ACCEL Library Manager Contents

This table of contents lists available help information for ACCEL Library Manager.   Click on an underlined term to access information on that subject or subject area.

For Help on Help, press ***F1***.

Series II Commands

In Series II, the Library Editor was a part of Tango-PCB and Tango-Schematic. With ACCEL, it is a separate, independent, product. For a list of Tango-PCB pattern and component creation commands and their ACCEL equivalents, refer to ACCEL PCB on-line help. For a list of Tango-Schematic symbol and part creation commands and their ACCEL equivalents, refer to ACCEL Schematic on-line help.

Shortcut and Special Use Keys

There are numerous shortcut and special use keys to speed your work with ACCEL. To review these keys, click on the desired key type.

Shortcut Keys
Special Use Keys

Component Menu Commands

New
Open
Save
Save As
Validate
Exit

Component New (Ctrl+N)

Allows you to create a new component for later addition into a component library. You can use the toolbar icon for Component New as a shortcut.

When you run this command, the Open dialog appears.

In the Open dialog you can select a library in which you want to save the new component and in which the components symbol and pattern information reside.

When you click *OK* on the Open dialog, the Component Information dialog appears. From this dialog, you can attach a pattern, set the component type, set the number of gates in the component, set alternate representations of the component, and set the Refdes prefix.

Creating a New Component
Drag and Drop Library Load

Component Open (Ctrl+O)

Opens an existing component from a library.   You can use the Component Open button from the toolbar as a shortcut.

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.

To Open a Component

1. Choose Component Open to display the dialog.
2. If the library you want to access is not current (does not appear in the Library field), click the **Library...** button to display a dialog to access the appropriate library (a Windows common dialog).

   You can also drag and drop a library file (.LIB) into the Library Manager.
3. When you have the library name displayed, either enter the component name in the Component textbox, or click on the component name from the list of components within the listed library.

   A display of the attached symbol, if any symbol is attached, will appear in the right side of the dialog.
4. Click **OK**.   The Component Information dialog appears.

You can now open different views of the component and attach symbols or patterns and edit component properties.

Component Save (Ctrl+S)

Saves the current component (in the spreadsheet) to its associated library file.   You can use the Component Save button on the Toolbar as a shortcut.

To save the component under a different name, use the Component Save As command.

Component Save As

Saves the component to a different library or with a different component name or reference designator prefix. If you simply want to save the current component to the current library, use the Component Save command.

When you run this command, the Component Name dialog appears.

In the **Component Name** drop down listbox, you can specify a new name for this component or select one from the drop down list.

Component Validate (Ctrl+K)

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.

Component Exit (Alt+F4)

Exits the Library Manager program.

If you have an unsaved component, you will be prompted as to whether you want to save changes before you exit the program.

Edit Commands

Undo Spreadsheet Change
Cut Spreadsheet Selection
Copy Spreadsheet Selection
Paste Spreadsheet Selection
Slide Selection Up
Slide Selection Down
Select Symbols
Select Pattern
Component Attr

Edit Undo Spreadsheet Change (Ctrl+Z)

Undo reverses your last *completed* action to the Pins View, Pattern View, and Symbol View dialogs, (but not in the Component Information dialog). Many File commands, such as File New, Save, Print, etc. *cannot* be undone. If an action cannot be undone (or there is nothing to undo), Undo appears grayed on the Edit menu.

The undo button on the toolbar and the *Z* key are equivalent to the Edit Undo command.

Edit Cut Spreadsheet Selection (Ctrl+X)

Moves the data from all selected cells to the clipboard, overwriting the previous clipboard contents. Use the toolbar icon equivalent as a shortcut. You must have one or more cells selected before you can cut.

note:
Edit Cut Spreadsheet Selection does not function within the Component Information dialog.

Use Paste to copy the clipboard data to another cell or cells within the spreadsheet. Data in the clipboard can also be pasted into a commercial spreadshee where it can be edited and pasted back into the Library Manager.

Edit Copy Spreadsheet Selection (Ctrl+C)

Copies the data from all selected cells to the clipboard. The original data is not erased from the cells. Use the toolbar icon equivalent as a shortcut.

Edit Copy Spreadsheet Selection does not function within the Component Information dialog. You must have one or more cells selected before you can copy.

Use Paste to copy 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 the Library Manager.

Edit Paste Spreadsheet Selection (Ctrl+V)

Copies the clipboard data to cells or cells within the spreadsheet. Use the toolbar icon equivalent as a shortcut.

Edit Paste Spreadsheet Selection does not function within the Component Information dialog. You can use the Windows clipboard to copy data from a commercial spreadsheet and paste it into the Library Manager using the Edit Paste Spreadsheet Selection command.

Edit Select Symbols

Allows you to select one or more symbols from the current library to attach to the current component. You can attach any symbols that reside in the same library as the component.

This command also can be run 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 run this command, the Library Browse dialog appears. All symbols in the current library are listed in the **Symbol** listbox.

After you have selected a symbol, 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

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.

This command also can be run by clicking the **Select Pattern** button.

When you run this command, the Library Browse dialog appears.   All patterns in the current library are listed in the **Pattern** listbox.

After you have selected a pattern, 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 Attr

You can view, add, modify, or delete a collection of component attributes. The dialog contains a two-column table showing a collection of component attributes. Within the collection, each attributes name and value appear in the column.

- **Adding an Attribute**: To add an attribute, click the **Add** button to open the Attribute Property dialog (shown below). Enter and 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:** To view or change an attributes properties, select an attribute from the table and click the **Properties** button (or double-click the attribute) to open the Attribute Property dialog.
- **To Delete an Attribute**: Highlight an attribute in the table and click **Delete**, or press the *Del* key.

Attribute Property Dialog

The following information appears in the dialog:

- **Category Listbox:** Displays a list of all attribute categories, All, Component, Net, Router, and SPECCTRA. Selecting a category brings up a list of pre-defined attributes for that category.
- **Name Listbox:** Displays all pre-defined attributes for the specified category. The first entry in the list is *User-defined*.
- The currently-selected attribute also appears in the **Name** edit box, unless *User-defined* is selected. In that case, the Name edit box is blank so that you can enter a user-defined attribute name.
- **Name Edit Box:** For user-defined attributes, enter a name for the attribute.

note:

If the dialog is accessed for an attribute that already has a name, then the Category listbox, Name listbox, and Name edit box are filled in, but grayed. If the attribute doesnt have a name, these controls are enabled.

- **Value:** Use this edit box to enter a value for the attribute.
- **Visible:** This checkbox indicates whether or not the attribute is visible.

The following fields are grayed out in Library Manager:

- **Location:** This area shows the X and Y coordinates of the components reference point.
- **Text Style:** This area lets you select the attribute text style. Text styles appear in the **Text Style** drop down listbox. To change the selected Text Style, click on the text style you want from the listbox. To modify the text style, click the **Text Style** button.
- **Rotation**: Shows the rotation amount if the pattern has been rotated.
- **Flipped** Box: This box indicates whether or not the pattern has been flipped.
- **Justification**: Under **Justification** are nine buttons, which allow you to change text justification by setting the *reference point* of the text string. For example, if you enable the middle button, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

View Commands

View Component Info

When you run the View Component Info command, the Component Information dialog appears.

This dialog is also available from the toolbar and from the **Component Info** button.

From this dialog, you can attach a pattern, set the component type, set the number of gates in the component, set alternate representations of the component, and set the Refdes prefix.

This dialog allows you to 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.

However, the more information you provide, the more automatic entries will be filled in for you.

- When you attach a pattern, pin designators are filled in.
- When you select symbols, gate numbers and pin numbers for that symbol are automatically added.
- Gate equivalence is automatically applied in Pins View to the gates gate equivalence.

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** box is automatically filled in. Alternatively, attaching gates automatically increases the number of pins, if no pattern is attached.

Gates

If you set the **Component Type** 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 gate is greater than 0, you can select an attached symbol using the **Select Symbol** button. IEEE and DeMorgan equivalents are available if you select them in the **Alternate Views** box.

View Pins View

The Pins View dialog allows you to assign pin data.

When you run the View Pins View command, the Pins View dialog appears. This dialog also is available from the toolbar and from the **Pins View** button.

If a pattern is attached, the rows of the spreadsheet are arranged by pad sequence. If no pattern is attached, the rows of the spreadsheet are arranged by the pin sequence of each successive gate. If no pattern and no symbols are attached, you should order the rows of the spreadsheet in a way that is logical for the pin designators.

If there is an attached pattern, the information in the current row also appears in the Pattern View dialog. The corresponding pad is highlighted 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 box also appears in the Symbol View dialog. The corresponding pin and gate are highlighted in the Symbol View dialog. Changes made to row data are automatically updated in the Symbol View dialog.

You can use any of the Microsoft Windows resizing features to vertically resize the Pins View dialog allowing you to see more rows of pin data.

View Pattern View

The Pattern View dialog allows you to assign pin data to a component.

When you run the View Pattern View command, the Pattern View dialog appears. This dialog is also available from the toolbar and from the **Pattern View** button.

note:
If you use the mouse while holding down the *Shift* key, you can expand the dialog vertically to display additional spreadsheet rows which are hidden.

