You can also add a new line width in the *Options Current Line* dialog.
- **Default Pin Name**: The default pin name of the current pin.
- **Default Pin Designator**: The default pin designator for the current pin.
- **Current Pin Number**: The current pin number. The slide arrows scroll through each pin number, allowing you to assign pin names and Default Pin Designators.

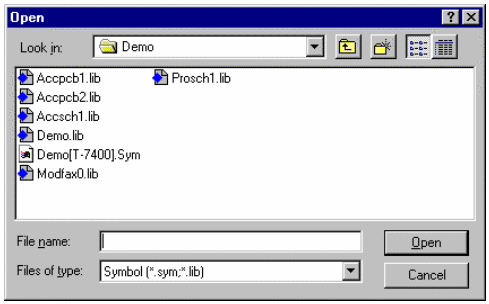
- **Length**: Selects the pin length of all pins in the symbol. Select **User** and enter a value to set a custom pin length.
- **Display**: Turns on or off the display of the Pin Name and Pin Des.

Working with Symbol Files

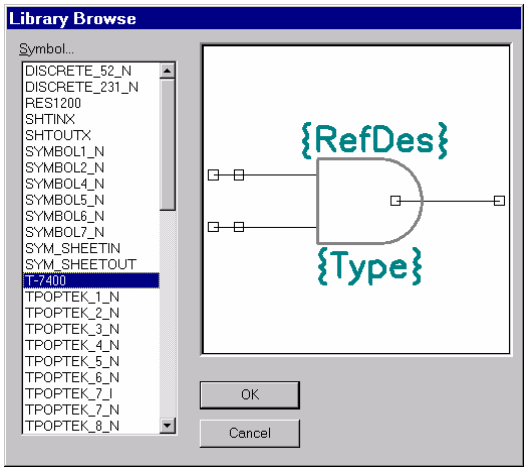
Symbols can be saved as symbol files (`.sym`), allowing you to save partially created symbols and to create symbol templates, which you can use to create new symbols. Before a symbol can be copied to a library, it must pass a validation test. If it fails, you can save it as a `.sym` file and return to it at a later time.

Loading a Symbol from a Library

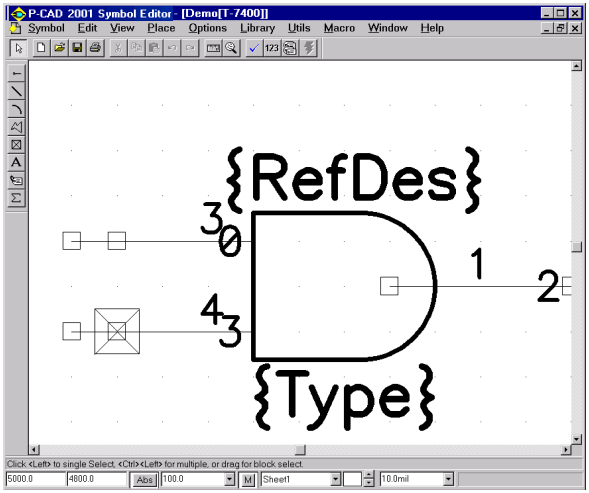
You can load an existing symbol from a library and modify it to create a new symbol. When you choose **Symbol » Open**, the *Open* dialog appears.



Notice that this dialog allows you to open symbol files (`.sym`) and library files (`.lib`). When you choose a library file and click **Open**, the *Library Browse* dialog opens.



Select a symbol from the list (e.g. T-7400) and click **OK**. The symbol is loaded into Symbol Editor. When a symbol is loaded from a library, the library and symbol name appear in the title bar. The symbol name and source library appear in the title bar even though this symbol has not been saved as a `.sym` file.



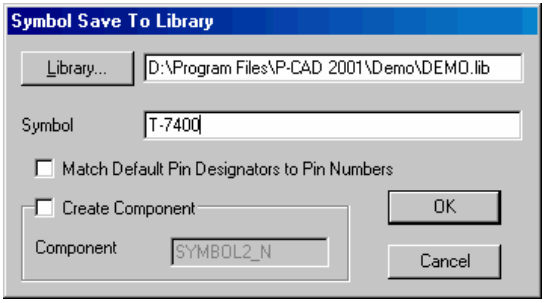
Saving a Symbol to a Library

Once a symbol is created, you can save it to a library after it has been validated.

1. Choose **Utils » Validate**.

   If your symbol fails validation, an error message appears. For a list of error messages, see *Appendix B, System Messages (page 275)*.

2. Choose **Symbol » Save** to open the *Symbol Save To Library* dialog, shown below.



3. Select the library you want to save the symbol to and then specify a symbol name. If the symbol was loaded from a library, that library appears in the Library box and the symbol name appears in the Symbol box.

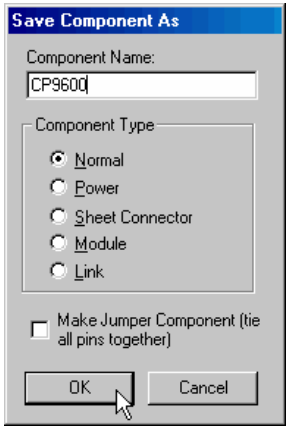
4. If you want to just create a symbol (not a component), click **OK**.

   To create a component automatically that corresponds to the symbol, select the **Create Component** check box before you click **OK**. If the existing default pin designators are correct for the new symbol, you do not have to change them. You can set the default pin designators to **Match Default Pin Designators to Pin Numbers** if you select this check box.

   If the **Create Component** box is cleared, an electrically complete component can be generated with this symbol attached.

5. Click **OK**. The symbol is saved to the library.

6. If the **Create Component** check box is selected, the *Save Component As* dialog opens.



7. Type a component name in the Component Name box, select a component type and click **OK**.

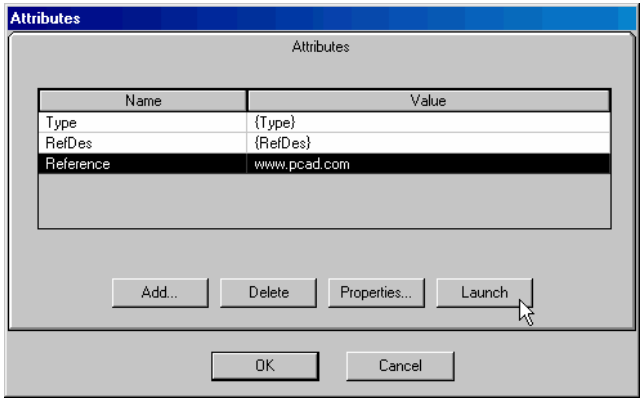
Saving a Symbol as a Symbol file

Choose the **Symbol » Save to File** or **Symbol » Save to File As** command to save a symbol as a symbol file (`.sym`), a unique file type that allows you to keep symbols for later use, without having to save them to a library.

Symbol Attributes

All symbol attributes, including invisible attributes like *RefDes* and *Type*, can be viewed and modified by choosing the **Symbol » Attributes** command.

When you choose **Attributes**, the *Attributes* dialog opens.



In the *Attributes* dialog you can view, add, modify or delete attributes. The dialog contains a two-column table showing the name of the attribute in the left column and corresponding value in the right column.

- **Adding an Attribute**: Click the **Add** button to open the *Place Attribute* dialog. Select an **Attribute Category** and **Name** or, if defining a new {user-defined} attribute, enter a **Name** and then set the **Value** for the attribute. Click **OK** and the attribute is added to the table.
- **Viewing or Changing Attribute Properties**: Select an attribute from the table and click the **Properties** button (or double-click the attribute) to open the *Attribute Properties* dialog to view the attribute's properties or change the attribute's value.
- **To Delete an Attribute**: Select an attribute in the table and click **Delete**, or press the **DELETE** key.
- **Launching a Reference link**: When the special attribute *Reference*, whose value is a reference link, is added to the pattern, you can select the Reference attribute and click the **Launch** button to start a program or web address to display a document or web site.

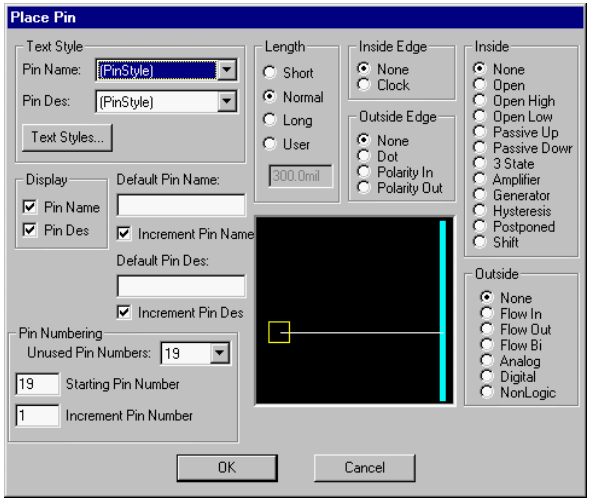
For details about this function, refer to the *P-CAD Schematic User's Guide*.

Pin Placement

The **Place » Pin** command shows you the next available pin number and lets you increment pin numbers.

To place a pin:

1. Choose **Place Pin** or click the toolbar **Pin** button to open the *Place Pin* dialog.



2. In the Text Style frame, set the text style for the pin name and pin designator. Click the **Text Styles** button to open the *Options Text Style* dialog, where you can add or modify existing text styles for your design.

3. In the Display frame, select the **Pin Name** and **Pin Des** check boxes to turn on the display of the pin name and pin designator. Clear the check boxes to turn off the display.

4. Type the default pin name, if desired, in the Default Pin Name box. The default pin name is a placeholder for the real pin name. Use this default label to change the orientation or position of the pin name. The default pin name, like a pin number, cannot be edited once the pin is attached to a symbol. Select the **Increment Pin Name** check box so that each time this pin is placed the pin name is one greater than the last pin placed.

5. Type the default pin designator, if desired, in the Default Pin Designator box. The default pin designator can be incremented each time a pin is placed by selecting the **Increment Pin Des** check box.

6. The Unused Pin Number box defaults to the next unused pin number; unused pins are shown in the list that appears when you click the Down Arrow. Choose a number or choose **>#** to begin numbering from that number (#) onwards.

7. The Starting Pin Number box defaults to the next available pin number. To change the default value, type a new number in the box.

   If you place a pin with a duplicate pin number, you hear a warning beep.

8. Enter the number (positive or negative) by which you want to increment pin numbers in the Increment Pin Number box.

9. Select the desired pin length option button from the options in the Length frame.

10. Select the desired display characteristics by selecting the **Inside Edge**, **Outside Edge**, **Inside** and **Outside** option buttons.
11. Click **OK**.
12. Move the cursor to the location in the active window where you want to place the pin and click to place it. Drag and drop (release) to place it more accurately. (An alternate method for drag-and-drop is **ALT**+**click** and then release the **ALT** .You can then move the pin freely with the mouse without having to keep the button depressed.) To cancel the pin placement, right mouse-click.

    The Status Line shows the pin number of the pin about to be placed and also indicates if the pin number is a duplicate.

    The **N** key increments the pin about to be placed by one; **SHIFT**+**N** decrements the number. If 6 is the next available pin number, pressing **N** would result in the pin being assigned number 7; pressing **SHIFT**+**N** would result in the pin being assigned number 5.

    You can continue to place similar pins by clicking in additional locations. Pin attributes previously specified (in the *Place Pin* dialog) are also displayed.
13. To stop, right mouse-click, or press **ESC,** or click the **Select** toolbar button.

    The **Undo** command removes the entire sequence of pins. The **BACKSPACE** key removes the last placed pin.

    Pin names and pin designators can be subselected for moving, flipping and rotating.

Rotate or Flip a Pin

In Symbol Editor, pins can be flipped as well as rotated during placement.

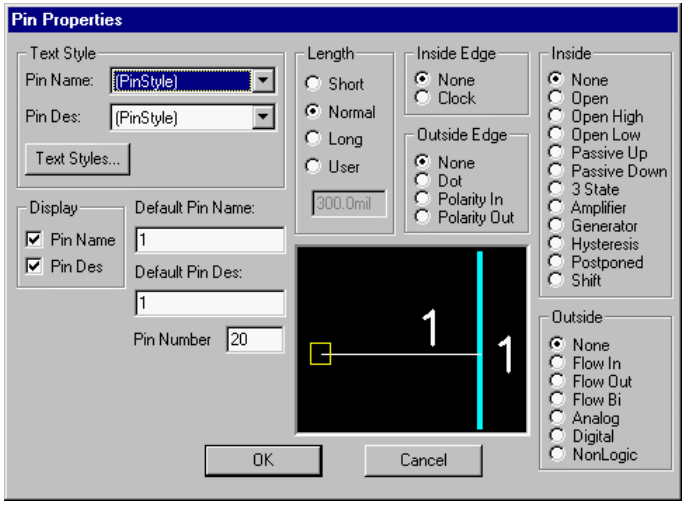
- To flip a pin as you are placing it, press **F** while the pin is ghosted before final placement.
- To rotate a pin as you are placing it, press the **R** key while the pin is ghosted before final placement. The resulting rotation angle is applied to the next pin you place.
  Refer to your *P-CAD Schematic User's Guide* for additional information.
- To rotate or flip a pin after it has been placed, select it and press **R** to rotate or **F** to flip. The **R** key rotates the pin 90 degrees.

Pin Properties

The *Pin Properties* dialog allows you to view or change pin properties.

> If two or more selected pins have differing properties, those properties appear in gray or blanked out. When you select a value for one of these properties it is applied to all the selected pins.

When you select a pin and choose **Properties**, the *Pin Properties* dialog opens.



Changing Pin Properties

The following options are available in the *Pin Properties* dialog:

- **Text Style**: Change the text style of the Pin Name and Pin Des by selecting one of the existing styles in the **Pin Name** or **Pin Des** lists.

  To modify or add a new text style, click the **Text Styles** button to open the *Options Text Style* dialog.

- **Display**: Select the **Pin Name** and **Pin Des** check boxes to display the pin name and/or pin designator.

- **Default Pin Name**: Enter or modify a default pin name.

- **Default Pin Designator**: Enter or modify a default pin designator.

- **Pin Number**: Changing the pin number is useful when you only want to change one or two pins. If you want to renumber a series of pins, choose **Utils » Renumber** for smoother and faster action.

- **Length**: Select **Short** (100 mil), **Normal** (300 mil), **Long** (500 mil), or set your own length by clicking the **User** option and entering a length in the box.

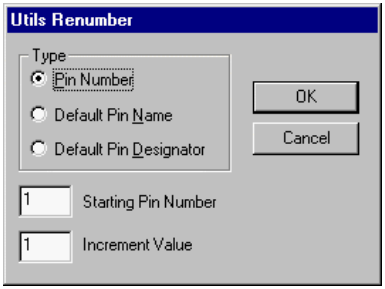
- **Inside Edge**, **Outside Edge**, **Inside**, **Outside** frames: Select the option buttons to turn on various display characteristics. These display characteristics include all attribute symbols that may be attached to a pin for design clarification and are for graphical appearance only.

Renumbering Pins

The renumbering function in Symbol Editor, unlike Schematic, supports single select pin renumbering and drag select for multiple pin renumbering.

To renumber the Pin Numbers:

1. Choose the **Select** tool.
2. Choose **Utils » Renumber** to open the *Utils Renumber* dialog.



3. Clear the **Display Default Pin Des** check box in the **Miscellaneous** tab of the *Options Display* dialog.
4. Select the **Pin Number** option button.
5. Specify **Starting Pin Number** and **Increment Value**, as appropriate.
6. Click **OK**. Notice that the cursor changes to the crosshair shape.

   You are in a temporary mode of assigning numbers, so every time you hold down the left-button and drag it over a pin, you renumber that pin. You can also click a pin to renumber it. For example, the first pin you drag the mouse over would be number 1 (if Starting Pin Number was specified as 1), the second pin number 2 (if the Increment Value was specified as 1).

   As you drag the mouse over a pin while in the Renumber mode, it shows that a number has been assigned. Once a pin is renumbered, you cannot change its number without choosing the **Renumber** command again.

   The Status Line information area displays the pin number every time you (re)number a pin.

   The **N** key increments the pin about to be renumbered by one; **SHIFT**+**N** decrements the number. If 6 is the next available pin number, pressing **N** would result in the pin being assigned number 7; pressing **SHIFT**+**N** would result in the pin being assigned number 5.

   You can use the unwind feature to reverse the renumbering process. The **BACKSPACE** key unwinds the renumbering.

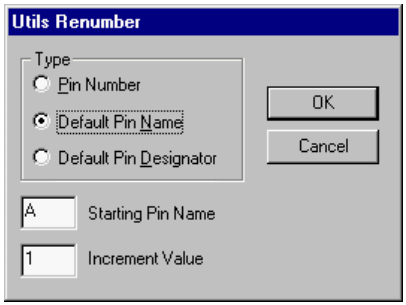
7. Right-click, or press **ESC**, to end the Renumber sequence.

   The **Undo** command reverses the entire renumbered sequence.

Renumbering Default Pin Names

You can renumber Default Pin Names by following these steps:

1. Choose the **Utils » Renumber** command to open the *Utils Renumber* dialog.

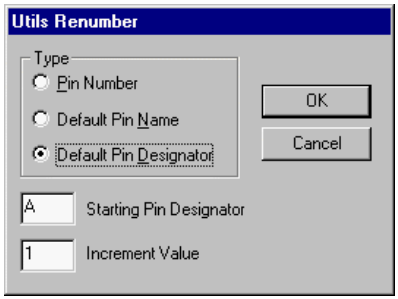


2. Select the **Default Pin Name** option button. Notice that the Starting Pin Name value changes to A, instead of 1. You can enter any alphanumeric prefix as the Starting Pin Name.
3. Follow the steps outlined in *Renumbering Pins (page 88)* to renumber each pin's Default Pin Name as you click on each pin.

Renumbering Default Pin Designators

You can renumber the Default Pin Designators by following these steps:

1. Choose **Utils » Renumber** to open the *Utils Renumber* dialog.



2. Select the **Default Pin Designator** option button. Notice that the Starting Pin Designator value changes to **A**, instead of **1**. You can enter any alphanumeric character as the Starting Pin Designator.
3. Follow the steps outlined in *Renumbering Pins (page 88)* to renumber each pin's Default Pin Designator as you click each pin.

You can also select a pin, choose **Properties** and then assign a pin number, pin name and pin designator.

Placing Reference Points

The **Place » Ref Point** command places a reference point on a symbol before it is saved to a library.

Symbol Editor automatically places a Ref Point on a pin or at center of pins if you save a symbol to a library without a Ref Point.

Refer to your *P-CAD Schematic User's Guide* for additional information.

Keyboard Shortcuts

This section contains a list of keyboard shortcuts unique to Symbol Editor.

Press these keys:	To do this:
BACKSPACE (Unwind)	Used as unwind command while placing objects with multiple segments (e.g. lines, polygons, pins). Each **BACKSPACE** stroke unwinds the previously placed item.
N and **SHIFT**+**N** (increment and decrement)	**N** increments a pin by one when placing or renumbering pins. **SHIFT**+**N** decrements a pin by one when placing or renumbering pins.

CHAPTER 7

The Source Browser

Using the Source Browser

The Source Browser displays component data currently available in P-CAD Library Executive in an easy-to-use Graphical User Interface (GUI). It is organized in a tree like structure that shows the relationship between these component data sources. The Source Browser provides access to all the information and many of the commands available to you while using Library Executive.

The Source Browser contains any or all of the following items:

- **P-CAD library sets**: A library set contains a group of P-CAD libraries. Many of the Library Executive commands can operate on either an individual library or a group of libraries simultaneously by selecting a library set. For more information, refer to *Library Sets (page 95)*.
- **P-CAD libraries**: A P-CAD library can be accessed through the Source Browser. Libraries in the Source Browser are always contained within a Library Set. You can open the library to view or edit the components, patterns and symbols it contains. For more information, refer to *Browsing Components, Patterns and Symbols (page 96)*.
- **External source file**: A component data file from a non-P-CAD source can be imported into Library Executive. The component attributes of the external source file can be used for many of the Library Executive functions, including updating a P-CAD library. For more information, see *Importing Data from an External Source (page 121)*.
- **Query results**: A search, or query, can be conducted on any source for a component or component attribute satisfying specified criteria. A Query Result is added to the Source Browser after a Query is complete. The Query Result contains all components or component attributes from the selected source that satisfy the search criteria. For more information, see *Querying Libraries (page 103)*.
- **Cross link results**: Component attributes in two different sources can be linked together with the **Cross Link** command. A Cross Link Result is added to the Source Browser after a **Cross Link** is complete. This Cross Link Result contains a combination of the attributes in both sources. For more information, see *Understanding Cross Linking (page 130)*.

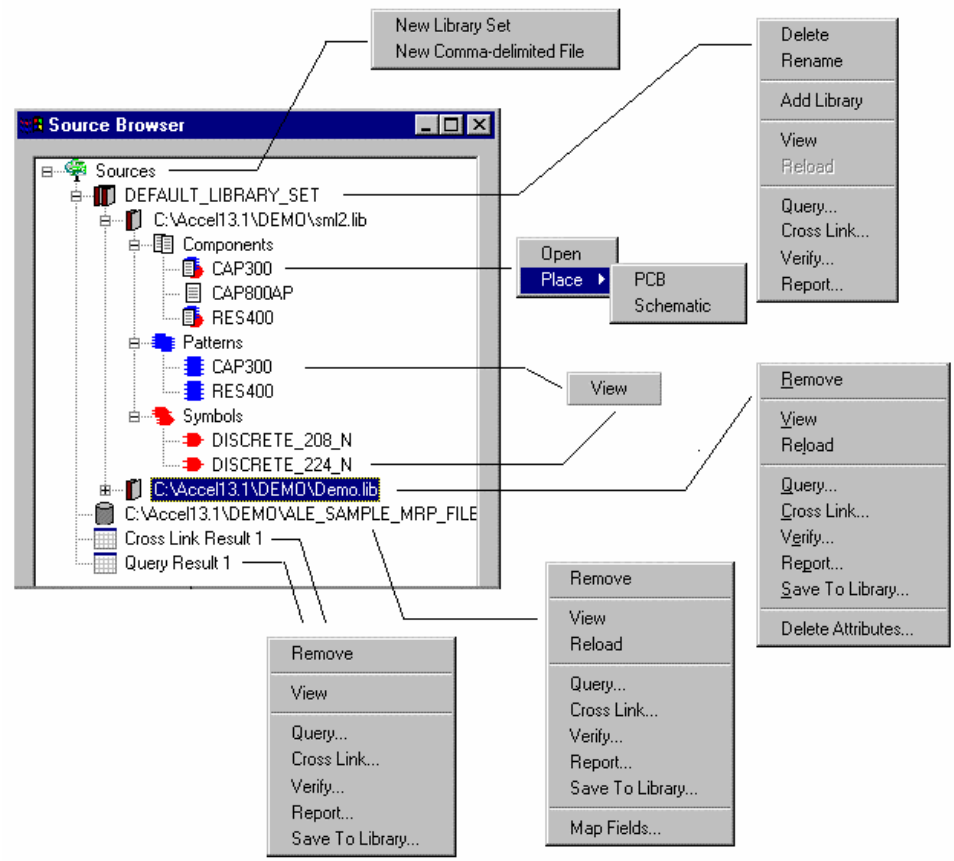
From the Source Browser, you can access almost all of the Library Executive commands. These commands are detailed in *Source Browser Shortcut Menu Commands (page 17)*. A brief summary of each can be found in *Source Browser Commands (page 92)*.

Source Browser Commands

When you select a source in the Library Executive Source Browser and right mouse-click, a shortcut menu opens, providing access to common commands performed on a selected object. This menu changes depending on the object selected. The commands available are summarized in the following table.

Choose this Command:	To do this:
Add Library	Add a library to the selected library set.
Cross Link	Open the *Cross Link* dialog.
Delete	Delete the selected library set. You can also press the **DELETE** key.
Delete Attributes	Delete selected user-defined attributes from a P-CAD library.
Map Fields	Open the *Map Fields* dialog.
New Comma-delimited File	Open the *Import Comma-delimited File* dialog.
New Library Set	Add a new library set.
Open	Open the *Component Information* dialog.
Place	Open the *Place Component* or *Place Part* dialog in the selected P-CAD application.
Query	Open the *Query* dialog.
Reload	Reload an external file, a library, or a library set's contents into P-CAD Library Executive.
Remove	Remove the selected source from the Source Browser. You can also press the **DELETE** key
Rename	Rename the library set.
Report	Open the *Report* dialog.
Save to Library	Open the *Save Source* dialog.
Verify	Open the *Verify* dialog.
View	Open the selected pattern, symbol or Viewer.

The following diagram illustrates the shortcut commands available for each source in the Source Browser.



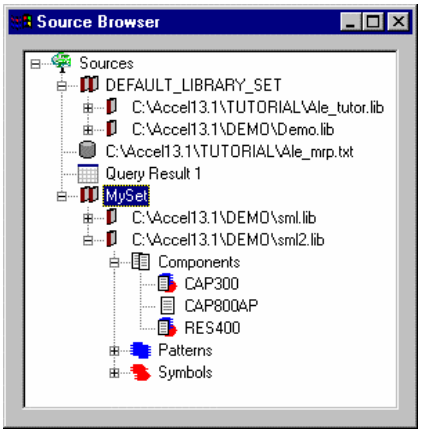
This chapter summarizes the Source Browser, its contents and its function as the take off point to the other Library Executive features. For additional information about the use of Query, Cross Link, Verify, Reports, Place and Save To Library from the Source Browser, refer to *Accessing Other Features (page 99)*.

The Source Browser Interface

To open the Source Browser, click the **Source Browser** button on the *Library Executive* toolbar or choose **View » Source Browser**.

The *Source Browser* dialog displays information in a tree structure, or hierarchy. You can browse the tree to view its contents. The source files in the Source Browser can be a library set, a library, an external source file, a query result, or a cross link result.

Right mouse-clicking on a selected item in the tree opens the Library Executive shortcut command menu. Refer to *Source Browser Commands (page 92)* and *Accessing Other Features (page 99)* for additional information.



The tree structure allows you to view all Library Executive source files and their contents at various levels of detail using the following commands and functions:

- Groupings containing collapsed levels are shown with a **+** sign. To expand the grouping, simply click the **+** sign. Expanded groupings are shown with a **–** sign. To collapse the grouping, simply click the **–** sign.

  When modifications are made to the library contents from outside Library Executive Source Browser, the graphics representing the library contents may not be current.

- To refresh the Source Browser contents, collapse the tree from the modified level. For example, when a component is removed from a library, collapse both the component level and the library level to refresh their contents. Expand the levels again to view their current contents.

- To refresh the Library Executive memory, so that the *Viewer* contents reflect recent changes to a library or library set, choose **Reload**. Since Query, Verify, Cross Link and Report functions operate on the source in memory, choosing the **Reload** command assures you that you are operating on the most current library contents.

Library Sets

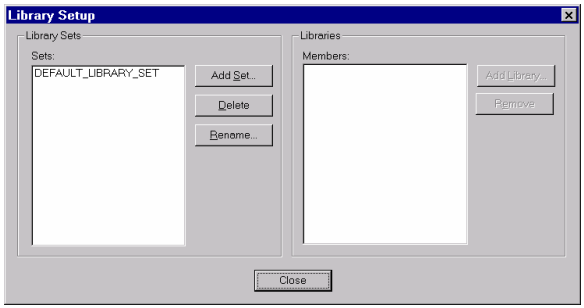
A library set is a group of P-CAD library files. In the Source Browser, you can open up a library set to view its contents including the components and their associated symbols and patterns. You can also perform operations, such as query and verify, on the library set or its contents.

This section discusses the creation and use of library sets. Operations on library sets and other Source Browser contents are summarized in *Accessing Other Features (page 99)*.

Library Setup

The *Library Setup* dialog provides the methods to organize and manage library file names and library sets.

1. Choose **Library Setup** to open the *Library Setup* dialog.



2. Select a library set from the *Library Sets* list. The libraries contained in the set are listed in the *Members* list.

To be usable in P-CAD Library Executive, all available libraries must be in one or more library sets. A default library set, called `DEFAULT_LIBRARY_SET`, includes any open library files that do not have a set specified explicitly.

Library sets defined in the *Library Setup* dialog are saved in the initialization file, `cmp.ini`, for the next time you start Library Executive.

Adding, Deleting and Renaming Library Sets

To add a new library set:

1. Click the **Add Set** button.
2. Type the name of the new library set in the box.

To delete a library set:

1. Select a library set from the *Library Sets* list.
2. Click the **Delete** button.

3. Deleting a set removes the set and the libraries it contains.

To rename a library set:

1. Select a library set from the *Library Sets* list.
2. Click the **Rename** button.
3. Modify the set name in the box.

Modifying Library Set Contents

To add a library to the selected library set:

1. Click the **Add Library** button.
2. Navigate to and select the desired library.
3. Click **Open**.

Selected libraries can be removed from a library set by clicking **Remove**.

Modifying Library Sets from the Source Browser

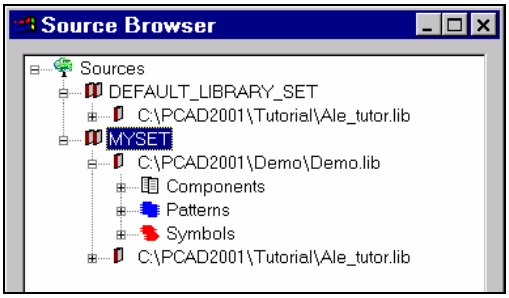
You can add, rename and delete selected libraries and library sets directly from the *Source Browser* by choosing those commands from the shortcut menu.

Other operations, including Query, Verify, View and Reload, are also accessible through the shortcut menus.

Browsing Components, Patterns and Symbols

From the Source Browser, you can view the contents of a library, including its components, patterns and symbols, by following these steps:

1. Expand a library set and one of the libraries contained within it by clicking the **+** sign to their left. Inside a library are three items: components, patterns and symbols. In the Source Browser tree, shown in the following figure, some of these items have been expanded to expose the contents of the library.



The component icon in the Source Browser indicates whether the component has an attached pattern or symbol, as shown below:

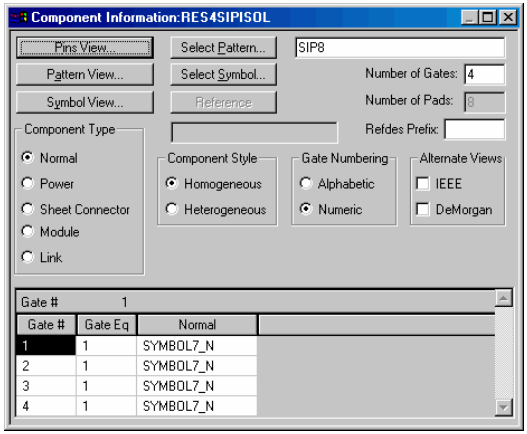
Component Icon	Description
	A component with no pattern or symbol attached.
	A component with only a pattern attached.
	A component with only a symbol attached.
	A component with both a pattern and symbol attached.

2. If you select a component in the Source Browser tree, you can choose the **Open** or **Place** commands from the shortcut menu to open or place the selected item.
3. You can view a selected pattern or symbol by choosing the **View** command from the shortcut menu.

Opening a Component

To open a component, follow these steps:

1. Select a component in the Source Browser tree.
2. Choose the **Open** command from the shortcut menu. You can also open a component by double-clicking on the component name in the Source Browser tree. The *Component Information* dialog opens.



The *Component Information* dialog provides general information about a component. From this dialog, you can complete many functions including setting the component type as homogeneous or heterogeneous, selecting alternate views, setting the number of gates and gate numbering style, and defining the total number of pins.

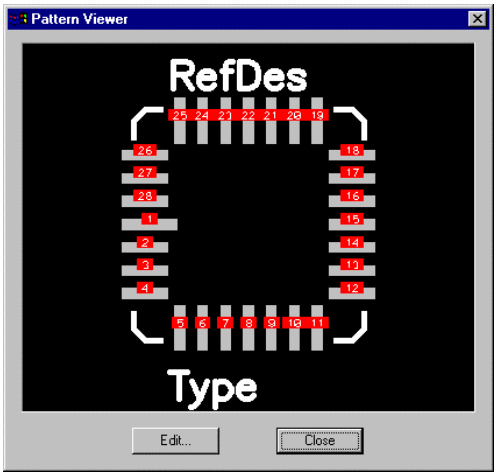
For more information about the *Component Information* dialog, see the **View » Component Info** command in *View Component Info (page 255)*.

Viewing and Editing a Pattern or Symbol

To view or edit a pattern or symbol, follow these steps:

1. Select a pattern or symbol in the Source Browser.
2. Choose the **View** command from the shortcut menu. You can also view a pattern or symbol by double-clicking its name in the Source Browser tree.

   When you view a pattern, for example, the *Pattern Viewer* dialog opens:



   The P-CAD Pattern Editor can be opened by clicking the **Edit** button on the *Pattern Viewer* dialog. See *The Pattern Editor (page 59)*, for information on using the Pattern Editor

   The P-CAD Symbol Editor can be opened by clicking the **Edit** button on the *Symbol Viewer* dialog. See *The Symbol Editor (page 77)*, for information on using the Symbol Editor.

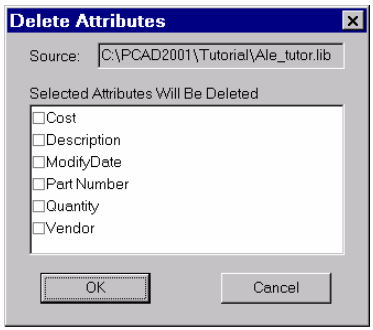
Deleting Library Attributes

From the Source Browser, you can remove user-defined attributes from a P-CAD library.

To delete attributes from a P-CAD library:

1. Select the library in the Source Browser tree.

2. Choose **Delete Attributes** to open the *Delete Attributes* dialog.



The name of the selected library appears in the Source box. The user-defined attributes that may be deleted from the library appear in the Selected Attributes Will Be Deleted box.

3. Select the check box to the left of the attribute you want to delete from the library. All selected attributes will be deleted from the library. Clear the check boxes of the attributes you want to leave in the library.

4. Click **OK**. The selected attributes are automatically removed from the library.

Accessing Other Features

From the Source Browser you can perform queries, import data from an external source, cross link and update or create libraries. You can even place components selected in the Source Browser directly to a design in PCB or Schematic.

The shortcut menus vary depending on the item selected in the Source Browser. This section lists the features accessible from selected items in the Source Browser.

Query

With Query, you can search for components with particular specifications. You can designate a wide variety of component characteristics, including cost, manufacturer, number of pins, etc. Searches can be conducted both on P-CAD libraries and on external sources, such as a company-wide database.

You can access Query from the Source Browser by selecting a library set, a library, an external source file, a cross link result, or another query result and choosing the **Query** command.

Query results are stored on the Source Browser tree, where you can view the query spreadsheet, generate a report, cross link, save the query to a library or query again.

For more information about Query see *Querying Libraries (page 103)*.

Importing Data from an External Source

Component data from a non-P-CAD source can easily be imported into Library Executive.

To import an external source from the Source Browser, select the root of the Source Browser tree (Sources) and choose **New Comma Separated** File.

The **Map Fields** and **Cross Link** commands are available from the Source Browser shortcut menu when you select an external source file.

For more information about importing component data from a non-P-CAD source, see *Importing Data from an External Source (page 121)*.

Cross Linking

Cross Linking allows you to quickly combine component attribute information from two different sources. Refer to the section *Understanding Cross Linking (page 130)* for details about the cross link feature.

The Cross Link feature is available from the Source Browser by selecting a library set, a library, an external source file, a cross link result, or another query result and choosing the **Cross Link** command from the shortcut menu.

Cross Link results, stored on the Source Browser tree, provide access to view the cross link spreadsheet, generate a report, query, cross link again, or save the cross link result to a library by choosing the appropriate commands in the shortcut menu.

Updating or Creating Libraries

You can create or update P-CAD libraries from several sources. A new library can be created from an external source file and updated as the external source file changes. You can also use query results or cross link results to create or update libraries.

To create or update libraries from the Source Browser, select the external source file, cross link result, or query result from the Source Browser tree and choose the **Save To Library** command.

Verify

P-CAD Library Executive supports four consistency verifications from the Source Browser:

- Between a P-CAD library or library set and another P-CAD library or library set.
- Between an external source file and a P-CAD library or library set.
- Between a Query Result and a P-CAD library or library set.
- Between a Cross Link Result and a P-CAD library or library set.

A difference report file is generated from the Source Browser tree, by selecting a P-CAD library or library set, the external source file, a cross link result, or a query result and choosing the **Verify** command.

For more information on verification, see *Library Verification (page 137)*.

Reports

To generate a report using a P-CAD library, an external source file, a cross link result, or a query result from the *Source Browser*, select the item from the *Source Browser* tree and choose the **Report** command.

For additional information about generating reports, refer to *Reports (page 145)*.

Component Placement

To place components into P-CAD PCB or P-CAD Schematic from the Source Browser:

1. Select the component from the Source Browser tree.
2. While PCB or Schematic is running, choose the **Place** command.
3. Select the desired application, **PCB** or **Schematic**.

For more information about placing components in a P-CAD PCB or P-CAD Schematic design, see *Component Placement (page 163)*.

CHAPTER 8

Querying Libraries

Using Query

With P-CAD Library Executive, you can search for components using a wide variety of search criteria. For example, you may be searching for a component with a prescribed functionality that is available at a low price. In Library Executive, you simply specify these criteria in an easy-to-use spreadsheet form and a list of the ideal component(s) appears.

You can query P-CAD libraries or library sets, an external source file, like the corporate component database, or another query result to narrow your component search by additional criteria.

A query result can be used to update or create a P-CAD library. It can also be used to verify component data sources, finding differences between a library and a select set of components.

This chapter details how to use the Library Executive Query.

Fields and Data Types

To both search and enter data into P-CAD Library Executive, it is useful to understand the commonly used component attributes, or fields, and how they are stored in memory.

This section discusses both the fields that are immediately recognized by P-CAD Library Executive and their data types. Additional user-defined fields may also be included.

Fields

Fields are the names, recognized by P-CAD Library Executive, for component attributes. Fields include attributes and property names such as ComponentName, RefDesPrefix and NumberOfPins.

> In general, a property is an intrinsic building block of a component. For example, the number of pads on the component is a property. An attribute is additional component information that may be relevant to the design, manufacture or production of the printed circuit board. Throughout this document, the term attribute is used to generally describe both component properties and attributes.

A list of field names is included in the following table:

Field Names	Read- only	Field Type	Data Type
ComponentName	Yes	Primary key	String
RefDesPrefix	No	Predefined property	String
PatternName	Yes	Predefined property	String
NumberOfPads	Yes	Predefined property	Integer
NumberOfPins	Yes	Predefined property	Integer
NumberOfParts	Yes	Predefined property	Integer
ComponentLibrary	Yes	Predefined property	String
ComponentType	Yes	Predefined property	ComponentType
Homogeneous	Yes	Predefined property	Boolean
AlphaNumeric	Yes	Predefined property	Boolean
HasIEEE	Yes	Predefined property	Boolean
HasDemorgan	Yes	Predefined property	Boolean
Value	No	Predefined attribute	String
ComponentHeight	No	Predefined attribute	String
Description	No	Predefined attribute	String
Link	Yes	Predefined attribute	String
NoSwap	No	Predefined attribute	String
Part Number	No	Predefined attribute	String
SwapEquivalence	No	Predefined attribute	String
Tolerance	No	User-defined attribute	String
Wattage	No	User-defined attribute	String
PowerConsumption	No	User-defined attribute	String
Manufacturer	No	User-defined attribute	String
Supplier	No	User-defined attribute	String
Cost	No	User-defined attribute	String
Leadtime	No	User-defined attribute	String

Field Names	Read- only	Field Type	Data Type
CreateDate	Yes	User-defined attribute	DateTime
ModifyDate	Yes	User-defined attribute	DateTime
VerifyDate	Yes	User-defined attribute	DateTime

There are three types of Field names:

- Primary key
- Predefined properties and attributes
- User-defined attributes

There is only one primary key, ComponentName. This attribute is required for all P-CAD library components. The ComponentName field contains the name of the component and is the fundamental field used for component selection in P-CAD PCB, Schematic and Library Executive.

Predefined properties and attributes are fields that P-CAD libraries use to define the structure and function of the component. These predefined fields are specifically listed in the *Attribute Properties* dialog. See *Edit Commands (page 249)* for details.

The user-defined attributes are field names that are recognized by P-CAD, but are not specifically listed in the *Attribute Properties* dialog. Although any user-defined attribute may be defined, an assortment of preset attributes, listed in the above table, are available in the Field column list (see *Selecting Search and Display Fields (page 106)*. These attributes may be useful in determining the feasibility for manufacturing a board, such as cost and supplier.

The data types of these fields impacts how you enter and search for component attributes. See *Data Types (page 105)* for a summary of data types available in Library Executive.

Fields that are read-only have values that cannot be modified in Library Executive Viewers.

Data Types

Each field contains attribute or property values. These values are stored in memory as a particular data type. The following data types are supported in Library Executive:

- **String**: String data is a collection of alphabetic or numeric characters. All user-defined fields are stored in the String data type, with the exception of the Timestamps.
- **Integer**: Integer data values do not contain a decimal point. For example, the numbers 3, 6 and 9 are integers, but 8.5 is not.
- **Boolean**: Boolean data can be one of two values. True or False are Boolean. True and False data values are also represented by the numbers 1 and 0, respectively.
- **DateTime**: DateTime data specifies the time an event occurred. It can include the day, month, year, hour, minute and second.

- **ComponentType**: The ComponentType data can be one of the following values: Normal, Power, Sheet Connector, Module or Link.

  The *Data Type* of the field is important for searching, comparing, sorting and displaying component attribute values.

Setting up a Query

To complete a query, you must:

- Set up the search and display fields.
- Specify search criteria.
- Specify the desired output.
- Execute the query.

The following section discusses defining your search and display fields, as well as setting up your search criteria using query operators. It is followed by a brief summary of how to generate the query output. Query output is detailed in *Using Query Results (page 116)*.

The *Query* dialog is a spreadsheet. Refer to *Library Basics (page 3)* for details on manipulating spreadsheets.

The **Query** command searches for components from the current contents of Library Executive memory, as displayed in the Viewer. Query does not search the original source file. If changes have been made to a library, choose **Reload** to refresh the library components in Library Executive's memory before conducting the Query.

Selecting Search and Display Fields

For a Query, you must select the fields you want to search, as well as the fields to be displayed in your query result. The search fields are used to specify criteria important to your component search. Display fields contain the information you want to view in your query result.