The attached pattern appears in a browse window. The Pattern View dialog may be resized to increase the display area of the browse window. If you use the mouse while holding down the *Shift* key, you can expand the dialog vertically to display additional spreadsheet rows which are hidden.

A pad may be selected in the browse window using the mouse cursor. When a pad is selected, the following occurs:

- The pad is highlighted in the Pattern View browse window.
- The pads corresponding pin is highlighted in the Symbol View browse window.
- The corresponding spreadsheet row is highlighted in the Pattern View and Symbol View dialogs.

Prev Pad/Next Pad

The **Prev Pad** and **Next Pad** buttons automatically select and highlight the next and previous pads. The order of the pads is defined by the pad number sequence set during pattern creation. Using the *Up* key in any field or selecting the **Prev Pad** button selects and highlights the previous pad. Using the *Down* key in any data field or selecting the **Next. Pad** button selects and highlights the next pad.

Component Creation

Use the **Select Pattern** button to select the attached pattern. Changes made in this dialog are reflected in the Component Information dialog.

View Symbol View

The Symbol View dialog allows you to assign pin data to a component.

When you run the View Symbol View command, the Symbol View dialog appears. This dialog is also available from the toolbar and from the **Symbol View** button.

The attached symbol appears in a browse window. The Symbol View dialog may be resized to increase the display area of the browse window. If you use the mouse while holding down the *Shift* key, you can expand the dialog vertically to display additional spreadsheet rows which are hidden.

note:
> If you use the mouse while holding down the *Shift* key, you can expand the dialog vertically to display additional spreadsheet rows which are hidden.

A pin may be selected in the browse window using the mouse cursor. When a pin is selected, the following occurs:

- The pin is highlighted in the Symbol View browse window.
- The pins corresponding pad is highlighted in the Symbol View browse window.
- The corresponding spreadsheet row is highlighted in the Pattern View, Pins View, and Symbol View dialogs.

Prev Pin/Next Pin

The **Prev Pin** and **Next Pin** buttons automatically select and highlight the next and previous pins within the symbol. The order of the pins is defined by the pin sequence set during symbol creation.

Prev Sym/Next Sym

If there is more than one sym, the **Prev Sym** and **Next Sym** buttons automatically select the next and previous syms. The **Prev Sym** button goes to the *last* pin of the previous sym; the **Next Sym** button goes to the *first* pin of the next sym.

Component Creation

Use the **Select Symbol** button to select the attached symbol. Changes made in this dialog are reflected in the Component Information dialog.

View Toolbar

Allows you to either display or hide the toolbar. A check mark appears next to the command on the menu when it is enabled (displayed). Current visibility is saved to the CMP.INI file and restored in subsequent sessions.

The toolbar pushbuttons are shortcuts for commonly-used commands.

View Prompt Line

Allows you to either display or hide the prompt line. A check mark appears next to the command on the menu when it is enabled (displayed). Current visibility is saved to the CMP.INI file and restored in subsequent sessions.

Library Commands

Library New

Library New allows you to create a new library.   The new library is empty.

When you choose Library New, the Library New dialog appears.   In the dialog you can specify the filename of your new library.

Library Alias

An *alias* is an alternate name for a component, pattern or symbol. You can create multiple names for the same item with this command.

An alias is equivalent to creating a new item except the actual data is only referenced not copied. Thus, when you create aliases 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 re-invoke any of the commands during the same design session.

Aliases allow you flexibility in using a variety of naming conventions for components, patterns, or symbols *without renaming them*. For example, for what ACCEL calls an SN7400N, you may want to use a generic alias of 7400. Or, if you are using 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.

Creating an Alias

1. Choose Library Alias to display the Library Alias dialog.
2. Specify either the **Component** or **Pattern** or **Symbol** radio button in the **Alias Item** box.
3. If the appropriate library is not current, then click the **Library** button and the Library Select dialog is displayed.

   Select the library you want, click **OK**, and the Alias dialog is redisplayed.
4. You can click the button (**Component...** or **Pattern...**or **Symbol...**) below **Library...** to display the Library Browse dialog. Highlight the item you want from the list and click **OK**.
5. When you return to the Library Alias dialog, the Pattern, Component, or Symbol you chose is now listed in the **Pattern...**, **Component...**, or **Symbol...** listbox. You can enter the alias name you want in the **New Alias** textbox. Click **Add** and the new alias will be listed in the **Aliases** listbox.

Library Copy

Copies an item from one file to another (either in the same or in a different library).

When you copy a component, you will be prompted whether you want to copy the pattern and symbols that it references.

The dialog allows you to select the source library and item name as well as the destination library and item name.

The source and destination libraries that you use with Library Copy will remain current if you re-invoke this command during the same ACCEL session.

If you are copying a component or pattern or symbol but are not changing its name, you can leave the **Destination Name** box blank.

Copying a Library Item

1. Select the Library Copy command to open the dialog.
2. Select the **Copy Item** (**Component** or **Pattern** or **Symbol** radio button) for your copy action.
3. Click the **Source Library** button. The Library Select dialog is displayed, from which you can select the source library of your copy action.

   After you choose the library from Library Select, the source library name is displayed in the Library Copy dialog.
4. Click the item button (**Pattern..., Component...,** or **Symbol...**) and a list of items in the library are displayed in the Library Browse dialog. Select the item you want to copy, and the item name will be displayed in the Library Copy dialog.
5. Click the **Destination Library** button. The Library Select dialog is displayed, from which you can select the destination library of your copy action. The library name is displayed in the Library Copy dialog.
6. Specify the **Destination Name** for the new item. If you want to keep the same name, you can leave the textbox blank.
7. Click the **Copy** button. If you choose **Component**, you are prompted whether you want to copy the pattern and symbols or not when copying from one library to another.

   All the boxes (except the Source library path and the Destination library path) become blank. This way you can continue to copy items between the same source and destination libraries.
8. Press **Close** to exit the dialog. If you click **Close** before **Copy**, then the box will close without completing the copy action.

Library Delete

Deletes a 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 the item has only one name and you delete it, then the item itself is deleted from the library. Use the Library Alias command to check whether an item has aliases.

The library that you use in the execution of Library Alias, Library Delete, or Library Rename will remain current if you re-invoke any of the commands during the same ACCEL session.

warning

If you delete a pattern or symbol, then all of the components in the library that reference that pattern or symbol will be incomplete, and therefore unplaceable. Normally you would want to delete a pattern alias or symbol alias only, which is typically not as dangerous.

Deleting a Library Item

1. Choose Library Delete to display the dialog.
2. Select the **Delete Item** type (**Component**, **Pattern**, or **Symbol** radio button) for your delete action.
3. Click the **Library...** button. The Library Select dialog is displayed, from which you can select the library in which you want to delete an item.

   The library you selected in Library Select is displayed in the Library Delete dialog.
4. Click the item button (**Pattern..., Component...,** or **Symbol...**) and the items within the displayed library will be listed in the Library Browse dialog. Select one and it will then be listed in the Library Delete dialog.
5. Click the **Delete** button and the item box becomes blank. This way you can continue to delete items from the same library.
6. Click **Close** to exit the dialog. If you click **Close** before **Delete**, then the box will close without completing the delete action.

Library Rename

Renames a pattern, symbol, or a component.

warning:

If you rename a pattern or symbol, then all of the components in the library that reference that pattern or symbol by the original name will be incomplete and be unplaceable. If you want to use a different naming convention for a pattern, symbol, or component, then create an alias for the pattern or symbol (Library Alias command) and use that alias name.

Renaming a Library Item

1. Select the Library Rename command to display the dialog.
2. First select the item type (**Component, Pattern,** or **Symbol** radio button) in the **Rename Item** area to specify what you will rename.
3. Click the **Library** button to display the Library Select dialog, where you can choose the library to access, which will be displayed in the Library Rename dialog.
4. Click **Pattern..., Component...**, **Symbol...** and the items within the displayed library will be listed in the Library Browse dialog. Select one and it will then be listed in the Library Rename dialog.
5. In the **New Name** section type the new name of your item, then click **Rename**. Both the old name and new name disappear if the rename action is successful. Then you can continue renaming items in the same library.
6. Click **Close** to exit the dialog. If you click **Close** before **Rename**, then the box will close without completing the rename action.

Library Update

Converts TangoPRO version 1.x libraries into TangoPRO version 2.0 libraries.   This command is necessary for converting only libraries used by TangoPRO PCB version 1.x into libraries which are compatible with TangoPRO PCB and Schematic version 2.0 and above.   Once converted, these libraries are also compatible with ACCEL PCB and ACCEL Schematic.

1. Select the Library Update command to display the Library Update dialog:
2. Click the **Source Library** button to display the Library File Listing dialog, where you can choose the library to convert.
3. Click the **Destination Library** button to display the Library File Listing dialog, where you can choose the destination library.
4. Click **Update** to begin the conversion process.
5. When the library has been converted, click **Close** to exit the dialog.

Library Translate

Translates libraries in the following formats into ACCEL binary- and ASCII-formatted libraries.

- ACCEL Binary
- ACCEL ASCII
- TangoPRO Binary
- TangoPRO ASCII
- Tango-Schematic (DOS)
- Tango-PCB (DOS)
- SCHEMA (DOS)
- SCHEMAx (Windows)
- PDIF

ACCEL uses one integrated library, which contains components, symbols, and patterns. These integrated libraries are used by all ACCEL 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 fundamentally the same, ACCEL PCB reports that the components do not match, and will not allow you to add the component to the design. For this reason, we make the following recommendations:

- PCB designs created using previous versions of ACCEL PCB or Tango-PCB using pattern libraries should be loaded into ACCEL PCB with only pattern libraries open.
- If you encounter a component cache error after using the Library Manager 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. Use the Force Update command to replace all occurrences of a component of a single type.
- The Maintain Rotation option in the Utils Force Update command does not maintain rotations for components in designs loaded from Tango-PCB.

Using the Command

1. Select the Library Translate command to display the Library Translate dialog:
2. Click the **Source Library** button to display the Library File Listing dialog, where you can choose the library to translate.
3. Select the source library type by clicking a **Source Format** radio button.

   If you select **ACCEL Binary**, select a destination format by clicking a **Destination Format** radio button.
4. Click the **Destination Library** button to display the Library File Listing dialog, where you can choose the destination library or input to a new library name.
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.

See also:

SCHEMA Translation
PDIF Translation

Library Merge Patterns

Copies patterns from one or more ACCEL libraries containing patterns into another ACCEL library that contains pattern references, but not actual patterns.

After you translate a Tango Schematic symbol library and Tango pattern library to ACCEL 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.

This command is useful for merging entire libraries. If you want to integrate an individual component, use the Library Copy command. Attach the appropriate pattern to the component using the **Select Pattern** button in the Component Information dialog.

1. Select the Library Merge Patterns command to display the Library Merge Patterns 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 for the components in the destination library.

   Highlight a library and click the **Remove Library** button to remove a library from the **Source Libraries** box.
3. Click the **Destination Library** button to display the Library File Listing dialog, where you can choose the destination library. This is typically a symbol library which has been converted to ACCEL 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.

   When the process is complete, 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.

   If a library has been translated from a Tango Series II library, and it has components with no pattern attribute, or the pattern that is referenced doesn't exist in any of the source libraries, it will write an error to the log file.
5. Click **Close** to exit the dialog.

Utils Commands

Query
ACCEL PCB
ACCEL Schematic

Utils ACCEL PCB

Runs ACCEL PCB if that program is installed on your computer.   Refer to your PCB documentation for additional information.

Utils ACCEL Schematic

Runs ACCEL Schematic if that program is installed on your computer. Refer to your Schematic documentation for additional information.

Creating Components

In this chapter we describe the process of creating components by creating a hypothetical 7400 component (see component specifications for details), using PCB, ACCEL Schematic, and the Library Manager. You will save this component to TUTOR.LIB found in C:\ACCELEDA\LIB. This chapter also describes common pins and creating heterogeneous components.

This procedure involves the following steps:

Step	See Topic
1. Create and save the pattern to a library, using PCB.	Creating a Pattern
2. Create and save the symbols to a library, using ACCEL Schematic.	Creating a Symbol
3. Create the component, using the Library Manager.	Creating a Component
4. Attach the pattern and symbols to the component, using the Library Manager.	Attaching a Pattern
5. Add Component Properties using the Library Manager.	Adding Component Properties
6. Attach the symbols to the component, using the Library Manager.	Attaching a Symbol
7. Edit the component spreadsheet (Pins View).	Editing the Pins View Spreadsheet
8. Save the component to a library, using the Library Manager.	Saving the Component

You should be familiar with PCB and Schematic before trying this tutorial. If you don't have one of these programs, you can read along and practice those sections which apply to the programs you own.

Libraries

This help topic provides important information about the structure and characteristics of ACCEL libraries, including:

- The relationship between components, patterns, and symbols.
- Using aliases instead of renaming.
- Pad numbers vs. pin designators.

Shortcut Keys

Alt+F4 **(Component Exit)**

Shortcut for Component Exit, which exits the Library Manager 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.

Ctrl+N **(Component New)**

A shortcut for Component New. This command clears the Spreadsheet to load the new component.

Ctrl+O **(Component Open)**

Displays the Component Open dialog, from where you can choose a component.

Ctrl+S **(Component Save)**

Saves changes to the current component without closing it. To save the component to a different name or location, use Component Save As. To clear the spreadsheet, use Component New.

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. Cuts text from the selected cell(s) in the spreadsheet to the clipboard.

Special Use Keys

F1 (Help)

Displays the Help Contents window, from which you can access help information on Library Manager commands and tutorials.

F2 (Edit)

Moves the cursor to the edit box where you can edit cell 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 move only in the edit box.

Shift+arrow keys extends the selection.

Delete

Deletes items from the selected cell or cells

Home

When the cursor is in the edit box, moves to the beginning (left side) of the edit box. When the cursor is in the spreadsheet. moves to the beginning of the row.

End

When the cursor is in the edit box, moves to the end (right side) of the edit box. When the cursor is in the spreadsheet, 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.

PgDn

Scrolls the spreadsheet one page down.

Creating a Pattern

PCB provides the editing tools to either create patterns from scratch, or to *explode* and modify existing components if a pattern with the correct graphics doesnt already exist. In this section, you will create a pattern from scratch for the 7400 component. This process involves the following basic steps:

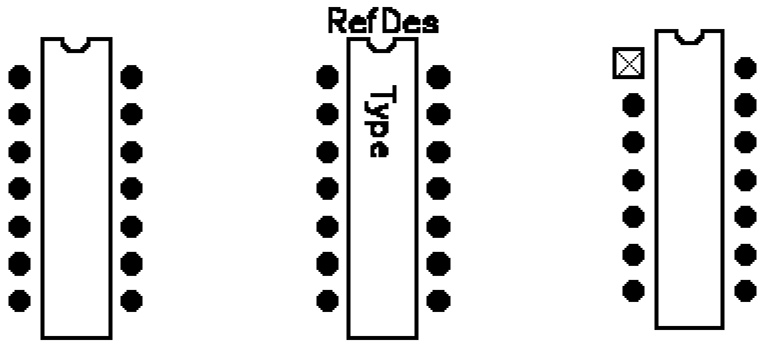
1. Place objects (lines, pads, etc.) to create the pattern.
2. Place RefDes and Type attributes on the pattern.
3. Renumber the pads on the pattern.
4. Add a reference point to the pattern.
5. Save the pattern to a library.

Creating a Pattern for the 7400

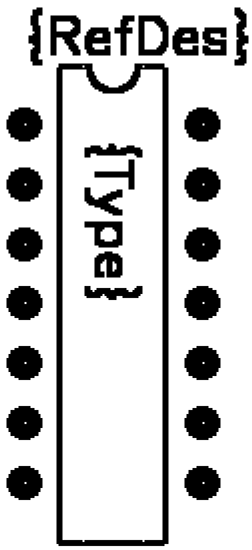
In this section you'll use the PCB editor to place lines, pads, attributes, and a reference point to create a pattern. You'll be creating a 14-pin DIP pattern called TUT14. Later, in the Library Manager, you'll attach TUT14 to the new component.

Before you begin, run PCB and open TUTOR.LIB using the Library Setup command.

1. First you need to set up the appropriate grid spacing so that pads can be placed or moved on grid with the help of the *snap to grid* feature. Use the Options Grids command or the grid toggle button to set your grid spacing to 100 mil.
2. Use the View Zoom In command to get a closer view, or press the plus key (+). Place fourteen pads using the **Default** pad style in two columns of seven (Place Pad or Toolbar pad button). Place the pads 100 mils agate with 300 mils between columns. You can place one pad and then use the Edit Copy Matrix command to create the two columns.
3. Change grid to 10 mils, change to the Top Silk layer, then draw the silk items between the pads. Place lines and arcs on the Top Silk layer appropriately (Place Line, Place Arc). See the following figure:

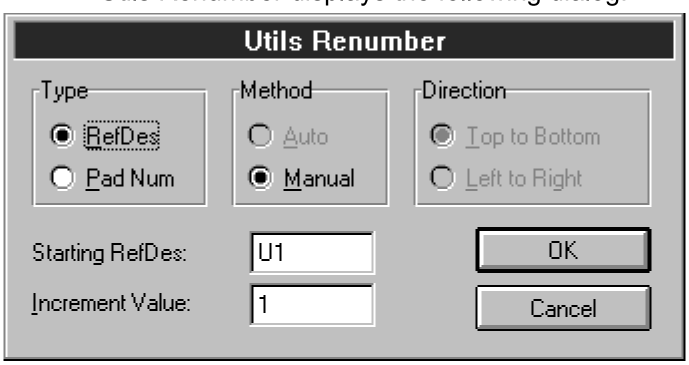


4. Place RefDes and Type Attributes on the Top Silk layer (Place Attribute) as shown below:

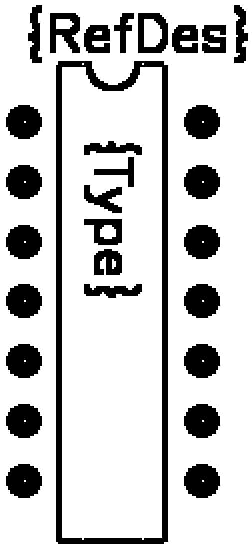


5. When you place a pad, it is given a default pad number of 0. Use the Utils Renumber command to renumber the pads. You need to run the Edit Select tool and be on the Top layer before trying to renumber.