When conducting a Query, both search and display fields should be in your *Field* list in the *Query* dialog. To put a field in the Field list, you can do one of the following:

- Click the **Select All** button. All of the fields defined for the present query source appear in the Field list.
- Select desired fields from the pick list. When you click the **Field** column a down arrow appears to the right. Click the **Down Arrow** and the field list appears, where you can select the desired field. The field is automatically placed on the spreadsheet.
- The **Clear All** button removes all fields from the Field list.
- To add additional fields to a filled spreadsheet, click the **Add Row** button. A new row appears at the end of the Field list.

Search Fields

A search field is used to specify your component search criteria. To make a field a search field, you must define search criteria in the Criteria(And) and Or columns of the *Query* dialog. See *Selecting the Search Criteria (page 107)* and *And Versus Or (page 112)* for more information.

Display Fields

A display field is included in the query results. To make a field a display field, select the **Show** check box next to the field name in the Field list.

You can search for a field and not display its value in the Query Result table. To do so, specify a search criterion for the field, but clear its **Show** check box.

Selecting the Search Criteria

To complete any component search, you must specify what you are looking for. Search criteria define a condition that a component attribute must satisfy in order for the component to be acceptable.

To specify search criteria, you must determine three things:

- The Field you wish to search.
- The Data Type of that Field.
- The Query operator and values to limit the Field values.

This section discusses query operators, including wildcard characters. These are the fundamentals needed to create any component search.

Special Search Criteria by Data Type (page 109) details how the search expression is constructed when searching for a unique data type: Boolean, ComponentType, or DateTime. When searching for a String or Integer data type, only the information available in this section is required.

A general search expression usually has the form of a query operator followed by a value. A combination of search expressions on multiple fields can limit the component search even further, making the search for the ideal component quick and easy. For details on how to combine more than one search expression, refer to *And Versus Or (page 112)*.

Query Operators

To precisely define your search criteria, the Query function uses a set of operators selected from a list of available operators.

When you click the spreadsheet in the Criteria(And) and Or columns, a down arrow appears to the right. Click the **Down Arrow** to display the operator list and then click on the operator you want to use. The operator is automatically placed on the spreadsheet. Operators may also be typed directly into the spreadsheet.

The available operators are listed in the following table.

Operator	Function
=	Exactly equal to If used with a wildcard operator, * or ?, this operator becomes literal. It searches for a set of characters with, for example, a question mark at the end.
<	Less than
>	Greater than
<=	Less than or equal to
>=	Greater than or equal to
<>	Not equal to
IsLike	If used with a wildcard operator, IsLike means is similar to. For example, IsLike 5* could be 50, 510, 5, etc. If not used with a wildcard operator, IsLike is equivalent to =.
IsNotLike	If used with a wildcard operator, IsNotLike means is not similar to. For example, IsNotLike 5* could be 14 or 20 or 42, but not 50, 510, or 5. If not used with a wildcard operator, IsNotLike is equivalent to <>.
Exist	The attribute exists.
NotExist	The attribute does not exist.
AnyValue	The attribute exists and it has some value.
NoValue	The attribute exists, but it is assigned no value.

To specify search criteria, the operator is generally followed by alphanumeric characters, as shown in the following spreadsheet.

	Field	Show	Criteria (And)	Or
1	ComponentName	☒	IsLike DIP2*	
2	Alias	☐		
3	ComponentLibrary	☒		
4	ComponentType	☒	Exist	
5	NumberOfPads	☒		
6	NumberOfPins	☒		
7	NumberOfParts	☒	>=6	
8	Homogeneous	☐	= True	
9	AlphaNumeric	☐		
10	HasIEEE	☐		

The four operators, Exist, NotExist, AnyValue and NoValue, are not followed by alphanumeric characters.

For the above query, one component found could be DIP24, a homogeneous component with 6 parts and a ComponentType attribute. Notice that you can search for a homogeneous component without displaying the Homogeneous field value in the query result. Because of the search criteria, all components in the query result will have a value True for the Homogeneous field. This value, however, is not displayed in the Query Result because its **Show** box is cleared.

If a search criteria entry is not preceded by an operator, the operator is assumed to be *IsLike*.

Wildcard Characters

There are two wildcard characters recognized by Query: ? and *. Wildcard characters can represent any alphanumeric character. The wildcard character ? represents exactly one character; the wildcard character * can more generally represent zero to any number of characters.

For example, DIP*, could be DIP24, DIP16 or DIP8. Whereas DIP?4 must remain five characters long and end with the number 4, such as DIP14 or DIP24.

Special Search Criteria by Data Type

How you specify search criteria depends on what you're searching for. Knowing the data type of the field you want to search, you can narrow the field of results.

When searching for an Integer or String, all you need to know to create a successful search criteria is summarized in *Selecting the Search Criteria (page 107)*.

When searching for a Boolean, ComponentType, or DateTime data type, additional details are helpful to understand how to define a successful search. This section discusses how to search for Boolean (True/ False), ComponentType and DateTime data values.

True and False

Several standard field types defined in P-CAD libraries have a Boolean data type. Boolean data is either True or False. This can also be represented numerically as the numbers 0 or 1, for False and True respectively.

The following predefined fields are Boolean:

- **Homogeneous**: Whether or not the component is homogeneous. In a homogeneous component, all gates use the same symbol.
- **AlphaNumeric**: Whether the gate numbering in the reference designator is alphanumeric. If True, the gate numbering contains letters, such as U1: A and U1: B. If false, the gate numbering is numeric only, such as U1: 1 and U1: 2.
- **HasIEEE**: Whether the gate has an IEEE representation.
- **HasDemorgan**: Whether the gate has a DeMorgan representation.

To search for True/False values using the Query search criteria, you can type a variety of expressions in the *Query* dialog. These expressions are summarized in the following table.

Search Description	Expressions
Find the True values for this Boolean field.	True = True 1 = 1 > 0
Find the False values for this Boolean field.	False = False 0 = 0 < 1

No wildcards are allowed when searching for a True or False value.

ComponentType

The ComponentType attribute has five possible values: Normal, Power, Sheet Connector, Module, or Link. The ComponentType attribute is not stored as a character string. To search for any of these ComponentType values, the full value must be entered in the search expression. Wildcard characters are not allowed. The search is not sensitive to case.

For example, to search for a ComponentType of Power, the following entries could be made in the Criteria(And) column: Power, = Power, power, or = power. But you cannot search for = Pow*. The wildcard would be taken literally and no components would be found.

Dates and Times

You can Query for the CreateDate, ModifyDate and VerifyDate fields. For example, you may want to find all components in a P-CAD library that have been modified after a certain date. This feature may be particularly useful if you are not the only one using and modifying the same library files.

The CreateDate, ModifyDate and VerifyDate attributes can be automatically added to and updated in your P-CAD libraries. If the Query operator requires a value, the query expression for the above fields must contain at least the date. Both date and time are optional. There are many date and time formats acceptable to the Query utility.

Date

When entering the month and day in a Criteria (And) or Or column, use the following formats:

Description: Month and Day	Value
A three letter month abbreviation followed by the day number.	Jul 23
The full month's name followed by the day number.	July 23
The month number followed by the day number, divided by a slash.	7/23

The date requires both a month and day. Including the year in the date is optional. If a year is included, the following formats may be used:

Description: Month, Day and Year	Value
If the month is either a three letter abbreviation or the full name, the day number must be followed by a comma. The year is either a 2 digit or 4 digit entry.	Jul 23, 2001 Jul 23, 01
If the month and day are number values, the year number follows divided by a slash.	7/23/2001 7/23/01

If the year is not included, the current year is assumed. For example, if today's date is January 2nd, 2001, then a query expression containing 12/31 will automatically be December 31st 2001, not 2000.

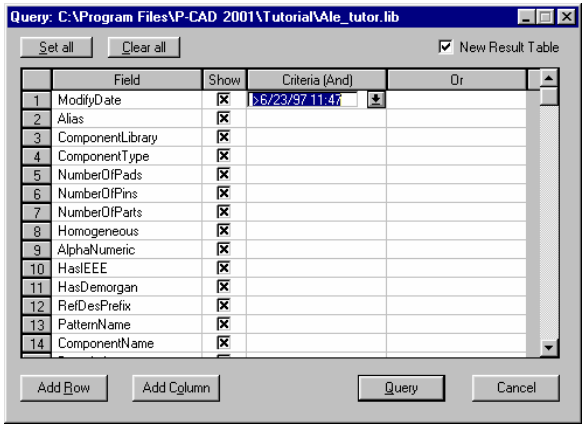
Time

The time entry, if included, follows the date. The Date and Time values must be divided by a space. Time requires only an hour value. Minutes and seconds are optional. When entering the time and date in a Criteria (And) or Or column, the time is always entered as a series of numbers divided by colons.

Time value: HH:MM:SS

- **HH**: The hour value on a 24 hour clock (0-24). The hour is required if a time value is used in the query expression.
- **MM**: The minute value (0-59).
- **SS**: The second value (0-59).

For example, to find a component that was modified in a library after 11:47 am on the July 23, 1997, the following search criteria may be entered in the *Query* dialog:



And Versus Or

In the *Query* dialog there are several columns where you can enter search criteria, the *Criteria (And)* column and the *Or* columns. When combining more than one search expression in a single query, both columns may be used to precisely define the desired component. This section details how different combinations of two or more search criteria can be used to construct a wide variety of component searches

When you enter your search criteria, its spreadsheet location defines your intended search. If you want to find a component that satisfies criteria X AND criteria Y, place X and Y in the same column. If you want to find a component to either satisfy criteria X OR criteria Y, place X and Y in different columns.

You can only use one search criteria per spreadsheet cell. Spreadsheet rows can be added to contain duplicate Field names if more than one search criteria is required on a single attribute.

For example, in the following query, the search is for a component that has one pin and whose name begins with the number 4. After executing this search, a one pin component with the name 4DHX57 might appear on the Query Results table.

	Field	Show	Criteria (And)	Or
1	ComponentName	☒	IsLike 4*	
2	NumberOfPins	☒	=1	

On the other hand, in the following query, the search is for a component that has one pin or whose name begins with the number 4. After executing this search, a one pin component with the name 4DHX57 might again appear on the Query Results table. However, it could be followed by the two pin component 4DXF42S and the one pin component 6HGD32.

	Field	Show	Criteria (And)	Or
1	ComponentName	☒	IsLike 4*	
2	NumberOfPins	☒		-1

You can put a field name in more than one row. This is needed for finding a component that has between 5 and 10 pins, for example:

	Field	Show	Criteria (And)	Or
1	NumberOfPins	☒	>5	
2	NumberOfPins	☐	<10	

There can be multiple *Or* columns if you want to search for components with criteria X or Y or Z. You can add *Or* columns to the spreadsheet, by clicking the **Add Column** button. Additional columns are included to the right of present *Or* columns.

You can also have multiple entries in an *Or* column if you want to search for components with the criteria **X** and **Y** or criteria **L** and **M**.

Specifying Output

You can specify the Query output in two ways:

- Which fields are displayed in the query result.
- Whether or not the present query overwrites the previous result.

To display a particular field in a query result, select the **Show** check box to the right of the field name.

To preserve the present query results, select the **New Results Table** check box. When the new query is executed, both the original query result and the new query result tables remain in memory. Easy access to all query result tables is available through the Source Browser. Refer to *The Source Browser (page 91)*, for additional information.

Executing a Query

After setting up your query, click the **Query** button. P-CAD Library Executive will search the selected source and automatically display the components satisfying your search criteria in a Query Result Viewer table.

For more information about the Query Result Viewer and how you can use query results in Library Executive, see *Query Results (page 113)*.

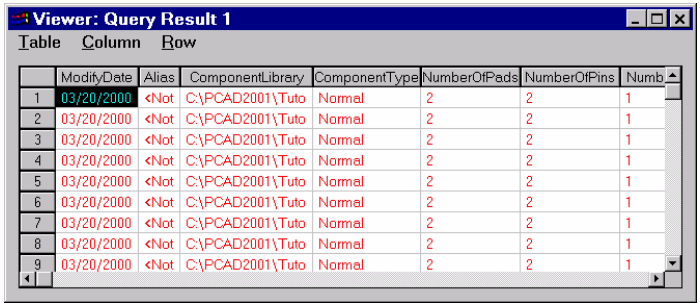
Query Results

When you execute a Query, the components found in the search appear on the Query Results Viewer.

This section describes the Query Result Viewer spreadsheet and how you can use it.

Query Result Viewer

After executing a Query, the Query Result Viewer spreadsheet appears.



	ModifyDate	Alias	ComponentLibrary	ComponentType	NumberOfPads	NumberOfPins	Numb
1	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1
2	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1
3	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1
4	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1
5	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1
6	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1
7	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1
8	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1
9	03/20/2000	<Not	C:\PCAD2001\Tuto	Normal	2	2	1

The Query Result Viewer is a simple spreadsheet form; it contains all of the components in the queried source that satisfy the selected criteria. Across the top of the table are the selected display fields. Their values, for each of the components found, are listed in the appropriate column.

The *Query Result Viewer* dialog is a spreadsheet. Refer to *Library Basics (page 3)* for details on manipulating spreadsheets.

Individual read-only field values cannot be modified, although read-only fields may be added or deleted from the table. A read-only field is displayed in red on the Viewer.

Query Result Menu Commands

The following reference section describes how to use all of the menu commands for the Query Result Viewer. These commands are listed in the order that they appear on the dialog menu: **Table**, **Column** and **Row**.

Table Commands

The Table commands allow you to access other features of P-CAD Library Executive, including Query, Cross Link, Verify, Report and Save To Library. They also allow you to rearrange the contents of the table by using cut, copy and paste. The following table describes the Table commands.

Command	Description
Cut	Deletes any selected component row from the Query Result Viewer and moves it to the Clipboard. This command will not work on a read-only field.
Copy	Copies selected item(s) from the Query Result Viewer onto the Clipboard.
Paste	Pastes items from the Clipboard onto the Query Results Viewer. The Paste command will not work on a read-only field.
Query	Opens the *Query* dialog where you narrow the component list of the present Query Results Viewer by including more search criteria.
Cross Link	Opens the *Cross Link* dialog where you can link component attributes in the selected file with their corresponding components in another source. The Cross Link Results Viewer displays a combination of the attributes in the two source files for the linked components.
Verify	Opens the *Verify* dialog where you can compare the components on the Query Results Viewer with components in a P-CAD library.
Report	Opens the *Report* dialog where you can generate a report on the components included in the Query Result Viewer.
Save To Library	Opens the *Save Source* dialog where you can save the components of the Query Result Viewer to a P-CAD library.
Close	Closes the Query Result Viewer.

For more information about Query, Verify, Report and Save commands applied to a Query Result Viewer, see *Using Query Results (page 116)*.

Column Commands

The Column commands are available from the **Column** or shortcut menus when a column is selected from the Query Result Viewer. With the Column commands you can add, remove, rename and sort the columns of fields and their values.

The following table describes the Column commands.

Command	Description
Add	Adds a column to the Query Result Viewer to the left of the currently selected column. Displays a dialog prompting you for the new field name.
Delete	Removes the currently selected column from the Query Result Viewer.
Rename	Opens a dialog where you can designate a new name for the currently selected column.
Sort	There are two methods available to sort columns: **Allow Duplicates** and **Resolve Duplicates**. The **Allow Duplicates** command sorts the field values, grouping duplicate values together. If you want unique field values, use the **Resolve Duplicates** command. The suffixes _1, _2, _3, etc. are added to the end of repeated values. The **Resolve Duplicates** command may be preferred, for example, when sorting the ComponentName field, since P-CAD requires a unique name for each component. The **Resolve Duplicates** command is available only for fields with data type String.

Row Commands

The Row commands are available when a row is selected from the Query Result Viewer either from the **Row** menu or the shortcut menu. With the *Row* commands, you can remove the component in the selected row. You can place a component in a P-CAD PCB or Schematic design. You can also open a component to view its component information.

Command	Description
Delete	Removes the currently selected row (component) from the Query Result Viewer.
Place	Places the selected component in a running PCB or Schematic design. Makes the program current with the *Place Part* or *Place Component* dialog open and the component selected.
Open	Opens the *Component Information* dialog.

The **Open** command is only available if the row contains the ComponentName and ComponentLibrary field. The **Place** command requires the row represents a valid component with an attached symbol or pattern.

For more information about placing a component from a Query Result Viewer into a PCB or Schematic design commands, see *Using Query Results (page 116)*.

Using Query Results

After completing a Query, all of the components satisfying the search criteria are displayed in a Query Results Viewer. You can use these components as you would any other source in Library Executive and place components selected in the Query Results Viewer directly to a design in PCB or Schematic.

This section lists the primary features you can access from the Query Results Viewer.

Query

Choose **Query** from the **Table** menu to open the *Query* dialog. With Query, you can further limit the component list of your Query Result by specifying additional search criteria.

Query results are stored on the Source Browse*r* tree. The shortcut menu on a query result allows you to view the query spreadsheet, generate a report, save the query to a library, or query again.

Cross Link

Choose **Cross Link** from the **Table** menu to open the *Cross Link* dialog. With Cross Link, you can link a component attribute in the query result with a matching attribute in a P-CAD library or library set. All attributes, from both the query result and the library, are displayed in the Cross Link Results Viewer.

Cross Link results are stored on the Source Browser tree. The shortcut menu on a cross link result allows you to view the cross link *Viewer* spreadsheet, generate a report, save the result to a library, query, or cross link again.

Refer to *Understanding Cross Linking (page 130)* for details about the cross link feature.

Verify

From the Query Results Viewer, you can verify the consistency between the queried components and a set of P-CAD libraries. A report is generated if any differences are found.

Choose **Verify** from the **Table** menu or from the shortcut menu in the Source Browser, after selecting the query result from the Source Browser tree.

For more information see *Library Verification (page 137)*.

Reports

To generate a report on a query result from the Query Result Viewer, choose **Report** from the **Table** menu or from the shortcut menu when a query result is selected in the Source Browser tree.

For additional information about generating reports, refer to *Reports (page 145)*.

Updating or Creating Libraries

You can use query results to create or update libraries. If a library is created from the Query Result Viewer, the library will include only the queried components. If a library is updated from the Query Result Viewer, only the select group of queried components in the table will be modified in the library. The attributes in the Query Result Viewer will be created or updated in the P-CAD library for components with the same name.

From the Query Result Viewer, choose the **Save To Library** command from the **Table** menu. You can also create or update libraries from the Source Browser by selecting the query result from the Source Browser tree and choosing **Save To Library** from the shortcut menu.

Placing Components

To place a component into P-CAD PCB or P-CAD Schematic from the Query Result Viewer:

1. Select the row containing the desired component information.
2. Choose **Place** in the *Row* menu.
3. Drag the mouse to the right over the arrowhead to select the desired application, PCB or Schematic.

> If you have accessed Query from either a *PCB Component Properties* dialog or a *Schematic Parts Properties* dialog, a **Replace** button appears in the upper left corner of the Query Result Viewer. To replace the current component with a selected component in the Query Result Viewer, click the **Replace** button.

You can also select the component from the Source Browser tree and choose **Place PCB** or **Place Schematic** from the shortcut menu.

For more information about placing components in a P-CAD PCB or P-CAD Schematic design, see *Component Placement (page 163)*.

Accessing Query and Query Results

There are several locations in Library Executive from which you can access the *Query* dialog and the Query Result Viewers.

Accessing Query from the Source Browser

Query results are stored on the Source Browser tree. The shortcut menu on a query result allows you to view the query Viewer spreadsheet, generate a report, save the query to a library, or query again.

You can access the *Query* dialog from the Source Browser by selecting a library set, a library, an external source file, a cross link result, or another query result and choosing **Query** from the shortcut menu.

Accessing Query from the Utils Menu

Choose **Utils » Query** to open the *Query* dialog containing the most recent Query conducted.

Accessing Query from the File Menu

Choose **File » Query** to open the *Query* dialog containing the most recent Query, if any, conducted on the external source file.

The **File » Query** command only queries a non-P-CAD data source file.

Using Query from PCB and Schematic

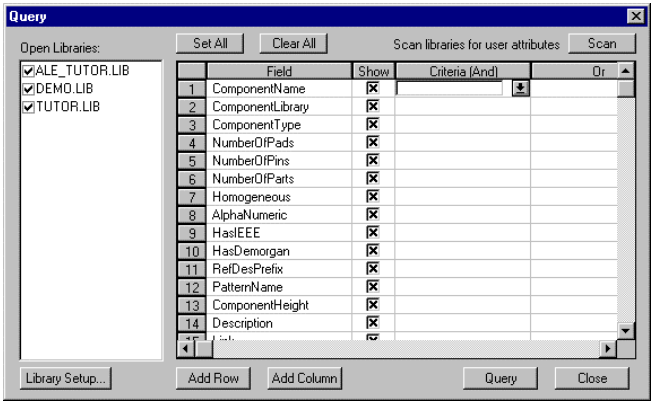
When Library Executive is installed, a Query button appears on the *Place Part* dialog in Schematic and the *Place Component* dialog in PCB.

1. Click the **Query** button to access the *Query* dialog.
2. To access Query from PCB or Schematic, you can also choose the **Query** command that appears in the application's Library menu.

The appearance and function of the *Query* dialog, when embedded in PCB or Schematic, is slightly modified. This section describes the changes to the *Query* dialog and how it is used from PCB and Schematic.

Embedded Query Dialog

When accessing the Library Executive Query embedded within Schematic or PCB, the *Query* dialog has a slightly modified appearance and function. The embedded *Query* dialog is shown below.



The first difference you might note is the list of open libraries on the left hand side. Selecting the check box next to the library name indicates that you would like to search that library using Query. You can open additional libraries by clicking the **Library Setup** button.

A second difference is the **Scan** button in the upper right hand corner. Clicking the **Scan** button searches all open libraries for user-defined component attributes and adds them to the pick list for the **Fields** column. By default the pick list contains only the primary field, ComponentName, and the other predefined attributes. User-defined component attributes may also be typed directly into the Fields column if you know the name of the attribute.

For the embedded query, there can be only one Query Result table.

Using the Embedded Query

This section contains a quick summary of using the Query embedded in PCB and Schematic.

1. Choose **Library Query** or click the **Query** button in the *Place Component* or *Place Parts* dialog to open the *Query* dialog.
2. Select the check boxes next to the open libraries that you wish to query for components.
3. Specify the search and display fields in the **Fields** column.

   To include user-defined attributes in the pick list, click the **Scan** button. All open libraries are searched for user-defined attributes; the attributes are added to the list.
4. Set up the search criteria in the **Criteria (And)** and **Or** columns.
5. Click **Query**. The Query Results Table appears with the components satisfying the search criteria in each row. The component(s) can then be placed in the PCB or Schematic design or queried again.

CHAPTER 9

Importing Data from an External Source

One of the most exciting features of P-CAD Library Executive is its ability to create and update a P-CAD library file from almost any source, including your company's component database. Once imported, the imported attributes can be used to create a P-CAD library, to verify and update a P-CAD library, or to search for the optimal component to place directly into a PCB or Schematic design.

With P-CAD Library Executive you can easily keep your P-CAD libraries current to changes in component attributes, availability and pricing. The ability to import data from any non-P-CAD source, combined with the other powerful features of P-CAD Library Executive, create a quick, convenient, no-fail system to find the best PCB design at the lowest cost.

What Can Be Imported?

Any component library or component database can be imported into Library Executive. For example, you may have a corporate component database that changes regularly to reflect the variable inventory and cost of components. This component database can be easily imported into P-CAD, updated regularly and searched for the optimal component to place in a design.

Before importing component data or a component library from a non-P-CAD source, it must be converted to a simple ASCII file format. The character-delimited ASCII file understood by Library Executive is a basic, inclusive format that can be created from any component database. Most spreadsheet applications allow you to output an ASCII file containing the spreadsheet data in a character-delimited file format. See *Character Separated File Format (page 121)* for details.

Character Separated File Format

There are two different, but closely related, character-delimited file formats that can be imported into Library Executive. The difference between the two is solely on the file's first line. The first line can contain one of the following:

- A list of character-separated field names.
- A list of character-separated field values, with no field name headers.

In the first format, line 1 contains a list of character -separated field names. Each field name is translated as a component attribute. Line 2 and greater contain a list of character-separated values corresponding to the attribute.

For example, the first few lines of a delimited file (using the letter x as its delimiter) to be imported into Library Executive could be:

```
"Part Type"x"Part Number"x"Cost (in $)"x"Vendor"x"Quantity"

"7400SL"x"0412345"x"0.25"x"Intel"x"10"

"RAM128KX8"x"0324314"x"50.0"x"National Semiconductor"x"25"
```

In the second format, there is no list of field names. Line 1 and greater simply contain the list of comma- separated values. In the absence of the field name list, default field names are generated for each column of the file.

In the second format, the above example would be simply:

```
"7400SL"x"0412345"x"0.25"x"Intel"x"10"

"RAM128KX8"x"0324314"x"50.0"x"National Semiconductor"x"25"
```

For a character-separated file of the latter format, default field name headers will be attached when imported.

In both cases, the imported field name headers can be mapped to field names recognized by P-CAD.

Import Limitations

There are a few limitations regarding what may be imported into Library Executive. First, there are maximum limits on how long an attribute name or value may be. Second, list separators and comment characters, with the exception of the double quote, can be changed to a character of your choice.

- **Attribute Names and Values**: A maximum number of characters that may be included in attribute names and values when importing them into P-CAD Library Executive. The *ComponentName* field value has a maximum length of 30 characters. In general, attribute names can have a maximum length of 20 characters. An attribute value can be up to 250 characters in length.

- **Using and Importing Special Characters**: Special characters, except the double quote, may be imported. The double quote is used exclusively to enclose a string of characters. Any character can be part of the characters inside a double quoted string of characters.

  For example, you can include the following component description attribute: "`3 Input NAND Gate, .01 uF`" with a comma delimiter. To import this description, the comma-delimited file should contain "`3 Input NAND Gate\, .01 uF`". The comma becomes part of the attribute value, as opposed to signifying the beginning of the next attribute.

Importing a Character Separated File

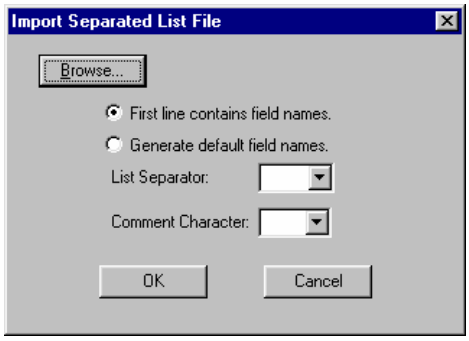
For any set of component attributes, either in a P-CAD library or in an external source file, there is an attribute (or combination of attributes) that indicate which component the set of attribute values belongs to. In a P-CAD library, the component name is the identifier that references only one unique component with that name in the library. When a ComponentName field is specified, all component attributes associated with that particular P-CAD library component are automatically included. The ComponentName in P-CAD is the key that references the rest of that component's attributes.

Depending on how your external component database is organized, a specific attribute or set of attributes identifies a component and all its associated component attributes. This attribute may or may not be the component name as used by P-CAD. In this document, the attribute that identifies the component and its attributes is called the primary key when referring to P-CAD's ComponentName. The attribute(s) used to identify the component and its attribute in an external source file is called an index identifier.

The P-CAD ComponentName must be associated with the imported data so that Library Executive can determine which library or design component the imported attributes belong to. This association of the ComponentName field with the imported component data can be completed with one of two commands: **Map Fields** or **Cross Link**. The command of choice depends on the original contents of the imported character-delimited file.

To import a character-separated file containing component attribute data, follow these steps:

1. Choose the **File** » **Import** command to open the *Import Separated List File* dialog.



> You can also access the *Import Separated List File* dialog from the Source Browser. Select the root of the Source Browser tree, Sources, and choose **New Comma-Separated File** from the shortcut menu.

2. Click the **Browse** button to navigate to and select the file containing the character-delimited file.
3. Click **Open**. The name of the file to be imported is displayed next to the **Browse** button.

4. If the first line of the character-delimited file contains field names, select the First line contains field names option button.

   If the first line of the character-delimited file does not contain field names, select the **Generate** default field names option button. Default field names will be attached to the imported data.

   The primary key, ComponentName, is the only attribute required for the import process. This attribute tells Library Executive which component the imported attributes belong to.

   To include the ComponentName field in your imported data file, there are two possible processes depending on the contents of your external source file: field mapping or cross linking. Field mapping is appropriate for character-delimited files containing the P-CAD primary key, ComponentName. On the other hand, cross linking is optimal for importing external source files that do not contain this essential information.

   Refer to the *Field Mapping (page 124)* or *Cross Linking (page 126),* for details.

   In both cases, any imported field name header can be mapped to a field name recognized by P-CAD. For example, the imported field name Component Price may be mapped to the P-CAD-recognized field name, Cost. Any field name can be mapped by choosing the **File » Map Fields** command.

5. Choose the desired separator character from the list in the List Separator box or type the character you want to use (with the exception of the double quote).

6. Select the character to be used to denote comments from the Comment Character list or type the character you want to use (except a double quote).

7. Click **OK** to open the *Viewer* dialog.

Field Mapping

P-CAD Library Executive has knowledge of a set of field names, including a primary key, predefined properties and attributes and user-defined attributes. These fields are listed in *Fields and Data Types (page 103)*.

When importing data from an external source, the original field names may or may not be recognized by P-CAD. If unrecognized, Library Executive assumes the data within the field are user-defined attribute values. If recognized, the component data within those fields are understood by P-CAD.

Both user-defined and predefined attributes can be used throughout Library Executive, including library update, design and library verification and query. If you want to compare, for instance, the number of pads on the component in the P-CAD library and the external source file, both must be named the P-CAD recognized name *NumberOfPads*. Mapping fields in the imported source file connects the attributes and their values between the P-CAD library and external source.

The **File » Map Fields** command maps the field name to predefined attribute or property names recognized by P-CAD or to other user-defined attribute names.

The primary key, ComponentName, is the only attribute required to complete the import process. This attribute tells Library Executive which component the imported attribute values belong to. If the imported file contains the ComponentName field, the import process can be completed by

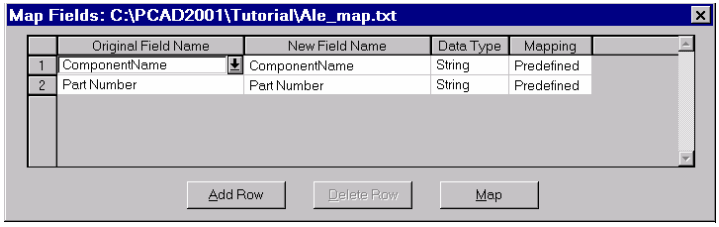
field mapping. If the imported file does not contain a component name attribute, refer to *Cross Linking (page 126)* for details on how to include this required ComponentName field.

The mapped fields are properties and attributes that form the building blocks of a complete, intelligent component.

Map Fields maps the original character-delimited file. Any modifications made to the file contents within the Viewer prior to mapping will be lost when the **Map Fields** command is chosen. To save these modifications, choose **Save To Library** before mapping.

To map the imported fields:

1. Choose **Map Fields** from the **Table** menu of the Viewer to open the *Map Fields* dialog.



You can also choose **Map Fields** from the **File** menu on the main *Library Executive* dialog or from the shortcut menu when the imported external file is selected.

The fields in the *Map Fields* dialog are:

Original Field Name: The field names from the imported separated list file or the default field names if generated by P-CAD Library Executive.

New Field Name: The field name to be used by P-CAD.

Data Type: The field type that defaults to String for any unrecognized field name.

Mapping: Shows if the attribute has been mapped to a recognizable predefined P-CAD field name or to a user-defined name.

2. To map the imported fields to names recognized by P-CAD, click in the **New Field Name** column. A down arrow appears at the right of the selected field name. Click the **Down Arrow** and the field list appears.

3. Select the desired P-CAD field name from the list. The field is automatically placed on the spreadsheet. The field type is automatically updated in the Data Type column for the P-CAD recognized name.

4. Modify the *Data Type* from the list of data types available in the third column. Modifying the data type from its default value may be desired for two reasons:

   Sort order: The Integer and String data types have different sorting orders. See *Viewer Menu Commands (page 131)* for more information.

Display: Selecting Boolean data type displays True or False attribute values. Selecting Integer displays these attributes as the integers 0 or 1.

The mapped field values are attributes that can be used to update, verify, or search components in the imported file or library. Read-only attributes are for search and display purposes only. The default data type, String, is always used when saving the remaining attributes to a P-CAD library.

Read-only attributes cannot be modified nor saved to a P-CAD library. These attributes are displayed in red on the imported file Viewer.

When the Original Field Name is mapped to a New Field Name the field type is automatically updated in the Mapping column. The field type indicates whether the attribute is a primary key, a predefined attribute, or a user-defined attribute.

5. You can map the original field name to more than one P-CAD recognized field name. Click the **Add Row** button to add an additional row to the end of the table. A list of original field names from the separated list source file is available in the first column.

   This one-to-many mapping may be useful when your corporate database is indexed differently than a P-CAD library. For example, the database contains two 7400 components produced by two different manufacturers. The database field Part Number can be mapped to both a ComponentName field, which must be unique for P-CAD products, and a predefined Part Number field, which is the original name 7400.

   Click the **Delete Row** button to delete a selected row.

   You can only delete rows that have been added to the table.

6. When all Original Field Names have been mapped to their desired New Field Name, click the **Map** button. You are reminded that the original imported file is mapped, any changes you have made to the file contents in the Viewer are lost during the mapping process.

7. The Viewer reappears with the fields mapped to the chosen P-CAD values. The word, mapped, will appear next to the mapped file name.

Library Executive remembers the specified field mapping for the latest mapped file. For example, changes are made to your corporate component database every week. With Library Executive mapping, you can periodically update your component libraries. If you map the fields today, then next week when you import the database again to verify your P-CAD library, the mapping is completed automatically.

If you import a file with the same filename but different field names than the previously imported file, the original mapping will be overwritten.

Cross Linking

The Cross Link command links attributes from two sources and combines them into one. Although the Cross Link command can link any two source files in Library Executive, it can be particularly useful when importing an external source file that does not include a ComponentName field.

Combining component attributes from a P-CAD library and an external source file can help you easily include the required ComponentName field in the imported data.

In *Field Mapping (page 124)* the field mapping of an imported separated list file was discussed. Field mapping is the optimal method of including the required ComponentName field in the imported file, if the external source file contains the ComponentName field. For example, the source file field Name of Component may be mapped to the P-CAD primary key ComponentName using the **Map Fields** command.

If the external source file does not contain the name of the components, the Cross Link command is the optimal method to include a ComponentName field. For instance, the component information you want to import is organized with a unique Part Number as the database index identifier. There is a component name field, but the component names are not unique. There may be, for instance, several components named 7400 produced by different manufacturers. These components are distinguished by a unique Part Number in the database.

The Cross Link feature of P-CAD Library Executive allows you to link the unique Part Number directly to its corresponding component in the P-CAD libraries. The imported separated list file does not need to have a unique ComponentName field.

The Cross Link command can link the Part Number in the external source file to the ComponentName field in a P-CAD library. The result of the Cross Link command is a table containing a combination of component attributes from the two linked component data sources. When importing an external source file, the common sources to be linked are the external source file, a P-CAD library and a map file. The Cross Link command, the above mentioned source files and how they are used in the process of importing an external source file, is discussed in *Attaching the Index Identifier (page 127*) which details how a map file is used to link the Part Number to the ComponentName, followed by *Import By Linking the Index Identifier (page 128)* which describes how to complete the import process once the ComponentName has been included with the external source file's component attributes.

The cross linked component attributes are displayed in a file *Viewer* dialog and are stored in a Cross Link Result. This Cross Link Result can be accessed through the Source Browser and used as the source for any P-CAD Library Executive function. For example, cross linked component attributes can easily be used to update or verify P-CAD libraries. Components in a Cross Link Result may also be placed directly in a PCB or Schematic design, if the components are complete.

The power and flexibility of the Cross Link feature aids far more than simply importing external source files. Cross Link is a quick way to combine component attributes from a variety of sources. Refer to *Understanding Cross Linking (page 130*) to maximize the cross link power at your fingertips.

Attaching the Index Identifier

In order to cross link an external source file, you must associate the index identifier (one or more fields which uniquely identify each component) in your character-delimited file to its respective component in a P-CAD library. If your database component attributes are organized by Part Number and each Part Number is unique, for example, you will link each Part Number to its P-CAD ComponentName.

Making this link between the components in the external source file and the P-CAD library can be accomplished in many ways. You can choose to enter the Part Number as a user-defined attribute in your P-CAD library by typing the index identifier attribute(s) directly into a Library Executive spreadsheet. You can also choose to use a map file to automatically connect the Part Number to the ComponentName by choosing the **Cross Link** command, which is the recommended method.

The end result of whatever process you choose is a single source file containing both the P-CAD ComponentName and the component's index identifier as a user-defined attribute(s). For example, cross linking a map file with a P-CAD library will result in a source containing all of the component properties and attributes stored in the library, including ComponentName and the index identifier for each component from the map file, for example Part Number. If your database uses a combination of Part Number and Manufacturer attributes to reference its components, then the source file would contain columns for both these fields and ComponentName.

For example, start by importing a map file that contains the two fields: ComponentName and Part Number. The Cross Link command links the ComponentName of the map file to a P-CAD Library. The cross link result then has all of the P-CAD library data and an additional user-defined attribute column, Part Number for each component. The cross link result may now be used to save the Part Number attribute to the library, or to link to the Part Number attribute of another external source file.

Import By Linking the Index Identifier

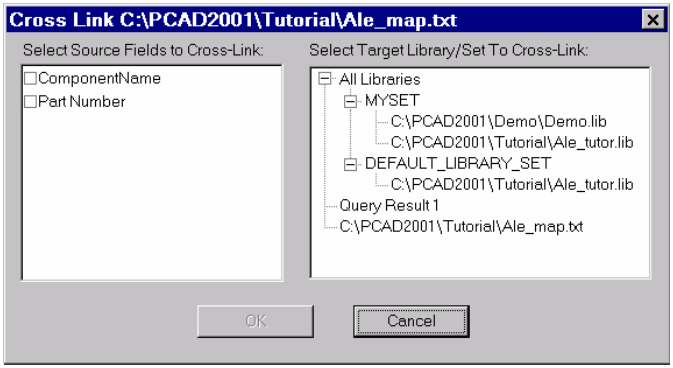
The Cross Link command links two source files by matching component attributes. Beginning with the newly created source file that contains the ComponentName and the index identifier, you can cross link with an external source containing only the index identifier. This link adds the ComponentName field required by P-CAD to the appropriate components of the external source. In the example begun in *Attaching the Index Identifier (page 127),* you want to link the Part Number in the imported character- delimited file to the Part Number in the source file that contains both the ComponentName and Part Number.

One source, the external source file, contains all of the attributes you wish to import from your company component database and the index identifier, Part Number. This external source does not have the essential ComponentName field that identifies a P-CAD component. The other source, either a P-CAD library or a cross link result, contains other component attributes, including the *Part Number* index identifier and the P-CAD primary key, ComponentName.

The *Cross Link* command links the attributes of these two source files by matching the unique Part Numbers of the two sources. The result of the cross link is displayed in a Viewer spreadsheet. The cross link result will contain every component found in both source files. All component attributes from both sources will be included in the result.

To cross link the imported field, an initial association must be made between the unique index identifier(s) in the external source file and the corresponding components in the P-CAD libraries as described in *Attaching the Index Identifier (page 127)*. The following steps complete the cross link for the imported fields:

1. Choose the **Cross Link** command from the **Table** menu of the Viewer to open the *Cross Link* dialog.



You can also choose the **Cross Link** command from the **File** menu on the main *Library Executive* dialog or from the shortcut menu when the imported external file is selected in the Source Browser.

2. In the **Select Source Fields to Cross Link** box, select the check box to the left of the field name(s) to link to the source file created in *Attaching the Index Identifier (page 127)*. The Part Number is the unique index identifier for the corporate component database and the check box to the left of the Part Number field should be selected.

3. In the **Select Target Library/Set to Cross Link** box, expand the groupings by clicking the **+** sign to their left. Navigate to the desired target source, library or library set and select the target.

   In this case, a P-CAD component library has been updated to include the index identifier of the imported external source. If you do not wish to save the index identifier to your P-CAD libraries this target may also be a Cross Link result.

4. Click **OK**.

   The selected target is searched for the chosen source field, in this case the index identifier Part Number. When a component with a matching source field is found in the target, all component information in both sources is displayed in the Cross Link Result Viewer.

Viewer: Cross Link Result 2

Table Column Row

	Part Number	Quantity	Vendor	Cost	ComponentName	Alias	Componer
1	238704	32487	Panaso	.29	CAP300	<Not	C:\Accel13.1
2	238709	4218	Panaso	.29	CAP450	<Not	C:\Accel13.1
3	238710	40458	Panaso	.29	CAP500	<Not	C:\Accel13.1
4	238711	4244	Plessey	.34	CAP500AP	<Not	C:\Accel13.1
5	238715	3489	Plessey	.34	CAP1000AP	<Not	C:\Accel13.1
6	238716	8766	Plessey	.34	CAP1200AP	<Not	C:\Accel13.1
7	238801	1279	Plessey	.28	POLCAP	<Not	C:\Accel13.1
8	438802	8743	CTS	.82	RES6SIPISOL	<Not	C:\Accel13.1
9	438803	200481	CTS	.82	RES7DIPISOL	<Not	C:\Accel13.1
10	438804	54843	CTS	.82	RES8DIPISOL	<Not	C:\Accel13.1
11	438809	65438	CTS	.82	RES10SIPDT	<Not	C:\Accel13.1
12	438811	7000	CTS	.82	RES14DIPB	<Not	C:\Accel13.1
13	438812	23511	CTS	.82	RES14DIPDT	<Not	C:\Accel13.1
14	438813	65943	CTS	.82	RES14DIPI	<Not	C:\Accel13.1
15	438814	13083	CTS	.82	RES16DIPB	<Not	C:\Accel13.1
16	437909	35081	Yageo	.33	RES400	<Not	C:\Accel13.1
17	437914	456892	Yageo	.33	RES850	<Not	C:\Accel13.1
18	437915	97561	Yageo	.33	RES900	<Not	C:\Accel13.1
19	437916	54324	Yageo	.33	RES1200	<Not	C:\Accel13.1
20	437917	44571	Yageo	.33	RES1300	<Not	C:\Accel13.1

The Cross Link Result Viewer is a simple spreadsheet form; attributes found in both the source file and the target are displayed. Across the top of the table are the fields found in the source and target. If a field name is found in both source and target, values from the target field are distinguished by their column header TARGET:FieldName. The field values, for each of the components found, are listed in the appropriate column.

For details on manipulating the Cross Link Result Viewer spreadsheet, see *Library Basics (page 3)*.

Understanding Cross Linking

Cross linking is one of the most powerful features of P-CAD Library Executive. You can cross link between any two sources in P-CAD Library Executive: an external source file, a P-CAD library or library set, a query result and a cross link result. The output is a result in the Viewer spreadsheet that contains a combination of the component attributes originally in both sources.

> The Cross Link command links the source files as they are in memory. If changes have been made to a library used as the cross link source or target, the **Reload** command refreshes the library components in Library Executive's memory before the Cross Link is performed.

To understand how cross linking can be used to combine data from various sources, consider the following examples.

Setting up a Cross Link

The examples in this section both work with the same two source files: Source #1 contains 3 components (A, B and C); source #2 contains only two components (A and C). Each of these sources contains different component attributes. The only requirement for cross linking is that the two

sources have one common attribute. This common attribute will be used to link the sources together. In this example, the common attribute will be ComponentName, but it could be anything.

With these two source files you could:

- Combine the attributes in Source 1 and 2 to produce a component containing complete attribute information from both sources that can be placed on a PCB or Schematic design.
- Use a combination of attributes from both sources to verify or update components in a P-CAD library.
- Generate a report about the components, which includes the attributes from both sources.

Source #1 is a P-CAD library that has all the component attributes needed for an integrated library component and contains four attributes: ComponentName, Attr1, Attr2 and Attr3.

Source #2 contains component attributes from an external source and also includes information about cost and availability of some of the library (Source #1) components. Specifically, source #2 has the ComponentName and Attr3 attributes. Instead of Attr1 and Attr2, this source contains Attr5 and Attr6.

Cross Link Example #1

In example number one, both ComponentName and Attr3 are used to link the fields between the two source files. The result of this cross link is a file containing only components matching the ComponentName and Attr3 field values.

Cross Link Example #2

In this example only the field ComponentName is used. This second cross link produces a new file containing all six attributes. The result differs from the first Cross Link Result in that there will be two Attr3 attributes, one from Source #1 and another from Source #2. In this case, there is no attempt at comparing the repeated attributes, so components with matching or unmatching Attr3 values are included.

Cross Link Results are stored in the Source Browser. You can select a result at any time in the future to use a particular combination of components and component attributes. With a cross link result you can update a library, search for a component with specific attributes, generate a report, verify attributes in a P-CAD library and even cross link again to another Source file.

Viewer Menu Commands

The following reference section describes how to use all of the menu commands for the Viewer. These commands are listed in the order that they appear on the dialog menu: **Table**, **Column** and **Row**.

The Viewer displays the contents of both an external source file and the cross link results. The Viewer for the external source file has the additional Table command **Map Fields**.

Table Commands

The Table commands allow you to access other features of P-CAD Library Executive, including Query, Cross Link, Verify, Report and Save To Library. They also allow you to rearrange the contents of the table by using cut, copy and paste. Of particular importance for importing files from an external source are the **Map Fields** and **Cross Link** commands which can be accessed from the **Table** menu.

Command	Description
Cut	Deletes any selected component row from the Viewer and moves it to the Clipboard.
Copy	Copies selected item(s) from the Query Result Viewer onto the Clipboard.
Paste	Pastes items from the Clipboard onto the Viewer.
Query	Opens the *Query* dialog where you can narrow the component list of the imported file by including search criteria.
Cross Link	Opens the *Cross Link* dialog where you can link component attributes in the selected file with their corresponding components in another source. The Cross Link Results Viewer displays a combination of the attributes in the two source files for the linked components.
Verify	Opens the *Verify* dialog where you can compare the components in the Viewer with components in a library.
Report	Opens the *Report* dialog where you can generate a report on the components included in the *Viewer*.
Save To Library	Opens the *Save Source* dialog where you can save the components in the Viewer to a P-CAD library
Map Fields	Opens the *Map Fields* dialog where you can map the field names imported from an external source file to field names recognized by P-CAD Library Executive.
Close	Closes the Viewer.

The **Verify** and **Save To Library** commands are only available if the primary field, ComponentName, is present.

For more information about Query, Cross Link, Verify, Report and Save To Library commands applied to the imported file, see *Using the Imported Source File (page 134)*.

Column Commands

The Column commands are available from the **Column** menu and the shortcut menu when a column is selected in the Viewer. These commands allow you to add, remove and sort the columns of fields and their values.

Command	Description
Add	Adds a column to the *Viewer* to the left of the currently selected column. Opens a dialog where you can choose a new field name.
Delete	Removes the currently selected column from the Viewer.
Rename	Opens a dialog where you can specify a new name for the currently selected column.
Sort	Two methods are available to sort columns: **Allow Duplicates** and **Resolve Duplicates**. The **Allow Duplicates** command sorts the field values, grouping duplicate values together. The **Resolve Duplicates** command sorts the fields uniquely. The suffixes _1, _2, _3, etc. are added to the end of repeated values. The **Resolve Duplicates** command may be preferred when sorting the ComponentName field, since P-CAD requires a unique name for each component. The **Resolve Duplicates** command is available only for fields with data type String.

There is a different sorting method for String and Integer data types. The default data type, String, is always used when saving attributes to a P-CAD library. Only modifiable attributes (not read- only) are saved to the library.

Row Commands

The Row commands are available from the **Row** menu or the shortcut menu when a row is selected in the Viewer. With the Row commands, you can add a row or remove the component in the selected row. You can place a component in a P-CAD PCB or Schematic design. You can also open a component to view its component information.

Command	Description
Delete	Removes the currently selected row from the Viewer.
Place	Places the selected component in a running PCB or Schematic design. Brings up the application with the *Place Part* or *Place Component* dialog open and the component selected.
Open	Opens the *Component Information* dialog.

The **Open** command is only available if the row contains the ComponentName and ComponentLibrary fields. The **Place** command requires the row represents a valid component with an attached symbol or pattern.

For more information about placing a component from the Viewer into a PCB or Schematic design commands, see *Using the Imported Source File (page 134)*.

Using the Imported Source File

From the Viewer, the command menus access the Library Executive features. You can even place components selected in the Viewer directly to a design in PCB or Schematic.

This section lists the primary features you can access from the Viewer.

Query

You can access Query from the Viewer by choosing **Query** from the **Table** command menu. With Query, you further limit the component list of the imported source file by specifying desired search criteria.

Query results are stored on the Source Browser tree. The shortcut menu on a query result allows you to view the query spreadsheet, generate a report, save the query to a library, or query again.

Cross Link

You can access Cross Link from the Viewer by choosing **Cross Link** from the **Table** command menu. With Cross Link, you can link a component attribute in the mapped imported source file with a matching attribute in any target source: a P-CAD library, query result, or cross link result. All attributes, from both the source file and the library, are displayed in the Cross Link Results Viewer.

Cross Link results are stored on the Source Browser tree. The shortcut menu on a cross link result allows you to view the cross link Viewer spreadsheet, generate a report, save the cross link result to a library, query, or cross link again.

Refer to *Understanding Cross Linking (page 130)* for details about the cross link feature.

Verify

From the Viewer, you can verify the consistency between the imported components and a set of P-CAD libraries. A report is generated if any differences are found.

The **Verify** command is available from the **Table** menu or from the shortcut menu for a selected, imported separated list file in the Source Browser tree.

For more information on verification, see *Library Verification (page 137)*.

Reports

To generate a report on an imported source file from the Viewer, choose **Report** from the **Table** menu or select an imported file from the Source Browser tree and choose **Report** from the shortcut menu.

For additional information about generating reports, refer to *Reports (page 145)*.

Updating or Creating Libraries

You can use imported separated list files to create or update P-CAD libraries. A new library can be created from an external source file. Later, you can easily update this library as the external source file changes.

To create or update P-CAD libraries from an imported file, choose **Save To Library** from the **Table** menu in the Viewer. You can also create or update libraries from the Source Browser by selecting the external source file in the Source Browser tree and choosing **Save To Library** from the shortcut menu.

Placing Components

To place components into P-CAD PCB or P-CAD Schematic from the Viewer, select the row containing the desired component information. Choose **Row » Place** and select the desired application, PCB or Schematic.

You can also select the component from the Source Browser tree and choose **Place » PCB** or **Place » Schematic** from the shortcut menu.

For more information about placing components in a P-CAD PCB or P-CAD Schematic design, see *Component Placement (page 163)*.

Accessing the Imported File

You can access the imported source file from several locations in Library Executive.

Accessing from the Source Browser

Imported source files are stored on the Source Browser tree. The shortcut menu on an imported file allows you to view the Viewer spreadsheet, map fields, cross link, generate a report, save the imported component attributes to a library, or query.

Cross link results are also stored on the Source Browser tree. The shortcut menu on a cross link result allows you to view the cross link spreadsheet, generate a report, query, cross link again, or save the cross link result to a library.

Accessing from the View Menu

Choosing the **View » Comma-separated File** command displays the imported source file in the Viewer.

CHAPTER 10

Library Verification

With P-CAD Library Executive you can easily check for discrepancies between components in different sources. You can check for differences between the following sources:

- An external source file and a P-CAD library
- A PCB or Schematic design and a library
- A Query Result and a library
- A Cross Link Result and a library
- P-CAD libraries

Perhaps you have created a P-CAD library from your corporate component database. With library verify you can quickly check whether the database has since changed. And how it has changed.

Perhaps you have been working on a design that may contain some of the altered components. With design verify you can validate the design directly and update it to reflect any database changes.

This chapter describes the library and design verification tolls of P-CAD Library Executive.

Verifying a P-CAD Library

From P-CAD Library Executive, you can verify difference between a P-CAD library or library set and the following sources:

- an external source file
- a query result
- a cross link result
- another P-CAD library or library set.

Depending on the desired source, the library verify tool is started in distinct ways. Refer to *Accessing the Verify Command (page 138)* for details.

The basic setup of the verification tool is identical, independent of the source. Customizing the common verification dialog is covered in *The Verification Report (page 140)*.

The differences between source and library are logged in a report file. This difference report is described in *The Verification Report (page 140)*.

The **Verify** command compares component attributes only. Attributes of the attached symbols and patterns are not included in the difference report.

Accessing the Verify Command

Choose the **Verify** command to compare a component source to a P-CAD library or library set. The Verify command can be accessed from the Source Browser, the **File** menu or the **Table** menu. The location from which you choose the Verify command defines the components you intend to use as a source for the comparison. For example, choosing **Verify** from a Query Result table generates a report of the differences between the components listed in that Query Result and a P-CAD library or library set.

This section describes how and where to choose the **Verify** command to get the results you want.

From the Source Browser

Choose **Verify** from the shortcut menu for a selected library, library set, query result, or external source file in the Source Browser tree. The following list describes the results for each selected item:

- **Library**: The components listed in that library are compared to components in another specified P- CAD library or library set.
- **Library set**: The components listed in all libraries in that set are compared to components in another specified P-CAD library or library set.
- **Query result**: The components listed in that Query Result Viewer are compared to components in the specified P-CAD library or library set.
- **Cross link result**: The components listed in that Cross Link Result Viewer are compared to components in the specified P-CAD library or library set.
- **External source file**: The component attributes listed in the file Viewer are compared to components in the specified P-CAD library or library set.

From the Table Menu

Choose **Verify** in the **Table** menu of the Viewer to verify imported source files, query results, cross link results and P-CAD libraries.

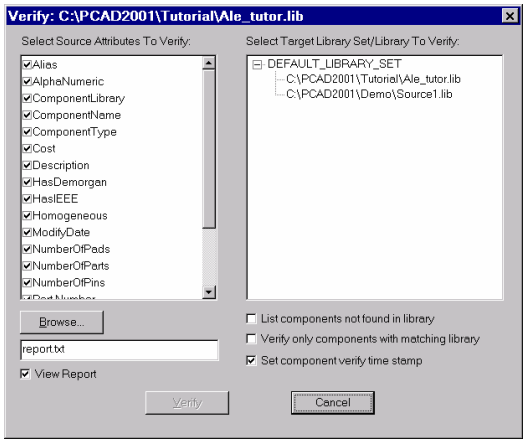
From the File Menu

Choose **Verify** from the **File** menu to verify the mapped, imported source file against a specified P-CAD library or library set.

Setting up the Verify Dialog

To compare a P-CAD library or library set to a source, follow these steps:

1. Choose **Verify** to open the *Verify* dialog.



   See *Accessing the Verify Command* above for details on how to choose the Verify command to check the consistency of different sources.

2. The Select Source Attributes to Verify box lists the attributes in the selected source. All selected attributes are verified against a P-CAD library or library set. Selected attributes are indicated by the check mark.

   The Verify command uses source component attributes from the current contents of Library Executive memory, as displayed in the Viewer. Verify does not use the original source file for comparison. If changes have been made to a library used as a source for the Verify command, choose Reload to refresh the Library Executive's memory before conducting the verification.

3. The tree structure in the Select Target Library Set/Library to Verify box allows you to view all Library Executive library sets and their contents at various levels of detail by expanding or collapsing the branches of the tree.

   Groupings containing collapsed levels are shown with a + sign. To expand the grouping, simply click the +. Expanded groupings are shown with a – sign. To collapse the grouping, simply click the – sign.

   Select the library or library set which you want to verify. If a library is selected, its contents are compared to the selected source attributes chosen in step 2. If a library set is selected, all libraries within that set are compared.

   If the **List components not found in library** box is cleared, components in the source that are not in the selected library are listed in the difference report.

If the **Verify only components with matching library** box is selected, Library Executive searches the source for components whose ComponentLibrary attribute matches the selected library. Only matched components are compared to that library.

If the selected source does not have a ComponentLibrary attribute, the **Verify only components with matching library** check box is ignored. All attributes in the selected source are verified.

When verifying a library, the component's time stamp attribute VerifyDate, if present, is automatically updated with the current date and time. If the **Set component verify timestamp** check box is selected, and the VerifyDate attribute does not exist, the VerifyDate attribute is added to the component library.

4. Type the desired report file name in the box or click the **Browse** button to navigate to and select a report file name. The selected report file name is displayed.
5. Click **Verify** to generate the difference report.

   If you checked the **View Report** checkbox, the difference report is displayed on the screen.

The Verification Report

The verification report contains differences in the components or component attributes between the source and the target library. These differences are listed in an easy-to-read ASCII format.

The first line of the report includes the verification source, which can be a library, library set, query result or external source file. It also names the first target library against which the source is being compared. If the selected target was a library set, the verification report format is repeated for second and higher numbered target libraries in the set.

The differences between the source and target(s) are reported, including the following information: component name, attribute, source value and library value. For each component, multiple attributes and values may be reported depending on the attributes selected in the *Select* Source Attributes to Verify list of the *Verify* dialog.

The following file excerpt is a simple example of a verification report.

```
report.txt - Notepad
File  Edit  Search  Help
Verify Source: C:\PCAD2001\Demo\Source1.lib  Against Target Library: C:\PCAD2001\Tutorial\Ale_tutor.lib.

 Component                        Property                Source                          Target


=================================================================

CAP1000AP
                                  ComponentLibrary        C:\PCAD2001\Demo\Source1.lib
                                  NumberOfPads            0                               2
                                  NumberOfPins            0                               2
                                  NumberOfParts           0                               1
                                  AlphaNumeric            True                            False
                                  RefDesPrefix                                            C
                                  PatternName                                             CAP1000AP
                                  CreateDate              03/30/2000 12:00:22             <Not Exist>
                                  ModifyDate              03/30/2000 12:00:22             03/20/2000 12:11:08
                                  Part Number             435-5487                        238715
RES
                                  ComponentLibrary        C:\PCAD2001\Demo\Source1.lib
                                  NumberOfPads            0                               2
                                  NumberOfPins            0                               2
                                  NumberOfParts           0                               1
                                  AlphaNumeric            True                            False
                                  RefDesPrefix                                            R
                                  PatternName                                             RES300
                                  CreateDate              03/30/2000 12:00:22             <Not Exist>
                                  ModifyDate              03/30/2000 12:00:22             03/20/2000 08:20:54
                                  Part Number             334-4589                        479802
```

The verification report is a difference report. It does not include attributes that are the same in both the source and the target library. It also does not include attributes that exist in the source, but not in the library.

The Verify command only reports differences between the source and target library. You can also choose the **Verify** command directly from P-CAD PCB and P-CAD Schematic. In this case, Verify reports on differences between the current design and the target library. Using Verify from a P-CAD design, you can optionally choose to update the writeable component attributes and properties from an open library. For details, refer to *Using Verify from PCB and Schematic (page 141)*.

Using Verify from PCB and Schematic

With P-CAD Library Executive, you can access the **Verify** tool from both P-CAD PCB and P-CAD Schematic to verify your current design. You can also specifically check the design's component attributes in comparison to a set of open libraries.

With the Design Verifier, you can easily keep your design current - incorporating updates in the corporate component database as it reflects the changing marketplace.

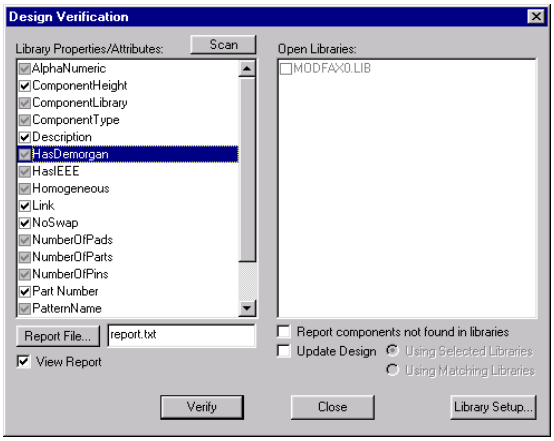
> As with library verification, the Design Verify command compares component attributes only. Attributes of the attached symbols and patterns are not included in the difference report.

Verifying a Design

Verifying a design from P-CAD PCB or P-CAD Schematic is similar to using the Verify command in Library Executive. This section covers in detail the process of verifying a design.

To verify a design from Schematic or PCB:

1. Choose **Verify Design** from the **Library** menu while the design is in the current workspace to open the *Design Verification* dialog.



2. The Library Properties/Attributes list displays only the primary field, ComponentName, and the other predefined properties and attributes.

   To add user-defined attributes to the **Library Properties/Attributes** list, click the **Scan** button. Clicking the **Scan** button searches all open libraries for user-defined attributes and adds them to the list.

3. Select the attributes in the Library Properties/Attributes list you want to verify between the design and any open libraries.

   A grayed check box indicates that the property or attribute is read-only. Read-only attributes can be verified, but not updated in the design.

4. The Open Libraries list box contains all open libraries in the current design. Design verification reports on differences between components in the design and all open libraries.

   To add, remove, or rearrange libraries in the Open Libraries list, click the **Library Setup** button. See your PCB or Schematic User's Guides for additional information.

   If the **Report components not found in library** option button is selected, components in the design that are not in the open libraries are listed in the difference report.

   If the **Update Design** option button is selected, the difference report is generated and the design is updated to correspond to the open libraries.

   The *Design Verification* dialog does not update the VerifyDate attributes. The VerifyDate time stamp marks when the library components, not the design components, have been verified.

5. Type the desired report file name in the box or click the **Report File** button to navigate to and select a report file name. The selected report file name is displayed.

6. Click **Verify** to generate the difference report.

   If you selected the **View Report** check box, the difference report is displayed on the screen.

Updating the Design

Design verification not only generates a difference report; it also provides the option of updating your design. In three easy steps you could import your corporate database, update a P-CAD library based on changes to that database and update the design you are working on to instantly reflect those database changes.

By selecting the **Update Design** option button, you can choose to update the design's component attributes with values from one open library. Two options are available in the *Design Verification* dialog to allow you to select which open library is used to update the component attributes:

- **Using selected libraries**: If the Using selected libraries option button is selected, the design verifier uses the selections made in the Open Libraries list to determine the library used to update the component attributes.

  If more than one library is selected then the first occurrence of the component found is used for the design update. The libraries are searched in order from top to bottom of the list. Click the **Library Setup** button to add, delete, or rearrange the libraries in the Open Libraries list.

- **Using matching libraries**: If the Using matching libraries option button is selected, then the check boxes in the Open Libraries list do not apply. Components in the design with the ComponentLibrary attribute matching the name of one of the open libraries are updated.

These options are available only if the **Update Design** option button is selected.

Only selected attributes in the Library Properties/Attributes list are updated in the design from an open library. Read-only attributes, marked by a gray colored check box, cannot be updated.

CHAPTER 11

Reports

With P-CAD Library Executive there are multiple reporting options and enhancements. These enhancements span the P-CAD products: PCB, Schematic and Library Executive.

From P-CAD Library Executive, two report options are available: the basic source component reporting feature and the advanced library publisher. From PCB and Schematic, the Bill of Materials has been enhanced, with the purchase of Library Executive, to include imported component attributes.

With the Library Executive report command, you can generate a report file containing details of the components in the selected source. An ASCII or comma-delimited report can be generated of the following sources:

- An external source file
- A P-CAD library or library set
- A query or cross link result

The Library Publisher command of P-CAD Library Executive generates a detailed report of components, patterns and symbols in a P-CAD library. The report is output to Microsoft® Word 97™ and can contain a variety of reporting options, including pictures.

P-CAD Library Executive also provides enhanced reporting capabilities for PCB and Schematic. When generating a Bill of Materials, you can select and organize the component attributes to include in the report output. These attributes can even be imported from an external source file.

This chapter describes the reporting capabilities available with P-CAD Library Executive.

Reports in P-CAD Library Executive

From Library Executive, you can generate a report file from the following sources:

- an external library source file
- a query result
- A cross link result

- Another P-CAD library or library set

Depending on the desired source, the **Report** command is chosen in distinct ways. Refer to *Accessing the Report Command (page 146)* for details.

The basic setup of the report dialog is identical, independent of the source. Customizing this common report dialog is covered in *Generating a Report (page 146)*.

Accessing the Report Command

Choose the **Report** command to generate a report file on any Library Executive source. The **Report** command can be accessed from the Source Browser, the **File** menu and the **Table** menu. The location from which you choose the Report command defines the components listed in the report output. For example, choosing **Report** from a Query Result table generates a report of the components listed in that Query Result.

This section describes how and where to choose the Report command to get the output you want.

From the Source Browser

Choose the **Report** command from the shortcut menu after selecting a library, library set, query result, cross link result, or external source file in the Source Browser tree.

The components in the source you select are listed in the report output.

From the Table Menu

Choose the **Report** command from the **Table** menu when in the Viewer to generate reports on imported source files, query results, cross link results and P-CAD libraries.

From the File Menu

Choose the **Report** command from the **File** menu to generate a report on the imported source file.

Generating a Report

To generate a report on a query result, a cross link result, an imported file, or a P-CAD library:

1. Choose the **Report** command to open the *Report* dialog.

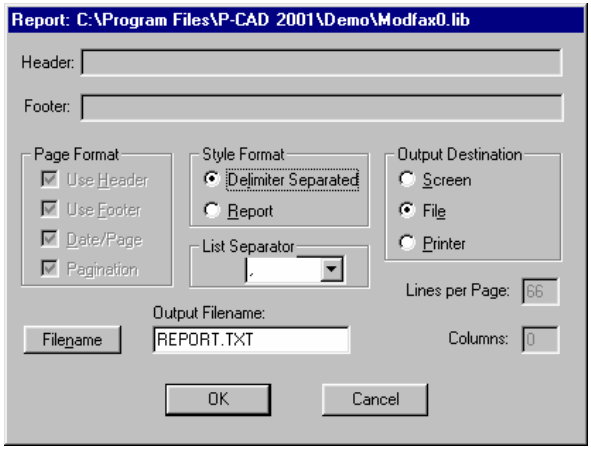
   When reporting on an imported file, choose **File » Report**.

   Other reports can be generated by selecting the item (query result, cross link result, imported file, or P-CAD library) in the Source Browser tree and choosing **Report** from the shortcut menu.

   A report on the current Viewer contents can be made by choosing the **Table Report** command.

   > The Report command reports on the current contents of Library Executive memory, as displayed in the Viewer. Report does not report on the original source file. If changes have been made to a library, choose **Reload** to refresh the library components in Library Executive memory before generating the report.

   The *Report* dialog displays.



2. Select the report format in the Style Format frame:
   - **Delimiter Separated**: A delimiter separated report output with formatting specifications.
   - **Report**: Generates an ASCII report file. The ASCII report is more readable and formatting specifications can be applied. The Page Format frame becomes available and you can designate a Header, Footer, Lines per Page and Columns.
3. Select the desired file **Output Destination**:
   - **File**: Define the **Output Filename** by typing the name in the box or clicking the **Filename** button to navigate to and select the desired file.
   - **Screen**: Puts the report out to the screen.
   - **Printer**: The report output is sent directly to the current printer.
4. Click **Generate**.

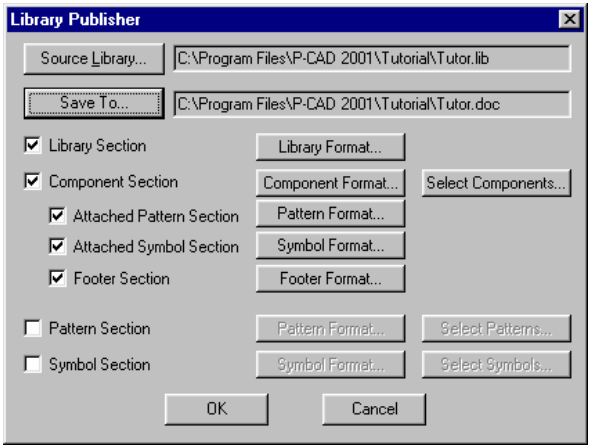
Publishing a P-CAD Library

Library Publisher is a powerful Library Executive feature that generates complete, customizable reports directly in Microsoft® Word™. These reports can include pictures of the patterns and symbols in the library.

Microsoft® Word 97™ is required for the *Library Publisher* utility.

Setting up the Library Publisher Dialog

Choose **Library » Publisher** to open the *Library Publisher* dialog.



This dialog is the point of access for the content and format options of the Library Publisher.

To set up the *Library Publisher* dialog, follow these steps:

1. Click the **Source Library** button to navigate to and select the directory and filename of the P-CAD library you want to publish.

2. Click the **Save To** button to navigate to and select the directory and filename of the output file.

3. Select the **Library Section** check box if you want to include a report on the selected P-CAD library in the publisher output.

   When this check box is selected, the **Library Format** button becomes available. Click the **Library Format** button to select the contents of this section, including file name, size, component name list, etc. Refer to *Format Options (page 149)* for additional details.

4. Select the **Component Section** check box if you want to include a report on the components in the selected P-CAD library in the publisher output.

   When this option is selected, the **Component Format** button becomes available. Click the **Component Format** button to select the contents of this section, including component name, pin table, attribute table, etc. Refer to *Format Options (page 149*) for additional details.

   The **Select Components** button becomes available. Click the **Select Components** button to choose which components from the selected library will be included in the report output. Refer to *Selecting Components, Patterns and Symbols (page 154)* for additional details.

5. If the **Component Section** check box is selected, several additional sections are available, including Attached Pattern Section, Attached Symbol Section and Footer Section check boxes.

   If you select the **Attached Pattern Section**, **Attached Symbol Section**, or **Footer Section** check boxes, the appropriate section is added to your publisher output. For instance, selecting the **Attached Pattern Section** check box includes a report of the pattern immediately after the report on the component to which it is attached.

The footer section is global, so if this check box is selected, specified attributes are included in the footer of all pages in the publisher output.

If any of these sections are included, their corresponding format button is available. Click the item's **format** button to select the formatting options for that section. Refer to *Format Options (page 149)* for additional details on the *Pattern Format*, *Symbol Format* and *Footer Format* dialogs.

6. Select the **Pattern Section** check box if you want to include a report on the patterns in the selected P-CAD library in the publisher output.

   When selected, the **Select Patterns** button becomes available, which allows you to select the patterns included in this section. All selected patterns from the library are included, whether or not they are attached to any library components. Refer to *Selecting Components, Patterns and Symbols (page 154)* for additional details.

   The **Pattern Format** button is also available. Click the **Pattern Format** button to select the contents of this section, including pattern name, attribute table, a picture of the pattern, etc. Refer to *Format Options (page 149)* for additional details.

7. Select the **Symbol Section** check box if you want to include a report on the symbols in the selected P-CAD library in the publisher output.

   When this check box is selected, the **Select Symbols** button becomes available, allowing you to select the symbols included in this section. All selected symbols from the library are included, whether or not they are attached to any library components. Refer to *Format Options (page 149)* for additional details.

   The **Symbol Format** button is also available. Click the **Symbol Format** button to select the contents of this section, including symbol name, attribute table, a picture of the symbol, etc. Refer to *Format Options (page 149)* for additional details.

8. When all options have been specified as desired, click **OK**. The published document includes a report on the specified library with the selected sections, items and formatting options.

   Microsoft® Word 97™ works in the background to generate a published library document of the specified name. Open the document in Word™ to view, to make changes, or to print. To open the document, you can select the file name in the recently used file list on Word's **File** menu.

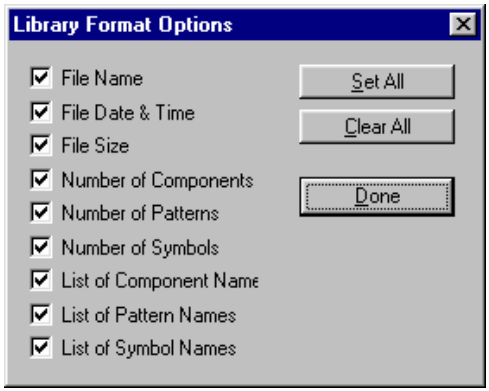
Format Options

If you choose to include a section in your published output by selecting its check box, you can modify the contents of the section by clicking its corresponding **format** button. The format options available for each output section are detailed in *Library Format Options (page 150)*.

Library Publisher uses Word's `Normal.dot` template as a basis for the published output. To select the font type, font size and page layout for the published library, modify your `Normal.dot` template accordingly.

Library Format Options

When including the Library Section, click the **Library Format** button to open the *Library Format Options* dialog.



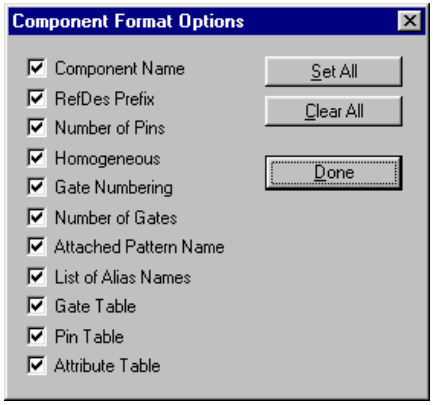
This dialog allows you to specify the contents of the library section by selecting each option's check box.

When all desired library format options are selected, click **Done**. The selected options are saved and you are returned to the *Library Publisher* dialog.

For details about the Library Format options, refer to *Library Commands (page 211)*.

Component Format Options

When including the Component Section, click the **Component Format** button to open the *Component Format Options* dialog.



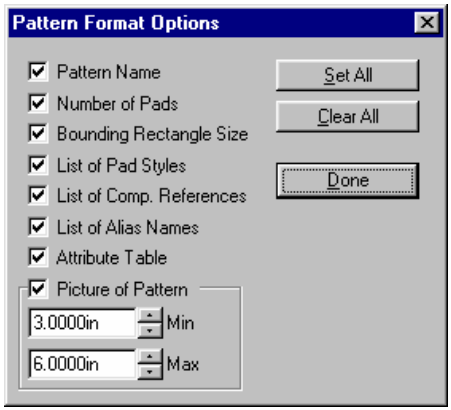
This dialog allows you to specify the contents of the component section by selecting each option's check box.

When all desired component format options are selected, click **Done**. The selected options are saved and you are returned to the *Library Publisher* dialog.

For details about the Component Format options, refer to *Library Commands (page 211)*.

Pattern Format Options

When including a Pattern Section in your published library, click the **Pattern Format** button to open the *Pattern Format Options* dialog.



This dialog allows you to specify the contents of the pattern section by selecting each option's check box.

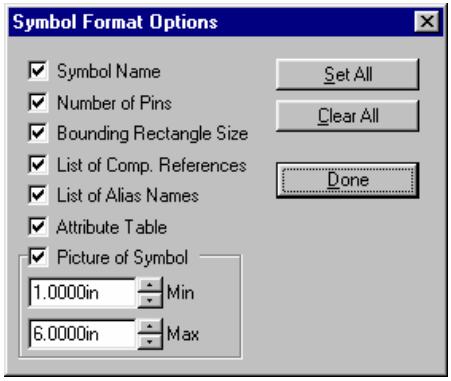
The pattern section check box can be selected in two locations. If you select the **Attached Pattern Section** check box within the Component Section, the pattern section will directly follow the component section of the component to which it is attached. If you select the **Pattern Section** check box outside the Component Section, the pattern section will follow after the component section for all components is completed. In this case, the pattern section will contain all of the selected patterns in the library consecutively, independent of their associated component.

Each of these two pattern sections can be formatted separately. Click the **Pattern Format** button to open the *Pattern Format Options* dialog.

When all desired pattern format options are selected, click **Done**. The selected options are saved and you are returned to the *Library Publisher* dialog. For details about the Pattern Format options, refer to *Library Commands (page 211)*.

Symbol Format Options

When including a Symbol Section in your published library, click the **Symbol Format** button to open the *Symbol Format Options* dialog.



This dialog allows you to specify the contents of the symbol section by selecting each option's check box.

The symbol section check box can be selected in two locations. If you select the **Attached Symbol Section** check box within the Component Section, the symbol section will directly follow the component section (and attached pattern section, if selected) of the component to which it is attached. If you select the **Symbol Section** check box outside the Component Section, the symbol section will follow after the component section (and pattern section, if selected) for all components is completed. In this case, the symbol section will contain all of the selected symbols in the library consecutively, independent of their associated component.

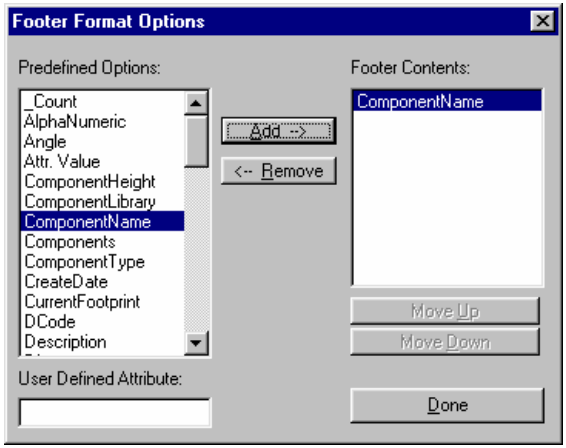
Each of these two symbol sections can be formatted separately. Click the **Symbol Format** button to open the *Symbol Format Options* dialog.

When all desired symbol format options are selected, click **Done**. The selected options are saved and you are returned to the *Library Publisher* dialog.

For details about the Symbol Format options, refer to *Library Commands (page 211)*.

Footer Format Options

When including the Footer Section, click the **Footer Format** button to open the *Footer Format Options* dialog.



This dialog allows you to specify the footer contents displayed on each page of the component section by selecting the desired properties or attributes.

To select the attributes you want displayed on the bottom of the component section's pages:

1. Select a component or attribute from the list of Predefined Options.
2. Click **Add** to move the selection to the Footer Contents list.

   To include an attribute that is not predefined, type the attribute name into the **User Defined Attribute** box. Click **Add**. The attribute will appear in the Footer Contents box. The attribute value for the particular component will be found in the library and listed in the footer section. You can also double-click the attribute to add it.

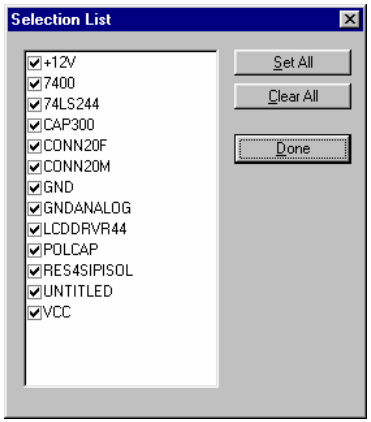
   To remove an attribute from the Footer Contents box, select the attribute and click **Remove**. You can also double-click the attribute to remove it.

   Text can be included in the footer. For example, the words "Components Designed By:" can become a footer for your published document. To include text, type the text into the User Defined Attribute box. Click **Add**. Since the name is not recognized as an attribute, an attribute value will not appear in the footer.
3. The footer contents are the attributes and their values listed in the order that they appear in the Footer Contents box. These attributes appear in order on the bottom of the pages in the component section from left to right.

   To organize the Footer Contents box, select the property or attribute you want to move, click **Move Up** or **Move Down**. These commands rearrange or modify the contents of this box by moving the selected attribute higher or lower in the list.
4. Click **Done**. The properties and attributes in the Footer Contents box are used to create a footer for the published library document.

Selecting Components, Patterns and Symbols

When a Component, Pattern, or Symbol section is included in your published library, you can select the items you want to include. The **Select Components**, **Select Patterns** and **Select Symbols** buttons open their corresponding component, pattern, or symbol selection list dialogs. A sample selection list dialog appears as shown in the following dialog.



A list of all components, patterns, or symbols in the selected library is displayed. If an item is selected, the corresponding component, pattern, or symbol is included in the published output.

1. To select all check boxes, click the **Select All** button.

   To clear all check boxes, click the **Clear All** button.

   Click an item to toggle its check box between selected and cleared.

2. When the desired items are selected, click **Done**. The selected items will be published in their component, pattern, or symbol section.

The selection dialog is not available for attached patterns and symbols in the component section. If **Attached Pattern Section** or **Attached Symbol Section** is selected, the patterns and symbols published are those attached to the selected component(s).

Generating the Report Output

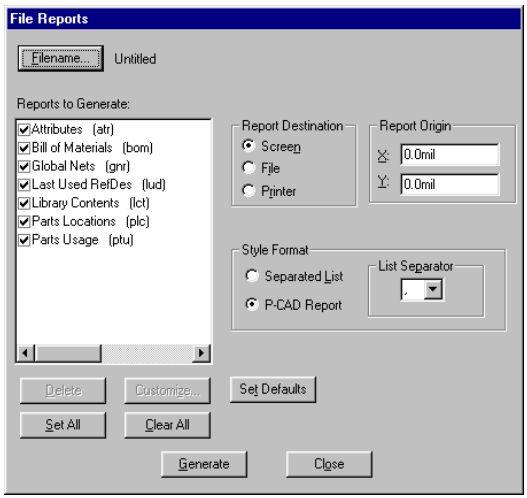
You can generate the published library by clicking **OK** in the *Library Publisher* dialog after having specified the Library Publisher output: filenames, sections, formatting and item selection

Microsoft® Word 97™ works in the background to generate a published library document of the specified name. Open the document in Word™ to view, make changes, or to print. To open the document, you can select the file name in the recently used file list on Word's **File** menu.

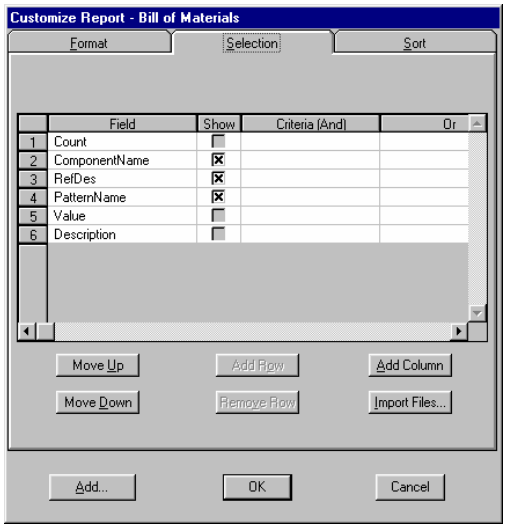
> If you run Word™ in the foreground after the Library Publisher has been started, be careful not to quit the Word™ application. Exiting Word™ before Library Publisher has finished will halt the publishing process.

Bill of Materials from PCB and Schematic

The *File Reports* dialogs of P-CAD PCB and P-CAD Schematic provide enhanced capabilities for the Bill of Materials report when P-CAD Library Executive is installed. The BOM report may be customized to include selected component attributes, including component attributes from an external source file. The *File Reports* dialog provides the initial entry to the Custom Report feature and appears.



Select the **Bill of Materials** report and click the **Customize** button to open the *Customize Reports* dialog.



The Bill of Materials typically lists the component type, pattern, component value (if any), number of components with matching characteristics and the reference designator (RefDes) assigned to each component.

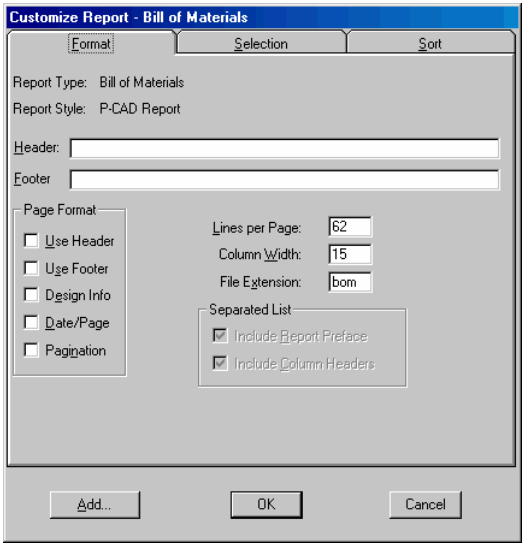
In the Selection tab, you can select and organize the component attributes you want to include in the report, along with setting selection criteria for all fields except Count. All attributes in the design are available to be displayed in the report by selecting the **Show** check box for the specific fields. The **Selection** tab also provides access to the Import Files feature, which allows you to import component attributes from a non-P-CAD source.

See *Setting up a Custom BOM Report (page 156)*, to learn how to customize your Bill of Materials report.

For information about importing component data from an external source to include in your Bill of Materials report, refer to *Using an External Source File (page 160)*.

Setting up a Custom BOM Report

When you click the **Customize** button on the *File Reports* dialog, the *Customize Report* dialog is opened with the **Format** tab selected.



Format Tab

In the **Format** page you can select the following options:

Header: Enter the name of the heading for the report if you have chosen the P-CAD Report Style Format.

Footer: Enter the name of the desired footer if the Style Format of P-CAD Report has been selected.

Page Format: The Page Format frame applies to reports whose Style Format is the P-CAD Report. Choose the desired options by selecting the appropriate check boxes for:

- Use **Header** to include the information you specified in the Header box.
- Use **Footer** to include the footer information entered in the Footer box.
- **Design Info** includes the information you entered in the **File Design Info** command and dialog.
- **Date/Page** includes the current date and the page number.
- **Pagination** allows you to create your own pagination (lines per page).

Lines per Page: Enter the number of lines you want printed on each page.

Column Width: Defines the number of characters, up to 2000, used to display an attribute.