   Utils Renumber displays the following dialog:



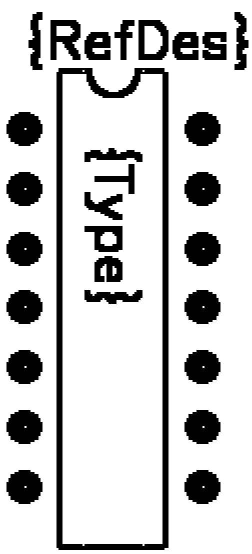
   In the dialog, enable **Pad Num** for Type, set **1** for the sequence in Starting Number, and **1** for Increment Value. Click *OK*, then click on the pads one-by-one, assigning them numbers 1 through 14 in the order shown in the following picture. Click the right mouse button to complete renumbering.



Check the pad numbering by enabling Select mode (Edit Select), clicking on a pad, clicking the right mouse button to display the Select popup menu, and choosing Modify from the popup to enable Modify Pad (Edit Modify). The dialog lists the pad number of the selected pad.

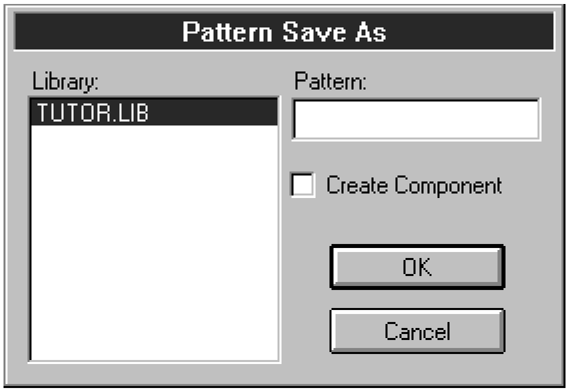
You can also check the pad numbering by zooming in with the Display Pin Designators option enabled; use the Options Display (Misc) dialog to enable the option. This option controls the display of pad numbers on primitives and pin designators on components.

6. Using the Place Ref Point command, place a reference point over the top-left pad in the pattern, as shown.



7. Do a block select of all the objects which will be in the pattern.

   Use Library Pattern Save As to display the dialog. To save and name the pattern to a library, name the pattern TUT14 and save it to the TUTOR.LIB library. Do <u>not</u> enable the **Create Component** check box as you'll be creating the component in the Library Manager.



Now that you have created a pattern in PCB, you can go into ACCEL Schematic and create a symbol. This process is presented below.

Creating a Symbol

Now you'll go through the steps needed to create a symbol for the component in ACCEL Schematic. You can create a symbol by exploding and modifying an existing symbol, but for this exercise you are going to create a symbol from scratch. This step is unnecessary if you find an existing symbol with the correct graphics.

Symbol creation involves the following basic steps:

1. Place objects to create a symbol.
2. Place RefDes and Type attributes on the symbol.
3. Renumber the pins in the symbol.
4. Add a reference point to the symbol.
5. Save the symbol to a library.

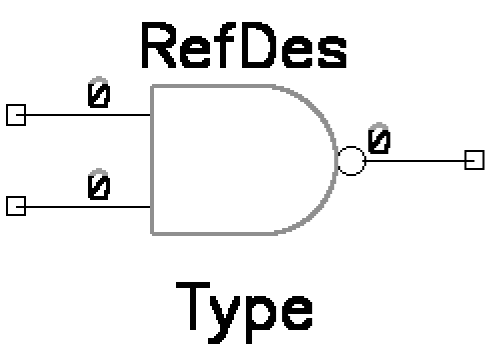
Creating a Symbol for the 7400

In this section you'll use the Schematic editor to place lines, pins, arcs, and a reference point to create a symbol. You'll be creating a simple NAND symbol called TUTNAND. Later in the Library Manager, you'll attach TUTNAND to the new component.

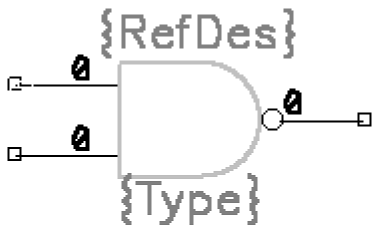
Before you begin, run ACCEL Schematic and open TUTOR.LIB using the Library Setup command.

1. First you need to set up the appropriate grid spacing so that pins can be placed or moved on grid with the help of the *snap to grid* feature. For example, use the Options Grids command or the grid toggle button to set your grid spacing to 100 mil.
2. Use the View Zoom In command to get a closer view. Use the appropriate place commands to place lines, an arc, and pins to create the following symbol. Be sure to enable **Display Pin Name** and **Pin Des** when placing the pins.

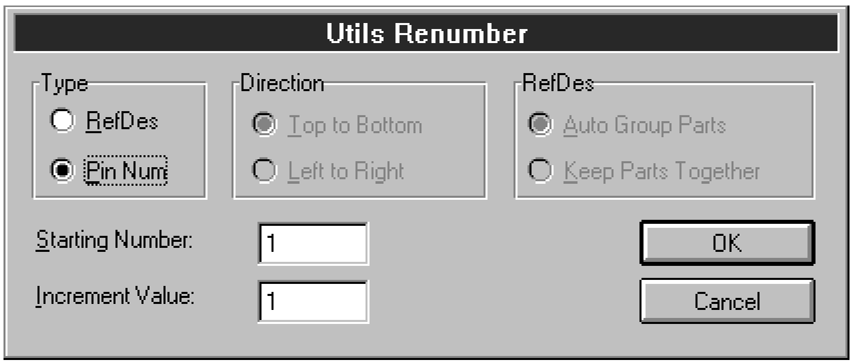
note:
When creating the output pin, set the display characteristics for the outside edge to Dot.



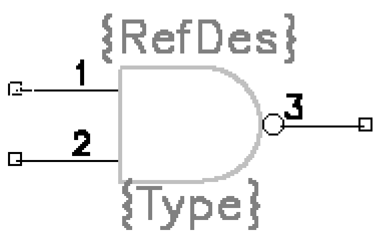
3. Place RefDes and Type Attributes (Place Attribute) as follows:



4. Use the Utils Renumber command to renumber the pins. You need to enable the Edit Select tool first before renumbering.



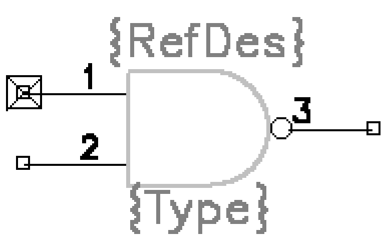
In the dialog, enable **Pin Num** for **Type** and enter 1 as the values for **Starting Number** and **Increment Value**. For symbols, pins must be numbered sequentially, starting with one. Click **OK**, then assign the numbers in the order shown below by clicking on the pins one-by-one.



note:

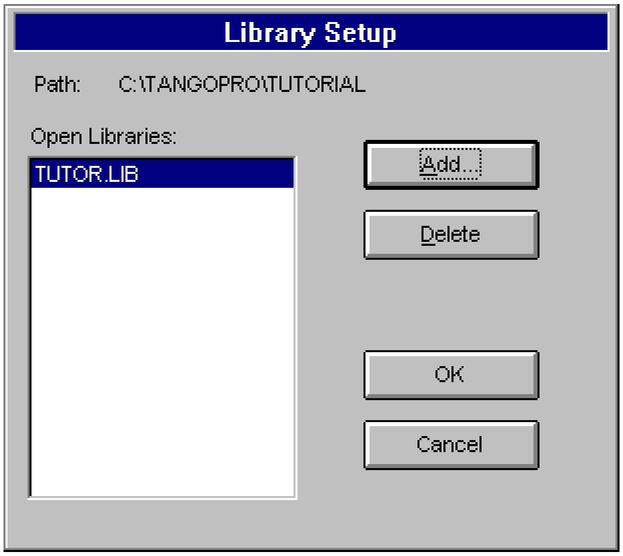
When creating symbols that have common pins, you should renumber the pins starting with the common pin. This allows easier creation of the component later on if the symbol is intended for a heterogeneous component.

5. Use the Place Ref Point command to place a reference point on the gate.



6. Do a block select of all objects to be in the symbol.

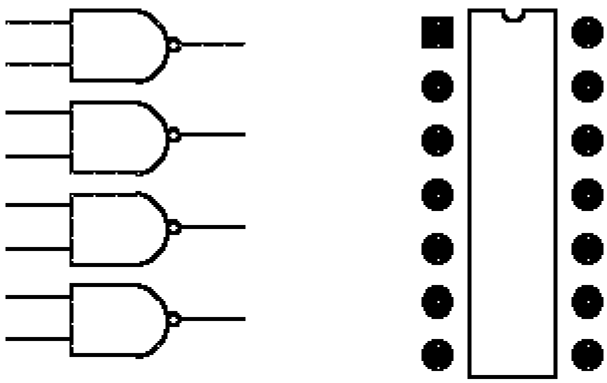
   Use Library Symbol Save As to display the dialog. To save and name the symbol to a library, name the symbol TUTNAND and save it to the TUTOR.LIB library.   Do not enable the **Create Component** check box as you'll be creating the component in the Library Manager.



Now that you have created the pattern and symbol for the component, you can go into the Library Manager to create the component and attach the pattern and symbol.   These steps are described in the next section.