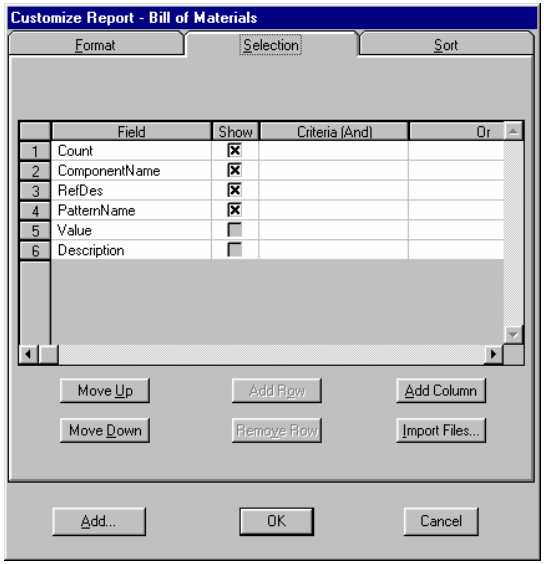
File Extension: The File Extension edit box displays the default extension for the selected report. You can also enter a new extension if desired.

Separated List: The Separated List frame is available when the Separated List report style has been selected. In this frame you can **Include Report Preface** and/or **Include Column Headers** by selecting the appropriate check box.

Selection Tab

The Selection tab lists the report fields, allows you to choose which are displayed in the report and provides the ability to define selection criteria.

The Customize Report Selection tab appears.



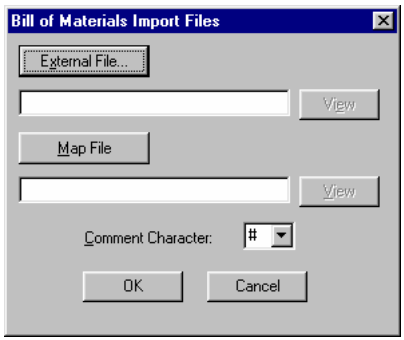
The columns in the Selection spreadsheet are:

- **Field**: The list of fields specific to the selected report.
- **Show**: The **Show** option turns on the display of selected fields in the report. Cleared fields do not appear in the report output.
- **Criteria (And)**: Contains the selection criteria used to filter the report data.
- **Or**: Additional selection criteria can be entered here, if necessary.

Operators used in the Criteria (And) and Or columns are identical to those used in the *Query* dialog.

The columns in the spreadsheet can be resized by moving the column separation lines right or left. You can append another **Or** column by clicking the **Add Column** button. Fields can be repositioned in the list by using the **Up** and **Down** buttons, which determines the order in which they are output to the report.

If you want to import information from a non-P-CAD source, click the **Import Files** button to open the *Import Files* dialog.



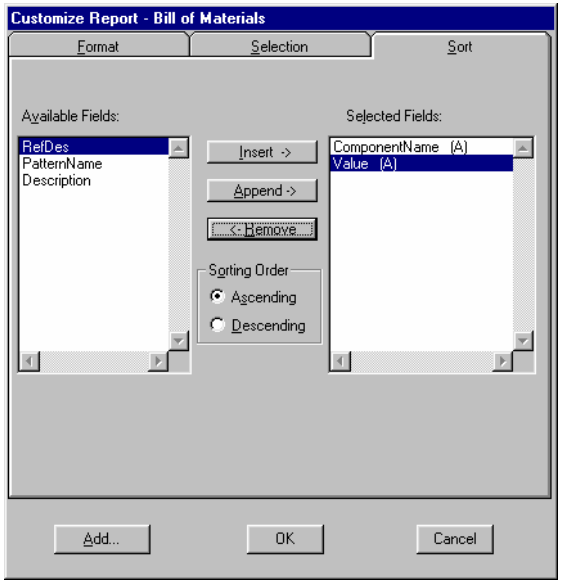
The **External File**, **Map File**, **View** and **Import** buttons are used to add component attributes to the Bill of Materials report. A specific character can be designated as the start of a comment line from the options in the list or by entering the desired character in the box.

For additional information about this option, refer to *Using an External Source File (page 160)*.

Sort Tab

In the Sort tab you can select the field(s) used to sort the report output and choose a sort order.

The **Customize Report Sort** tab is shown in the following figure.

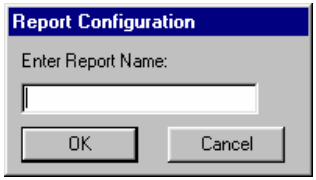


Options available on the **Sort** tab are:

- **Available Fields**: The list of fields in the selected report.
- **Selected Fields**: The fields used to sort the data, the order in which the sort is applied and the field's sorting order, (A) for Ascending or (D) for descending, is displayed in the Selected Fields list.
- **Insert**: Click the **Insert** button to move a selected field from the Available Fields list and insert it above the selected field in the Selected Fields list.
- **Append**: Click the **Append** button to move the selected field to the bottom of the Selected Fields list.
- **Remove**: To move fields from the Selected Fields list back to the Available Fields list, select the field and click the **Remove** button.
- **Sorting Order**: The Sorting Order is set by selecting the desired option button to sort in Ascending or Descending order before moving a field to the Selected Fields list.

Adding a Custom Report

Once the desired options are selected, click **Add** to open the *Report Configuration* dialog.



Enter the **Report Name** and click the **OK** button. You are returned to the *File Reports* dialog where the new report is listed in the *Reports to Generate* area. You can now select the reports and generate output by clicking the **Generate** button.

Using an External Source File

You can import component attributes from a non-P-CAD source to be included in the Bill of Materials report. For example, you may have a company component database that includes additional information about the components not contained in your P-CAD libraries. These imported attributes can be included in your Bill of Materials report directly from the **File » Report** command in both P-CAD PCB and Schematic. The imported file must be in a simple separated list format.

If your external source file contains the ComponentName field, refer to *Importing a Source File with a ComponentName (page 160)*. This field is essential for P-CAD libraries since it is used as the basis for both search and placement of all library components in all P-CAD products. With the ComponentName in your external file, component attributes in the file are easily associated with their appropriate component for the Bill of Materials report.

Many external component databases are organized without a unique component name. For example, the attribute Part Number may be the index that labels and identifies individual components. If your external source file does not contain the ComponentName field, an additional file, called a map file, is needed to match the imported attributes with their associated components. Importing attributes from an external file without the ComponentName is discussed in *Linking with a Map File (page 161)*.

Importing a Source File with a ComponentName

If the imported source file contains the ComponentName field, then the attributes are automatically linked with their corresponding component in your P-CAD library. In this case, follow these steps to generate a Bill of Materials with imported component attributes:

1. In the *Selection* dialog, click the **External File** button.
2. Navigate to and select the external source file.
3. Click **OK**. The filename is displayed in the box.
4. Click **View** if you wish to verify the file's contents. The file is displayed on the screen.
5. To import the file, click **Import**. The attributes contained within the external file appear in the Selected fields will be included in the report box.
6. Select and organize the attributes for the report contents.

7. Click **OK** to return to the *File Reports* dialog.

Linking with a Map File

If the source file does not contain the P-CAD primary key, ComponentName, then a Map file can also be imported to map the component index identifier from the source file to its associated library component. This map file is a simple separated list file that contains the ComponentName in one column and the identifying attribute in the remaining column(s), which are used in the external source, for example Part Number.

In this case, you must load the map file. The attributes are automatically linked with their corresponding component in your P-CAD design.

Follow the steps in *Importing a Source File with a ComponentName (page 160)* to generate a Bill of Materials with imported component attributes. After importing the external file, click the **Map File** button and choose the separated list map file. Click the **View** and **Import** buttons to view and import the map file, respectively.

When imported, the map file is used to map the design's component names with their corresponding index identifier. The now-recognized index identifier in the external file automatically attaches the imported attributes to their appropriate P-CAD components.

CHAPTER 12

Component Placement

With P-CAD Library Executive, you can place components directly into a PCB or Schematic design from several sources. These sources include a Query Result, a Cross Link Result, an imported library source file and any P-CAD library.

The component placement feature of P-CAD Library Executive is a perfect companion to Query. Imagine the advantage you will achieve by being able to search for a component by various criteria, including Cost. And, after finding the ideal component, you can convenient place that component directly into your design.

Component Placement Requirements

To place a component in a design, the component must be in a P-CAD library. The component must also be complete, including all of the required graphical and electrical information.

If the component is chosen from the Source Browser, the ComponentName and ComponentLibrary fields are known. A complete component with the required electrical and graphical information may be placed in a design. If the component has an attached pattern, it includes the required graphical information to be placed in P-CAD PCB. If the component has an attached symbol, it may be placed in P-CAD Schematic.

If the component is selected in the Viewer, the Viewer must include the ComponentName and ComponentLibrary fields. If the component in the specified library has complete electrical and graphical information, Library Executive quickly accesses the component and places it in the current design.

Accessing Component Placement

Choose the Place command to place a component to a P-CAD PCB or Schematic design. The Plac**e** command can be accessed from the Source Browser and the **Row** menu. The item selected when you choose the **Place** command defines the component you intend to place in the design. For example, choosing **Place » PCB** from a Query Result table while a component row is selected, opens the *Place Component* dialog in the current PCB design with the component selected.

This section describes how and where to choose the **Place** command to get the desired results.

From the Source Browser

From the Source Browser you can open a P-CAD library directly and select a component to place into a design.

Select a component contained within a P-CAD library in the Source Browser tree and choose **Place** from the shortcut menu. Drag the mouse to the right to select the desired application name, PCB or Schematic.

Refer to *Placing Components in P-CAD PCB (page 164)* and *Placing Parts in P-CAD Schematic (page 164)* for additional details.

From the Viewer

You can choose to **Place** a component from a Query Result Viewer or a Cross Link Result Viewer. You can also choose to **Place** a component from a file Viewer, while viewing either a P-CAD library or an imported library from an external source.

To place a component from the Viewer, the Viewer must include the ComponentName and ComponentLibrary fields. Select the row on the table corresponding to the desired component by clicking the **row number** to its left. While the row is selected, choose the **Place** command from the *Row* menu. Drag the mouse to the right to select the desired application name, PCB or Schematic.

Placing Components in P-CAD PCB

To place a component in a PCB design, P-CAD PCB must be running with the design in the workspace. The *Place Component* dialog appears with the component selected. Refer to your PCB documentation or online help for details on placing a component.

Component Placement from P-CAD PCB

Library Executive assists with component placement from P-CAD PCB. If you search for a component from within the PCB application by running an embedded query, you can choose to place a component from the Query Result table.

To access the embedded Query, a **Query** button appears on the *Place Component* dialog. The **Library** menu of P-CAD PCB also contains the **Query** command.

The components satisfying the search criteria are displayed in a Query Result Viewer. Refer to *Component Placement Requirements (page 163)* for details about the requirements for placing a component from a Viewer spreadsheet.

Placing Parts in P-CAD Schematic

To place a component in a Schematic design, P-CAD Schematic must be running with the design in the workspace. The *Place Part* dialog appears with the component's parts selected. Refer to your Schematic documentation or online help for details on placing a part.

Part Placement from P-CAD Schematic

Library Executive assists with part placement from P-CAD Schematic. If you search for a component from within the Schematic application by running an embedded query, you can choose to place a component's parts from the Query Result table.

To access the embedded Query, a **Query** button appears on the *Place Part* dialog. The **Library** menu of P-CAD Schematic also contains the **Query** command.

The components satisfying the search criteria are displayed in a Query Result Viewer. Refer to *Component Placement Requirements (page 163)* for details about the requirements for placing a component from a Viewer spreadsheet.

CHAPTER 13

Extended Library Features

This chapter covers using advanced library features for importing and exporting and converting libraries form other sources, such as MRP and Tango. It contains a series of simple lessons that will teach you how to work with the extended Library Executive features, including:

- Importing and Updating a Library from an external source file.
- Importing an external source file without the ComponentName field.
- Using Query to selectively update a library.
- Using Query to find and place a component.
- Reporting on component library updates.
- Verifying your design against a library.
- Verifying your libraries against an external source file.
- Creating a component from the imported file, with help from the Pattern and Symbol Editors.
- Converting libraries from other sources, e.g. Tango.

Getting Setup for Learning

The following information is needed to perform the lessons you will learn in these lessons.

The Sample Files

Four sample files are used in the following lessons. These files include two external source files, a map file and one P-CAD library.

All of the sample files can be found in the `Tutorial` folder in the P-CAD installation directory. Make a backup of the original sample files before beginning these lessons. Modifications made to the source file, such as during field mapping, are saved automatically.

The external source files contain attributes that are associated with many of the components included in the sample P-CAD library. For example, one such attribute is Quantity. The values for this and other attributes have changed recently in the company database, or MRP, to reflect the current inventory. For lessons 1 and 2, you would like to import this data from the company database to update your library. The updated library is used for many of the remaining lessons in this chapter.

External Source Files

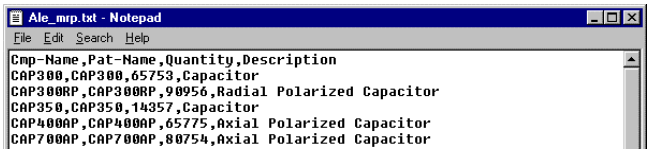
When importing data from a non-P-CAD source, such as an MRP database, the first step is to generate a simple comma-delimited file containing the database information. In these example files, this step has already been completed for you.

In fact, two comma-delimited files are provided, with the essential difference being whether or not the file contains the primary key, ComponentName. The first lesson guides you through the steps of importing the MRP data with the primary key using the Map Fields command; lesson 2 uses the Cross Link command to import the MRP file without the ComponentName field.

Refer to the *Importing Data from an External Source (page 121*) for additional information on how to convert your component database information to this simple format.

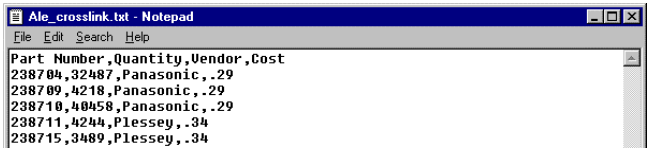
- `Ale_mrp.txt`

  This file is a comma-delimited external source file that contains the primary key, ComponentName. The actual field name, Cmp-Name, must be mapped to the field name recognized by P-CAD using the **Map Fields** command.



- `Ale_crosslink.txt`

  This file is a comma-delimited external source file that, in contrast to the file above, does not contain the ComponentName field. The Part Number field from this company database will be used to link this file to the sample library, `Tutor.lib`.



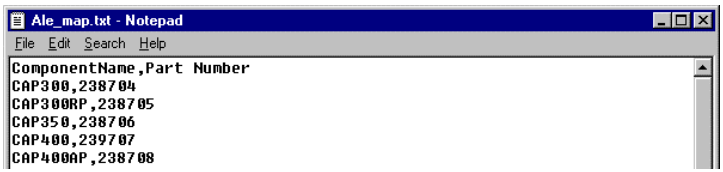
Map File

In order to import an external source file, such as `Ale_crosslink.txt`, that does not contain the ComponentName, an additional step is required to set up the automated import process. A simple, comma-delimited file will be used to initially associate the library ComponentName with a unique MRP database index, in this case the Part Number field. This additional file is called a map file.

The map file will only be used once, to insert the database index, Part Number, into the component library. The Part Number can then be used alone as the import identifier.

- `Ale_map.txt`

  This map file is a simple comma-delimited text file that contains two columns: the primary key ComponentName and the database index identifier Part Number. This map file is used to enter the Part Number attribute into the appropriate library component's attributes.

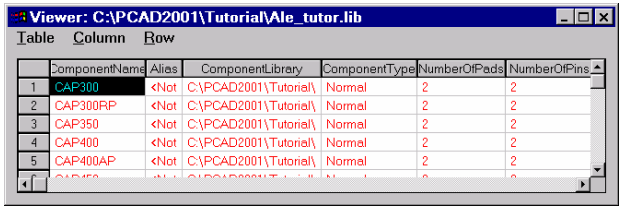


P-CAD Library File

The remaining sample file is a P-CAD library. This library contains many of the same components found within the sample MRP databases. These library components are complete and integrated, with an attached pattern, attached symbol, and electrical information.

- `Ale_tutor.lib`

  This file is a P-CAD library containing the components referenced in the external source file.



	ComponentName	Alias	ComponentLibrary	ComponentType	NumberOfPads	NumberOfPins
1	CAP300	<Not	C:\PCAD2001\Tutorial\	Normal	2	2
2	CAP300RP	<Not	C:\PCAD2001\Tutorial\	Normal	2	2
3	CAP350	<Not	C:\PCAD2001\Tutorial\	Normal	2	2
4	CAP400	<Not	C:\PCAD2001\Tutorial\	Normal	2	2
5	CAP400AP	<Not	C:\PCAD2001\Tutorial\	Normal	2	2

Lesson 1: Importing and Updating a Library from an MRP File

This section illustrates the process of importing component data from a non-P-CAD source. It shows the initial comma-delimited source file, importing and field mapping. The imported data will then be used to update the library, `Ale_tutor.lib`.

In this lesson, you will be importing the sample comma-delimited file, `Ale_mrp.txt`. This sample file contains the primary key, ComponentName. It will be used to introduce the Map Fields function of Library Executive.

The MRP file `Ale_mrp.txt` is an external source file that contains the ComponentName field. This field is essential for P-CAD libraries since it is used as the basis for both search and placement of all library components in all P-CAD products. With the ComponentName field in your MRP file, it is only a few quick, easy steps to update a P-CAD library.

Many MRP databases are organized without a unique component name. For example, the attribute Part Number may be the index that labels and identifies individual components. If your external

source file does not contain the ComponentName, an additional process is needed to initially set up and automate the import process.

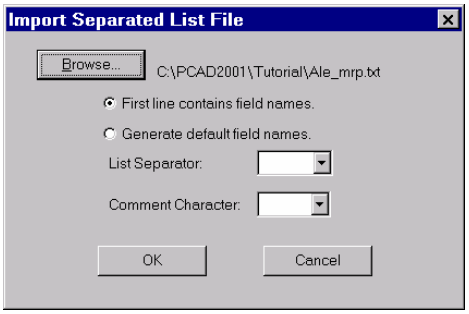
Using the sample files, follow these steps to learn how to import and update a library from an MRP file:

Step 1: Import an MRP with a ComponentName

The first step in importing any external source file is to put the data into a simple comma-delimited file format. The source file `Ale_mrp.txt` is already in the appropriate file format and can be imported directly into Library Executive. The *Importing Data from an External Source (page 121)* contains additional information on how to convert your component database information to this simple format.

To import the sample MRP file, follow these steps:

1. Choose **File » Import** to open the *Import Separated List File* dialog.



2. Click the **Browse** button. Navigate to the `Tutorial` folder in the P-CAD installation directory and select the comma-delimited source file `Ale_mrp.txt`.
3. Click the **First line contains field names** button since the first line of the sample file contains the field names that correspond to the file contents.

   If the first line of the external source file did not contain field names, **the Generate default field names** button should be selected.
4. Enter or select a character in the **List Separator** box.
5. Enter or select a character in the **Comment Character** box.
6. Click **OK**. The external source file Viewer appears.

Viewer: C:\PCAD2001\Tutorial\Ale_mrp.txt

Table Column Row

	Cmp-Name	Pat-Name	Quantity	Description
1	CAP300	CAP300	65753	Capacitor
2	CAP300R	CAP300	90956	Radial Polar
3	CAP350	CAP350	14357	Capacitor
4	CAP400A	CAP400	65775	Axial Polariz
5	CAP700A	CAP700	80754	Axial Polariz
6	CAP800A	CAP800	4362	Axial Polariz
7	CAP1000	CAP100	4759	Axial Polariz
8	CAP1200	CAP120	321	Axial Polariz
9	POLCAP	CAP100	9874	Polarized Ca
10	CAP400	CAP400	8903	Capacitor
11	RES500	RES500	22357	Resistor
12	RES600	RES600	64532	Resistor
13	RES700	RES700	7897	Resistor
14	RES1300	RES130	26659	Resistor
15	RES2200	RES220	486	Resistor
16	RES6SIP	SIP6	80654	Five Bussed
17	RES6SIPI	SIP6	579806	Three Isolat
18	RES6SIPI	SIP6	1956	Three Isolat
19	RES8SIP	SIP8	65967	Twelve Dual
20	RES8SIPI	SIP8	12	Four Isolate
21	RES10SI	SIP10	35437	Nine Bussed
22	RES10SI	SIP10	24723	Sixteen Dual
23	RES14DI	DIP14	67887	Seven Isolat

The field names are the column headers with the field values filling in the table contents.

Step 2: Mapping Fields

Now that you have the external source file imported, you can use the Map Fields command to map the field names to those recognized by P-CAD. You will be mapping the Cmp-Name and Pat-Name fields to their P-CAD counterparts ComponentName and PatternName.

The field mapping is remembered. So, if you are regularly importing the same external source file, this mapping step need only be completed once. Subsequent imports of the same file will be automatically mapped.

To use the Map Fields command on the imported source file:

1. From the **Table** menu of the file Viewer, choose **Map Fields** to open the *Map Fields* dialog.

Map Fields: C:\PCAD2001\Tutorial\Ale_mrp.txt

	Original Field Name	New Field Name	Data Type	Mapping
1	Cmp-Name	Cmp-Name	String	User-define
2	Pat-Name	Pat-Name	String	User-define
3	Quantity	Quantity	String	User-define
4	Description	Description	String	Predefined

Add Row Delete Row Map

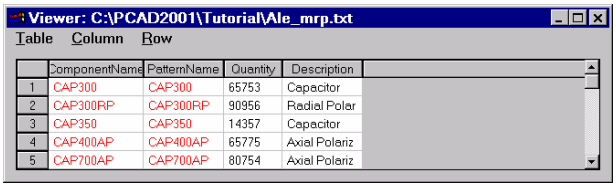
2. Click the **Cmp-Name** entry in the **New Field Names** column to display the list of component names.
3. Click the down arrow to display the list of P-CAD recognized field names. Select **ComponentName**. This list disappears and ComponentName is displayed in the column.

Press ENTER. The field is recognized by P-CAD when the Mapping column changes from user-defined to Predefined.

4. Repeat steps 2 and 3 to map the Pat-Name field to the P-CAD-recognized field, PatternName.

5. Click the **Map** button. You are reminded that the original imported file is mapped, any changes you have made to the file contents in the Viewer are lost during the mapping process.

   The file Viewer appears with the two fields mapped.



	ComponentName	PatternName	Quantity	Description
1	CAP300	CAP300	65753	Capacitor
2	CAP300RP	CAP300RP	90956	Radial Polar
3	CAP350	CAP350	14357	Capacitor
4	CAP400AP	CAP400AP	65775	Axial Polariz
5	CAP700AP	CAP700AP	80754	Axial Polariz

   All fields in the sample source file are now mapped to their P-CAD predefined values, if available. In this case, the predefined fields ComponentName and PatternName are read-only, as indicated by their red color on the file Viewer.

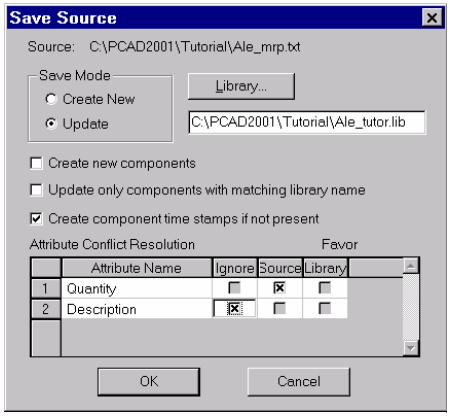
Step 3: Updating a P-CAD Library with MRP Data

The number of components available in your company inventory has changed since you created the component library. To get a quick turn-around on your design, you would like to search for some components based on their availability. You have now imported that new data into Library Executive. In this section, you will use the imported file to selectively update the Quantity attribute in a P-CAD library.

This section assumes that you have already set up the `Ale_tutor.lib` so it is available for Library Executive. If not, open the Source Browser and select the DEFAULT_LIBRARY_SET. Choose Add Library to add the sample library to the default set.

To update the Quantity attribute of the `Ale_tutor.lib` library, follow these steps:

1. From the Table command of the file Viewer, choose **Save to Library**. The *Save Source* dialog appears.



2. Select the **Update** option button.

3. Click the **Library** button, navigate to the `Tutorial` folder in the P-CAD installation directory and select the `Ale_tutor.lib` library. Click **Open**. The selected library name displays in the edit box.

   There are no new components in the imported MRP file. Leave the **Create new components** box cleared.

   Since the MRP file does not contain a ComponentLibrary attribute, leave the **Update only components with matching library name box** cleared.

4. Select the **Create component time stamps if not present** check box when adding or updating the **ModifyDate** attribute in the component library.

5. All attributes that will be updated in the library are listed in the Attribute Conflict Resolution section. In this section, you can specify how to handle discrepancies between the source and the target library. Since you wish to update only the Quantity attribute, click the **Source** box to its right. The MRP file data will be favored and the Quantity attribute will be updated.

6. Click the **Ignore** box to the right of all other attributes.

7. Click **OK**. The library has been updated to reflect the new Quantity attribute values.

Lesson 2: Importing an MRP File Without a ComponentName

This lesson also illustrates the process of importing data from a non-P-CAD source. The imported data will again be used to update the P-CAD library, `Ale_tutor.lib`, using ComponentName as the primary key.

The difference between the two lessons is the data contained within the MRP file. Lesson 2 imports a comma-delimited source file that does not contain the primary key, ComponentName. Instead, the data in this file is referenced by a unique Part Number.

In this lesson, you will be importing the sample comma-delimited file, `Ale_crosslink.txt`. This external source file does not contain a ComponentName field. Importing this file will be used to

introduce the Cross Linking function of Library Executive. For additional information about the Cross Link feature, see the *Importing Data from an External Source (page 121)*.

Because the imported file does not contain a ComponentName, a few additional steps are necessary in comparison to Lesson 1. The first step serves the purpose of inserting the MRP database index identifier, Part Number, as an attribute of its associated component. Since this attribute is saved with the library, this step need only be completed once.

The remaining steps are the import and update process for an external file without a ComponentName. These steps are completed every time you wish to update the P-CAD library components from this external source file.

This lesson assumes that you have already set up the `Ale_tutor.lib` so it is available for Library Executive. If not, open the Source Browser and select the DEFAULT_LIBRARY_SET. Choose **Add Library** to add the sample library to the default set.

You can follow along the next few sections, using the sample files to complete each of these steps yourself.

Step 1: Adding the Part Number Attribute to a Library

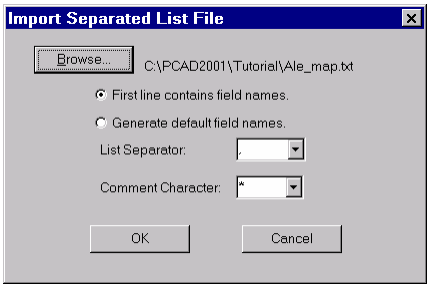
When importing an MRP file that does not contain the ComponentName field, some connection must be defined that associates a P-CAD library component with its MRP entry. This association can be completed in a variety of ways. The most automated technique is through the use of a map file.

In this section, you will be completing the miniature import/update process of adding the Part Number attribute to a P-CAD component library. This attribute is the unique index identifier of the MRP database. Once added, Library Executive will automatically know which component you are referring to when you import a component identified solely by a Part Number.

The map file is a simple, comma-delimited external source file. This file provides the cross reference between the ComponentName and Part Number fields. With the information in the map file, you will merge the external file's data with the associated component data in the P-CAD library.

To add the Part Number to the library:

1. Choose **File » Import** to open the *Import Separated List File* dialog.



2. Click the **Browse** button. Navigate to the `Tutorial` folder in the P-CAD installation directory and select the comma-delimited source file `Ale_map.txt`.

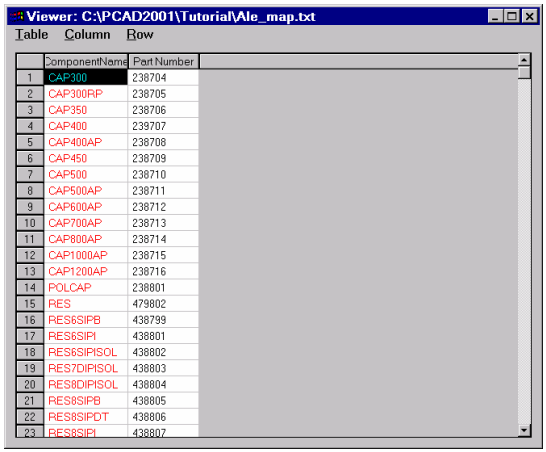
3. Click the **First line contains field names** button since the first line of the sample file contains the field names that correspond to the file contents.

   If the first line of the external source file did not contain field names, the **Generate default field names** button should be selected.

4. Enter or select a character in the **List Separator** box.

5. Enter or select a character in the **Comment Character** box.

6. Click **OK**. The external source file Viewer appears.



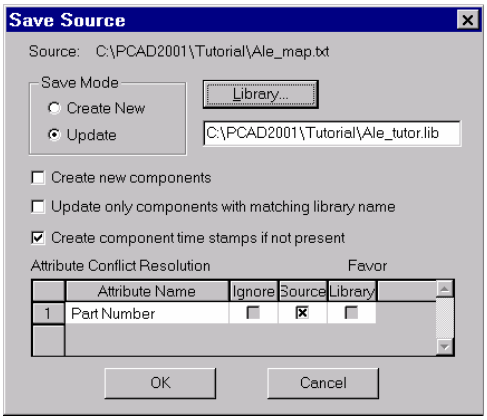
	ComponentName	Part Number
1	CAP300	238704
2	CAP300RP	238705
3	CAP350	238706
4	CAP400	239707
5	CAP400AP	238708
6	CAP450	238709
7	CAP500	238710
8	CAP500AP	238711
9	CAP600AP	238712
10	CAP700AP	238713
11	CAP800AP	238714
12	CAP1000AP	238715
13	CAP1200AP	238716
14	POLCAP	238801
15	RES	479802
16	RES6SIPB	438799
17	RES6SIPI	438801
18	RES6SIPISOL	438802
19	RES7DIPISOL	438803
20	RES8DIPISOL	438804
21	RES8SIPB	438805
22	RES8SIPDT	438806
23	RES8SIPI	438807

The field names are the column headers with the field values filling in the table contents.

Step 2: Updating the Library

The Viewer appears with the two column contents of the map file, ComponentName and Part Number. At present the components in the P-CAD sample library, `Ale_tutor.lib`, are only referenced by the ComponentName and the MRP database is only referenced by the Part Number. The following steps will link the Part Number to an associated ComponentName within the library.

1. Save the Part Number attribute to the library by choosing the **Save to Library** command from the Table menu. From the Table command of the file Viewer, choose **Save to Library**. The *Save Source* dialog appears.



2. Select the **Update** option button.

3. Click the **Library** button, navigate to the `Tutorial` folder in the P-CAD installation directory and select the `Ale_tutor.lib` library. Click Open. The selected library name displays in the edit box.

   There are no new components in the imported MRP file. Leave the **Create new components** box cleared.

   Since the MRP file does not contain a ComponentLibrary attribute, leave **the Update only components with matching library name box** cleared.

4. Select the **Create component time stamps if not present** check box when adding or updating the ModifyDate attribute in the component library.

5. All attributes that will be updated in the library are listed in the Attribute Conflict Resolution section. For Part Number, select **Source** and then click **OK** to apply the update.

   The Part Number and the ComponentName are now linked. This allows you to reference data by the ComponentName or the Part Number fields.

6. Click **OK**. The library has been updated to reflect the new PartNumber attribute value.

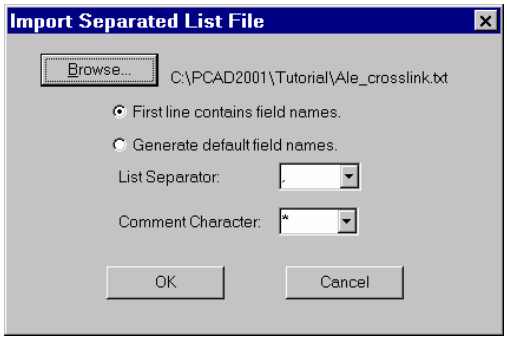
Step 3: Importing an MRP without a ComponentName

Once the Part Number has been added to the sample library, you never have to repeat the above step again. The library components are now associated with the Part Number index of the company database. From here on out, to import the MRP sample file without a ComponentName, you can start with the step you just completed.

The `Ale_crosslink.txt` file is a comma-delimited external source file. This sample file has been generated from a component database and includes the following attributes: Part Number, Vendor, Quantity, and Cost. Note that there is no ComponentName field. The Part Number, which was added to the library in the step above, will be used from this point forward to reference the library components.

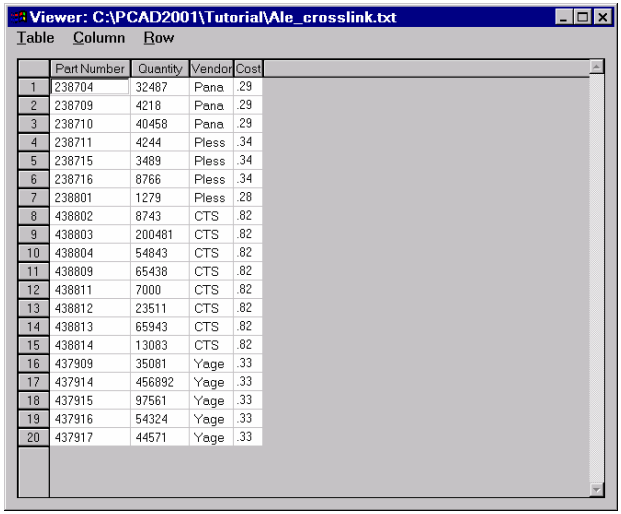
Import the map file by following these steps:

1. Choose **File » Import** to open the *Import Separated List File* dialog.



2. Click the **Browse** button. Navigate to the `Tutorial` folder in the P-CAD installation directory and select the comma-delimited source file `Ale_crosslink.txt`.
3. Click the **First line contains field names** button since the first line of the sample file contains the field names that correspond to the file contents.

   If the first line of the external source file did not contain field names, the **Generate default field names** button should be selected.
4. Enter or select a character in the List Separator box.
5. Enter or select a character in the Comment Character box.
6. Click **OK**. The external source file Viewer appears.



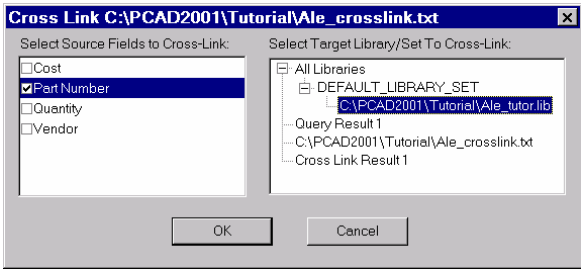
	Part Number	Quantity	Vendor	Cost
1	238704	32487	Pana	.29
2	238709	4218	Pana	.29
3	238710	40458	Pana	.29
4	238711	4244	Pless	.34
5	238715	3489	Pless	.34
6	238716	8766	Pless	.34
7	238801	1279	Pless	.28
8	438802	8743	CTS	.82
9	438803	200481	CTS	.82
10	438804	54843	CTS	.82
11	438809	65438	CTS	.82
12	438811	7000	CTS	.82
13	438812	23511	CTS	.82
14	438813	65943	CTS	.82
15	438814	13083	CTS	.82
16	437909	35081	Yage	.33
17	437914	456892	Yage	.33
18	437915	97561	Yage	.33
19	437916	54324	Yage	.33
20	437917	44571	Yage	.33

The field names are the column headers with the field values filling in the table contents.

Step 4: Cross Linking the Files

To cross link the MRP file with the P-CAD sample library:

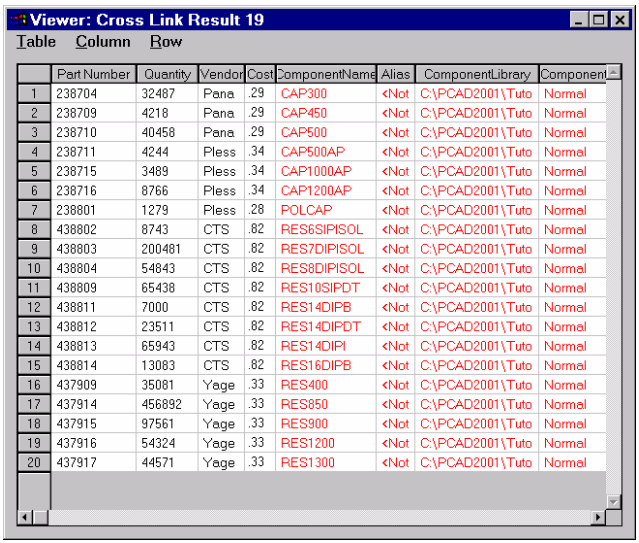
1. From the **Table** menu of the MRP file Viewer, choose **Cross Link** to open the *Cross Link* dialog.



2. In the **Select Source Fields to Cross Link** box, select the box to the left of the Part Number field name.

3. In the **Select Target Library/Set to Cross Link** box, select the library `Ale_tutor.lib` file under DEFAULT_LIBRARY_SET.

4. Click **OK**. The Cross Link Result Viewer appears with the combination of the library and source file information for components with matching Part Number attributes.



	Part Number	Quantity	Vendor	Cost	ComponentName	Alias	ComponentLibrary	Component
1	238704	32487	Pana	.29	CAP300	<Not	C:\PCAD2001\Tuto	Normal
2	238709	4218	Pana	.29	CAP450	<Not	C:\PCAD2001\Tuto	Normal
3	238710	40458	Pana	.29	CAP500	<Not	C:\PCAD2001\Tuto	Normal
4	238711	4244	Pless	.34	CAP500AP	<Not	C:\PCAD2001\Tuto	Normal
5	238715	3489	Pless	.34	CAP1000AP	<Not	C:\PCAD2001\Tuto	Normal
6	238716	8766	Pless	.34	CAP1200AP	<Not	C:\PCAD2001\Tuto	Normal
7	238801	1279	Pless	.28	POLCAP	<Not	C:\PCAD2001\Tuto	Normal
8	438802	8743	CTS	.82	RES6SIPISOL	<Not	C:\PCAD2001\Tuto	Normal
9	438803	200481	CTS	.82	RES7DIPISOL	<Not	C:\PCAD2001\Tuto	Normal
10	438804	54843	CTS	.82	RES8DIPISOL	<Not	C:\PCAD2001\Tuto	Normal
11	438809	65438	CTS	.82	RES10SIPDT	<Not	C:\PCAD2001\Tuto	Normal
12	438811	7000	CTS	.82	RES14DIPB	<Not	C:\PCAD2001\Tuto	Normal
13	438812	23511	CTS	.82	RES14DIPDT	<Not	C:\PCAD2001\Tuto	Normal
14	438813	65943	CTS	.82	RES14DIPI	<Not	C:\PCAD2001\Tuto	Normal
15	438814	13083	CTS	.82	RES16DIPB	<Not	C:\PCAD2001\Tuto	Normal
16	437909	35081	Yage	.33	RES400	<Not	C:\PCAD2001\Tuto	Normal
17	437914	456892	Yage	.33	RES850	<Not	C:\PCAD2001\Tuto	Normal
18	437915	97561	Yage	.33	RES900	<Not	C:\PCAD2001\Tuto	Normal
19	437916	54324	Yage	.33	RES1200	<Not	C:\PCAD2001\Tuto	Normal
20	437917	44571	Yage	.33	RES1300	<Not	C:\PCAD2001\Tuto	Normal

5. In the Viewer table, there are two columns containing values for the Quantity attribute: one from the source and one, with the header TARGET:Quantity, from the library. The same is true for the Vendor and Cost attributes.

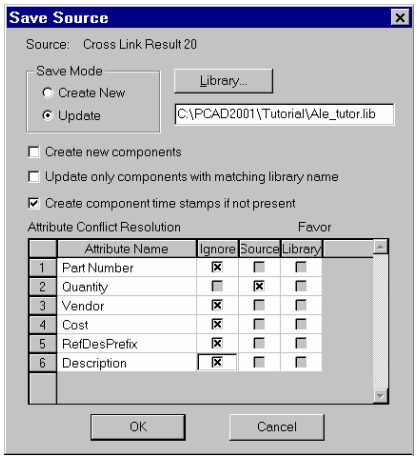
   Select the column **TARGET:Quantity** by clicking the header. While holding the SHIFT key, click and hold the left mouse button and drag the column until a black line appears adjacent to the Quantity column. Release the mouse button and the TARGET:Quantity column is placed next to the Quantity column.

6. With the two columns containing component Quantity information adjacent to each other you can visually compare changes in inventory.

   Select the **TARGET:Quantity**, **TARGET: Cost** and **TARGET:Vendor** columns by clicking the column headers with the SHIFT key pressed. Click DELETE to eliminate the original library Quantity values. The updated attribute values are in the file Viewer and ready to be saved.

Step 5: Updating the Library

1. Update the Library by choosing the **Save to Library** command from the **Table** menu to open the *Save Source* dialog.



2. Select the **Update** option button.

3. Click the **Library** button, navigate to the `Tutorial` folder in the P-CAD installation directory and select the `Ale_tutor.lib` library. Click **Open**. The selected library name displays in the edit box.

   There are no new components in the imported MRP file. Leave the **Create new components** box cleared.

   Since the MRP file does not contain a ComponentLibrary attribute, leave the **Update only components with matching library name** box cleared.

4. Select the **Create component time stamps if not present** check box when adding or updating the ModifyDate attribute in the component library.
5. Select the **Source** check box only to the right of the attribute Quantity, since only that attribute will be updated.
6. Click **Ignore** to the right of all remaining attributes.
7. Click **OK**.

   Notice that the Cost attribute for many of these components has changed. You will be updating the component Cost in lesson 3.

Lesson 3: Importing Without Saving

If for any reason you do not want to add the index identifier to your P-CAD library, you can still import a MRP file without a ComponentName. To do so, you will be using the Cross Link feature of Library Executive in a slightly different way. For more information about the versatility and power of cross linking, refer to the *Importing Data from an External Source (page 121)*.

Although a complete lesson is not presented here, you may complete all of the following steps using the same three sample files as above: `Ale_map.txt`, `Ale_crosslink.txt` and `Ale_tutor.lib`.

The imported file does not contain a ComponentName field. The basic problem is to associate the components in a P-CAD library with those of the imported file. As with lesson 2 above, the first step serves the purpose of inserting the MRP database index identifier as an attribute for its associated component.

In this case, however, you do not wish the attribute to be saved. So the association between ComponentName and Part Number must be repeated for each import process. It only requires one quick, easy step. When the index identifier is not saved to the library, the import process for an external file without a ComponentName becomes:

1. Cross Link the `Ale_map.txt` map file ComponentName field with the P-CAD library. The Cross Link result then contains the combination of the Part Number and the library components.
2. Choose **File Import** to import the map file.
3. Add the Part Number by choosing the **Cross Link** command from the **Table** menu of the file Viewer.
4. Check the **ComponentName** field in the **Select Source Fields to Cross Link** box and select the `Ale_tutor.lib` library in the **Select Target Library/Set to Cross Link** box.
5. Click **OK** to display the combination of the library and map file. In this case, it contains the Part Number attribute as well as the component attributes inside the library.

   The data in this viewer can be used without saving it to a library until the Library Executive application is exited. The Source Browser lists the result of the cross link that you have just completed (e.g. Cross Link Result #1).

6. Import the `Ale_crosslink.txt` file.
7. Cross link the imported file with the library — map file Cross Link result generated in step 1. A new cross link result table is generated with a combination of all map file, imported file, and P-CAD library component information.

   This step is identical to cross linking, except that, instead of the updated library, the cross link result generated in step 1 is selected as the target.
8. Select the **TARGET:Quantity**, **TARGET:Cost** and **TARGET:Vendor** columns by clicking the column headers while pressing SHIFT. Click **DELETE** to eliminate the original library Quantity values. The updated attribute values are in the file Viewer and ready to be saved
9. Update the library from the new Cross Link results by choosing the **Save to Library** command.

   The cross link result of step 1 contains all the information of the library updated with the map file contents. The two are interchangeable. If you choose to update your library with the Part Number, then in step 3 cross link with the updated library. If you choose not to update your library with the Part Number, then in step 3 cross link with the map file cross link result.

   The only fundamental difference between the two methods is that saving the index identifier to the library saves you a step in subsequent import processes.

Lesson 4: Using Query to Selectively Update a Library

Query is a particularly useful tool when updating a library from an external source file. With Query, you can select components with particular characteristics to update. For instance, you could choose to update only components made by a particular vendor. Or, you could choose to update components modified after the date you last updated the library.

You can choose to query and update from any source in Library Executive: an external source, a query result, a cross link result or a P-CAD library. In this lesson, you will be updating components that are produced by a particular vendor. This vendor has just changed its component prices, so the Cost attribute in the library will be updated.

This lesson assumes that you have already set up the `Ale_tutor.lib` so it is available for Library Executive. If not, open the Source Browser and select the DEFAULT_LIBRARY_SET. Choose **Add Library** to add the sample library to the default set.

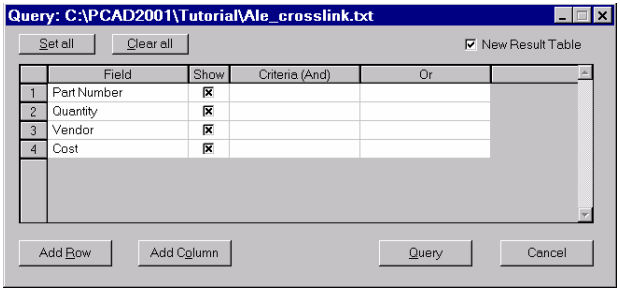
You can follow along the next few sections, using the sample files to complete each of these steps yourself.

Step 1: Select the Source to Query

Again you will be using the sample file `Ale_crosslink.txt` to update the sample library `Ale_tutor.lib`. If this file is not presently in your Source Browser, import the file.

To select the `Ale_crosslink.txt` file to Query:

1. Click the **Source Browser** toolbar button to open the Source Browser tree.
2. Select `Ale_crosslink.txt` and choose **Query** from the shortcut menu.



Step 2: Setting Up the Query Dialog

In this case, you want to update only components that are produced by CTS. The expression for this search criterion is:

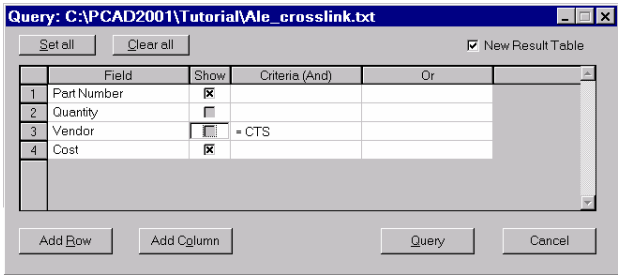
```
Vendor = CTS
```

Since there is only one criterion, it will be placed in the Criteria (And) column of the *Query* dialog.

To set up the *Query* dialog, follow these steps:

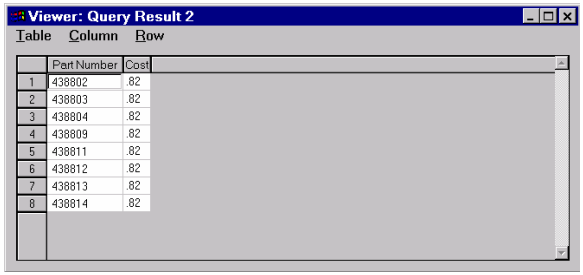
1. Click **Set All** to list the source fields in the Field column.
2. Click in the **Criteria (And)** column to the right of the Vendor attribute, an arrow appears. Click the arrow to display a pick list which can help you select the = operator. Enter the expression = `CTS`.
3. Since only the Cost attribute will be updated in the library, display only the Part Number and Cost attributes. The Part Number will be used to cross link to the corresponding library components made by CTS. Clear the **Show** check box to the right of the other attributes.
4. Check the **New Result Table** box, so that any previous query results are not overwritten.

   The *Query* dialog should look like the following:



Step 3: Updating the Library with Query Results

1. Click **Query** to display the Query Results Viewer that contains all source components satisfying the selected search criteria. In this case, the Query Results Viewer should appear as follows.



2. Choose **Cross Link** from the Query Results Viewer to link the Part Number to its corresponding component in the library `Ale_tutor.lib`.

3. Delete the TARGET:Cost column to eliminate the out-of-date library values by clicking the column heading and then pressing the DELETE key.

4. Update the library by choosing the Save to Library command. Select the **Source** check boxes to the right of all attributes. All attributes for the vendor's components will by updated in the selected library, including Cost.

Lesson 5: Using Embedded Query to Find and Place a Component

The Query utility in P-CAD Library Executive is a perfect aid in finding the ideal component for a design. For a simple example, you would like to place a capacitor on your PCB design. With P-CAD Library Executive, the perfect capacitor is right at your fingertips!

If you intend to place a component in a design, the source that you query must contain complete components. For example, you could query a P-CAD library and place any query result component directly into a design. If, however, the source is an external file, the imported component attributes must be attached to a pattern, a symbol, and electrical information before placement.

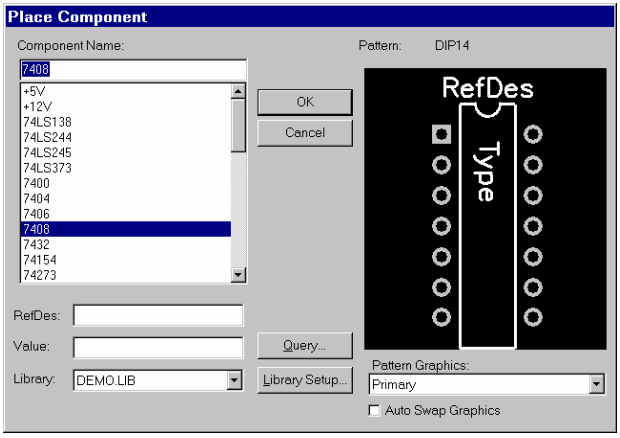
To place a component from a Query Result, you must also select to display the ComponentName and ComponentLibrary fields in the Query Result Viewer. Refer to the *Component Placement* chapter *(page 163)* for details on the requirements for placing components.

Although you can use Query from Library Executive, as in lesson 3, this example illustrates using the *Query* dialog directly from the PCB design in which you want to place the component. The Query utility is embedded in the Place Component and Place Part commands of PCB and Schematic. The basic structure of these embedded *Query* dialogs are slightly different than the Query accessed directly from Library Executive, however, its basic function is the same.

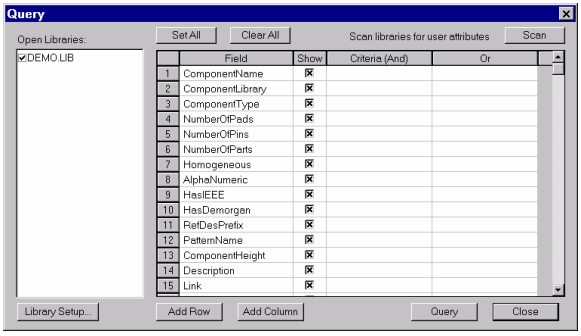
Step 1: Open the P-CAD Design

To access the embedded Query utility, open a P-CAD design after you start PCB. You will be placing the capacitor in a blank workspace. Placing a queried component in a design is just as easy.

1. Choose the **Place Component** command to open the *Place Component* dialog.



2. Click **Query** to search for a component to place in a design. The embedded *Query* dialog appears.



Step 2: Setting up the Query Dialog

In this case, you would like to place a capacitor. In the sample library all capacitors have the letters CAP somewhere in their component name. The expression for this search criteria is:

```
ComponentName IsLike *CAP*
```

The two * wildcards indicate that CAP can be in the beginning, end, or middle of the name. Since there is only one criterion, it will be placed in the Criteria (And) column of the *Query* dialog.

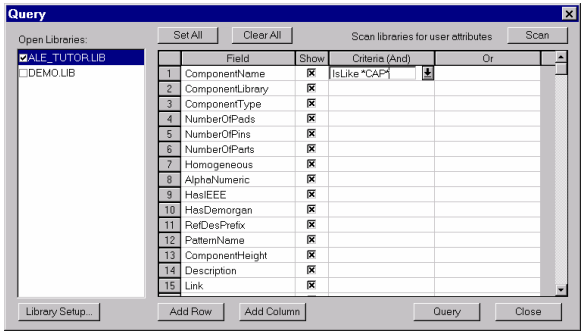
1. If the `Ale_tutor.lib` is not in the Open Libraries box, click the **Library Setup** button and add the sample library. Click **OK** to return to the *Query* dialog.

2. In the Open Libraries box, verify that the `Ale_tutor.lib` is selected. If there are any other open libraries, clear the check box to their left.

3. Click in the **Criteria (And)** column to the right of the ComponentName field, an arrow appears.

4. Click the arrow to display a list from which you should select the **IsLike** operator. Enter the expression: `IsLike *CAP*`.

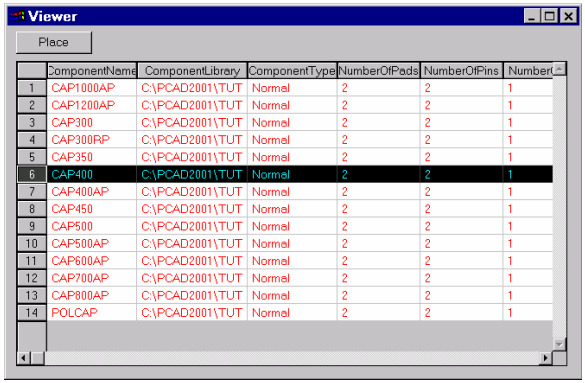
   The embedded *Query* dialog should look like the following:



Step 3: Using Query Results to Place a Component

When you click **Query**, the Query Results Viewer appears. This Query Results Viewer is identical to that which appears when you access Query directly from Library Executive. The Viewer contains all source components satisfying the selected search criteria.

In this case, the Query Results Viewer should look like the following:



You can use the Query Results to place a capacitor in your design. You will be choosing the CAP400 capacitor to place in the blank PCB workspace.

1. Select the row containing the capacitor CAP400 by clicking on the row number to its left.
2. Click the **Place** button. The *Place Component* dialog appears with the chosen component selected in the Component Name list.
3. Click **OK**. Move the ghosted component outline to accurately place the component on the workspace and click the left mouse button.

For more details, refer to the *Place Component* command in your *P-CAD PCB User's Guide*.

Lesson 6: Reporting on Component Library Updates

You have been working on a design for the past six weeks. It is almost complete, but you want to check that changes made to the component library do not critically impact your design.

In this lesson, you will learn how to generate a quick report of all components that have been updated since you loaded the netlist using the timestamp feature of Library Executive. With the component update report, you can easily verify that all of your design components are current.

Assume that your calendar indicates the netlist was loaded and most components were placed on June 16th. The steps to report on library changes after this date will be detailed below.

Step 1: Query the Library for a ModifyDate

To query the library for all components modified after June 16th, you will be using the timestamp feature of Library Executive. When creating, updating, and verifying a P-CAD library, you can use timestamp to add CreateDate, ModifyDate, and VerifyDate to the library components. These dates can then be used for checking the progress of the libraries and the design components that come from them. These timestamps must be added to your library before you can query for the timestamp attributes.

For more information about the timestamp feature, refer to *Time Stamps (page 47)*.

To query a library for a ModifyDate after June 16th:

1. Open the Source Browser to add the library to a library set. The library is then available for Library Executive utilities.

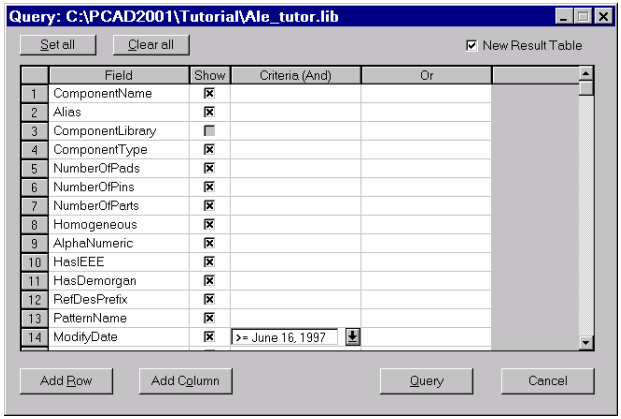
   If you would like to follow along with this lesson, you can use the `Ale_tutor.lib`. If you have used this file for any other lessons, load a new backup copy of the original library file. Any modifications made to this library during the above lessons will have caused the ModifyDate attribute to be updated.

2. Select the library in the Source Browser tree and choose **Query** from the shortcut menu. The *Query* dialog appears.

3. Click the **Set All** button to show the fields contained within the library. Scroll down the list to find the ModifyDate attribute.

4. In the **Criteria (And)** column, use the pick list to select the >= operator. After the operator, type: `June 16, 1997`

   The *Query* dialog appears.



5. Click **Query**. All components in the library modified on or after June 16th, 1997 will appear in the Query Results Viewer.

Step 2: Report on Updated Components

You can generate a report on these updated components. This report can be easily compared with a Bill of Materials from your design. You can determine which attribute changes, if any, impact your final board design.

1. From the Query Results Viewer, choose **Report** from the **Table** menu.
2. Select the **Report** option button in the Style Format frame.
3. Type in a header, footer, and output file name.
4. Select all the check boxes in the Page Format frame, as well as the **File** option button in the Output.
5. Click **OK**. Load the file into a text file viewer to view and print the report.

Lesson 7: Verifying Your Design Against a Library

In lesson 6, you learned how to generate a report on any library components changed after placement. In this lesson, you will learn how to verify the component attributes in the design and generate a difference report directly. The design components may be updated automatically to reflect these library changes.

Again you have been working on a design for the past six weeks. It is almost complete, but you want to check that changes made to the component library do not critically impact your design.

The report in lesson 6 lists all library components modified after the design placement date, whether or not they exist in the design. The difference report generated in this lesson lists only components with attribute discrepancies between the design and library.

Step 1: Open a PCB or Schematic Design

To access the Design Verification utility, open a P-CAD design by starting PCB. You will be generating a difference report from a PCB design against a P-CAD library.

1. Choose **File » Open**.
2. Open a design file, such as `Demo1_u.pcb` in the `Demo` folder in the P-CAD installation directory.
3. Choose **Library » Verify Design**. The *Design Verification* dialog appears.

Step 2: Set up the Design Verification Dialog

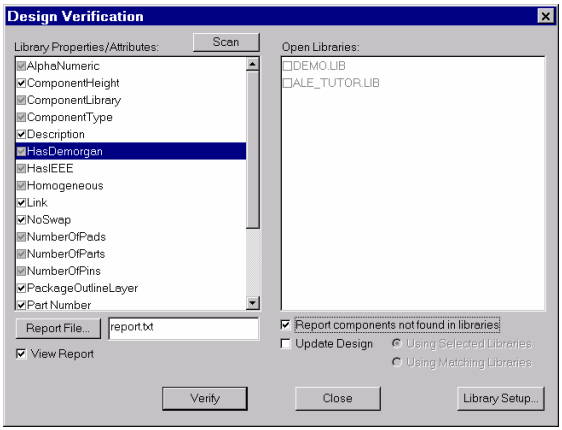
In this section, you will set up the *Design Verification* dialog to generate a difference report. The difference report will be displayed to the screen, as well as to a specified output file. This report will contain components that have differing attributes in the design and the library. It will also list the components that have been placed in the design that are not found in the open libraries.

You can also use the Design Verification utility to update component attributes in your design. For more information about updating your design using the Design Verification utility, refer to *Library Verification (page 137)*.

1. In the *Design Verification* dialog all predefined attributes are listed in the **Library Properties/ Attributes** box. Leave all attribute check boxes in this list selected. You will be verifying all component attributes.

   In this case, the library components that you are going to verify, have no additional user-defined attributes. If the library had additional attributes that you wanted to include in the difference report, you would click **Scan** to add them to the list.

   Design components are verified against all open libraries in the Open Libraries list. The *Design Verification* dialog appears.



2. Type the report file name in the box and select the **View Report** check box.

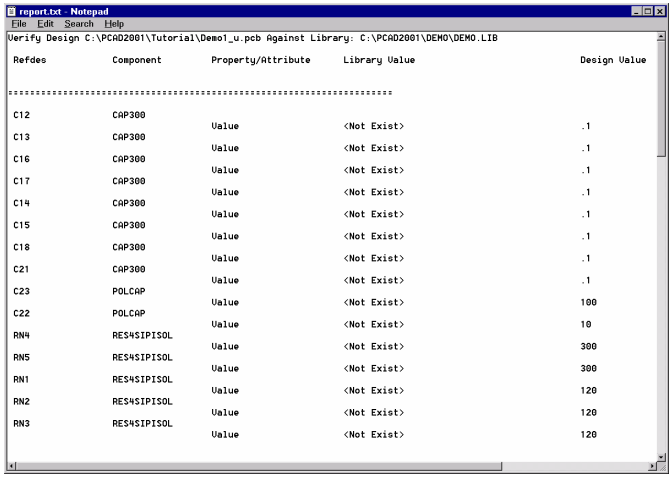
3. Select the **Report components not found in libraries** check box to list in the difference report design components that were not found in the open libraries.

Step 3: Generate a Difference Report

You are now ready to generate a difference report that includes both attribute differences between the design and library and design components not found in the open libraries.

1. Click **Verify**. A difference report is generated and displayed on the screen.

   The difference report is a simple text file. A sample difference report is shown below.



```
report.txt - Notepad
File  Edit  Search  Help
Verify Design C:\PCAD2001\Tutorial\Demo1_u.pcb Against Library: C:\PCAD2001\DEMO\DEMO.LIB

 Refdes          Component       Property/Attribute     Library Value                    Design Value

==================================================================

 C12             CAP300
                                 Value                  <Not Exist>                      .1
 C13             CAP300
                                 Value                  <Not Exist>                      .1
 C16             CAP300
                                 Value                  <Not Exist>                      .1
 C17             CAP300
                                 Value                  <Not Exist>                      .1
 C14             CAP300
                                 Value                  <Not Exist>                      .1
 C15             CAP300
                                 Value                  <Not Exist>                      .1
 C18             CAP300
                                 Value                  <Not Exist>                      .1
 C21             CAP300
                                 Value                  <Not Exist>                      .1
 C23             POLCAP
                                 Value                  <Not Exist>                      100
 C22             POLCAP
                                 Value                  <Not Exist>                      10
 RN4             RES4SIPISOL
                                 Value                  <Not Exist>                      300
 RN5             RES4SIPISOL
                                 Value                  <Not Exist>                      300
 RN1             RES4SIPISOL
                                 Value                  <Not Exist>                      120
 RN2             RES4SIPISOL
                                 Value                  <Not Exist>                      120
 RN3             RES4SIPISOL
                                 Value                  <Not Exist>                      120
```

Lesson 8: Verifying Your Libraries Against an MRP File

You can also check to see whether your libraries need updating when changes to the MRP database are made. If the MRP is equivalent to the library and the library is equivalent to components in the design, the components in the design will be up to date with the MRP.

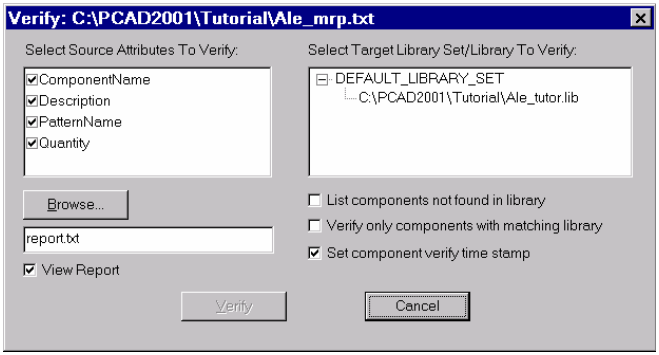
This section assumes that you have the MRP source imported. The MRP source component must be associated with its corresponding ComponentName for library verification. It also assumes that you have set up the libraries so that they are available for P-CAD Library Executive. Add the libraries to the default library set, or create your own library set to contain the libraries.

Step 1: Verifying the MRP Source

After importing the MRP source and setting up the chosen libraries as you did in Lesson 1, complete the following steps:

1. Select the MRP source file from the Source Browser.
2. Choose **Verify** to open the *Verify* dialog.

   The **Verify** command can also be accessed from the **Table** menu of the file Viewer.



Step 2: Setting up the Verify Dialog

In this section, you will set up the *Verify* dialog to generate a difference report. The difference report will be displayed to the screen, as well as to a specified output file. This report will contain components that have differing attributes in the MRP file and the library. It will also list the components that are in the MRP file, but are not found in the selected libraries.

In contrast to Design Verification, discussed in Lesson 7, the library Verification utility can generate or update a VerifyDate attribute in the library indicating when the library was last verified. You will be generating a VerifyDate timestamp with the following steps:

1. In the *Verify* dialog, all MRP source attributes are listed in the **Select Source Attributes to Verify** box. Leave all attribute check boxes in this list selected. You will be verifying all component attributes.
2. The source MRP components are verified against all selected libraries in the **Select Target Library Set/Library To Verify** list. Select the library or library set.
3. Type the report file name in the box and select the **View Report** check box.
4. Select the **List components not found in libraries** check box to list in the difference report design components that were not found in the open libraries.
5. Check the **Set component verify time stamp** box to generate or update the VerifyDate attribute in the selected libraries.

You can also choose to verify only components with matching ComponentLibrary attributes in the MRP source and P-CAD library. For more information, about this and other Verify selections, refer to *Library Verification (page 137)*.

Step 3: Generate the Difference Report

The Verify difference report is similar to that generated by the Design Verification utility discussed in Lesson 6.

Lesson 9: Library Creation Using Pattern and Symbol Editor

From any source, you can generate a P-CAD library. The only requirement is that it contain a ComponentName field. The other essential attributes that define the pattern, symbol, and electrical properties of the component are generated using the tools in Library Executive.

The process of creating an integrated component library is summarized in this lesson. For more information, references are included to other sections of this manual.

1. Select a source with a ComponentName field. This source could, for instance, be an imported MRP file. Display the source in the file Viewer.
2. Select a component from this source by clicking on the row number to its left. The component that you would like to make integrated is now highlighted.
3. Choose **Open** from the **Row** menu. The *Component Information* dialog is displayed. For additional information, refer to the *View Commands* chapter.
4. Click the **Pattern View** button. From this dialog you can attach a pattern to the component.

   To create a new pattern or edit an existing pattern, click **Edit Pattern**. The Pattern Editor appears. To select an existing pattern, click **Select Pattern**.
5. Click the **Symbol View** button. From this dialog you can attach a symbol to the component.

   To create a new symbol or edit an existing symbol, click **Edit Symbol**. The Symbol Editor appears. To select an existing symbol, click **Select Symbol**.
6. Return to the *Component Information* dialog to complete the integrated component. Completion includes attaching the pattern and symbol, adding component properties, and editing the pins view spreadsheet to designate the component's electrical information.
7. Validate and save the component to a P-CAD library by clicking the **Validate** button on the toolbar.
8. Then, when no errors are found, choose the **Component Save** command to save the component. This command saves the pattern, symbol, and electrical information, as well as all of the user-defined attributes included in the imported MRP source.
9. Repeat from step 2 for all components in the source.

Converting Libraries

Working with External Libraries

Unlike Tango-Schematic and Tango-PCB, which use separate symbol and pattern libraries, P-CAD applications use integrated libraries, which contain components, symbols, and patterns.

To use your Tango pattern and symbol libraries with P-CAD Schematic and P-CAD PCB, you must translate them first using Library Executive. Once your libraries have been translated, if you intend

to use them with both P-CAD applications, you need to merge the converted pattern libraries into the converted symbol libraries.

This section uses a simple example to walk you through three conversion scenarios:

- You want to convert Tango-PCB libraries into P-CAD format.
- You want to convert Tango-Schematic into P-CAD format.
- You want to create integrated P-CAD libraries from converted pattern and symbol libraries.

Conversion Example

For this example, make the following assumptions:

- You have two Tango-PCB pattern libraries: `pcb1.lib` and `pcb2.lib`.
- `Pcb1.lib` contains a pattern called DIP14.
- `Pcb2.lib` contains a pattern called LLC20.
- You have one Tango-Schematic library: `sch1.lib`.
- `Sch1.lib` contains two components called SN5400J and SN5400FK.
- Component SN5400J has a reference to a pattern called DIP14.
- Component SN5400FK has a reference to a pattern called LLC20.

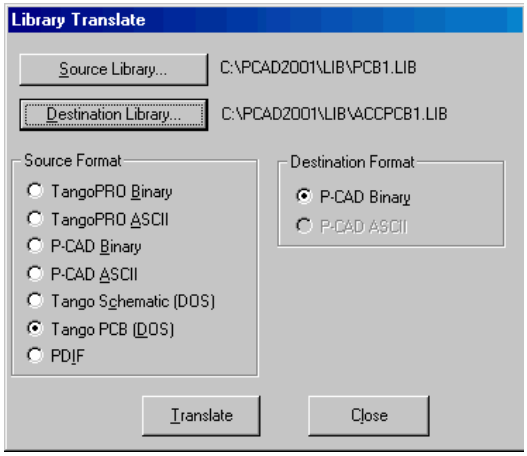
Tango-PCB Only

To translate your Tango-PCB pattern libraries, choose the **Library » Translate** command. The *Library Translate* dialog appears.

1. First, you would translate `pcb1.lib`.
2. Click the **Source Library** button to display the *Library File Listing* dialog, where you can choose your pattern libraries. In this example, choose `pcb1.lib`.
3. Select the **Tango-PCB (DOS)** option button in the Source Format frame.

   Note that the only available destination format is P-CAD Binary.
4. Click the **Destination Library** button to display the *Library File Listing* dialog, and type a name for your destination library. In this example, type `accpcb1.lib`.

   Your dialog now appears as follows.



5. Click T**ranslate** to begin the translation process.
6. Repeat the process for each pattern library you want to convert. In this example, translate `pcb2.lib`, and name it `accpcb2.lib`.
7. Click **Close** to exit the dialog.

The translated libraries are now ready to use with PCB, but not with P-CAD Schematic, because they don't have symbol information. You can add symbols to the library, or you can merge this library with another library having symbols as described in the **Library** » **Merge Patterns** command.

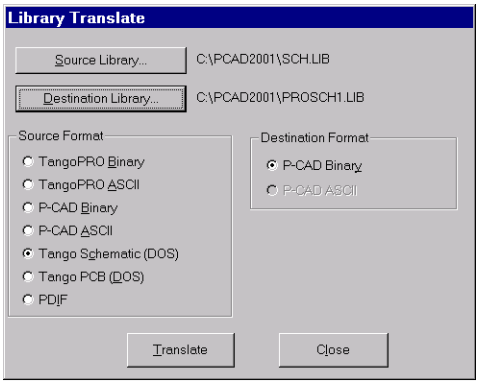
Tango-Schematic Only

To translate your Tango-Schematic symbol libraries, choose **Library** » **Translate**. The *Library Translate* dialog appears.

1. For this example, we'll translate `sch1.lib`.
2. Select the **Source Library** option button to display *the Library File Listing* dialog, where you can choose your libraries. Choose `sch1.lib`.
3. Select the **Tango-Schematic (DOS)** option button in the Source Format frame.

   The only available destination format is **P-CAD Binary**.
4. Click the **Destination Library** button to display the *Library File Listing* dialog, and type a name for your destination library. Type `prosch1.lib`.

   Your dialog now appears as shown in the following figure.



5. Click **Translate** to begin the translation process.
6. Repeat the process for each symbol library you want to convert.
7. Click **Close** to exit the dialog.

The translated library is now ready to use with P-CAD Schematic, but not with PCB because it doesn't have pattern information. You can add patterns to the library or you can merge this library with another library having patterns as described in the *Library Commands* chapter.

Naming Symbols

Symbol names in translated Schematic libraries now have the destination library as a name prefix. Each symbol in a translated library is assigned a unique symbol name as follows:

```
DESTLIB_XX_A
```

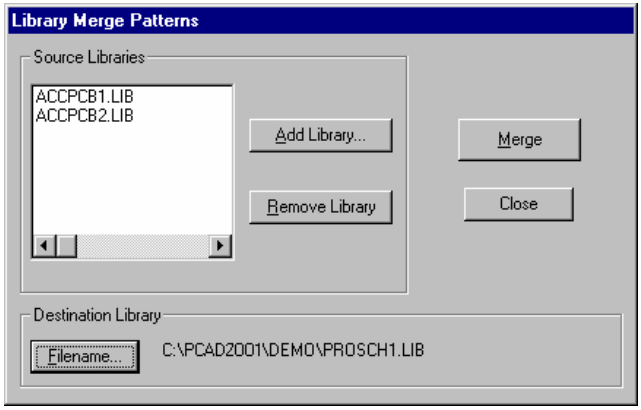
where DESTLIB is the name of the destination library, XX is a unique number, and A equals N(ormal), I(EEE), or D(eMorgan).

Tango-PCB and Schematic

When all your pattern and symbol libraries have been converted to P-CAD format, choose Library Merge Pattern to merge pattern and symbol libraries into integrated pattern/symbol/component libraries which can be used by all P-CAD applications.

In this example, merge `accpcb1.lib` and `accpcb2.lib` into `prosch1.lib`.

1. Choose **Library » Merge Patterns** to display the following dialog.



2. Click the **Add Library** button to display the *Library File Listing* dialog, where you can choose libraries containing patterns. The library names you add appear in the **Source Libraries** box.

   You need to select all the pattern libraries that contain the patterns referenced in all components in the destination library. In this example, choose `accpcb1.lib` and `accpcb2.lib`.

3. Click the **Destination Library** button to display the *Library File Listing* dialog, where you can choose one of your symbol libraries as the destination library. In this example, you would choose `prosch1.lib`.

4. Click **Merge** to begin the process. For every component in the destination library, the program searches the pattern libraries for the corresponding pattern. The libraries are searched in the order they appear in the list.

5. Repeat the process for each symbol library.

6. Click **Close** to exit the dialog.

   Your destination libraries should now be fully integrated and ready to use with both P-CAD Schematic and PCB. Following this example, `prosch1.lib` is a fully integrated library.

CHAPTER 14

File Commands

In P-CAD Library Executive, the File menu commands allow you to import component information from any component library or database in a simple character-delimited file format. A P-CAD library of the imported components can be generated or updated, if desired. Importing data from an external source is one of the many Library Executive enhancements.

Any other Library executive features can be accessed from the File menu, including Query, Verify and Reports.

File Import

In P-CAD Library Executive, you can import a file containing component information from a variety of sources, including your corporate component database. The incoming file must be a simple separated list format. The separated list file format is described in *Importing Data from an External Source (page 121)*.

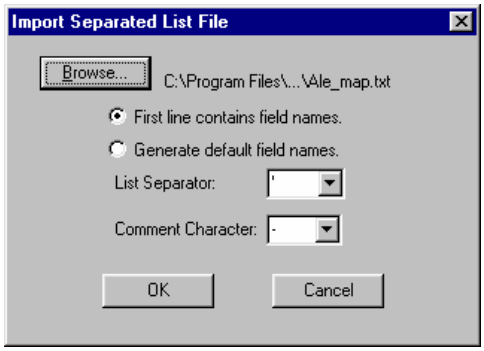
The **File Import** command completes the first step of importing component data from an external source. The second step is to convert, or map, the file contents to attributes recognized by P-CAD by choosing either the **Map Keys** or **Cross Link** commands.

The imported component data file creates or updates a P-CAD library; and you can Query, Verify and Report on the imported components.

The Import Separated List File Dialog

The *Import Separated List File* dialog is opened using one of the following methods:

- Choose **File** » **Import**.
- Select the root of the Source Browser tree and choose **New Character-delimited File** from the shortcut menu.



The *Import Separated List File* dialog allows you to specify the external source file and its organization with the following options:

- **Browse**: Opens a dialog where you can navigate to and select the external source file to import.
- **First line contains field names**: Select the **First line contains field names** option button if the first line of the separated list source file contains the field name headers for the field values. These names appear on the file Viewer.
- **Generate default field names**: Select the **Generate default field names** option button if the first line of the separated list source file does not contain field name headers. Default field names appear on the file Viewer.
- **List Separator**: Select from the list or enter the character you want to use as a delimiter in the List Separator box.
- **Comment Character**: Select from the list or enter the character you want to use as the Comment Character.

Importing a Separated List File

To import a separated list file into P-CAD Library Executive, follow these steps:

1. Choose **File** » **Import** to open the Import *Separated List File* dialog.
2. Click **Browse** to navigate to and select the file containing the separated list file.
3. Click **Open**. The name of the file to be imported is displayed next to the **Browse** button.
4. If the first line of the separated list file contains field names, select the First line contains field names option button.

   If the first line of the separated list file does not contain field names, select the **Generate default field names** option button. Default field names are attached to the imported data.

   In both cases, the imported field name headers must be mapped to field names recognized by P- CAD in the second stage of the import process.
5. Click **OK**. The *Viewer* dialog appears with the file contents in a spreadsheet format.

File Map Fields

When importing a separated list file from a non-P-CAD source, the imported field names can be associated with field names recognized by P-CAD. These fields are properties and attributes that define a complete, intelligent component. This association can be completed by choosing the **File » Map Fields** or **File » Cross Link** commands.

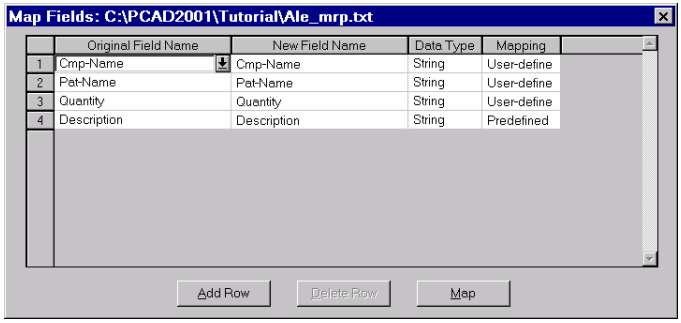
Map Fields is preferable when the imported file contains the P-CAD primary key, ComponentName. With **Map Fields**, the mapping is remembered so that subsequent external source file imports are mapped automatically.

Cross Link is preferable when the database does not contain a ComponentName field. **Cross Link** initially requires that the component database index number be linked to the corresponding P-CAD library component. Once this association has been completed, an external source file without a ComponentName may be directly connected to P-CAD library components. Refer to *File Cross Link (page 201)* for additional information.

The Map Fields Dialog

The *Map Fields* dialog is opened when you choose **Map Fields** from the **File** menu or **Viewer Table** menu, or by selecting the external source file in the Source Browser tree and choosing **Map Fields** from the shortcut menu.

The *Map Fields* dialog appears as shown in the following figure.



The *Map Fields* dialog is a spreadsheet. Refer to *Library Basics (page 3)* for information about spreadsheet editing and manipulation.

The *Map Fields* dialog allows you to specify the new field names and data types to which the original fields are mapped. The spreadsheet contains the following columns:

- **Original Field Name**: Lists the field names of the imported external source file along with the original field names. New rows can be added to generate a one-to-many mapping. For details, refer to *Using the Map Fields Command (page 200)*.
- **New Field Name**: Lists the field names to which the fields of the imported external source file are mapped along with the P-CAD-recognized field names.

- **Data Type**: Contains the data type of the field to be mapped: Integer, String, or Boolean. Modifying the data type is used for search and display purposes only.

Using the Map Fields Command

After the external source file has been imported into P-CAD Library Executive, you can map the field names to the predefined field names recognized by P-CAD.

Map Fields maps the original separated list file. Any modifications made to the file contents within the Viewer will be lost when the **Map Fields** command is used. To save these modifications, choose **Save To Library** before mapping.

To map the imported fields:

1. Choose **File » Map Fields** to open the *Map Fields* dialog.
2. Click in the **New Field Name** column. A down arrow appears at the right of the selected field name.
3. Click the **Down Arrow** to display the list of field names.
4. Select the desired P-CAD field name. The field is automatically placed on the spreadsheet and the field type is automatically updated in the Data Type column.

   You can modify the Data Type from the list of data types in the third column. Modifying the data type from its default value may be desired for two reasons:

   - **Sort order**: The Integer and String data types have different sorting orders.
   - **Display**: Selecting **Boolean** data type changes Integer 0/1 values to True/False and vice versa.

   Read-only attributes are for search and display purposes only. The default data type, String, is always used when saving the remaining attributes to a P-CAD library.

   Read-only attributes cannot be modified or saved to a P-CAD library. These attributes are displayed in red on the imported file *Viewer*.

   When the Original Field Name is mapped to a New Field Name, the field type is automatically updated in the Mapping column. The field type indicates whether the attribute is a primary key, a predefined attribute, or a user-defined attribute.

5. Click the **Add Row** button to append a row to the end of the table. A list of original field names becomes available in the first column. You can map the original field name to more than one P-CAD recognized field name.

   This one-to-many mapping is useful when your corporate database is indexed differently than a P- CAD library. For example, the database contains two 7400 components produced by two different manufacturers. The database field Part Number can be mapped to both a ComponentName field, which must be unique for P-CAD products, and a user-defined PartNumber field, which is the original name 7400.

Click the **Delete Row** button to delete a selected row.

> You can only delete rows that have been added to the table.

6. When all Original Field Names have been mapped to their desired New Field Name, click the **Map** button. The Viewer reappears with the fields mapped to the chosen P-CAD values. The identifier (mapped) appears next to the mapped file name.

   Library Executive remembers the specified field mapping for the latest mapped file. If changes are made to the corporate component database every week, and you map the fields today, then next week when you import the database to verify your P-CAD library the mapping is completed automatically.

   > If you map a file with the same filename but different field names than the previously mapped file, the original mapping will be overwritten.

File Cross Link

When importing a separated list file from a non-P-CAD source, the imported field names can be associated with field names recognized by P-CAD. These fields are properties and attributes that define a complete, intelligent component. This association can be completed by choosing either **File » Map Fields** or **File » Cross Link**.

Cross Link is preferable when the database does not contain a ComponentName field. With **Cross Link**, the initial association between the external source file index and its respective P-CAD library component is created before linking. The unique index identifier(s) are stored as user-defined attributes within the P-CAD library. The subsequent external source files need only contain this database index value to connect directly with their associated P-CAD library components. The primary key, ComponentName, is not necessary.

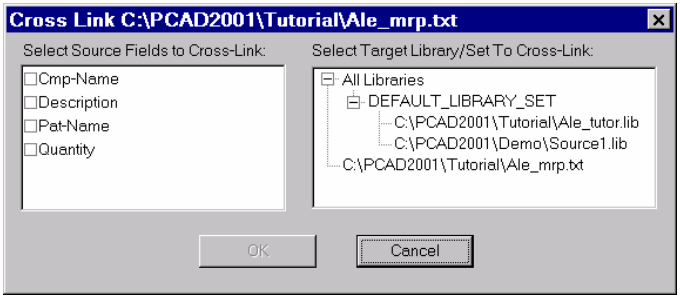
Map Fields is preferable when the imported file contains the P-CAD primary key, ComponentName. With **Map Fields**, the mapping is remembered so that subsequent external source file imports are mapped automatically. Refer to *File Map Fields (page 199)* for additional information.

The **Cross Link** command is also available from the **Viewer Table** menu or the Source Browser for imported source files, P-CAD libraries, Query Results and Cross Link Results. As with the **File » Cross Link** command, a selected attribute(s) in the cross link source is linked to the target P-CAD library or library set.

The Cross Link Dialog

The *Cross Link* dialog is opened when you choose **Cross Link** from the **File** menu or **Viewer Table** menu, or by selecting the external source file in the Source Browser tree and choosing **Cross Link** from the shortcut menu.

The *Cross Link* dialog appears as shown in the following figure.



The *Cross Link* dialog allows you to specify the field used as the unique external source file index, as well as the library or library set to which the source file is linked. The *Cross Link* dialog contains the following options

- **Select Source Fields to Cross Link**: Lists the field names of the imported external source file.
- **Select Target Library/Set to Cross Link**: Contains a tree of all libraries available to P-CAD Library Executive.

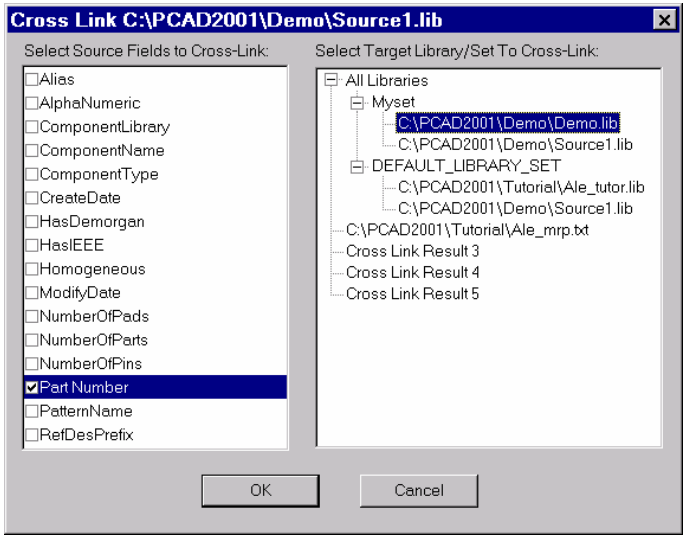
Using the Cross Link Command

To cross link the imported field, an initial association must be made between the unique index value(s) in the external source file and the corresponding components in the P-CAD libraries.