Creating a Component

In this section, we will create a fully-integrated component, suitable for use with both ACCEL Schematic and PCB. The component will have logical pin designators and pin data which corresponds to PCB pattern pads. For our 7400 component, that correspondence is shown as follows:

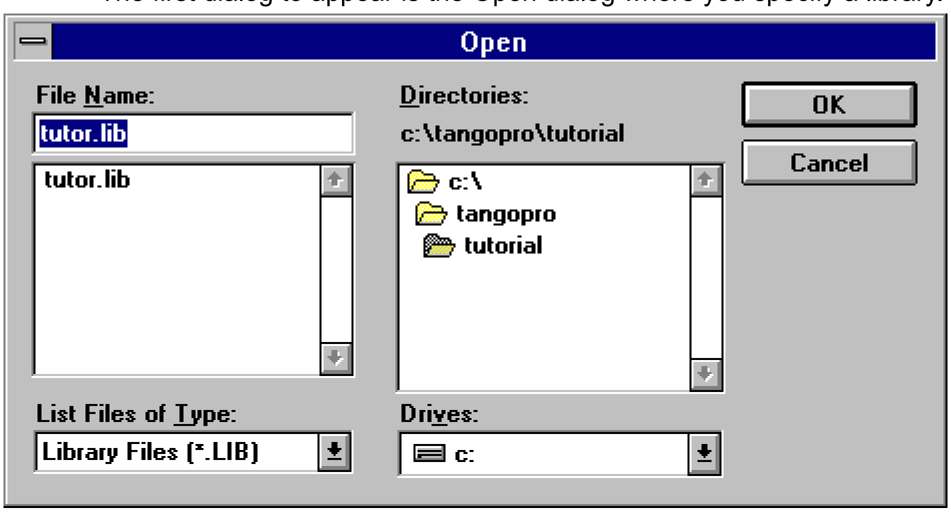


The following instructions show you how to use the Library Manager to create the 7400.

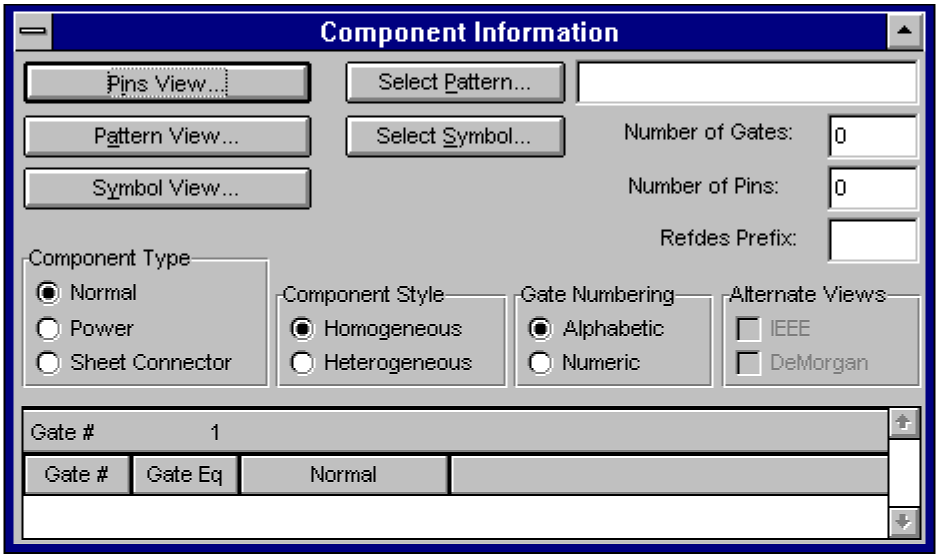
Creating a Component

1. Start the ACCEL Library Manager.
2. Run the Component New command.

   The first dialog to appear is the Open dialog where you specify a library.



3. You should select a library in which you want to save the new component and in which the components symbol and pattern information reside. So for our example, specify the tutorial library, TUTOR.LIB.
4. Click *OK*. The Component Information dialog appears.


Component Information
Pins View...
Select Pattern...
Pattern View...
Select Symbol...
Number of Gates:
0
Symbol View...
Number of Pins:
0
Refdes Prefix:
Component Type
Normal
Power
Sheet Connector
Component Style
Homogeneous
Heterogeneous
Gate Numbering
Alphabetic
Numeric
Alternate Views
IEEE
DeMorgan
Gate #
1
Gate #
Gate Eq
Normal

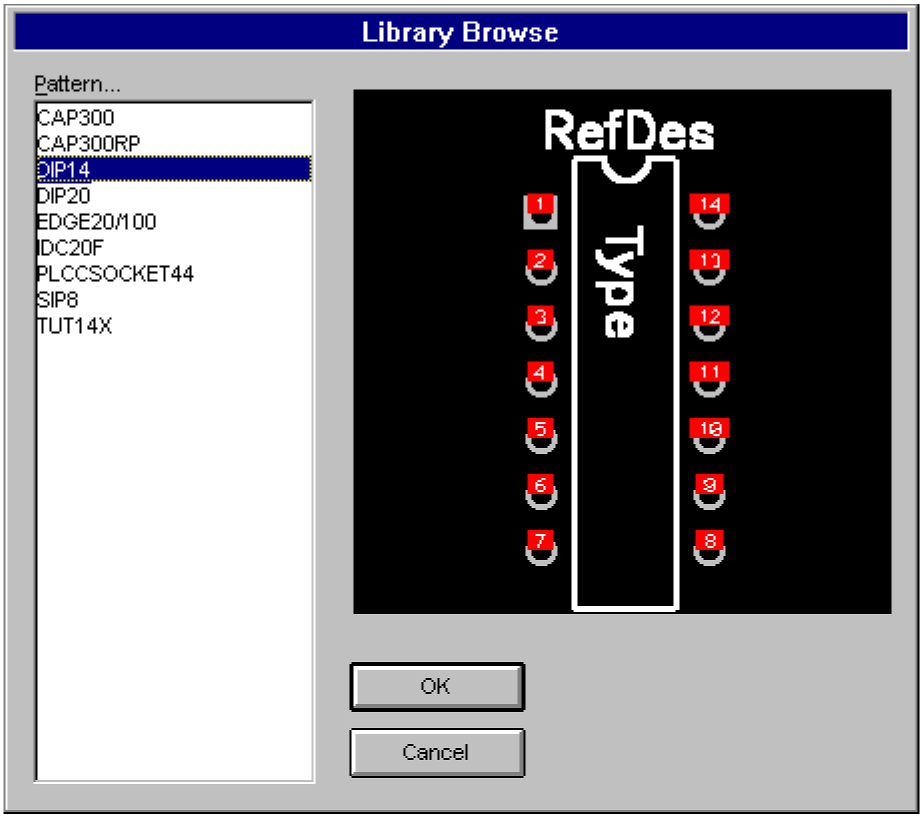
Attaching a Pattern

The next step is to attach the pattern. A pattern is the basic graphical structure that is used to display a component in PCB.

To attach a pattern, it must reside in the same library as the component. In our case, we will use TUT14, the pattern created and saved to TUTOR.LIB earlier in this chapter. If you didn't create TUT14, you can use DIP14, a pattern already in TUTOR.LIB.

1. From the Component Information dialog, click the **Select Pattern** button.

   The Library Browse dialog appears.



2. Attach pattern TUT14 (or DIP14) by selecting it from the list of patterns available in the current library (TUTOR.LIB).
3. After you have specified the pattern name, click **OK**. The Component Information dialog reappears and the pattern name appears in the edit box to the right of the Select Pattern button.

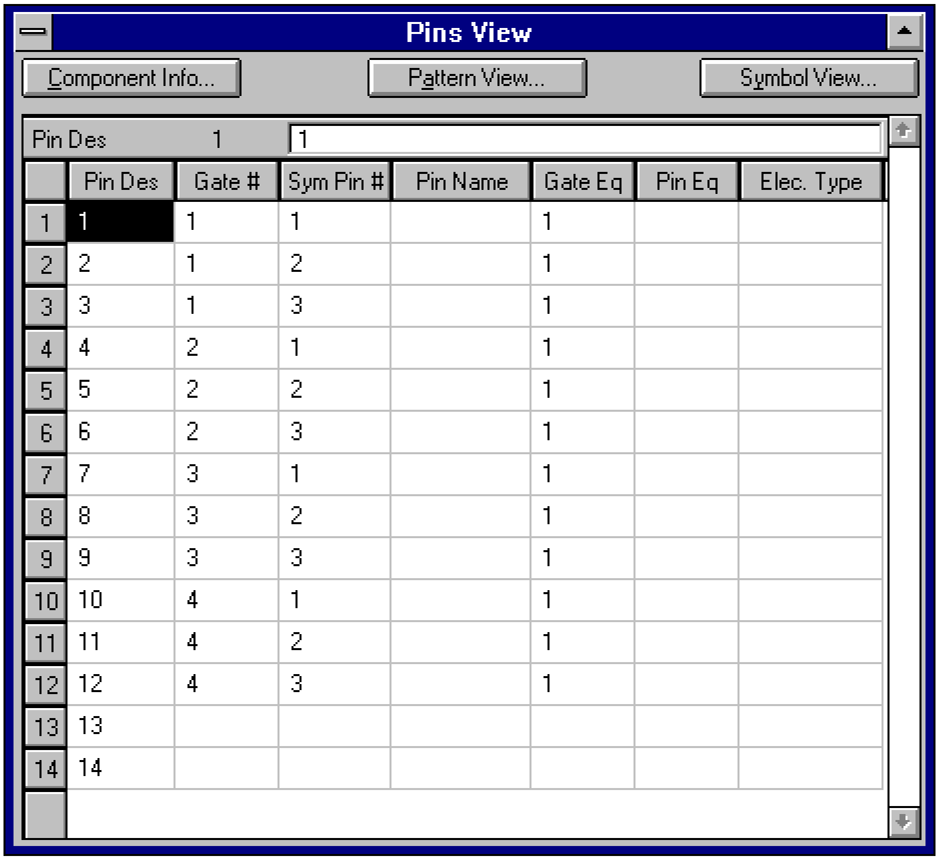
   Notice that the pin designator column automatically fills in the Pins View dialog. The pin designator is a unique name given to the pad/pin association.

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

Editing the Pins View Spreadsheet

Now, its time to update the Pins View spreadsheet.

Once you have finished assigning a symbol, the spreadsheet (Pins View dialog) should appear as follows:



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

The first step is to arrange the pin information so that it corresponds to the pads in a way that matches the 7400 component specifications.

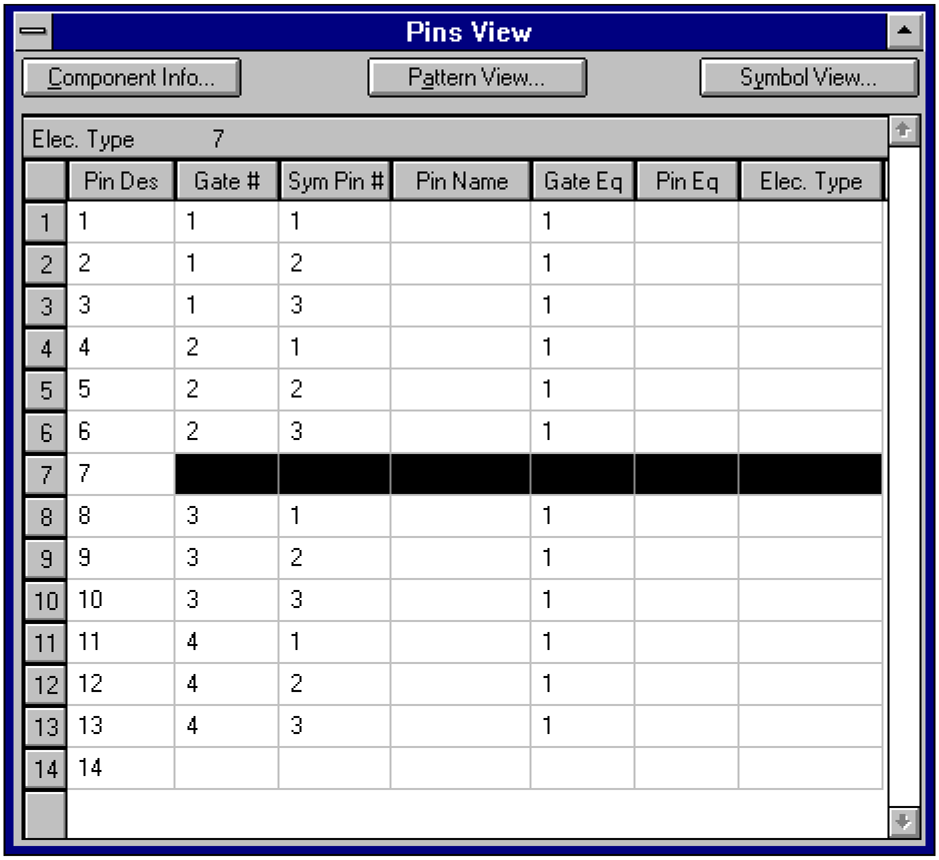
Notice that rows 13 and 14 are blank. This is because there are only 12 pins on the 4 NAND symbols associated with the pattern, but there are 14 pads on the DIP14. These two extra rows will be used to specify the power and ground pins which are associated with pads 7 and 14 respectively.

Lets move the empty pin information in row 13 up to row 7 as a place holder to be filled in later as a power pin.

1. To move the pin information from row 13 to row 7, select all the cells in row 13 except the Pin Designator. Then slide the selected cells up to row 7 using the SlideUp button in the tool bar or by using the Ctrl-UP key sequence.

   Notice that other rows will automatically move down as the selection moves up.

   The spreadsheet should now appear as follows:



Pins View

Component Info... | Pattern View... | Symbol View...

Elec. Type 7

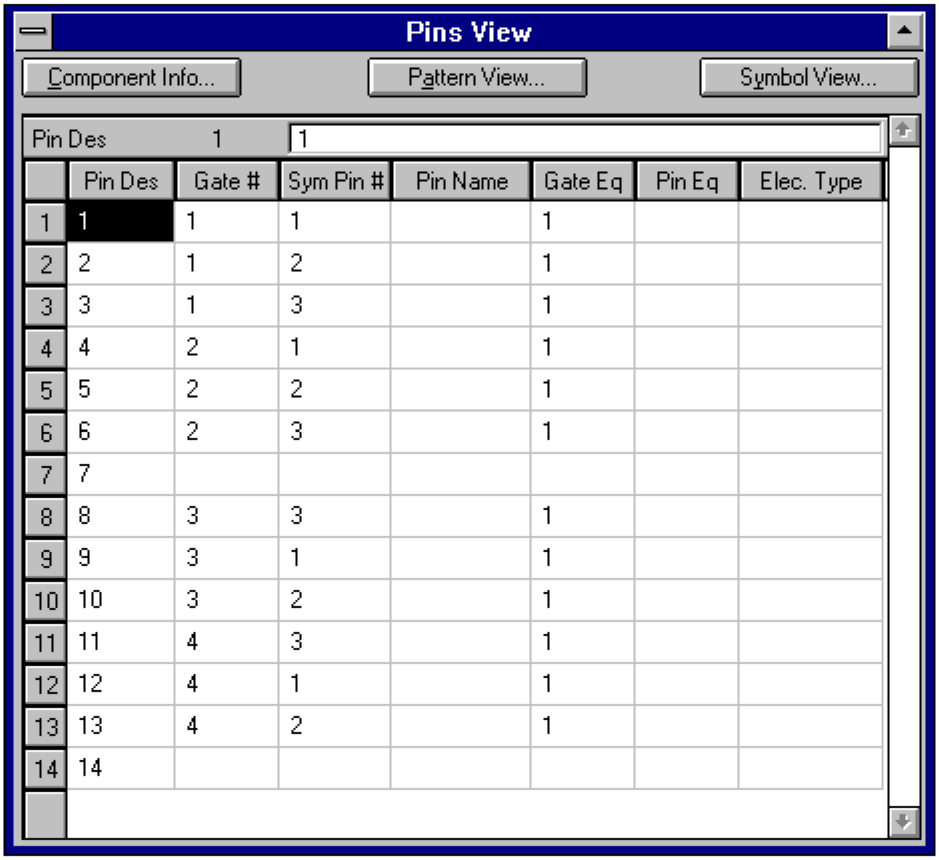
	Pin Des	Gate #	Sym Pin #	Pin Name	Gate Eq	Pin Eq	Elec. Type
1	1	1	1		1		
2	2	1	2		1		
3	3	1	3		1		
4	4	2	1		1		
5	5	2	2		1		
6	6	2	3		1		
7	7						
8	8	3	1		1		
9	9	3	2		1		
10	10	3	3		1		
11	11	4	1		1		
12	12	4	2		1		
13	13	4	3		1		
14	14						

Notice that the pin information for gates 3 and 4 do not correspond to the 7400 component specifications. The pin information for pin 3, gate 3 should be associated with pad 8 and the pin information for pin 3 gate 4 should be associated with pad 11.

You can correct this by simply moving rows of pin information.

4. To move pin 3 up and pins 1 and 2 down, you may either select the pin information for pins 1 and 2 and then slide them down, or you may select the pin information for pin 3 and slide it up. Remember not to select the Pin Designators since they are already correctly associated with the pads.