The **Cross Link** command links the source files as they are in memory. If changes have been made to a library used as the cross link source or target, choose **Reload** to refresh the library components in Library Executive's memory before conducting the **Cross Link**.

To cross link the imported fields:

1. Choose **Cross Link** from the **Table** menu of the Viewer to open the *Cross Link* dialog.



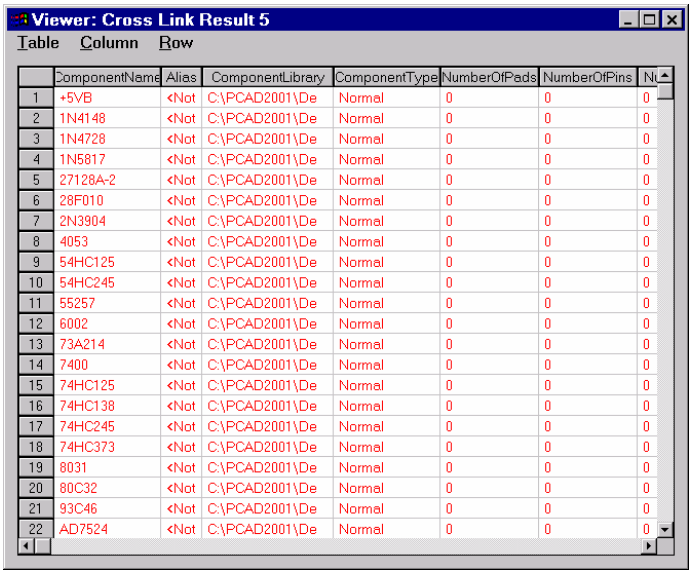
You can also choose the **Cross Link** command from the **File** menu on the main *Library Executive* dialog or from the shortcut menu when the imported external file is selected.

2. In the Select Source Fields to Cross Link list, select the check box to the left of the field name(s) you want to use to link to the P-CAD component libraries.

3. In the Select Target Library/Set to Cross Link list, expand the library groupings by clicking the **+** sign to its left. Navigate to and select the desired target library or library set.

4. Click **OK**.

The selected library is searched for the chosen source field. When a component with a matching source field is found in the library, the component information is displayed in the Cross Link Result Viewer.



Viewer: Cross Link Result 5

Table Column Row

	ComponentName	Alias	ComponentLibrary	ComponentType	NumberOfPads	NumberOfPins	N
1	+5VB	<Not	C:\PCAD2001\De	Normal	0	0	0
2	1N4148	<Not	C:\PCAD2001\De	Normal	0	0	0
3	1N4728	<Not	C:\PCAD2001\De	Normal	0	0	0
4	1N5817	<Not	C:\PCAD2001\De	Normal	0	0	0
5	27128A-2	<Not	C:\PCAD2001\De	Normal	0	0	0
6	28F010	<Not	C:\PCAD2001\De	Normal	0	0	0
7	2N3904	<Not	C:\PCAD2001\De	Normal	0	0	0
8	4053	<Not	C:\PCAD2001\De	Normal	0	0	0
9	54HC125	<Not	C:\PCAD2001\De	Normal	0	0	0
10	54HC245	<Not	C:\PCAD2001\De	Normal	0	0	0
11	55257	<Not	C:\PCAD2001\De	Normal	0	0	0
12	6002	<Not	C:\PCAD2001\De	Normal	0	0	0
13	73A214	<Not	C:\PCAD2001\De	Normal	0	0	0
14	7400	<Not	C:\PCAD2001\De	Normal	0	0	0
15	74HC125	<Not	C:\PCAD2001\De	Normal	0	0	0
16	74HC138	<Not	C:\PCAD2001\De	Normal	0	0	0
17	74HC245	<Not	C:\PCAD2001\De	Normal	0	0	0
18	74HC373	<Not	C:\PCAD2001\De	Normal	0	0	0
19	8031	<Not	C:\PCAD2001\De	Normal	0	0	0
20	80C32	<Not	C:\PCAD2001\De	Normal	0	0	0
21	93C46	<Not	C:\PCAD2001\De	Normal	0	0	0
22	AD7524	<Not	C:\PCAD2001\De	Normal	0	0	0

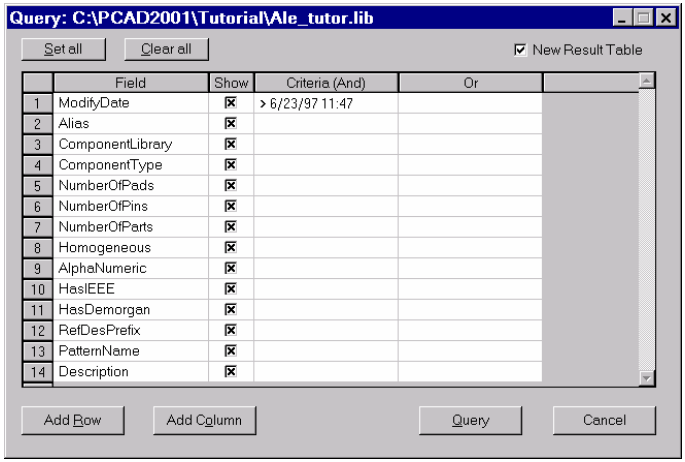
The Cross Link Result Viewer is a spreadsheet; attributes found in both the source file and the library are displayed. Across the top of the table are the fields found in the source and library. If a field name is found in both source and library, values from the library field are distinguished by their column header TARGET:FieldName. The field values for each of the components are listed in the appropriate column.

The *Cross Link Result Viewer* dialog is a spreadsheet. Refer to *Library Basics (page 3)* for details on manipulating spreadsheets.

File Query

The **File » Query** command opens the *Query* dialog. From this dialog, a query can be conducted on the components of the imported external source file. The *Query* dialog may also be accessed through the **Table** menu on the *Viewer* dialogs or from the shortcut menu in the Source Browser for query results, cross link results, imported files, or P-CAD libraries.

The *Query* dialog appears.



The *Query* dialog allows you to specify search criteria to look for components in the selected source. You can specify search and display fields, search criteria and output options.

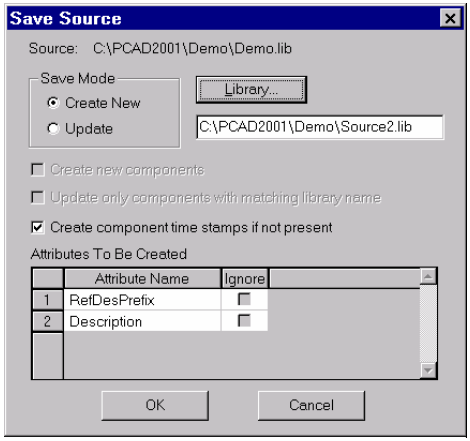
- **Set All/Clear All**: Click the **Set All** button to display all fields defined in the selected source in the *Field* column. Click **Clear All** to remove all fields from the *Field* column.
- **New Result Table**: Select this check box to generate an additional query result table. If the check box is cleared, the present query overwrites the previous result.
- **Query Spreadsheet**: The Field column contains both the search and display attributes for the query. The Show column is selected if the field is to be displayed in the query result. The Criteria (And) and Or columns display the query search criteria.
- **Add Row/Add Column**: The **Add Row** button adds a row to the query table if you wish to duplicate an attribute already on the *Query* dialog. The **Add Column** button adds another *Or* column to the *Query* dialog.
- **Query**: Click the **Query** button to search the selected source for components satisfying the specified criteria.

Refer to *Querying Libraries (page 103)* for more information.

File Save To Library

The **File » Save To Library** command opens the *Save Source* dialog. From this dialog, a P-CAD library can be created or updated from the components of the imported external source file. The *Save Source* dialog may also be accessed through the **Table** menu on the *Viewer* dialogs or from the shortcut menu in the Source Browser for query results, cross link results, imported files, or P-CAD libraries.

The *Save Source* dialog appears as shown in the following figure.



The *Save Source* dialog allows you to create or update a P-CAD library using components from the selected source. You can specify the library name, save mode and update options.

- **Source**: The source contains the components that are saved to the library. It is defined by the location from which you selected the **Save To Library** Command. If **Save To Library** is chosen from the **File** menu, the source is the imported external source file. If **Save To Library** is chosen from the Source Browser shortcut menus, the source could be a Query Result, a Cross Link Result, an external source file, or a P-CAD library.
- The **Save to Library** command saves only component attributes. To create a complete, integrated component both the electrical and graphical properties must be specified.
- **Save Mode frame**: **Create New** creates a new P-CAD library from the selected source. Update updates the selected P-CAD library according to the update criteria for specific attributes.
- **Library**: Click the **Library** button to select the name and directory of the P-CAD library to be updated or created. You can also type the name of the library directly into the box.
- **Create new components**: If selected, components in the source that are not found in the selected library are added to the library when updated.
- **Update only components with matching library name**: If selected, components are updated in the P-CAD library only if both the ComponentName and the ComponentLibrary field values match.
- **Create component time stamps if not present**: If selected, the time stamp attributes CreateDate and ModifyDate are added to the component with the current date and time value.
- **Attributes to be Created**: Lists the attributes in the component source.

  When creating a library, select the **Ignore** column for attributes you do not wish to generate in the library output.

  When updating a library, select the Ignore column for attributes you do not wish to compare to the target library. For the remaining attributes, select either the **Favor Library** or **Favor Source**

check boxes to resolve conflicting attributes. If attribute values differ in the source and library, selecting **Favor Source** updates the attribute, whereas selecting **Favor Library** does not. Attributes in the source that are not found in the library are added to the library if the **Ignore** column is cleared.

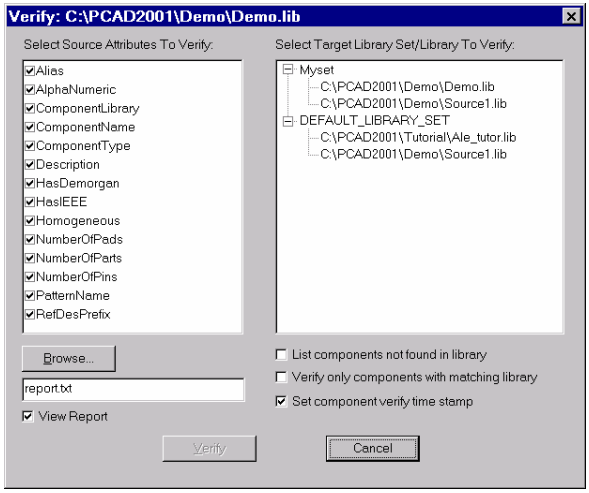
File Verify

The **File » Verify** command opens the *Verification* dialog. From this dialog, the components of the imported external source file can be compared to those of a P-CAD library and a difference report is generated.

The **Verify** command compares component attributes only. Attributes of the attached symbols and patterns are not included in the difference report.

The *Verify* dialog may also be accessed through the **Table** menu on the *Viewer* dialogs or from the shortcut menu in the Source Browser for query results, cross link results, imported files, or P-CAD libraries.

The *Verify* dialog appears.



The *Verify* dialog allows you to check for differences between selected attributes and a P-CAD library or library set. You can specify the attributes to verify, the target components and output options.

- **Select Source Attributes to Verify**: When the attribute in the **Select Source Attributes to Verify** check box is selected, the attribute is compared to the attributes of the components in the target library or library set.

- **Select Target Library Set/Library to Verify**: When the library or library set in the **Select Target Library Set/ Library to Verify** box is selected, the components in the library or library set are compared to the component attributes of the source.
- **Browse**: Click the **Browse** button to navigate to and select the directory and file name of the output difference report. The path to the desired file can also be typed directly into the box.
- **View Report**: Select the **View Report** check box to display the difference report file on the screen.
- **List components not found in library**: If selected, the components in the source file are listed in the difference report when a component with matching ComponentName is not found in the library.
- **Verify only components with matching library**: If selected, the source attributes are compared only in components that have identical ComponentName and ComponentLibrary field values.

  If the selected source does not have a *ComponentLibrary* attribute, this check box is ignored. All attributes in the selected source are verified.

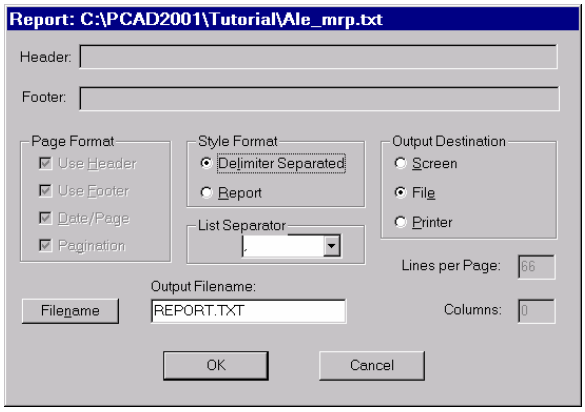
- **Set component verify timestamp**: When verifying a library, the component's time stamp attribute VerifyDate, if present, is automatically updated with the current date and time. If the Set component verify timestamp check box is selected and the VerifyDate attribute does not exist, the VerifyDate attribute is added to the component library.
- **Verify**: Click the **Verify** button to generate the difference report.

Refer to *Library Verification (page 137)* for more information.

File Report

The **File » Report** command opens the *Report* dialog where you can generate a report on the imported external source file. The *Report* dialog may also be accessed through the **Table** menu on the *Viewer* dialogs or from the shortcut menu in the Source Browser for query results, cross link results, imported files, or P-CAD libraries.

The *Report* dialog appears.



The *Report* dialog allows you to define the format of the report. You can specify the report format, the report style format and output options.

- **Header/Footer**: When the **Report Style Format** is selected, you can enter any header or footer text you want to appear on the query output.
- **Page Format frame**: Use Header and Use Footer include the information you specified in the header and footer dialog fields. Date/Page includes the current date and the page number. Pagination allows you to create your own pagination (lines per page). Page Format is activated when the Report Style Format is selected.
- **Style Format frame**: Delimiter Separated puts all data in separated list format. This format can be imported into other spreadsheet and database programs. **Report** produces a report format with columns and spaces, etc.
- **Output Destination frame**: The options in the Output Destination frame determine the source of your output. Screen sends the output to a file and opens the file using the selected File Viewer. File sends the output to a file. Printer sends the output directly to the current printer.
- **List Separator**: Choose a separator character from the list or enter the character in the box.
- **Lines per Page**: Allows you to specify the number of lines per page in your query report.
- **Column**: Allows you to specify the number of columns per line.
- **Output Filename**: Displays the name of the report if you choose to save it to a file. The default filename is `library.rpt`. Click the **Filename** button to change the default name.
- **Filename**: Click the **Filename** button to designate a report name other than the default name. The filename appears in the Output Filename box.

Generating a Report File

To generate a report on a query result, cross link result, an imported file, or a P-CAD library:

1. Choose the **Report** command to open the *Report* dialog.

Other reports can be generated by selecting the item (query result, cross link result, imported file, or P-CAD library) in the Source Browser tree and selecting **Report** from the shortcut menu.

2. Select the desired report format check box in the Style Format frame.

   If you choose **Delimiter Separated**, a character-delimited report output is generated automatically, with formatting specifications.

   If you choose **Report**, an ASCII report file is generated. The ASCII report is more readable and formatting specifications can be applied to the Page Format, Header, Footer, Lines per Page and Columns. Choose the desired page formatting by selecting the appropriate check boxes.

3. Select the desired file output destination, **Screen** or **File**.

   If **File** is selected, define the Output Filename by typing the name in the box or clicking the **Filename** button and navigating to the desired file.

4. Click **Generate**. The report file is generated and either saved to the specified file, displayed directly on the screen, or printed.

File Exit

The **File » Exit** command closes Library Executive.

If you have an unsaved component, you will be prompted as to whether you want to save changes before you exit the program.

CHAPTER 15

Library Commands

Using the Library Commands

With the Library commands you can create new libraries, add new item aliases, copy items from one library to another, delete library items, or rename library items. You can also update older version libraries, translate Tango format libraries into P-CAD format, and merge pattern and symbol libraries into integrated libraries.

P-CAD libraries combine patterns, symbols and components. These sections of a library represent all of the information for a component, such as component pin assignments and other electrical information, and the graphical pattern and symbol structure. Patterns and symbols referenced by a component must reside in the same library as the component.

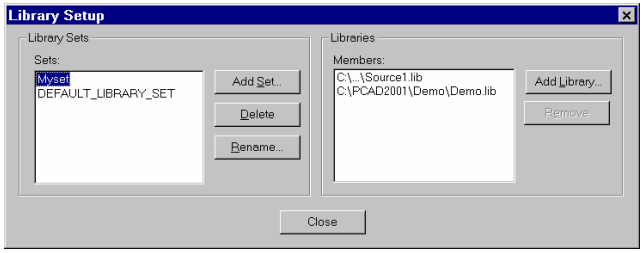
Library New

Library » New allows you to create a new, empty library.

Library Setup

A library set is a group of P-CAD library files. You can perform several Library Executive operations, such as query and verify, directly on a library set by choosing the **Library » Setup** command.

You can organize and manage library file names and library sets in the *Library Setup* dialog.



The *Library Setup* dialog allows you to define the libraries contained within a library set.

- **Library Sets frame**: Lists the presently defined library sets. Selecting a library set from the Library Sets list displays the libraries contained in the set, in the Members list. Within the Library Sets frame, the following activities are available:
  - **Add Set**: Allows you to specify the name of a new library set.
  - **Delete**: Deletes the selected library set. Deleting a set removes the set and the libraries it contains.
  - **Rename**: Allows you to rename the selected library set.
- **Members frame**: Displays the libraries that are contained within a selected library set. Within the Members frame the following activities are available:
  - **Add Library**: Adds a library to the selected set
  - **Remove**: Removes the selected library from the library set.

To be usable in P-CAD Library Executive, all available libraries must be in one or more library sets. A default library set, called `DEFAULT_LIBRARY_SET`, includes any open library files that do not have a set specified explicitly.

Library sets defined in the *Library Setup* dialog are saved in the initialization file, `cmp.ini`, for the next time you run Library Executive.

You can add, rename and delete selected libraries and library sets directly from the Source Browser by choosing **Delete** from the shortcut menus.

Library Alias

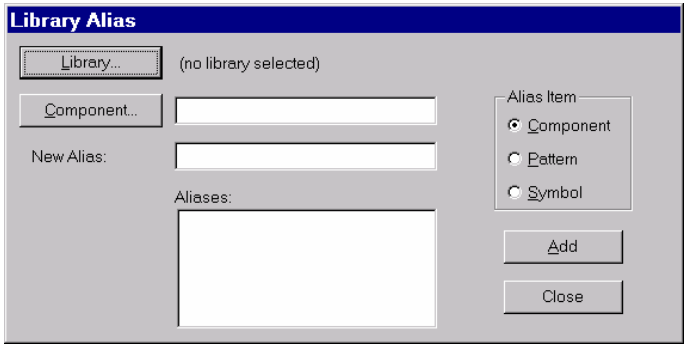
An *alias* is an alternate name for a component, pattern or symbol. You can create multiple names for the same item with the **Library » Alias** command. Aliases allow you the flexibility of using a variety of naming conventions for components, patterns, or symbols without renaming them. For example, for what P-CAD calls an SN7400N, you may want to use a generic alias of 7400. Or, if you have components from a vendor using a particular naming convention, and you want to continue using that system, you can use alias names and display them on your design as such.

An alias is equivalent to creating a new item except the actual data is only referenced not copied. When an alias is created for an item, it is not the same as creating copies or renaming. For copying or renaming, see the respective Library commands.

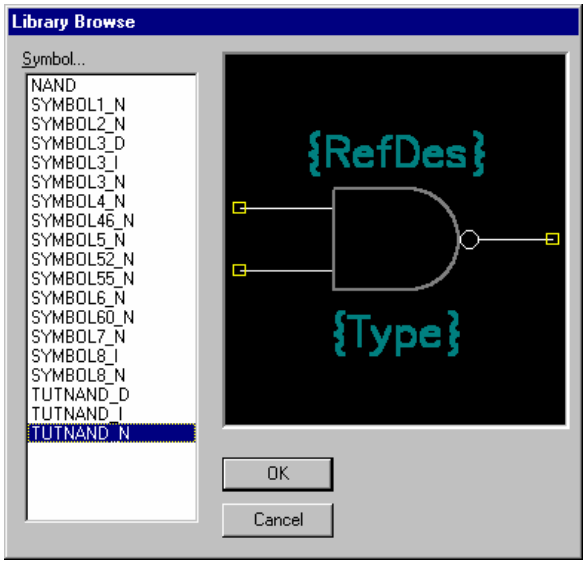
The library that you use in the execution of **Library » Alias**, **Library » Delete**, or **Library » Rename** will remain current if you invoke any of the commands during the same Library Executive session.

Creating an Alias

1. Choose **Library » Alias** to open the *Library Alias* dialog.



2. Select the **Component** or **Pattern** or **Symbol** option button in the Alias Item frame.
3. If the appropriate library is not current, click the **Library** button to navigate to and select the library you want, click **OK** and the *Alias* dialog is redisplayed.
4. Click the button below the **Library** button to open the *Library Browse* dialog where you can select the desired Component, Pattern or Symbol. Select the desired item and click **OK**.



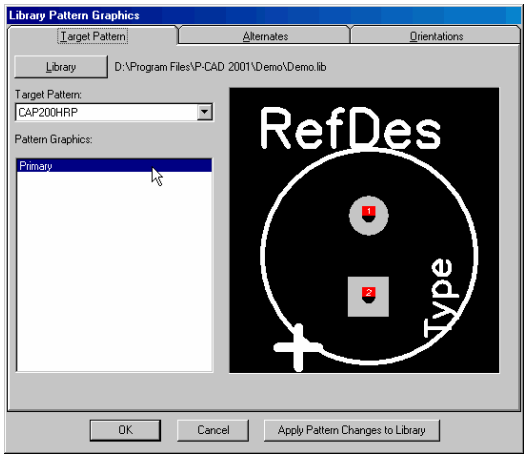
5. When you return to the *Library Alias* dialog, the Pattern, Component, or Symbol you chose is displayed in the Pattern, Component, or Symbol list. Enter the desired alias name in the *New Alias* box. Click **Add** and the new alias is listed in the Aliases list.

Library Pattern Graphics

When you choose **Library » Pattern Graphics**, the resulting *Library Pattern Graphics* dialog allows you to open a source library, choose a target pattern to which you want to add additional pattern graphics and then search through a list of all patterns in the library and select a pattern graphics to attach to the target pattern. This assignment of pattern graphics within a library can only be carried out using Library Executive. Orientations of the pattern graphics can also set here.

Target Pattern Tab

This tab of the *Library Pattern Graphics* dialog allows you to nominate a target pattern from a selected source library, for the purpose of attaching additional pattern graphics.



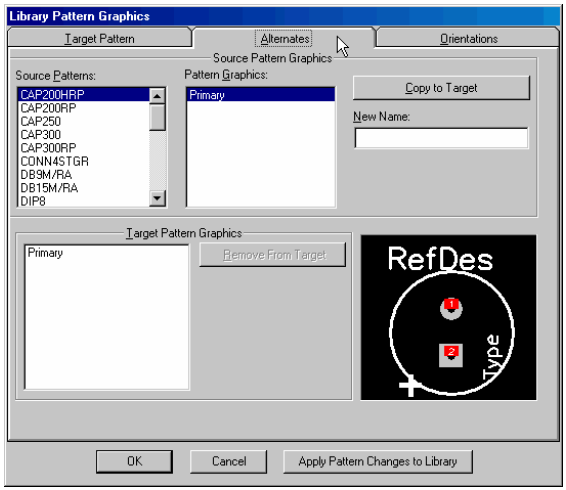
The following fields are available:

- **Library Button:** Click this button to launch the *Library Select* dialog, from where you can browse to open the required library. This is your selected source library. The name of the opened library is displayed to the right of the button.
- **Target Pattern:** This field shows the current, nominated target pattern, from the source library. The target pattern is the pattern that you will attach additional pattern graphics to. The dropdown list contains all the patterns in the selected source library. By default, the first pattern in the library is entered into the Target Pattern field. To nominate a different pattern, just click on an entry in the list.
- **Pattern Graphics List Window:** This list window shows all of the pattern graphics attached to the nominated pattern in the Target Pattern field. The default pattern graphic for a pattern is called 'Primary'. As more pattern graphics are added to the target pattern, this list updates to show all pattern graphics that are currently attached.

- **Pattern Graphics Visual Window:** This window gives a graphical representation of the pattern graphic selected in the Pattern Graphics list window. The default display is for the Primary pattern graphic.

Alternates Tab

This tab of the *Library Pattern Graphics* dialog allows you to browse through all of the patterns in the source library and attach any of the pattern graphics associated with these patterns to your nominated target pattern.



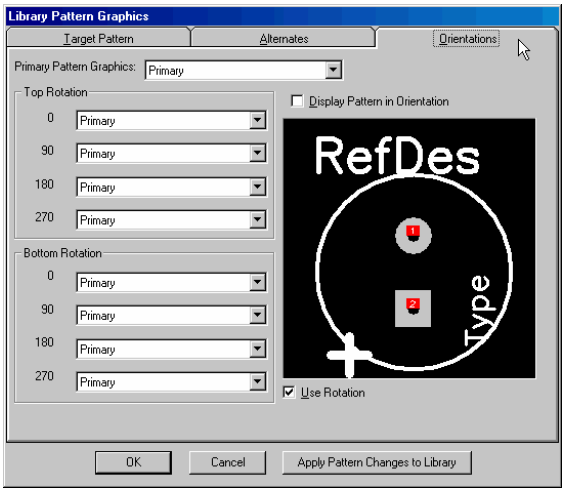
The following fields are available:

- **Source Pattern Graphics Frame:** This frame is used for browsing the patterns in the source library for pattern graphics that you want to attach to the target pattern. The available fields are:
  - **Source Patterns:** This scrollable list window contains all of the patterns within the source library. You can browse the pattern graphics that are attached to each pattern by selecting that pattern from the list. The attached pattern graphics are listed in the Pattern Graphics list window.
  - **Pattern Graphics:** This window lists all of the pattern graphics associated with the pattern selected in the Source Patterns window. Click on an entry to highlight it, if you wish to attach it to your target pattern.
  - **New Name:** This field shows the current name for the selected source pattern graphic. By default, the name is the same as the name of the graphic selected in the Pattern Graphics window list. If you want to call the pattern graphic you are attaching by a different name, just type the new name into the field, overwriting the default.
  - **Copy to Target Button:** Click this button to attach the selected source pattern graphic to your nominated target pattern graphic.

- **Target Pattern Graphics Frame:** This frame contains a window that displays all of the pattern graphics attached to the target pattern. As pattern graphics are copied from the source patterns, using the Copy to Target button, they appear in this window. If you decide that a certain pattern graphic that has been copied to the target is incorrect or not required, it can be removed using the Remove From Target button.
- **Pattern Graphics Visual Window:** This window gives a graphical representation of the pattern graphic selected. It is shared by both the Source Pattern Graphics and Target Pattern Graphics frames. By default, the Primary pattern graphic of the first pattern in the Source Patterns list is displayed.

Orientations Tab

This dialog allows you to specify which pattern graphics are to be used for a specific orientation.



The dialog contains the following fields:

- **Primary Pattern Graphics:** This field allows you to select which pattern graphics are the primary (or default) pattern graphics for the pattern. The drop down list shows all of the associated pattern graphics for the pattern. The default entry is 'Primary'. You can select any of the pattern graphics in the list to change the default primary pattern.
- **Top and Bottom Rotation Frames:** This frame contains editable fields for selecting which of the associated pattern graphics will be used for specific rotations. Up to eight different pattern graphics can be used to represent various component orientations, depending on whether the component is flipped (Bottom Rotation) or unflipped (Top Rotation) and whether it is rotated by 0°, 90°, 180° or 270°. By default, all fields will be filled with the name of the selected Primary pattern graphics. To select a different pattern graphic for a rotation, left-click on an entry in the drop down list.

The Primary pattern graphic will be used for all other rotations.

- **Display Window**: The display window on the right hand side of the tab gives a graphical representation of the pattern graphic selected. The window is shared by the Primary Pattern Graphics field and the individual rotation fields. Selecting an entry in any of these fields will display the corresponding pattern graphic in the window.
- **Display Pattern in Orientation:** This check box allows you to view the selected pattern graphic from the Rotation frames in terms of its actual orientation. If you check this box, you will be able to see how the graphic will appear when the component is placed with a particular rotation. If this box is unchecked, you will see the default, Top Rotation, 0° entry.
- **Use Rotation:** This check box allows you to enable/disable the orientations section of the page, for editing purposes. This is especially useful if you don't want to work on all specific rotations. If the box is checked, all of the specific rotation fields are enabled and you can set specific pattern graphics to specific orientations. If this box is unchecked, the Top Rotation 0° field is available only. All other specific orientation fields are disabled. This command, enabled or disabled, only applies whilst you are actually in the Orientations tab.
- **Apply Pattern Changes to Library Button:** When you have made all of the changes that you want to make to the pattern graphics orientations, click the Apply Pattern Changes to Library button to update and save to the source library, the changes you have made to the target pattern.
- **OK and Cancel Buttons:** Use the OK button to close the dialog after having updated and saved the pattern to the source library using the Apply Pattern Changes to Library button. Use the Cancel button to exit the dialog without effecting any changes made.

Library Copy

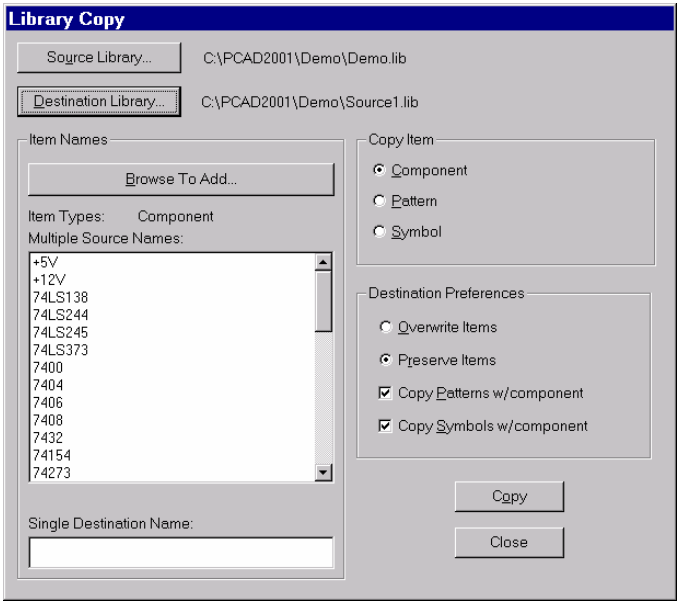
The **Library » Copy** command copies an item from one file to another. You can copy more than one object at a time from one library to another.

When you copy a component, you can also copy the pattern and symbol that it references.

The source and destination libraries designated remain current for the length of the P-CAD session.

Copying a Library Item

1. Choose **Library » Copy** to open the *Library Copy* dialog.



2. Click **Source Library** to open the *Library Select* dialog.
3. Select the source library and click **OK**.
4. Click **Destination Library** to open the *Library Select* dialog again.
5. Select the destination library and click **OK**. The *Library Copy* dialog displays the paths and filenames of the source and destination libraries you selected.
6. In the Copy Item frame select the type of object you want to copy: **Component, Pattern or Symbol**. The names of the available objects for the selected type appear in the Multiple Source Names box.
7. To select items to copy, choose one of the following key combinations:
   - **Left click** selects a single object. When a single object is selected, you can enter its destination name in the Single Destination Name box.
   - **SHIFT+Left Click** selects all objects between the last selected object and the next object you click on.
   - **CTRL+Left Click** adds or removes an object from the selection.
8. Click the **Browse to Add** button to add selected, single objects to the your selection and click **OK**.
9. Select your Destination Preferences: Overwrite Items in the destination file, Preserve Items to maintain the existing objects, Copy Patterns w/component or Copy Symbols w/component. The last two check boxes are available only when copying components.

10. Click **Copy**. The selected objects are copied from the source library to the destination library.

> If you copy an alias into a destination library, Library Executive creates a new component from that alias. The destination library now contains an alias that is an actual component.

Library Delete

The **Library » Delete** command removes a selected library item.

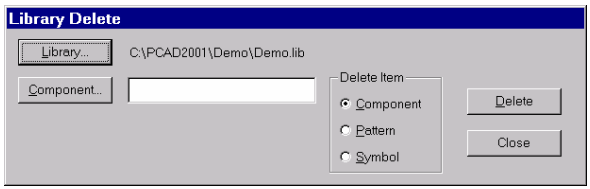
This command deletes the item in name only, if it has aliases. The alternate names (aliases) still exist unless you delete them. If an item has only one name and it is deleted, then the item itself is deleted from the library. The **Library » Alias** command provides a way to check an item for an alias.

The library that you designate in the execution of **Library » Alias**, **Library » Delete**, or **Library » Rename** remains current for the length of the Library Executive session.

> If you delete a pattern or symbol, the components in the library that reference that pattern or symbol become incomplete and cannot be placed. To prevent this, delete only a pattern alias or symbol alias.

Deleting a Library Item

1. Choose **Library » Delete** to open the *Library Delete* dialog.



2. Click the **Library** button to open the *Library Select* dialog where you can navigate to and select the library from which you want to delete an item. The selected library is displayed in the *Library Delete* dialog.
3. Select the desired **Delete Item** option button (**Pattern, Component** or **Symbol**) and the items within the displayed library will be listed in the *Library Browse* dialog. Select one to list it in the Component, Pattern or Symbol box of the *Library Delete* dialog.
4. Click the **Delete** button and the item box becomes blank.

   To continue deleting items, follow steps 3 and 4 until you have deleted all the desired items from the library.
5. Click **Close** to exit the dialog. If you click **Close** before **Delete**, the dialog closes and any pending delete is ignored.

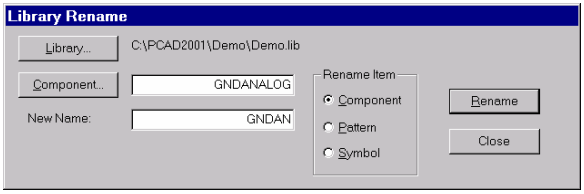
Library Rename

The **Library » Rename** command renames a pattern, symbol, or a component.

> If you rename a pattern or symbol, all components in the library that reference that pattern or symbol by the original name become incomplete and cannot be placed. To use a different naming convention for a pattern, symbol, or component, create an alias for the pattern or symbol by choosing the **Library » Alias** command and use that alias name.

Renaming a Library Item

1. Choose the **Library » Rename** command to open the *Library Rename* dialog.



2. Click the **Library** button to open the *Library Select* dialog, where you can navigate to and select the desired library.
3. In the Rename Item frame, select the type of item to be renamed (**Pattern, Component** or **Symbol**). Those types of items within the library are listed in the *Library Browse* dialog.
4. Select an item to display it in the item list of the *Library Rename* dialog.
5. In the New Name box, type the new name of your item.
6. Click **Rename**. Both the old name and new name disappear if the rename action is successful. You can continue renaming items in the same library by following steps 3 through 5.
7. Click **Close** to exit the dialog. If you click **Close** before **Rename**, then the dialog closes and ignores the pending rename.

Library Translate

Translates libraries in the following formats into P-CAD binary and ASCII-formatted libraries.

- P-CAD Binary
- P-CAD ASCII
- TangoPRO Binary
- TangoPRO ASCII
- Tango-Schematic (DOS)

- Tango-PCB (DOS)
- PDIF

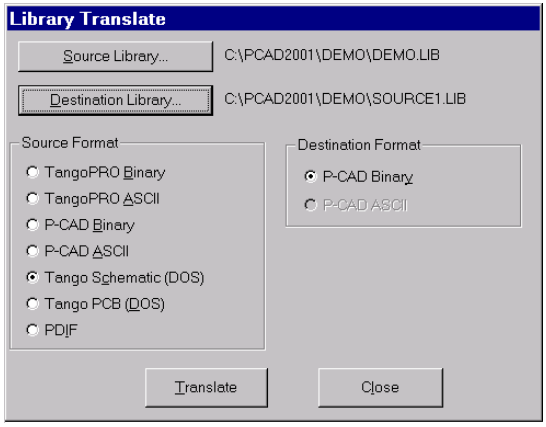
P-CAD uses one integrated library that contains components, symbols and patterns. These integrated libraries are used by all P-CAD applications.

PCB makes certain checks when components are added to a PCB design. In particular, the symbols, pattern and pin attributes like gate equivalence found in the library for a component are compared to those for any components of the same type already present in the design. If the components are not the same, PCB reports that the components do not match and does not allow you to add the component to the design. The following recommendations should be considered when adding components:

- PCB designs created using previous versions of P-CAD PCB or P-CAD PCB (6/400) using pattern libraries should be loaded into PCB with only pattern libraries open.
- If you encounter a component cache error after using Library Executive to change a component that is already placed in a PCB design, you must update all occurrences of that component in the design. This must be done before you can place any more instances of that component. Choose the **Force Update** command to replace all occurrences of a component of a single type.
- The **Maintain Rotation** option in **Utils Force Update** does not maintain rotations for components in designs loaded from P-CAD PCB (6/400).

Using the Library Translate Command

1. Choose **Library** » **Translate** to open the *Library Translate* dialog.



2. Click the **Source Library** button to open the *Library File Listing* dialog, where you can navigate to and select the library to translate.
3. Click the **Destination Library** button to open the *Library File Listing* dialog, where you can navigate to and select destination library.

4. Select the desired **Source Format** option button.

   If you select P-CAD Binary, select a **Destination Format** option button.

5. Click **Translate** to begin the translation process. When the library has been translated, an error log file is generated and automatically displayed in the Notepad utility. The name of this file is `Filename.err`, where **FILENAME** is the name of the output library.

6. Click **Close** to exit the dialog.

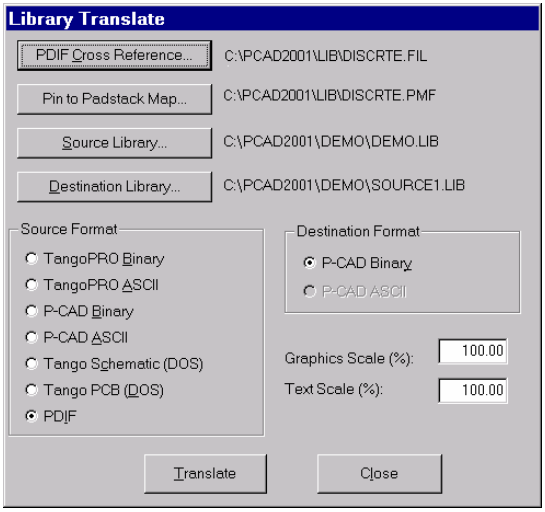
PDIF Translation

This section provides the steps you need to follow to create a fully integrated P-CAD library from a PDIF symbol library and its corresponding PDIF part library.

Translating PDIF Symbol Libraries

To create an integrated P-CAD library from PDIF symbol and part libraries, follow these steps:

1. Choose **Library** » **Translate** to open the *Library Translate* dialog.



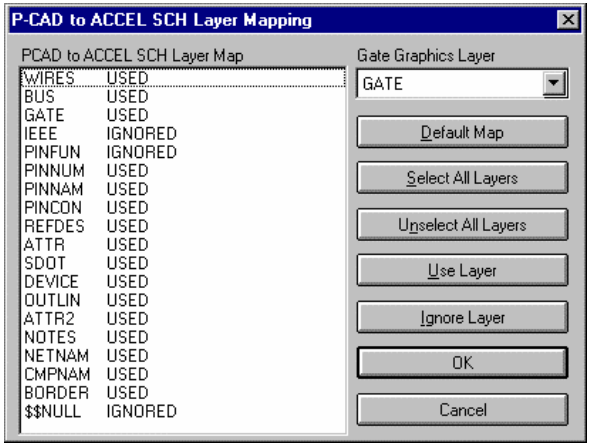
2. In the Source Format frame, select **PDIF**. The **PDIF Cross Reference** and **Pin to Padstack Map** buttons appear in the dialog.

   The **PDIF Cross Reference File** contains power and ground pin assignments. It also attempts to group heterogeneous components and is not required for translating libraries. If you don't specify a cross reference file, an error message appears, indicating that the number of components pins created for a component does not match the number of entries created for that component in the PDIF file and that power pins might be missing.

   The **Pin to Padstack Map** button lets you select a pin mapping file (`.pmf`) that contains pin mapping information used to translate P-CAD PDIF part library files. P-CAD Library Executive

uses this pin mapping file to change padstack information to P-CAD pad styles. The pin mapping file is not used when translating symbol libraries.

3. Click **Source Library**. The *Library File Listing* dialog opens where you can select the PDIF files available in the current directory.
4. Select the desired PDIF file and then click **OK**. The *Library Translate* dialog reappears and displays the name of the source library next to the **Source Library** button.
5. Click **Destination Library**. The *Library File Listing* dialog opens where you can type the path and name of the destination library.
6. Click **OK**. The *Library Translate* dialog reappears and displays the name of the destination library next to the **Destination Library** button.
7. Click **OK** again to open the *Master Designer to P-CAD SCH Layer Mapping* dialog.



8. Select the desired layers and map them accordingly.
9. Click **OK**.

   While loading, P-CAD Library Executive displays several messages indicating the system's progress. After translating the library, a message box appears indicating any errors or warnings. If there are none, you can click **Close**.

Translating PDIF Part Libraries

Master Designer PDIF part libraries are translated just as you translate symbol libraries. The difference is that you select a PDIF part library file as your source library, instead of selecting a symbol library.

> The layer status of the first component ever entered in a part library sets the status for all other parts in that part library.

You can also map pin types in the PDIF Part library file to the pad styles during translation. This is useful for Master Designer users who want to bring padstack information into P-CAD Library Executive. To translate Master Designer binary part libraries, do the following:

- Start the PIN2PAD utility in Master Designer. This program generates a pin mapping file (`.pmf`).
- Load the pin mapping file into P-CAD Library Executive. This step maps the pin types to pad styles in P-CAD Library Executive.

The PIN2PAD utility is a DOS program that is started from the command line. To start it you will need:

- an external ASCII aperture table file (optional)
- a special symbol file (`.ssf`)
- a Master Designer binary part library file (`.plb`)

To run the PIN2PAD utility, at the DOS prompt, enter:

```
pin2pad <options> -s <special symbol file> -p <library file>
```

where <options> are:

<Options>	Description
-a	specifies the aperture table file
-h	displays the help screen
special symbol file	specifies the special symbol file
library file	specifies the P-CAD binary part library file

If you do not use an aperture table when running PIN2PAD, the program only translates the graphic information over to P-CAD.