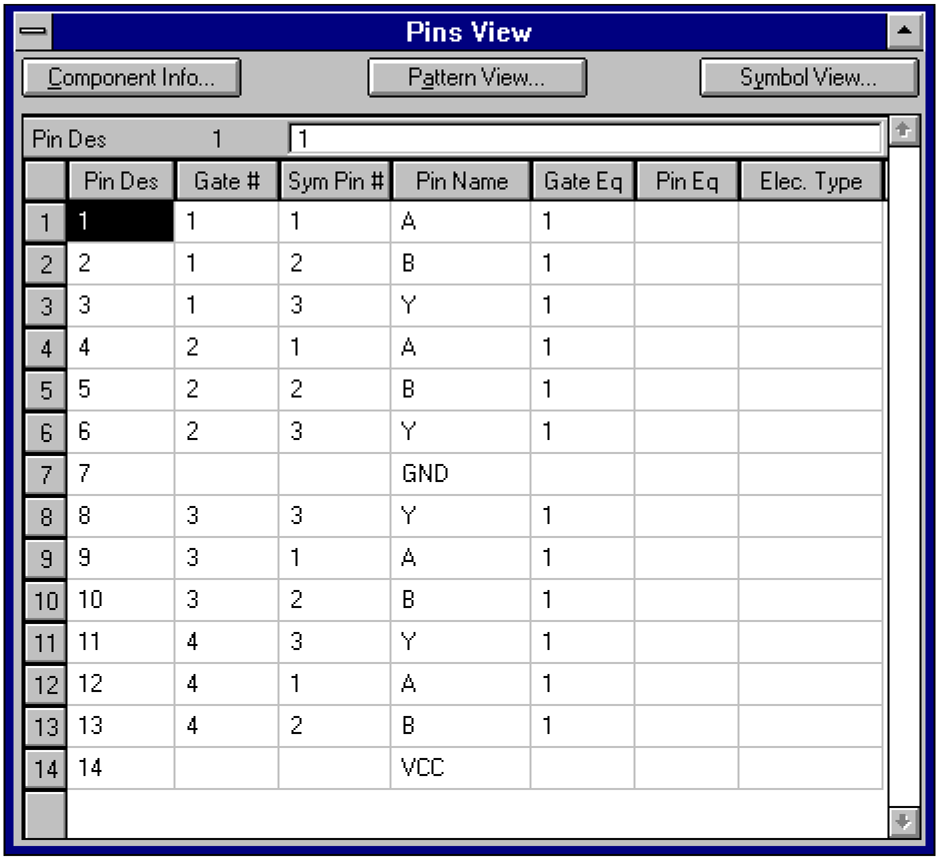
   The spreadsheet should now appear as follows:



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

5. Next, assign pin names for each pin so that they correspond to the names of the pins in our target component specifications.

- You may find it easier to fill in the spreadsheet once youve gotten used to some of the editing features.
- Try filling in the first gates pin names only. Then use the mouse (or cursor keys + Shift) to select those pin names.
- Use the copy button (or *Ctrl+C*) to make a copy of the pin names. Then move to the first pin name cell of the next gate.
- Use the paste button (or *Ctrl+V*) to duplicate the pin names for this gate.
- Move again to the first pin name cell of gates 3 and 4 and repeat the paste operation for each. Dont forget that for this component the pin names are slightly out of sequence for pin 3 in gates 3 and 4.
- Simply move the selection to the pin name cell in row 13 and use *Ctrl+UP* to slide the pin name up two rows.
- Move up to the pin name cell in row 10 and use Ctrl-UP to slide the pin name up two rows.
- Dont forget to fill in the pin names for power and ground in rows 7 and 14 respectively.



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

6. Assign pin equivalence values for the component.

   Pin equivalence values indicate which pins within a gate are logically equivalent and may be swapped using the Utils Optimize Nets pin swap commands. Equivalent pins may be swapped only within the same gate. The pin equivalence values must be non-zero and identical for a swap to occur between two pins. Non-swappable pins are indicated with a blank or zero value.

   For this component, we will indicate that the paired input pins are equivalent by giving each paired input pin the same value.

   You may do this by entering a value of 1 into each of the pin equivalence cells for the input pins.

   Alternatively, as an exercise in column copying, you may copy the gate equivalence column, which closely resembles the values needed for the pin equivalence column, and then remove the unwanted value for the output pins.

- Do this by selecting the **Gate Eq** button over the gate equivalence column.
- Then use the copy button (or *Ctrl+C*) to copy the contents of the selected column.
- Select the **Pin Eq** button over the pin equivalence column and use the paste button (or *Ctrl+V*) to paste the contents of the gate equivalence column into the pin equivalence column.
- Since output pins for this component are not equivalent, select the pin equivalence cell for each output pin and use the *Delete* key to remove the pin equivalence value.
- If you run into trouble, remember that the undo button (or *Ctrl+U*) may be used to undo the last modification to the spreadsheet.

7. Assign electrical types to each pin.

   Note that the electrical types may be entered quickly by 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 which starts with the same letter, typing the letter again will select the next one. For this particular component, the spreadsheet can be filled in quickly by typing:

I	*Enter*
I	*Enter*
O	*Enter*
I	*Enter*
I	*Enter*
O	*Enter*
P P P	*Enter*
O	*Enter*
I	*Enter*
I	*Enter*
O	*Enter*
I	*Enter*
I	*Enter*
P P P	*Enter*

Notice that the *Enter* key accepts the modified value of the cell and automatically moves to the next cell down allowing you to quickly enter a sequence of values for subsequent pins. Notice also that 3 Ps were needed to select the Power electrical type since Passive-H and Passive-L also start with P.

The available pin electrical types are as follows:

Unknown	The default pin type for all pins not having a pin type specifically assigned. Indicated by blank or Unknown.
Passive	A passive pin is typically connected to a passive device. A passive device does not have a source of energy.
Input	An input pin is one to which you apply a signal. For example, pins 1 and 2 on the 74LS00 NAND gate are input pins.
Output	An output pin is one to which the gate applies a signal. For example, pin on the 74LS00 NAND gate is an output pin.
Bidirectional	A bidirectional pin is either an input or an output. For example, pin 2 on the 74LS245 bus transceiver is a bidirectional pin. The value at pin 1 (an input) determines the active type of pin 2 as well as others.
Open-H	(Open Emitter) An open emitter gate omits the emitter pull-up. The proper resistance is added externally. ECL logic uses an open emitter gate and is analogous to an open collector gate. For example, the MC10100 has an open emitter gate.
Open-L	(Open Collector) An open collector gate omits the collector pull-up. This enables the collectors of several gates to be "wired-OR" together and to connect with a single pull-up resistor. For example, pin 1 on the 74LS01 NAND gate is an open collector gate.
Passive-H	A passive pin is typically connected to a passive device. A passive device does not have a source of energy. For example, a resistor lead is a passive pin. Passive-H is a passive pin tied high.
Passive-L	A passive pin is typically connected to a passive device. A passive device does not have a source of energy. For example, a resistor lead is a passive pin. Passive-L is a passive pin tied low.
3-State	A 3-state pin has three possible states: low, high, and high impedance.

When it is in its high impedance state, a 3-state pin looks like an open circuit. For example, the 74LS373 latch has 3-state pins.

Power A power pin expects either supply voltage or a ground. For example, on the 74LS00 NAND gate, pin 14 is VCC and pin 7 is GND (ground).

You will notice that in the last step, entering electrical types, that the power and ground pins were automatically filled in with a value of PWR for the gate number. Since all power pins have a gate number value of 0, the word PWR is used to more clearly indicate a power pin. A value of 0, blank, and PWR are all equivalent. You can save time by ignoring the gate number value of power pins until the electrical types have been entered.

Your finished spreadsheet should look like this:

Pins View

Component Info... | Pattern View... | Symbol View...

Pin Des 1 | 1

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

Saving the Component

After you have attached the pattern and symbols, run the Component Save command to save the component. Use the name TUT7400. The spreadsheet remains open.

If you have followed the complete tutorial, you have created and saved a 7400 component. You can now place the component in ACCEL Schematic or ACCEL PCB.

Pin Designator

The pin designator of each pin in the component. Used by **both** PCB and Schematic.

Pin Name

The pin name associated with that pin designator.   Used by **both** PCB and Schematic.

Pad

The number of the corresponding pad on the attached pattern. Pad numbers must be unique, and they must exist in the attached pattern.. Used by PCB.

Part

The part number defines the part that the pin is associated with. In multi-part components, the parts are uniquely numbered from 1 through n. Used by Schematic.

SymPin

The number of the corresponding pin on the attached symbol.   Pin numbers must be unique and must exist in the attached symbol.   Used by Schematic.

GateEq

The gate equivalence column defines which gates are equivalent. All gates with the same GateEq number are defined to be equivalent. This information is used by ACCEL Schematic when automatically incrementing reference designators and by ACCEL PCB to determine which gates can be swapped during manual or automatic gate swapping.

You cannot set the gate equivalency of pins in the same gate to be different. When you change a part number or gate equivalence for a gate, the spreadsheet updates the gate equivalence field of the other pins of that gate to match. Used by **both** PCB and Schematic.

PinEq

Indicates which pins within a gate are logically equivalent and may be swapped using the Utils Optimize Nets pin swap commands. The pin equivalence values must be non-zero and identical for a swap to occur between two pins. Non-swappable pins are indicated with a zero value. Used by **both** PCB and Schematic.

Elec Type

The type of pin.   Options appear in a popup menu, which is pictured below.   Used by Schematic.

Textbox

The textbox is where you will perform editing of whatever cell you have selected for change or assignment. When a cell is highlighted, the contents of the cell are copied to the textbox. Anything you type replaces the contents of the textbox. When you move to another cell by pressing the Tab key, the contents of the textbox are copied to the cell before moving. You must use the Tab key, the Enter key, or click in the selected cell with the mouse to update the cell; the arrow keys will not update the cell.

Using the Spreadsheet Cells

All the cells are editable, but there are certain restrictions and characteristics. For example, the columns **Pad #**, **Part #, SymPin**, **GateEq**, and **PinEq** only accept numeric values; if you attempt to place a letter or symbol in one of these cells, the value defaults to a number.