When you run PIN2PAD, the program creates the aperture section of the pin mapping file, which occurs only if you specify an aperture table file. Next, the program reads the special symbol file for the mapping between pin types and pad stacks. For each pad stack, the program loads the binary padstack file and writes the layer and graphic data. PIN2PAD also maintains a list of all the layers in all the pad stacks. The program uses the part library file's database unit as the database unit for the pin mapping file. If some padstacks in the special symbol file have different database units, PIN2PAD converts these database units to the part library file's database unit.

PIN2PAD displays a warning message if any of the following occurs:

- Any pin type that exists in the library file is not specified in the special symbol file.
- PIN2PAD is missing, unable to locate or unable to load a pad stack file.
- Apertures are specified in the pad stacks but not in the aperture table.

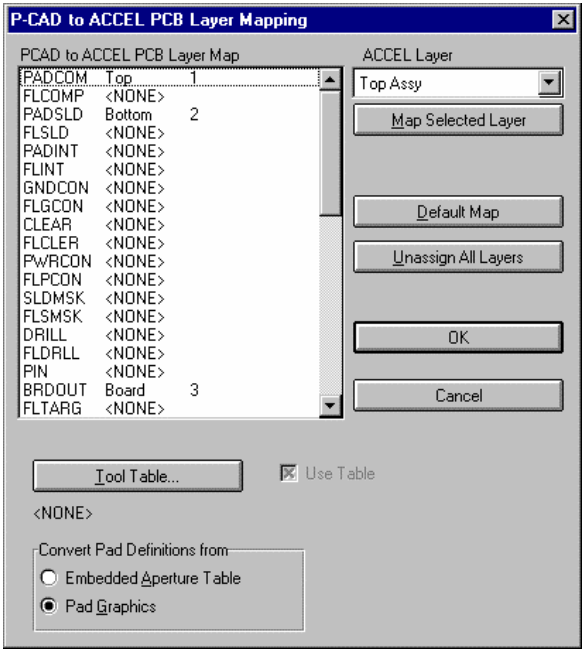
PIN2PAD saves the pin mapping file using the part library file's name. For example, if you run the program using the part file `Parts.plb`, the resulting pin mapping file is named `Parts.pmf`.

Now that you've created the pin mapping file, load it into Library Executive by following these steps:

1. Start **Windows** and then start **P-CAD Library Executive**.
2. Choose **Library » Translate** to open the *Library Translate* dialog.
3. In the *Source Format* frame select **PDIF**.

   The **PDIF Cross Reference** (`.fil`) and **Pin to Padstack Map** (`.pmf`) buttons appear.

   The **Pin to Padstack Map** button lets you select a pin mapping file (`.pmf`) that contains pin mapping information that the program uses for translating Master Designer PDIF part library files.
4. Click the **Pin to Padstack Map** button to open the *Library File Listing* dialog.
5. Select the desired pin mapping file, then click **OK** to return to the *Library Translate* dialog.
6. Click the **Source Library** button to open the *Library File Listing* dialog that displays the PDIF files available in the current directory.
7. Select the desired PDIF file, then click **OK**. The *Library Translate* dialog reappears and displays the name of the source library next to the **Source Library** button.
8. Click **Destination Library** to open the *Library File Listing* dialog.
9. Enter the name of the destination library, then click **OK**. The *Library Translate* dialog reappears and displays the name of the destination library next to the **Destination Library** button. Remember not to overwrite a symbol library with the same first name.
10. Click **OK**. The *Master Designer to P-CAD PCB Layer Mapping* dialog appears.



11. Select the desired layers and map them accordingly. Then click **OK**. The translation of the P-CAD PDIF part library begins.

    While translating, several messages are displayed indicating the system's progress. During translation, the program loads the pin mapping file, creates the new pad styles and maps the appropriate pins to the new pad styles. After translating the library, a message box appears indicating if there were any errors or warnings. If there are none, you can click **Close**.

Mapping Layers in P-CAD Library Applications

Mapping layers makes it easier for you to control the data that you carry across by selecting the desired layers. For each layer you select, the system translates the data on that layer. For each layer you don't select, the system ignores the data on that layer.

When mapping layers in Library Executive, you need to be aware of a few points:

- The system always maps the REFDES layer and brings over the data on that layer.
- Some library data has visibility control and some does not. Graphic objects, such as circles, lines and rectangles, do not have visibility control and cannot be hidden or displayed in Library Executive. For these objects, the data is translated, mapped to the appropriate layer and displayed in your design. Other data, such as such as reference designator, component type and value, have visibility control. In this case, the data is translated, mapped to the appropriate layer, and hidden or displayed based on the layer visibility setting in Master Designer. For example, if the first component ever entered in the library has an attribute on the ATTR layer in Master Designer and that layer's visibility is set to ON or ABL, Library Executive translates this

data and displays it. On the other hand, if the ATTR layer is set to OFF, Library Executive sets this visibility to OFF.

- When you translate a PDIF symbol library, the *Master Designer to P-CAD SCH Layer Mapping* dialog is opened. This dialog lets you select which layers to map and maps the desired layers.

Layer	Description
Gate Graphics Layer	Lists the target layers available for normal gate representation.
Default Map	Assigns all Master Designer layers to a default layer mapping structure.
Select All Layers	Selects all layers set to MAPPED or IGNORED.
Unselect All Layers	Deselects all layers set to MAPPED or IGNORED.
Set Layer Mapped	Instructs Library Executive to map this layer and translate its data.
Set Layer Ignore	Instructs Library Executive to ignore this layer and any data on it.
OK	Confirms your selections.
Cancel	Cancels the layer mapping and aborts loading the schematic design.

When you translate a PDIF part library, the *Master Designer to P-CAD PCB Layer Mapping* dialog opens.

Master Designer to P-CAD PCB Layer Map: Displays the Master Designer layer mapping assignments. A <*NONE*> next to the Master Designer layer name indicates that PCB will ignore data on that layer. A typical layer assignment looks like this:

```
PADCOM Top SA 1
```

where:

- PADCOM is the Master Designer layer.
- Top is the PCB layer assigned to the Master Designer layer PADCOM.
- S indicates the layer's status (S=signal, N=nonsignal, P=plane).
- A indicates the autorouter bias (A=auto, H=horizontal, V=vertical).
- 1 indicates the layer number.

P-CAD Layer: Lists the PCB layers available for mapping.

Map Selected Layer: Assigns the selected Master Designer layer to the selected PCB layer.

Create New Layer: Opens the *Options Layers* dialog so you can create a new layer or modify an existing one.

Default Map: Maps the PCB design signal to signal, plane to plane, and non-signal to non-signal to non-signal to non-signal. This option saves you having to create layers manually. The default layer map set varies, depending on whether you select embedded aperture tables or pad graphics to convert your pad definitions.

Auto Map All Layers: Automatically maps all unassigned Master Designer layers to PCB layers of the same name. If a layer does not have <*NONE*> next to it, PCB leaves the current assignment.

Unassign All Layers: Assigns <*NONE*> to all the Master Designer layers in the list.

Tool Table: Lets you select which tool table to use. Tool tables specify the hole size to use for each pad type in the converted design.

Use Table: Indicates whether or not to use the Tool Table. PCB needs the tool table to apply the correct hole sizes. If you don't specify a tool table, PCB uses a default hole size for the pad types.

Polygons: Indicates to convert polygons to polygons. You should select this option if you never ran the **Merge Polygon Voids** command on the Environment Menu in Master Designer 8.5.

Copper Pours: Instructs PCB to convert polygons to copper pours. Selecting this option automatically creates thermal ties to pads and vias in the same net. You should select this option if you want to preserve the voids in your design.

Embedded Aperture Table: Instructs PCB to convert pad definitions from the design's embedded aperture table.

Pad Graphics: Instructs PCB to convert pad definitions from the design's pad graphics.

Important Layer Mapping Tips

This section provides some important tips for mapping Master Designer layers to P-CAD layers. It also explains how Library Executive handles layers for which mapping is not recommended, as well as user- defined Master Designer layers.

Component Layers

- **Component Silk Layers (SLKTOP, SLKSCR)**: Map these layers to the appropriate P-CAD top or bottom silk layers, Top Silk or Bottom Silk.
- **Assembly or Fab Information (DEVICE)**: Map these layers to the appropriate P-CAD top or bottom assembly layers.

Pad Layers

- **Pad Shapes on Signal Layers (FLSOLD, FLCOMP, PINFTP)**: If your Master Designer pad shapes are defined using aperture flashes, map the flash layers to Top or Bottom, as appropriate. If your pad shapes are defined using graphics, map the graphics layers to Top or Bottom.
- **Paste, Mask and Drill Layers**: Library Executive does not map these layers automatically because the solder mask swell and paste shrink sizes are determined automatically from the pad shape.

- **Drill Symbol Layers (DRILL, FDRILL)**: Map these layers to <*none*>. Information on these layers is not used in pad definitions. Tool codes and Drill symbols can be assigned for each hole size for N/C Drill and Drill Symbol output in the PCB editor.
- **Paste and Solder Mask Layers (MSKGTP, SLDMSK, MSKGBT)**: Library Executive maps these layers to Top Mask or Bottom Mask. It is recommend that you map these layers to <*none*>.

Plane Layers

Plane layers are not mapped in Library Executive, so you get the default thermal on every plane layer that you create in your PCB design. Map these layers to <*none*>.

Recognizable but unmapped Layers

Layers that are recognized, but recommended not mapped, are mapped to <none>. It is recommended that you map internal signal layers to <*none*>.

Unrecognizable Layers

Any user-defined layers that are not in the list of standard Master Designer layer names are mapped to <*ignored*>. For instance, if you create a user-defined Master Designer layer TEST1, Library Executive maps this layer to <*ignored*>.

Merging the Libraries

To merge the libraries, choose **Library » Merge Patterns**. Select as the Source Libraries the **PCB** pattern libraries to be merged into the Destination Schematic library.

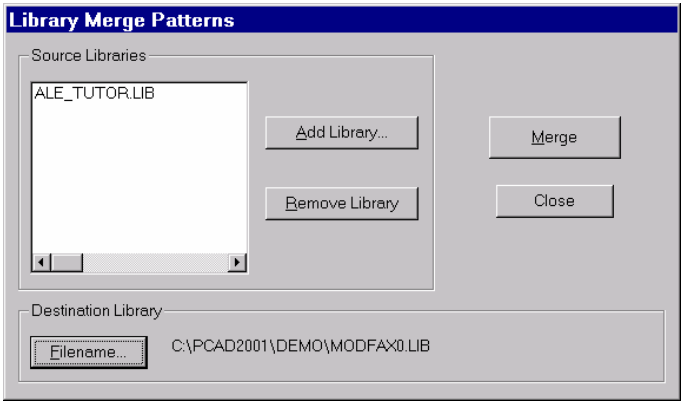
Once the libraries have been merged into an integrated P-CAD library, you can delete the pattern-only library or use it for PCB placement if you don't need integrated library components.

Library Merge Patterns

The **Library » Merge Patterns** command copies patterns from one or more P-CAD libraries containing patterns into another P-CAD library that contains pattern references, but not actual patterns. This command is useful for merging entire libraries. If you want to integrate an individual component, use the **Library » Copy** command.

After you translate a Tango Schematic symbol library and Tango pattern library to P-CAD format using the **Library » Translate** command, the **Library » Merge Patterns** command can be used as a way to merge the translated libraries into a single integrated library.

1. Choose **Library » Merge Patterns** to open the *Library Merge Patterns* dialog.



2. Click the **Add** Library button to open the *Library File Listing* dialog, where you can choose libraries containing patterns. The library names you add appear in the Source Libraries list. Select all the pattern libraries that contain the patterns for the components in the destination library.

   Select a library and click the **Remove Library** button to remove a library from the Source Libraries list.

3. Click the **Destination Library** button to open the *Library File Listing* dialog, where you can choose the destination library. This is typically a symbol library that has been converted to P-CAD format. The destination library shouldn't contain any patterns.

4. Click **Merge** to begin the process. For every component in the destination library, the program searches the source libraries for the corresponding pattern. The libraries are searched in the order they appear in the list.

   If any patterns exist in the destination library a warning message appears. If you choose to continue, these patterns will not be replaced with patterns from the source libraries during the merge process. However, the pin mapping will be updated with the information from the source libraries.

5. Click **Close** to exit the dialog.

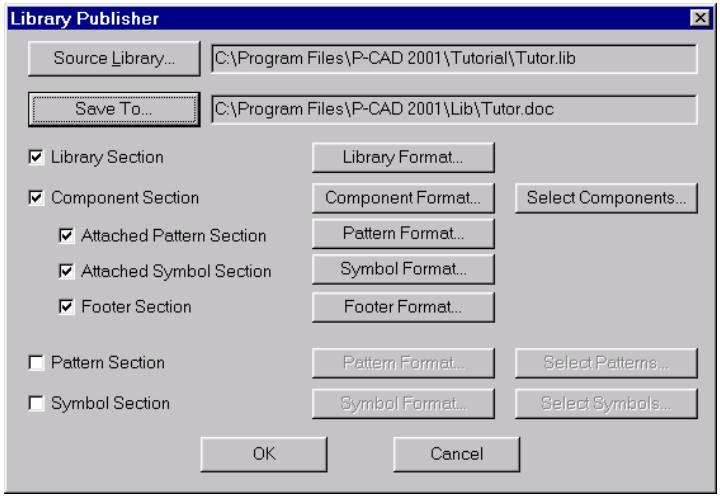
Library Publisher

Library » Publisher generates complete, customizable reports directly in Microsoft® Word™. These reports can include pictures of the patterns and symbols in the library.

Microsoft® Word 97™ is required for the *Library Publisher* utility.

The Library Publisher Dialog

When you choose **Library » Publisher**, the *Library Publisher* dialog opens.



This dialog is the point of access for the content and format options of the Library Publisher. These options are summarized below:

- **Source Library**: Click the **Source Library** button to navigate to and select the directory and filename of the library you want to publish.
- **Save To**: Click the **Save To** button to select the directory and filename of the output file.
- **Library Section**: Select the **Library Section** check box to include a report on the selected library in the publisher output. The **Library Format** button becomes available, allowing you to select the contents of this section, including file name, size, component name list, etc. Refer to *Formatting Options (page 232)* for additional details.
- **Component Section**: Select the **Component Section** check box to include a report on the components in the selected library in the publisher output. Several additional options become available, including the **Component Format** button, **Select Components** button, **Attached Pattern Section** check box, **Attached Symbol Section** check box and **Footer Section** check box.
- The **Component Format** button allows you to select the contents of this section, including component name, pin table, attribute table, etc. Refer to *Formatting Options (page 232)* for additional details.
- The **Select Components** button allows you to select which components from the selected library will be included in the report output. Refer to *Selecting Components, Patterns and Symbols (page 154)* for additional details.
- If you select the **Attached Pattern Section**, **Attached Symbol Section**, or **Footer Section** check boxes, the appropriate section is added to your publisher output. For instance, checking the **Attached Pattern Section** box includes a report of the pattern immediately after the report on the component to which it is attached. The footer section is global, so if this check box is selected, the selected attributes are included in the footer of all pages in the publisher output.

If any of these sections are included, their corresponding format button becomes available. Refer to *Formatting Options (page 232)* for additional details on the *Pattern Format*, *Symbol Format* and *Footer Format* dialogs.

- **Pattern Section**: Select the **Pattern Section** check box to include a report on the patterns in the selected library in the publisher output.
- The **Select Patterns** button becomes available, allowing you to select the patterns included in this section. All selected patterns from the library are included whether or not they are attached to any library components. Refer to *Selecting Components, Patterns and Symbols (page 154)* for additional details.
- The **Pattern Format** button becomes available allowing you to select the contents of this section, including pattern name, attribute table, a picture of the pattern, etc. Refer to *Formatting Options (page 232)* for additional details.
- **Symbol Section**: Select the **Symbol Section** check box to include a report on the symbols in the selected library in the publisher output.
- The **Select Symbols** button becomes available allowing you to select the symbols included in this section. All selected symbols from the library are included, whether or not they are attached to any library components. Refer to *Selecting Components, Patterns and Symbols (page 154)* for additional details.
- The **Symbol Format** button becomes available allowing you to select the contents of this section, including symbol name, attribute table, a picture of the symbol, etc. Refer to *Formatting Options (page 232)* for additional details.

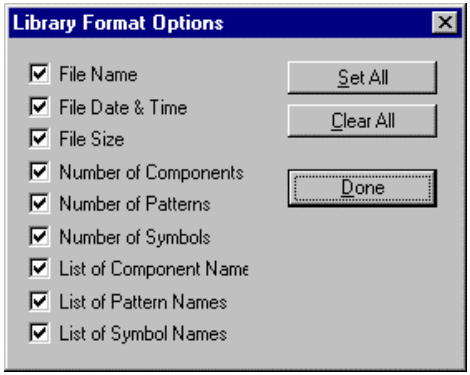
Formatting Options

If you choose to include a section in your published output by selecting its check box, you can modify the contents of the section by clicking its corresponding **format** button. The format options available for each output section are detailed below.

Library Publisher uses Word's `Normal.dot` template as a basis for the published output. To select the font type, font size and page layout for the published library, modify your `Normal.dot` template accordingly.

Library Format

When including the Library Section, you can click the **Library Format** button to open the *Library Format Options* dialog.

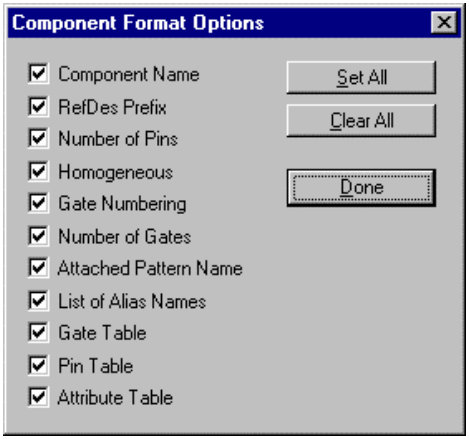


This dialog allows you to specify the contents of the library section. These options are described below:

- **File Name**: The full path and filename of the library being published.
- **File Date & Time**: The modified date and time of the library.
- **File Size**: The size of the library.
- **Number of Components**: The number of components contained in the library.
- **Number of Patterns**: The number of patterns contained in the library.
- **Number of Symbol**: The number of symbols contained in the library.
- **List of Component Names**: The names of the components contained in the library.
- **List of Pattern Names**: The names of the patterns contained in the library.
- **List of Symbol Names**: The names of the symbols contained in the library.
- **Set All**: Selects all options.
- **Clear All**: Clears all option selections.
- **Done**: Saves the selected options and returns to the *Library Publisher* dialog.

Component Format

When including the Component Section, click the **Component Format** button to open the *Component Format Options* dialog.

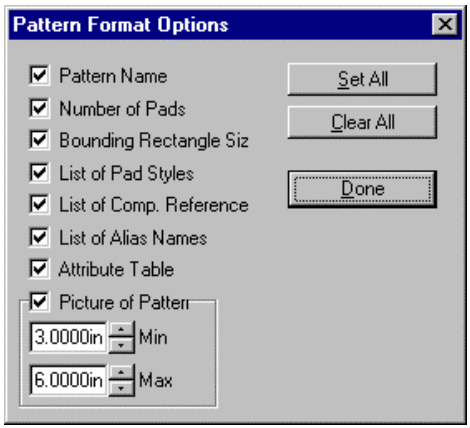


This dialog allows you to specify the contents of the component section. These options are described below:

- **Component Name**: The name of the component.
- **RefDes Prefix**: The reference designator prefix.
- **Number of Pins**: The number of pins on the component.
- **Homogeneous**: Whether or not the component is homogeneous. In a homogeneous component, all gates use the same symbol.
- **Gate Numbering**: Whether the gate numbering in the reference designator is alphabetic or numeric.
- **Number of Gates**: The number of gates in the component.
- **Attached Pattern Name**: The name of the pattern attached to the component.
- **List of Alias Names**: The alias names of equivalent components in the same library.
- **Gate Table**: Publishes the Gate Table for the component. This table is displayed in the *Component Information* dialog and includes the following information: Gate #, Gate Eq, Normal, IEEE and DeMorgan. Refer to *View Commands (page 255)* for additional details.
- **Pin Table**: Publishes the Pin Table for the component. This table is displayed in the *Pins View* dialog and includes the following information: Pin #, Gate #, Sym Pin #, Pin Name, Gate Eq, Pin Eq and Elec Type. Refer to *View Commands (page 255),* for additional details.
- **Attribute Table**: Lists the component attributes and their values.
- **Set All**: Selects all options.
- **Clear All**: Clears all selected options.
- **Done**: Saves the selected options and returns to the *Library Publisher* dialog.

Pattern Format

When including a Pattern Section in your published library, click the **Pattern Format** button to open the *Pattern Format Options* dialog.



This dialog allows you to specify the contents of the pattern section. The pattern section check box can be selected in two locations. If you select the **Attached Pattern Section** check box within the Component Section, the pattern section will directly follow the component section of the component to which it is attached. If you select the **Pattern Section** check box outside the Component Section, the pattern section will follow after the component section for all components and contain all selected patterns in the library consecutively, independent of their associated component.

The options on the *Pattern Format Options* dialog are:

- **Pattern Name**: The name of the pattern.
- **Number of Pads**: The number of pads in the pattern.
- **Bounding Rectangle Size**: The dimensions of the selection box surrounding the pattern graphics.
- **List of Pad Styles**: Lists all of the pad styles used in the current pattern.
- **List of Camp References**: A list of the component names in the current library that have this pattern attached.
- **List of Alias Names**: The alias names of equivalent patterns in the same library.
- **Attribute Table**: Lists the pattern attributes and their values.
- **Picture of Pattern frame**: Select this check box to include a picture of the pattern in the published output. The **Min Size** and **Max Size** boxes become available where you can select the size range of the picture output.

The minimum size sets the lower bound. If the pattern is smaller than this minimum size, it is enlarged. The maximum size sets the upper bound. If the pattern is larger than this maximum size, it is reduced.

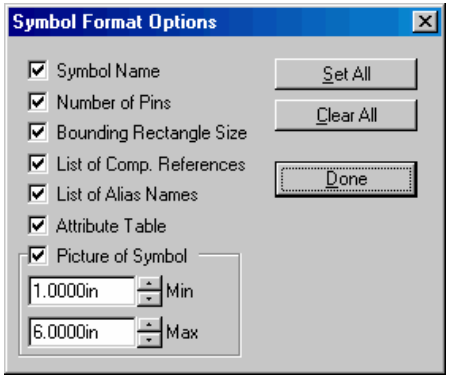
If the pattern picture is currently within the selected minimum - maximum range, it is fit to the next larger ½ inch increment. For example, a picture with a bounding rectangle size of 4.21 inches, would be fit to 4.5 inches.

The pattern pictures can be resized by selecting and modifying them in Word™. See your Microsoft® Word 97™ documentation for details.

- **Set All**: Selects all options.
- **Clear All**: Clears all selected options.
- **Done**: Saves the selected options and returns to the *Library Publisher* dialog.

Symbol Format

When including a Symbol Section in your published library, click the **Symbol Format** button to open the *Symbol Format Options* dialog.



This dialog allows you to specify the contents of the Symbol Section. The Symbol Section check box can be selected in two locations. If you select the **Attached Symbol Section** check box within the Component Section, the Symbol Section will directly follow the Component Section (and attached Pattern Section, if selected) of the component to which it is attached. If you select the **Symbol Section** box outside the Component Section, the symbol section will follow after the Component Section (and Pattern Section, if selected) for all components and contain all of the selected symbols in the library consecutively, independent of their associated component.

The options in the *Symbol Format Options* dialog are:

- **Symbol Name**: The name of the symbol.
- **Number of Pins**: The number of pins in the symbol.

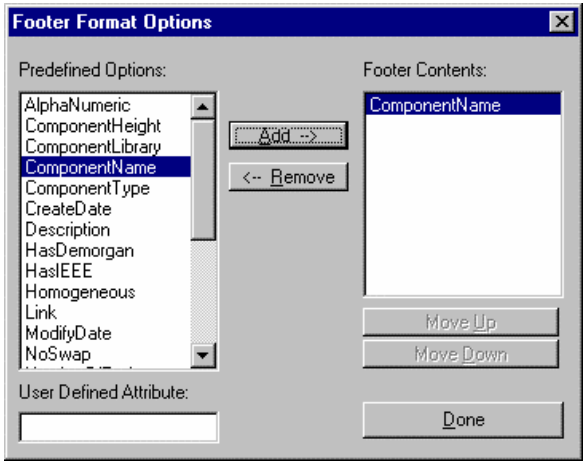
- **Bounding Rectangle Size**: The dimensions of the selection box surrounding the symbol graphics.
- **List of Comp References**: A list of the component names in the current library that have this symbol attached.
- **List of Alias Names**: The alias names of equivalent symbols in the same library.
- **Attribute Table**: Lists the symbol attributes and their values.
- **Picture of Symbol**: Select this option to include a picture of the symbol in the published output. The **Min Size** and **Max Size** boxes become available and you can select the size range of the picture output.

  The minimum size sets the lower bound. If the symbol is smaller than this minimum size, it is enlarged. The maximum size sets the upper bound. If the symbol is larger than this maximum size, it is reduced.

  If the symbol picture is currently within the selected minimum - maximum range, it is fit to the next larger ½ inch increment. For example, a picture with a bounding rectangle size of 4.21 inches, would be fit to 4.5 inches. The symbol pictures can be resized by selecting and modifying them in Word™. See your Microsoft® Word 97™ documentation for details.
- **Set All**: Selects all options.
- **Clear All**: Clears all selected options.
- **Done**: Saves the selected options and returns to the *Library Publisher* dialog.

Footer Format

When including the Footer Section, click the **Footer Format** button to open the *Footer Format Options* dialog.



This dialog allows you to specify the footer contents displayed on each page of the component section.

To set the Footer Format Options:

1. Select a component or attribute from the list of Predefined Options.
2. Click **Add** to move the selection to the Footer Contents box.

   To include an attribute that is not predefined, type the attribute name in the User Defined Attribute box. Click **Add**. The attribute will appear in the Footer Contents box. The attribute value for the particular component is found in the library and listed in the Footer Section. You can also double-click the attribute to add it.

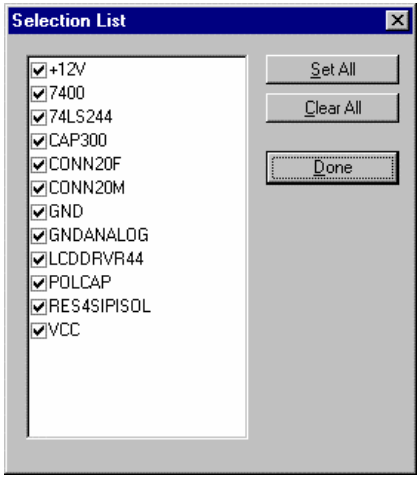
   To remove an attribute from the Footer Contents box, select the attribute and click **Remove**. You can also double-click the attribute to remove it.

   Text can be included in the footer. For example, the words "Components Designed By:" can be a footer for your published document. To include text, type the text into the User Defined Attribute box. Click **Add**. Since the name is not recognized as an attribute, an attribute value will not appear in the footer.
3. The footer contents are the attributes and their values listed in the order that they appear in the Footer Contents box. These attributes appear in order on the bottom of the pages in the component section from left to right.

   To organize the Footer Contents box, select the property or attribute you wish to move, click **Move Up** or **Move Down**. These commands rearrange or modify the contents of this box by moving the selected attribute higher or lower in the list.
4. Click **Done**. The properties and attributes in the Footer Contents box are used to create a footer for the published library document.

Selecting Components, Patterns and Symbols

When a Component, Pattern, or Symbol section is included in your published library, you can select the items you want to include. The **Select Components**, **Select Patterns** and **Select Symbols** buttons display their corresponding *Component*, *Pattern* or *Symbol Selection List* dialog. A sample selection list dialog appears.



The box lists all components, patterns, or symbols in the selected library. If selected, the corresponding component, pattern, or symbol is included in the published output.

To select or clear all check boxes, click the **Select All** or **Clear All** buttons. When the desired items are selected, click **Done**. The selected items will be published in their Component, Pattern, or Symbol Section.

The selection dialog is not available for attached patterns and symbols in the Component Section. If **Attached Pattern Section** or **Attached Symbol Section** is selected, the patterns and symbols published are those attached to the selected component(s).

Generating Publisher Output

You can generate the published library by clicking **OK** in the *Library Publisher* dialog after having specified the Library Publisher output options: filenames, sections, formatting and item selection.

Microsoft® Word 97™ works in the background to generate a published library document of the specified name. Open the document in Word™ to view, make changes, or to print. To open the document, you can select the file name in the recently used file list on Word's **File** menu.

If you run Word™ in the foreground after the Library Publisher has been started, be careful not to quit the Word™ application. Exiting Word™ before Library Publisher has finished will halt the publishing process.

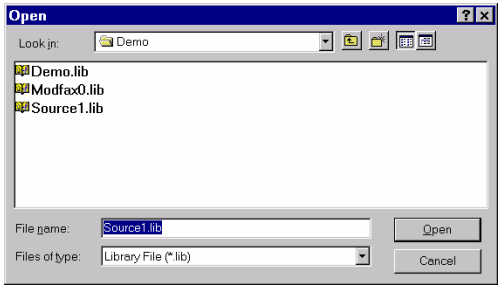
CHAPTER 16

Component Commands

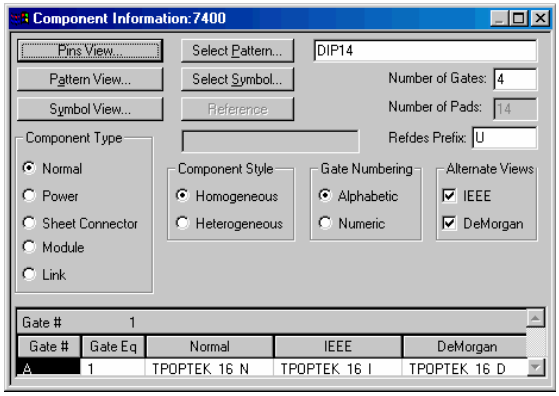
The Component commands allow you to create new components, open existing components, save components and validate components.

Component New

The **Component » New** command allows you to create a new component for later addition into a component library. When you choose the **Component » New** command or click the toolbar button, the *Open* dialog appears.



Navigate to and select a library where the new component will be stored and in which the component's symbol and pattern information reside. When you click **Open**, the *Component Information* dialog appears.



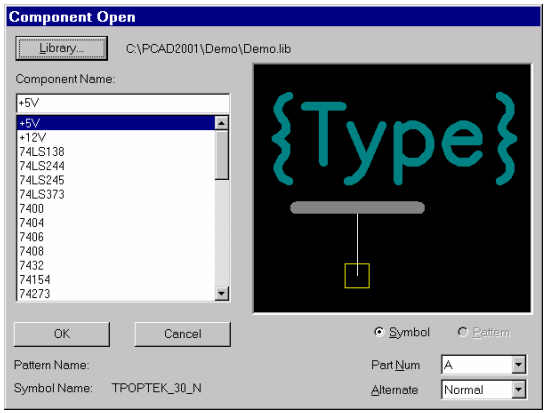
From the *Component Information* dialog you can attach a pattern and set the component type, the number of gates in the component, alternate representations of the component and the Refdes prefix.

Drag and Drop Library Load

You can also drag and drop a library file (`.lib`) into Library Executive. From the File Manager, click on a filename icon, drag it into the P-CAD window and release. The library file is loaded and the path established for use with the **Component** » **New** or **Component** » **Open** commands.

Component Open

The **Component** » **Open** command opens an existing component from a library. This command allows you to access a component for the purposes of attaching different patterns and symbols, changing pin assignments, pin equivalents, etc. for components you have created or for viewing the component data. When you choose the **Component** » **Open** command or click the toolbar button, the *Component Open* dialog appears.



If the library you want to access is not current (does not appear in the Library field), click the **Library** button to navigate to and select the desired library. You can also drag and drop a library file (`.lib`) into a P-CAD Library application.

Select the desired component and click **OK**. The *Component Information* dialog opens and displays information for the component. It also provides access to the different views of the component where you can attach symbols or patterns and edit component properties.

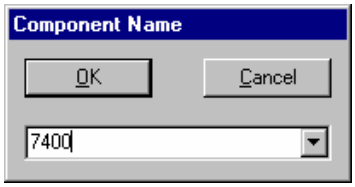
Component Save

The **Component » Save** command saves the current component to its associated library file. This command is also accessible by clicking the **Component Save** toolbar button.

Component Save As

The **Component » Save As** command saves the component to a different library or with a different component name.

When you choose **Component » Save As**, the *Component Name* dialog opens.



Enter the new name for this component or select one from the list.

Component Validate

The **Component » Validate** command verifies all fields for valid entries and reports any errors.

You cannot save a component using the **Save** or **Save As** command if an error exists.

CHAPTER 17

Pattern Commands

The Library Executive Pattern commands accesses the P-CAD Pattern Editor. In the Pattern Editor, you can quickly create a new pattern using the Pattern Wizard or edit an existing pattern. These patterns can, at some future time, be attached to a design component.

All Pattern commands are accessible in Library Executive only. The Pattern menu accesses the P-CAD Pattern Editor, an application that simplifies and automates pattern creation. The Pattern Editor is partnered with the Library Executive and is one of the many P-CAD Library Executive enhancements.

Pattern New

The **Pattern** » **New** command starts the Pattern Editor program with the Pattern Wizard in the workspace.

Pattern Open

The **Pattern** » **Open** command starts the Pattern Editor program and opens a dialog where you can navigate to and choose the pattern to be edited. The pattern can be in a pattern file (`.pat`) or a component library (`.lib`).

CHAPTER 18

Symbol Commands

The Library Executive Symbol commands accesses the P-CAD Symbol Editor. In the Symbol Editor, you can quickly create a new symbol using the Symbol Wizard or edit an existing symbol. These symbols can, at some future time, be attached to a design component.

All Symbol commands are accessible in Library Executive only. The Symbol menu accesses the P-CAD Symbol Editor, an application that simplifies and automates symbol creation. The Symbol Editor is partnered with the Library Executive and is one of the many P-CAD Library Executive enhancements.

Symbol New

The **Symbol » New** command starts the Symbol Editor program with the Symbol Wizard in the workspace.

Symbol Open

The **Symbol » Open** command starts the Symbol Editor program and opens a dialog where you can navigate to and choose the symbol to be edited. The symbol can be in a symbol file (`.sym`) or a component library (`.lib`).

CHAPTER 19

Edit Commands

The Edit commands are only available after you have created or opened a component (Component New or Component Open).

There are toolbar icon equivalents for Cut, Copy, Paste, Attach Symbols, Attach Pattern and Component Attr.

Edit Undo Spreadsheet Change

The **Edit » Undo Spreadsheet Change** command reverses your last completed action to the *Pins View*, *Pattern View* and *Symbol View* dialogs (but not in the *Component Information* dialog). Many component commands, such as **Component » New** and **Component » Save** cannot be undone. If an action cannot be undone (or there is nothing to undo), the **Undo** command appears grayed on the **Edit** menu.

The **Undo** button on the toolbar and the **CTRL+U** keys are equivalent to the **Edit » Undo Spreadsheet Change** command.

Edit Cut Spreadsheet Selection

The **Edit » Cut Spreadsheet Selection** command moves the data from all selected cells to the clipboard, overwriting the previous clipboard contents. You must have one or more cells selected before you can cut.

Edit » Cut Spreadsheet Selection does not function within the *Component Information* dialog.

The **Paste** (**CTRL+V**) command copies the clipboard data to another cell or cells within the spreadsheet. Data in the clipboard can also be pasted into a commercial spreadsheet where it can be edited and pasted back into Library Executive.

Edit Copy Spreadsheet Selection

The **Edit » Copy Spreadsheet Selection** command copies the data from all selected cells to the clipboard. The original data is not erased from the cells.

Edit » Copy Spreadsheet Selection does not function within the *Component Information* dialog. You must have one or more cells selected before you can copy.

The **Paste** (**CTRL+V**) command copies the clipboard data to another cell or cells. Data in the clipboard can also be pasted into a commercial spreadsheet where it can be edited and pasted back into Library Executive.

Edit Paste Spreadsheet Selection

The **Edit » Paste Spreadsheet Selection** command copies the clipboard data to a cell or cells within the spreadsheet.

Data copied to the Clipboard from a commercial spreadsheet can be pasted into one of the view dialog spreadsheets by choosing the **Edit » Paste Spreadsheet Selection** command.

Edit » Paste Spreadsheet Selection does not function within the *Component Information* dialog.

Edit Slide Selection Up

The **Edit » Slide Selection Up** command moves the selected spreadsheet information up one row. You can use this command to slide cells, partial or entire rows, and one or more complete rows.

Edit Slide Selection Down

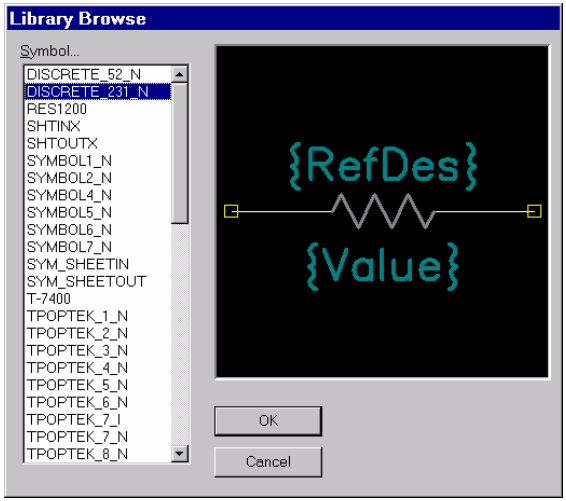
The **Edit » Slide Selection Down** command moves the selected spreadsheet information down one row. You can use this command to slide cells, partial or entire rows, and one or more complete rows.

Edit Select Symbols

The **Edit » Select Symbols** command allows you to select one or more symbols from the current library to attach to the current component. You can attach only symbols that reside in the same library as the component.

This command also can be chosen by clicking the **Select Symbol** button and by double-clicking in either the **Normal** or **Alternate Representations** columns of the *Component Information* dialog.

When you choose the **Edit » Select Symbols** command, the *Library Browse* dialog opens, displaying all symbols in the current library.

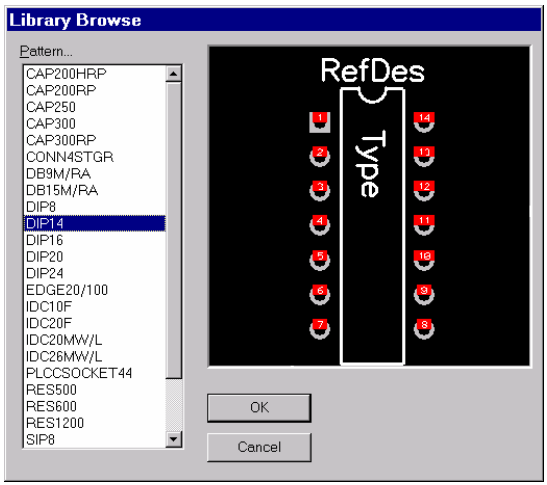


Select a symbol and click **OK**. For homogeneous components, the symbol is filled in for each gate in the component. For heterogeneous components, you need to assign a symbol to each gate.

Edit Select Pattern

The **Edit » Select Pattern** command allows you to select a pattern from the current library and attach it to the current component. You can attach a pattern if it resides in the same library as the component.

When you choose **Edit » Select Pattern** the *Library Browse* dialog opens, displaying all patterns in the current library.

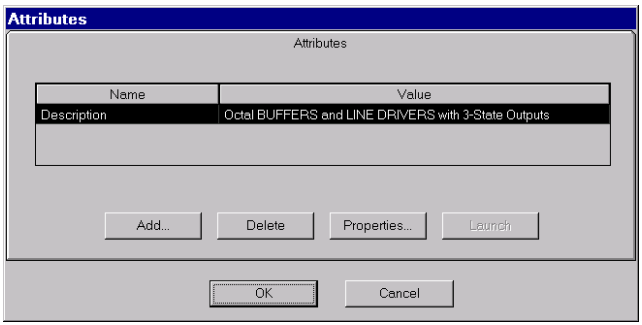


Select a pattern and click **OK**. The Pins View spreadsheet automatically adds a row for each pad in the pattern and automatically fills in the PinDes column.

Edit Component Attribute

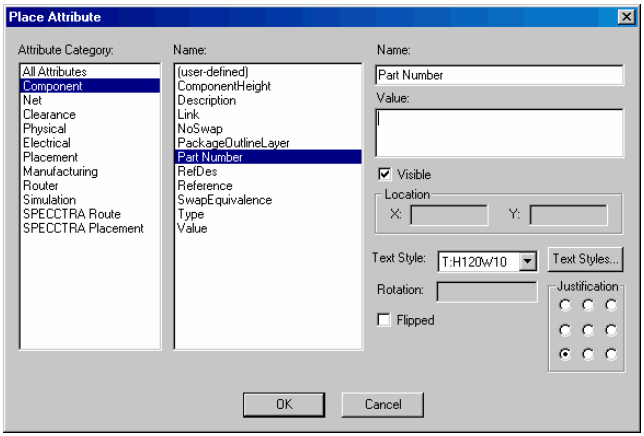
The **Edit » Component Attribute** command allows you to view, add, modify, or delete component attributes and values. If you assign an attribute to a component it is attached to that component when you place it in P-CAD PCB or P-CAD Schematic.

When you choose the **Edit » Component Attribute** command, the *Attributes* dialog opens.



The dialog contains a two-column table showing a collection of component attributes. Within the collection, each attribute's name and value appear in the column. You can view, add, modify, or delete a collection of component attributes and access reference links from the *Attributes* dialog.

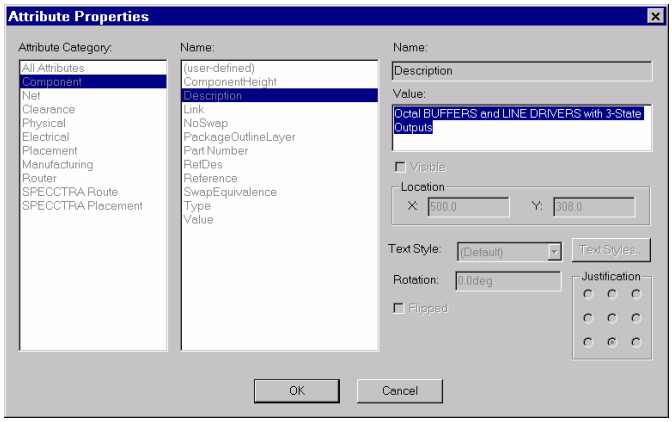
- **Adding an Attribute**: Click the **Add** button to open the *Place Attribute* dialog. Select the desired **Attribute Category** and attribute name from the Name list. If you are adding a (user-defined) attribute, enter its **Name** in the Name box and its value in the Value box. Click **OK** and the attribute is added to the table.



- **Viewing Attribute Properties**: Select an attribute from the table and click the **Properties** button (or double- click the attribute) to open the *Attribute Properties* dialog. This dialog is similar to the *Place Attribute* dialog except that only the *Value* field can be edited here. See *Attribute Property Dialog (page 253)* for more information.
- **Deleting an Attribute**: Select an attribute in the table and click **Delete**, or press the **DELETE** key.
- **Launching a Reference Link**: When the special attribute, Reference, whose value is a reference link, is added to the item, you can select the Reference attribute and click the **Launch** button to start a program, link to a web address or open a document to display the referenced information.

Attribute Property Dialog

The *Attribute Property* dialog, similar to the *Place Attribute* dialog.



The *Value* field is the only field that can be edited for an existing attribute. Information available in the *Attribute Properties* dialog includes:

- **Category List Box**: Lists the attribute categories: All, Component, Net, Clearance, Physical, Electrical, Placement, Manufacturing, Router and SPECCTRA.
- **Name List Box**: Lists the pre-defined attributes for the specified category. The currently-selected attribute also appears in the Name box, unless User-defined is selected. In that case, the Name box displays the user-defined attribute name.
- **Name Edit Box**: Displays the user-defined name of the attribute.
- **Value**: Enter a value for the attribute. See *Attributes and Their Values (page 254)* for an explanation of how component attribute values are placed in PCB and Schematic designs.
- **Visible**: This check box indicates whether or not the attribute is visible.
- **Location**: This frame shows the **X** and **Y** coordinates of the component's reference point.

- **Text Style**: The attribute text style.
- **Rotation**: Shows the rotation amount if the pattern has been rotated.
- **Flipped**: This check box indicates whether or not the pattern has been flipped.
- **Justification**: Shows the reference point of the text string. For example, if the middle button is enabled, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

Attributes and Their Values

The values given to Attributes become part of a design whenever the component is placed there. The following rules dictate which value is used when a component is placed in a PCB or Schematic design:

- If a component attribute has a value, that value is used when the component is placed in either a PCB or Schematic design.
- If a component attribute has no value, but the same named attribute in the pattern has a value, the pattern attribute's value is placed in a PCB design.
- If a component attribute has no value, but the same named attribute in the symbol has a value, the symbol attribute's value is placed in a Schematic design.

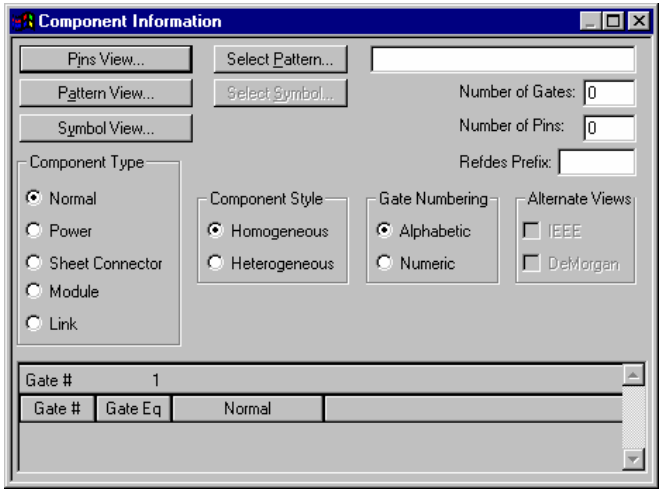
CHAPTER 20

View Commands

The View commands allow you to access the four modeless dialogs that provide different views of the current component. Additional view commands let you show or hide the toolbar and prompt line areas. The details of these features are provided below.

View Component Info

The **View » Component Info** command opens the *Component Information* dialog.



From the *Component Information* dialog, you can attach a pattern and set the component type, the number of gates in the component, alternate representations of the component and the RefDes prefix.

When creating a component, you can enter a minimum of information before entering more detailed information in one of the other dialogs. For example, the number of pins must be entered

before proceeding to the *Pins View* dialog, and the number of gates must be entered if you want to edit multiple gates from the *Symbol View* dialog.

Depending on how much information you provide, some entries are filled in for you:

- When you attach a pattern, pad numbers are supplied.
- Pin Des values are filled in when you attach a symbol or a pattern whose Default Pin Designators have been specified.
- When you select a symbol, gate numbers and symbol pin numbers are automatically added.
- Gate equivalence is automatically applied in Pins View.

Pins

If there is no attached pattern, then you can fill in the Number of Pins. If a pattern is attached, the Number of Pins is supplied. Attaching gates automatically increases the number of pins if no pattern is attached.

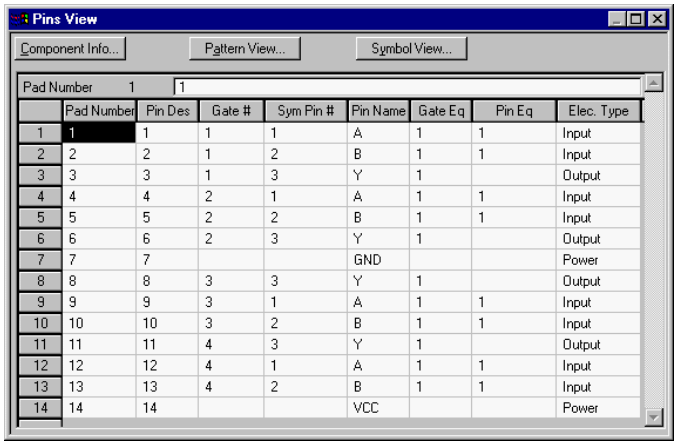
Gates

If you set the Component Style to **Homogeneous**, then all Gate Eq values are set to the first Gate Eq value.

Gate numbers appear in order, either alphabetically or numerically. If the number of gates is greater than 0, you can select an attached symbol by clicking the **Select Symbol** button. IEEE and DeMorgan equivalents are available if those check boxes are selected in the Alternate Views frame.

View Pins View

The **View » Pins View** command opens the *Pins View* dialog where you can assign pin data.



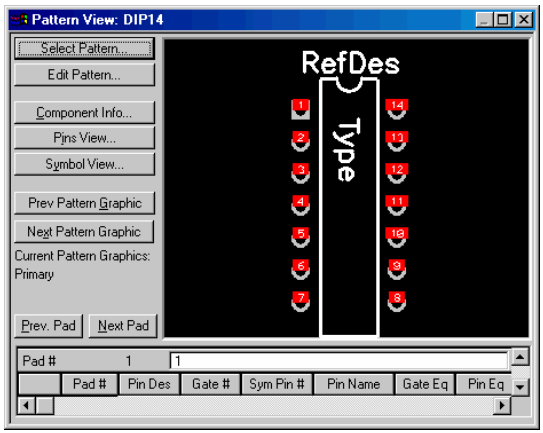
	Pad Number	Pin Des	Gate #	Sym Pin #	Pin Name	Gate Eq	Pin Eq	Elec. Type
1	1	1	1	1	A	1	1	Input
2	2	2	1	2	B	1	1	Input
3	3	3	1	3	Y	1		Output
4	4	4	2	1	A	1	1	Input
5	5	5	2	2	B	1	1	Input
6	6	6	2	3	Y	1		Output
7	7	7			GND			Power
8	8	8	3	3	Y	1		Output
9	9	9	3	1	A	1	1	Input
10	10	10	3	2	B	1	1	Input
11	11	11	4	3	Y	1		Output
12	12	12	4	1	A	1	1	Input
13	13	13	4	2	B	1	1	Input
14	14	14			VCC			Power

If there is an attached pattern, the information in the current row also appears in the *Pattern View* dialog. The corresponding pad is selected in the *Pattern View* dialog. Changes made to row data are automatically updated in the *Pattern View* dialog.

If there is an attached symbol, the information in the current row of the *Pins View* dialog also appears in the *Symbol View* dialog. The corresponding pin and gate are selected in the *Symbol View* dialog. Changes made to row data are automatically updated in the *Symbol View* dialog.

View Pattern View

The **View » Pattern View** command opens the *Pattern View* dialog where you can assign pin data to a component.



The attached pattern appears in a browse window. The *Pattern View* dialog may be resized to increase the display area of the browse window.

> To expand the dialog vertically to display additional spreadsheet rows that may not be displayed, press the **SHIFT** key while dragging an edge of the dialog to the left or right.

When a pad is selected in the browse window, the following occurs:

- The pad is selected in the Pattern View browse window.
- The pad's corresponding pin is selected in the Symbol View browse window.
- The corresponding spreadsheet row is selected in the *Pattern View* and *Symbol View* dialogs.
- If pad numbers are missing from the rows or misnumbered, the correspondence between rows and pads is not maintained.

Prev Pad/Next Pad

The **Prev Pad** and **Next Pad** buttons automatically select the next and previous pads. The order of the pads is defined by the pad number sequence set during pattern creation. Pressing the **UP** key when

the entire row is selected or clicking the **Prev Pad** button selects the previous pad. Pressing the **DOWN** key when the entire row is selected or clicking the **Next Pad** button selects the next pad. If the pad number is missing or misnumbered, the corresponding pad is not selected.

Pattern Selection

Click the **Select Pattern** button to select the attached pattern. Changes made in this dialog are reflected in the view dialogs.

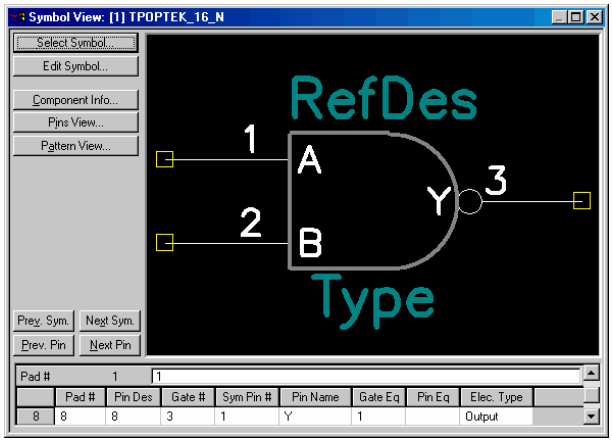
Attaching a Pattern

When you load a pattern into the spreadsheet, you have the option to overwrite the current information with the incoming data if a difference is detected. If you choose to overwrite the spreadsheet data the following conditions are applied:

- Incoming pads whose default pin designators match a row's pin designator are merged with the spreadsheet data and the pad number overwritten. A warning is issued when the pad numbers are different.
- Remaining pads are matched by pad number and the pin designator is overwritten. If pin designators are different a warning is issued.
- Rows already matched and merged are not considered for further matches.
- Unmatched pads are matched to the smallest remaining row number and the pad number and pin designator overwritten. If either the pad number or pin designator is different a warning is issued. This may cause a random sorting of the pad numbers and pin designators.

View Symbol View

The **View » Symbol View** command opens the *Symbol View* dialog where you can assign pin data to a component.



The attached symbol appears in a browse window. The *Symbol View* dialog may be resized to increase the display area of the browse window.

To expand the dialog vertically to display additional spreadsheet rows that may not be displayed, press the **SHIFT** key while dragging an edge of the dialog to the left or right.

When a pin is selected, the following occurs:

- The pin is selected in the Symbol View browse window.
- The pin's corresponding pad is selected in the Pattern View browse window.
- The corresponding spreadsheet row is selected in the *Pattern View*, *Pins View* and *Symbol View* dialogs.
- If pin or gate numbers are missing from the rows or misnumbered, the correspondence between rows and pins is not maintained.

Prev Pin/Next Pin

The **Prev Pin** and **Next Pin** buttons automatically select the next and previous pins within the symbol. The order of the pins is defined by the symbol pin number sequence set during symbol creation.

Prev Sym/Next Sym

If there is more than one gate, the **Prev Sym** and **Next Sym** buttons automatically select the next and previous gates. The **Prev Sym** button goes to the last pin of the previous gate; the **Next Sym** button goes to the first pin of the next gate.

Symbol Selection

Click the **Select Symbol** button to select the attached symbol. Changes made in this dialog are reflected in the other view dialogs.

Attaching a Symbol

When you load a symbol into the spreadsheet, you have the option to overwrite the current information with the incoming data if a difference is detected between the incoming and current pin designator. If you choose to overwrite the spreadsheet data, the following conditions are applied for single gate components:

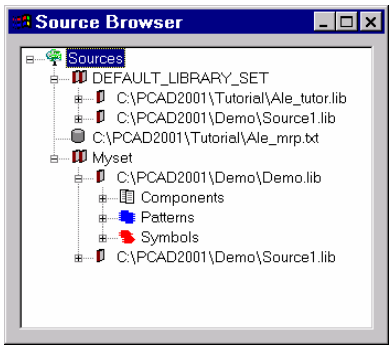
- Incoming pins whose default pin designators match a row's pin designator are merged with the spreadsheet data and the spreadsheet pin name and symbol pin number are overwritten.
- Remaining pins and rows that could not be matched using the pin designator are matched by pin name and the pin designator, symbol pin number and gate number are overwritten.
- Any pins remaining are matched using the symbol pin numbers and the pin designators, pin names and gate numbers are overwritten.

- Finally, any remaining unmatched pins and rows are written to the spreadsheet in the remaining blank rows on a first come, first served basis. No rows will be overwritten unless a match is found. If a pattern is not attached, additional rows are inserted to accommodate the remaining pins.

Multi-Gate components use the same matching criteria used for single gate components, except the gate number is used in conjunction with the matching criteria and is not overwritten unless the gate numbering can not be determined.

View Source Browser

The **View » Source Browser** command opens the Source Browser, shown in the following figure.

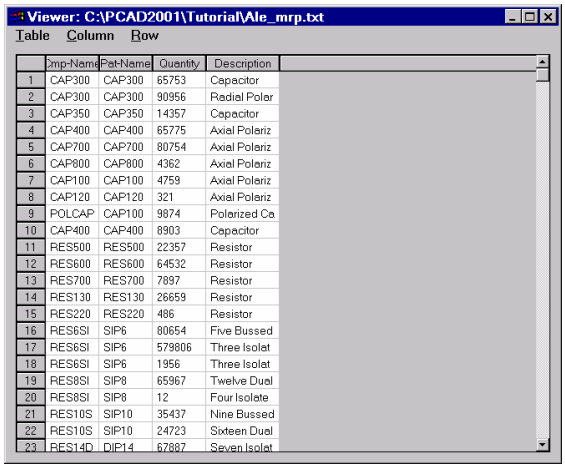


From the Source Browser, all data sources available in Library Executive can be accessed, including P-CAD libraries and library sets, query results, cross link results and an imported external source file.

Refer to *The Source Browser (page 91),* for details on the *Source Browser* and its function.

View Comma-separated File

The **View » Comma-separated File** command opens the Viewer containing the imported external source file.



	Cmp-Name	Pat-Name	Quantity	Description
1	CAP300	CAP300	65753	Capacitor
2	CAP300	CAP300	90956	Radial Polar
3	CAP350	CAP350	14357	Capacitor
4	CAP400	CAP400	65775	Axial Polariz
5	CAP700	CAP700	80754	Axial Polariz
6	CAP800	CAP800	4362	Axial Polariz
7	CAP100	CAP100	4759	Axial Polariz
8	CAP120	CAP120	321	Axial Polariz
9	POLCAP	CAP100	9874	Polarized Ca
10	CAP400	CAP400	8903	Capacitor
11	RES500	RES500	22357	Resistor
12	RES600	RES600	64532	Resistor
13	RES700	RES700	7897	Resistor
14	RES130	RES130	26659	Resistor
15	RES220	RES220	486	Resistor
16	RES6SI	SIP6	80654	Five Bussed
17	RES6SI	SIP6	579806	Three Isolat
18	RES6SI	SIP6	1956	Three Isolat
19	RES8SI	SIP8	65967	Twelve Dual
20	RES8SI	SIP8	12	Four Isolate
21	RES10S	SIP10	35437	Nine Bussed
22	RES10S	SIP10	24723	Sixteen Dual
23	RES14D	DIP14	67887	Seven Isolat

From the Viewer, all Library Executive functions applicable to an imported library file are available, including Query, Cross Link, Verify and Reports. One other important function for imported files, Map Fields, is also available from the Viewer.

Refer to *Library Basics (page 3)* for details on the Viewer and its function.

View Toolbar

The **View » Toolbar** command turns on or off the display of the toolbar. Current visibility is saved to the `cmp.ini` file and restored in subsequent sessions.

The toolbar buttons are shortcuts for commonly-used commands. Once a toolbar is visible on your screen, you can use your mouse to drag it to a new position.

Tool Tips explain each of the toolbar buttons. To activate a Tool Tip, place the mouse over the button. The Tool Tip pops up.

View Custom Toolbar

The **View » Custom Toolbar** turns on or off the display of the *Custom* toolbar.

If no tools have been added to the *Custom* toolbar, the display of the toolbar is turned off. As soon as the first custom tool has been added by choosing the **Utils » Customize** command, the *Custom* toolbar display is turned on. When custom tools exist in the *Custom* toolbar, you can turn off the display of the toolbar by choosing **View » Custom Toolbar**.

View Prompt Line

The **View » Prompt Line** command turns on or off the display of the prompt line. Current visibility is saved to the `cmp.ini` file and restored in subsequent sessions.

CHAPTER 21

Utils Commands

The Utils commands let you move quickly to other P-CAD customized applications.

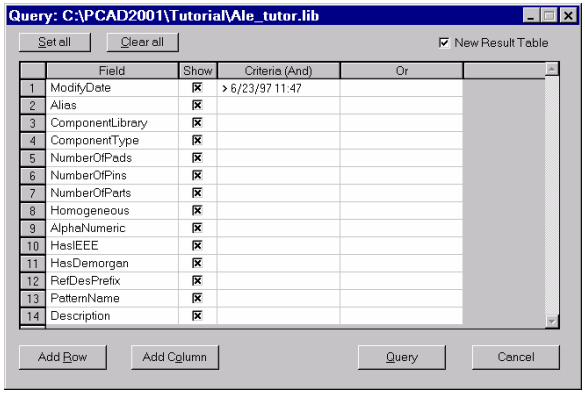
Utils Query

The **Utils » Query** command allows you to search for a wide variety of predefined and user-defined attributes in several sources — a P-CAD library or library set, an external library source file, a cross link result, or the result of a previous query. The **Query** utility can also aid in the search for a component to place into a PCB or Schematic design. The **Query** utility is also embedded directly in the P-CAD PCB and P-CAD Schematic programs.

The *Query* dialog may also be accessed through the **File** or **Table** menus on the *Viewer* dialogs or from the shortcut menu in the Source Browser.

The **Query** command searches for components from the current contents of Library Executive memory, as displayed in the Viewer. Query does not search the original source file. If changes have been made to a library, choose **Reload** to refresh the library components in Library Executive's memory before conducting the Query.

The *Query* dialog is shown in the following figure.



The following options are available in the *Query* dialog to specify search criteria, display fields and output options:

- **Set All/Clear All**: Click the **Set All** button to display all fields defined in the selected source in the **Field** column. Click **Clear All** to remove all fields from the Field column.
- **New Result Table**: Select this check box to generate an additional query result table. If the check box is cleared, the present query overwrites the previous result.
- **Query Spreadsheet**: The Field column contains both the search and display attributes for the query. Select the **Show** check box for each field to be displayed in the query result. The *Criteria (And)* and *Or* columns contain the query search criteria.
- **Add Row/Add Column**: The **Add Row** button adds a row to the query table. The **Add Column** button adds another *Or* column to the *Query* dialog.
- **Query**: Click the **Query** button to begin the search for components satisfying the specified criteria.

Additional information on the Query utility can be found in *Querying Libraries (page 103)*.

Utils Shortcut Directory

At the time you installed your P-CAD product suite, a sub-directory named `ShortcutDirectory` was created in the P-CAD directory. The ShortcutDirectory contains a list of web addresses for semiconductor manufacturers.

To access this directory:

1. Choose the **Utils » ShortcutDirectory** command to open the ShortcutDirectory in Windows Explorer.
2. Select the desired address and choose **Open** from the shortcut menu, or double click on the desired address to start the shortcut.

Shortcuts to any web site can be added to the ShortcutDirectory.

Utils PCB

Starts P-CAD PCB if that program is installed on your computer. Refer to your *P-CAD PCB User's Guide* for additional information.

Utils Schematic

Starts P-CAD Schematic if that program is installed on your computer. Refer to your *P-CAD Schematic User's Guide* for additional information.

Utils Pattern Editor

Starts P-CAD Pattern Editor.

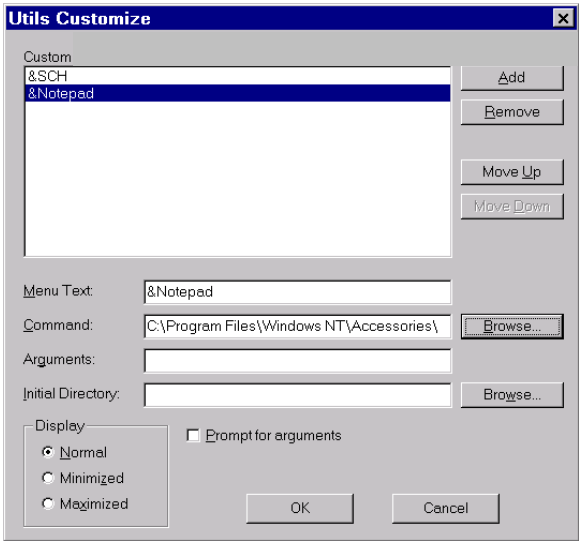
Utils Symbol Editor

Starts P-CAD Symbol Editor.

Utils Customize

The **Utils » Customize** command allows you to define access to other applications from P-CAD Library Executive by adding items to the *Custom Tools* toolbar and the *Utils* menu.

When you select **Utils » Customize**, the *Utils Customize* dialog opens.



The *Utils Customize* dialog is used to add new tools or delete and modify existing tools on the *Custom Tools* Toolbar and **Utils** menu. The fields on the dialog are as follows:

- **Custom Tools**: Lists the custom tools that currently exist.
- **Menu Text**: The description of the tool being added. The *Menu Text* field allows up to 40 characters. The buttons on the toolbar display as many letters as can fit on the button. If the name of a tool being added to the custom toolbar is too long to be displayed in its entirety, you can change the display by using lower case letters or shortening the name of the new tool.

  You may insert an ampersand (&) anywhere in the text string to designate a menu shortcut key. For instance, if the Menu Text entry is &Notepad, the menu shortcut key for the tool is the letter **N**.

The **Utils** menu displays the list of custom tools that have been added directly beneath the **Customize** command.

- **Command**: The path to the executable file of the new tool. Click the **Browse** button to navigate to and select the desired file. A warning is issued if a non-existent path is entered, but the entry will still be added.
- **Arguments**: Optional entry used to pass information into the targeted application, if desired.
- **Initial Directory**: Sets the initial working directory for the application.
- **Display**: Selects the way the application appears on the screen when initialized. Normal (the default) to display the application as a window in the workspace, Minimized to start the application and display it as an icon at the bottom of the screen, or Maximized to start the application and display it across the full screen.
- **Prompt for arguments**: Select this check box to automatically display the *Arguments* dialog where you will enter input to be passed to the program at execution time. The entry is saved and recalled the next time the program is run.
- **Add**: Click the **Add** button to begin adding a new tool. You can add a maximum of 16 tools to the Customized Toolbar.
- **Remove**: Select a tool from the custom area and click **Remove** to delete it from the customized toolbar.
- **Move Up/Move Down**: Select a tool from the Custom list and change its position in the list by clicking the **Move Up** or **Move Down** buttons.
- **OK**: Click **OK** to apply additions/modifications and exit the dialog.

Displaying the Custom Toolbar

The *Custom* Toolbar is not displayed in the workspace until a custom tool has been added. Once a tool has been added, the toolbar automatically appears with the other Library Executive toolbars.

The appearance of the *Custom* Toolbar in the workspace is turned on or off by choosing the **View » Custom Toolbar** command.

Executing a Custom Tool

To start an application added as a Custom Tool, choose one of these methods:

- Click the desired button on the *Custom* Toolbar.
- Choose the tool from the list at the bottom of the **Utils** menu.
- While the **Utils** menu is active, select a custom tool by pressing the menu shortcut key assigned to it. For instance, if Notepad was set up with the **N** shortcut key, press the **N** key.

CHAPTER 22

Help Commands

P-CAD Library Executive provide extensive online help that contains both reference and tutorial information, but differs from the manual in its format and accessibility.

The main advantages to online help are the availability of hypertext links between related subjects (an electronic cross-reference utility) and the keyword search function (an electronic index).

Help Topics

The **Library Executive Help Topics** command displays the online help for Library Executive. It includes the **Contents** tab, which is structured to match the order of commands as they appear in the product, and the **Index** tab, which lets you look up a specific concept or keyword.

How to Use Help

The **How to Use Help** command connects you to the Windows help system, where instructions on how to use the help system are provided.

About P-CAD Library Executive

The About P-CAD Library Executive command displays a screen that contains information about Library Executive, including the product version number, release date, memory used and memory available.

APPENDIX A

Importing Master Designer Libraries

This appendix discusses the differences between the Master Designer PDIF libraries and the P-CAD libraries and lists the considerations for importing PDIF libraries. You should read this section before importing any Master designer PDIF libraries.

PDIF Library Considerations

This section discusses the differences between Master Designer PDIF libraries and P-CAD libraries. You may encounter some of these differences when translating Master Designer PDIF libraries.

- **Internal components**: All component information must reside in the Master Designer PDIF file. External component references are not supported.
- **Library file names**: Because a P-CAD PCB library file and a P-CAD Schematic library file may have the same name after translating to Master Designer PDIF, be sure to place Master Designer PDIF Schematic libraries and Master Designer PDIF PCB libraries in separate directories.
- **Attached pattern names**: When translating a Master Designer PDIF Schematic library, P-CAD Library Executive takes the pattern name from the following sources, listed in order of priority:
  - the component's PRT attribute
  - the cross reference file
  - the name of the component with `.prt` added as a suffix.

  Attached pattern names are used when patterns from a PCB library are merged into a Schematic library. They are also used to determine the auto-placed pattern during PCB's netlist load operation.
- **Power pin net assignment**: Power and ground pins are specified either through a cross reference file or the component's PWGD attribute. The PWGD attribute takes precedence over an entry in the cross reference file.

  Padstack references: There are no padstacks in a Master Designer PDIF library. Component pads in a Master Designer PDIF library reference a padstack number only. During translation

from Master Designer PDIF to P-CAD libraries, component pads retain their padstack number. As in Master Designer, when a component is placed from the library, the padstack numbers is used to determine the correct padstack association.

If a component is placed onto an existing design that was translated from Master Designer PDIF and a padstack with the same number is found, then that padstack will be used. However, if no padstack with the corresponding padstack number is located, then the library Default padstack is used instead.

- **P-CAD integrated libraries**: P-CAD integrated libraries provide the ability to place the same component into both Schematic and PCB designs from the same library. An integrated component has both the pattern and symbol information and graphics.

  When translating a Master Designer PDIF library to a P-CAD library, the resulting library does not contain attached symbol or pattern information and is therefore not an integrated library. However, a Master Designer PDIF Schematic library can be translated into a P-CAD Schematic-only library and used to place symbols into P-CAD Schematic.

  Also, a Master Designer PDIF PCB library can be translated into a P-CAD PCB-only library and used to place patterns into P-CAD PCB.

- **Layer mapping differs from PCB:** When translating a Master Designer PDIF library to a P-CAD library, the default layer mapping from the following table is used.

P-CAD Layer	Library Layer
COMP	Top
PADCOM	Top
PINTOP	Top
SOLDER	Bottom
PADSLD	Bottom
PINBOT	Bottom
INT1	Mid-1
INT2	Mid-2
BRDOUT	Board
SLKSCR	Top Silk
DEVICE	Top Assy
ATTR	Top Silk
ATTR2	<NONE>
REFDES	Top Silk

P-CAD Layer	Library Layer
SLKTOP	Top Silk
SLKBOT	Bottom Silk
DVCTOP	Top Silk
DVCBOT	Bottom Silk
REFDTP	Top Silk
REFDBT	Bottom Silk
SLDMSK	Bot Mask
MSKGTP	Top Mask
MSKGBT	Bot Mask
PSTGPT	Top Paste
PSTGBT	Bot Paste

Library Executive maps all other layers to <*NONE*>. See *Mapping Layers* from P-CAD Binary and PDIF Files to PCB in the *P-CAD PCB User's Guide* for a full description of the layer mapping process.

- **Unused pads**: Components created from the translation of Master Designer PDIF Schematic libraries contain one pin for each pin found. If a pattern is attached to a Schematic component that has more pads than pins, the extra pads will have unassigned pins. These pads appear as unused in P-CAD PCB. Connections are not allowed to be placed to unused pads.

  Components created from the translation of Master Designer PDIF PCB libraries contain one pin for every pin that is matched to a pad. However, a place holder pin is created for every pad that did not have a corresponding pin. As a result, all pads have a pin and no unused pads are created during translation of Master Designer PDIF PCB design files or Master Designer PDIF PCB library files.

  Be aware that component patterns placed from the Schematic library may be different from the patterns already placed in the PCB design due to differences in unused pads. If there are problems placing component patterns from the Schematic library into an existing or imported PCB design, use the PCB library patterns.

- **No-connect padstacks**: A component pad translated from a Master Designer PDIF PCB file into P-CAD PCB may have an alternate no-connect padstack assigned to it. Since Master Designer PDIF libraries do not have padstack information, all components placed from the library have the connected padstacks assigned.

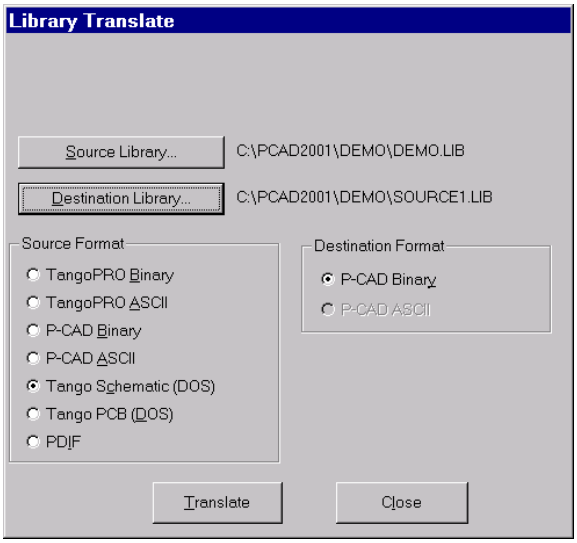
- **Heterogeneous Components**: See the section *Design Considerations in Using P-CAD Files* in the *P-CAD Schematic User's Guide* for details.

Translating Master Designer PDIF Libraries

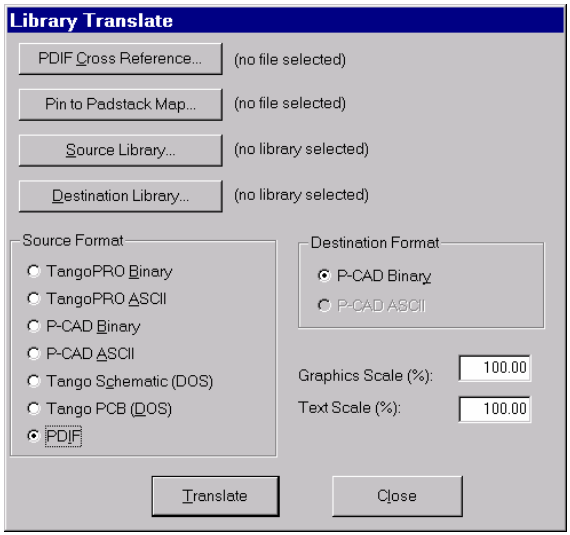
This section provides the steps used to translate a Master Designer PDIF PCB library or Master Designer PDIF Schematic library into a P-CAD library. Once translated, you may merge the resulting P-CAD libraries into a single P-CAD integrated library.

Translating PCB and Schematic Libraries

1. Choose **Library** » **Translate** to open the *Library Translate* dialog.



2. Select the **PDIF** check box as the Source Format type. When PDIF is selected, additional options appear in the *Library Translate* dialog.

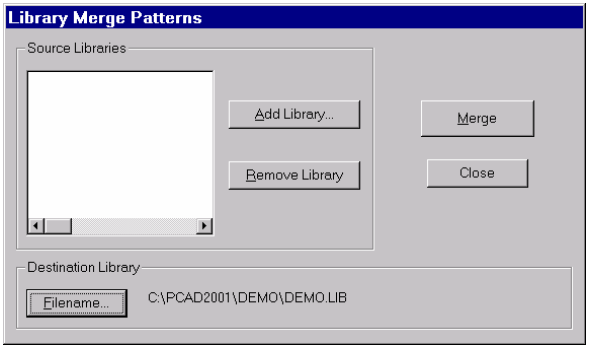


3. If you are using a cross reference file, click the **PDIF Cross Reference** button to open the *Library File Listing* dialog, where you can choose the PDIF cross reference file that corresponds to the library you want to translate.
4. Click the **Source Library** button to open the *Library File Listing* dialog, where you can choose the PDIF PCB library to translate.
5. Click the **Destination Library** button to open the *Library File Listing* dialog and choose the destination library name.
6. Click **Translate** to begin the translation process. Library Executive translates all components and adds new alias names. Library Executive also generates and displays an error log file in the Notepad utility. The name of this file is `Filename.err`, where **FILENAME** is the name of the output library.
7. Review the log file to make sure that all power and ground pins were defined and that important information has not been lost.
8. Repeat steps 3 through 7 to translate the appropriate Master Designer PDIF Schematic Library.
9. You should now have two P-CAD library files: one from a Master Designer PDIF PCB library; the other from a Master Designer PDIF Schematic library.
10. Click **Close** to exit the dialog.

Merging the Libraries

This section explains how to merge the translated Master Designer PDIF PCB and Master Designer PDIF Schematic libraries.

1. Choose **Library Merge Patterns** to open the *Library Merge Patterns* dialog.



2. Select the newly created P-CAD library translated from the Master Designer PCB PDIF library as the Source Libraries.
3. Specify the Destination Library as the P-CAD library translated from the Master Designer Schematic PDIF library.
4. Click the **Merge** button. Library Executive merges the Master Designer PDIF PCB library into the Master Designer PDIF Schematic library.

You can now use this library with both P-CAD PCB and P-CAD Schematic.

Import Considerations

This section lists considerations for importing into P-CAD Library Executive.

- Copper Pours in components are demoted to polygons.
- Cutouts in components are ignored.

APPENDIX B

System Messages

This appendix provides a listing of system messages by message number. Many of these messages relate to the verification process. A detailed explanation of each message is included.

Library Message Listing

Error Number	Message	Comment
5635	Duplicate pin des found in spreadsheet while loading pattern	Multiple pins have the same pin designators.
5636	Duplicate pad numbers found at rows __ and __.	Multiple pads have the same pad numbers.
5637	Pad number at row ___ is out of range	Valid pad numbers range from 1 to the number of rows, inclusively.
5700	Unable to allocate memory for pin data. Component not verified!	A small amount of memory is required (64K or less) to hold the pin information for the entire component.
5701	Gate Number out of range in row #. [-1 to 255]	No more than 5,000 gates may be used in a component. -1 or CMN specifies a common pin.
5702	Common pin in row # has a Gate Eq # that does not match any gates!	The gate equivalence of a common pin is used to specify the scope or range of gates that the common pin applies to. A common pin by definition must exist for at least 2 gates.
5703	Common pin in row # has a Gate Eq # that matches only 1 gate!	The gate equivalence of a common pin is used to specify the scope or range of gates that the common pin applies to. A common pin by definition must exist for at least 2 gates.

Error Number	Message	Comment
5704	Gate # missing symbol.	A gate was specified in the *Component Information* dialog but no symbol was attached. At least one symbol must be associated to every gate.
5705	Gate # missing symbol name.	A gate was specified in the *Component Information* dialog but no symbol was attached. At least one symbol must be associated to every gate.
5706	NORMAL Symbol name missing from library.	The attached symbol could no longer be found in the current library. The library is either corrupted or the symbol was removed by another operation while creating the component.
5707	IEEE Symbol name missing from library.	The attached symbol could no longer be found in the current library. The library is either corrupted or the symbol was removed by another operation while creating the component.
5708	DEMORGAN Symbol name missing from library.	The attached symbol could no longer be found in the current library. The library is either corrupted or the symbol was removed by another operation while creating the component.
5709	PATTERN name missing from library.	The attached pattern could no longer be found in the current library. The library is either corrupted or the pattern was removed by another operation while creating the component.
5710	Number of rows not equal to number of pads.	If a pattern is attached then the number of rows of pin information must exactly match the number of pads in the pattern. Rows may be left blank if a pattern pad has no corresponding pin associated with it.
5711	Missing PinDes in row #.	When there is no attached pattern then every row must correspond to a pin and every pin specified must have a pin designator. If the row is not needed then move the row to the bottom of the list and use the *Component Information* dialog to reduce the number of pins by 1.
5712	Missing PinDes in row #. Only unused pads are allowed to have a blank PinDes.	When there is an attached pattern then every row corresponds to a pad in that pattern. Only unused pads may have an blank pin designator. An unused pad is specified by a completely empty row.

Error Number	Message	Comment
5713	Invalid characters in PinDes in row # [Bad characters are COMMA, DASH, SPACE, TAB, BRACKET, PAREN, HI-ASCII]	The pin designator is not allowed to have any of the following characters: Comma “,” Dash “-” Bracket “[“ or “]” Paren “(“ or “)” Space Tab or characters whose ASCII value are larger than 128.
5714	POWER pin can't have missing PinName in row #.	A pin with electrical type, Power, must specify the pin name since it is used as the default net name of that pin.
5715	Invalid characters in PinName in row # [Bad character are SPACE,TAB,HI-ASCII]	The pin name is not allowed to have any of the following characters: Space, Tab, or characters whose ASCII value are larger than 128.
5716	Gate Number out of range in row #. [-1 to #]	The value entered in the Gate# column of the spreadsheet must be less than the number of gates specified in the *Component Information* dialog.
5717	Gate & Pin numbers must both be zero or both be non-zero in row #.	The combination of gate number and pin number uniquely specify a component pin. They must both be zero when specifying an empty row, unused pad, or power pin. They must both be set to a non-zero value to specify any other pin.
5718	Pin number _ in row _ exceeds the number of pads/ pins	The pin number in any row can never exceed the number of pads when a pattern is attached or can never exceed the total number of pins when no pattern is attached.
5719	The gate equivalence specified in Component Information for gate # does not match the pin's gate equivalence specified in row #.	It is possible to get the gate equivalence to differ between the spreadsheet values and the values specified in the *Component Information* dialog. Correct the values so that they match by reentering the gate equivalence for each gate in the *Component Information* dialog.
5720	Gate # has multiple Gate Eq values... first found in row #.	Multiple rows with the same gate number have differing gate equivalences.
5721	Equivalent gates # and # have a different number of pins.	Multiple gates with the same gate equivalence value (other than 0) must have the same number of pins.
5722	Gate # is missing. Gate numbers must be contiguous for homogeneous components.	The spreadsheet is missing information for a gate. There are no rows that correspond to the gate number found in the *Component Information* dialog.

Error Number	Message	Comment
5723	The same pin on equivalent gates must have the same Pin Eq. See row # and #.	Equivalent gates are considered to have exactly the same symbols and symbol information.
5724	The same pin on equivalent gates must have the same Electrical Type. See row # and #.	Equivalent gates are considered to have exactly the same symbols and symbol information.
5725	The same pin on equivalent gates of a homogeneous component must have the same Pin Name. See row # and #.	Equivalent gates are considered to have exactly the same symbols and symbol information.
5726	Equivalent pins can not have different electrical types in the same gate. See row # and #.	None.
5727	Gate Number # greater than number of gates # in row #.	The gate number specified in the spreadsheet is larger than the number of gates specified in the *Component Information* dialog.
5728	Pin Number # greater than number of pins # in row #.	The pin number specified in the spreadsheet is larger than the numbers of pins specified in the *Component Information* dialog.
5729	Duplicate Pin Number # found in gate # in row #.	The combination of gate number and pin number has already been specified. Make sure that a common pin's pin number does not match another pin's pin number.
5730	Pin Number # could not be found for gate # in row #.	The pin number specified in the spreadsheet could not be found in the symbol corresponding to the gate number.
5731	Common pin defined for gate number # is greater than number of gates # in row #.	The gate equivalence used to specify the scope or range that the common pin is defined for is larger that the number of gates.
5732	Pin Number # for gate # was never assigned in the spreadsheet.	A pin number found in the symbol could not be found in the spreadsheet.
5733	Duplicate pin number # not allowed in gate #. Pin numbers must start with 1 and must be contiguous.	None.
5734	Pin Number # is missing from gate #. Pin numbers must start with 1 and must be contiguous.	A pin number found in the symbol could not be found in the spreadsheet.

Error Number	Message	Comment
5735	Attempt to assign duplicate pin designator name.	Every pin designator must be unique.
5736	The jumper pin in row __ is not jumpered to any other pin.	A jumper pin, JMP-n, must be jumpered to another pin with the same -n value.
5737	Invalid jumper pin value in row __.	Check jumper pin values.
5786	Duplicate default pin designators are now allowed.	Default pin designators must be unique.
12039	Field name changes in file external source file name. The saved mapping in the `.ini` file is ignored.	Field mappings are saved so that if a source file of the same name is imported again, the fields are mapped automatically. When this warning appears, the field names in the external source file have been modified since last mapping and must be mapped again.
12070	Duplicate field name: field name.	The field name already exists in the source. Choose a different name for the added, renamed, or mapped field.
12080	Field mapping will reload the original imported file. Changes made to the table Viewer will be lost.	Any changes made to the imported source file in the file Viewer will be lost during field mapping. The **Map Fields** command always maps the original source file. Map the fields before modifying the Viewer contents if you would like those changes to be saved.
12132	Record at line # does not match with field description. Do you want to see additional errors? (Press **Cancel** to end the import.)	One or more of the rows in the imported comma-delimited file has too many or too few field entries. Check that all special characters, particularly commas within field values, have been properly escaped.
12133	No source is specified for the query. Select a source from the source browser, then select the right-mouse menu **Query**.	Conducting a Query from the Source Browser specifies a source to search. **Utils » Query** has no means of determining which source is desired. After the first Query has been conducted from the Source Browser, **Utils » Query** displays the spreadsheet of the previous Query.

Index

-D-

-E-

-F-

-G-

-H-

-I-

-J-

-K-

-L-

-M-

-N-

-O-

-P-

-Q-

-R-

-S-

-T-

-U-

-V-

-W-