Normally, you can edit a cell only by using the textbox at the top of the spreadsheet. When you highlight a cell, the current value appears in the textbox. You can type your change and it automatically replaces the previous one in the textbox. When you press *Tab* to access (or pick and highlight) another cell, the change you specified in the textbox takes effect. If you want to edit what is in the textbox rather than replacing it completely, move the cursor in the textbox and perform normal editing. You can position the cursor with the *Home*, *End*, *PgUp*, *PgDn*, and arrow keys, or use the mouse.

You can select a block of cells by clicking and dragging in the spreadsheet. Anything that is highlighted can be cut or copied. If you reduce the window size such that all the cells are not visible, then scroll bars will appear automatically.

The **Elec Type** cells support a popup menu. Click in the cell to select it, then click the right mouse button in the cell to display the popup menu. Select the electrical type you want from the popup menu and it will change in the cell. You can select:

Unknown
Passive
Input
Output
Bidirectional
Open-H
Open-L
Passive-H
Passive-L
3-State
Power

Components, Patterns, and Symbols

ACCEL integrated libraries contain components, patterns, and symbols.

A component contains the logical and electrical data for a device. Each component consists of one or more logical parts.

A *single* pattern is the basic graphical structure that is used in the creation and display of an *entire* component in PCB.

A symbol is the basic graphical structure that is used in the creation and display of a *single* part in Schematic.

Patterns and symbols contain no information except the shape of their graphic display.

PCB cannot use either a component or pattern by itself, but needs a component/pattern combination. Schematic needs a symbol/component combination. When a component is placed, it references the pattern or symbols which are 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 all reside in the same library.

It is very 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 very efficient. It is also potentially dangerous, as any changes to a pattern or symbol affect all components referencing that pattern or symbol.

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, any components referencing that pattern would reference a nonexistent pattern.

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

You could have a ACCEL library containing many components, but only one pattern and one symbols, and it would be complete as long as all the patterns and symbols that the components reference exist in the library.

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 without having to alter the original name.

Aliases are preferable for a number of reasons: it is 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.

Pin Numbers vs. Pad Numbers vs. Pin Designators

An important fact to understand about components and pattern or symbols is that ***pad numbers do not equal pin designator values*** and that ***symbol pin numbers do not equal pin designator values***, except sometimes by a convenient coincidence. Pad numbers are just identifiers for pads, used to cross-reference with the pin designators when assigning the pin designator values. Symbol pin numbers are just pin identifiers. Using an ordered, linear, pad or pin numbering system makes identifying pad numbers and 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 pin numbers, whereas components contain pin designators. Each pin designator references one pad number and one pin number.

Creating a New Component

1. Select Component New. The dialog of the same name is displayed.
2. Click the **Library** button to display a list of libraries to choose from. The library you choose will be the destination library for the component you are creating.
3. Type in the **Number of Pins**. Type in the **RefDes Prefix**, if necessary. Reference designator prefixes are useful for identifying and grouping components; for example, R for resistors, C for capacitors, Q for transistors, and U for ICs. Type in the name of the component in the **Component Name** box.

   Click **OK** and the Library Manager spreadsheet is displayed.
4. Your component is now created in memory, but you need to attach a pattern and symbols, assign pins, save the component, etc. Refer to Edit Select Pattern, Edit Select Symbols, and Component Save.

Drag and Drop Library Load

You can drag and drop a library file (.LIB) into the Library Manager. From your File Manager (or another Windows file management utility), click on a filename icon and drag it into the ACCEL window and release. The specified library file will be loaded. When you run Component New or Component Open, the library path will already be established.

PDIF Translation

This help panel provides the steps you need to follow to create a fully integrated ACCEL library from a PDIF Schematic library and its corresponding PDIF PCB library.

To create an integrated ACCEL library from PDIF schematic and PCB libraries, the schematic and PCB libraries must first be translated into ACCEL format, then merged into a single ACCEL library as described below.

1. Select the Library Translate command to display the Library Translate dialog.
2. Select PDIF as the source library type by clicking the **PDIF Source Format** radio button. The dialog displays:

PCB Library Translation

3. Click the **PDIF Cross Reference** button to display the Library File Listing dialog, where you can choose the PDIF cross reference file which corresponds to the library you want to translate.
4. Click the **Source Library** button to display the Library File Listing dialog, where you can choose the PDIF PCB library to translate.
5. Click the **Destination Library** button to display the Library File Listing dialog, and choose the destination library name.
6. Click **Translate** to begin the translation process. When all components have been translated and new alias names have been added, 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.

Schematic Library Translation

7. Repeat steps 3 through 6 to translate the appropriate PDIF Schematic Library.

   You should now have two ACCEL library files: one from a PDIF PCB library; the other from a PDIF Schematic library.
8. Review the log files to make sure that all power and ground pins were defined and that important information hasnt been lost.
9. Click **Close** to exit the dialog.

Merging the Libraries

To merge the libraries, run the Library Merge Patterns command discussed below. 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 ACCEL library, you can delete the pattern-only library or use it for PCB placement if you dont need integrated library components.

SCHEMA Translation

The library translator for SCHEMA displays the following behaviors:

- The section IDs for multiple part components are ignored and replaced with an alphabetic part numbering scheme. In SCHEMA and SCHEMAx, the section IDs are arbitrary, but in ACCEL they must be either numeric (1, 2, 3, 4) or alphabetic (A, B, C, D).
- If a component or multiple references a template that is in another library (that is, not in the current library), the resulting ACCEL component will not have an attached symbol and will have no pins. We suggest you copy the symbols into the library using the Library Copy command and retranslate the library.
- The eight visible user-defined attributes PARAM1...8 are no longer visible, but instead become part attributes. Place holders are created in the symbol for PARAM2...8 (PARAM1 becomes the Value attribute). They simply do not have values assigned.
- Pin and pin override swap groups are ignored. Multi-part, homogeneous components will have all parts in the same swap group.
- Connectors (components prefixed with "^^") translate into homogeneous, multi-part components with the maximum number of parts allowed in ACCEL (currently 255). Each part will contain one or more pins.
- Bitmaps are ignored in all library objects.
- Color values are ignored.
- Filled circles are translated as unfilled.

Converting Libraries

Unlike Tango-Schematic and Tango-PCB, which use separate symbol and pattern libraries, ACCEL applications use integrated libraries, which contain components, symbols, and patterns.

To use your Tango pattern and symbol libraries with ACCEL Schematic and ACCEL PCB, you must translate them first using the Library Manager. Once your libraries have been translated, if you intend to use them with both ACCEL applications, you need to merge the converted pattern libraries into the converted symbol libraries.

This help topic uses a simple example to walk you through three conversion scenarios:

1. You want to convert Tango-PCB libraries into ACCEL format.
2. You want to convert Tango-Schematic into ACCEL format.
3. You want to create integrated ACCEL libraries from converted pattern and symbol libraries.

Conversion Examples

For our 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
Tango-Schematic Only
Tango-PCB and Schematic

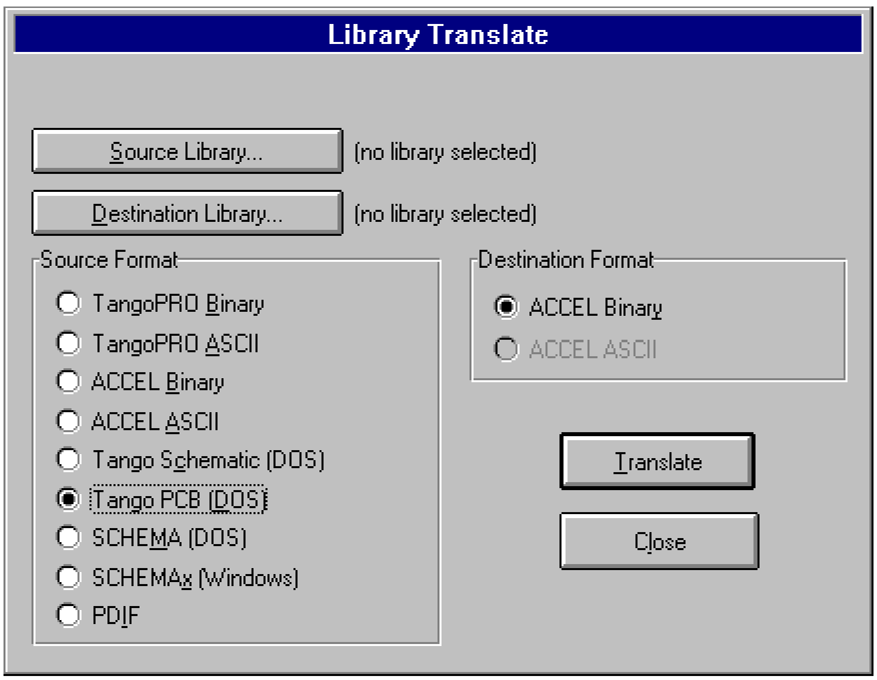
Tango-PCB Only (Library Conversion)

To translate your Tango-PCB pattern libraries, run the Library Translate command. The Library Translate dialog appears.

1. First, we 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 our example, we would choose PCB1.LIB.)
3. Click the **Tango-PCB (DOS)** radio button in the **Source Format** area.

   The only available destination format is **ACCEL Binary**.
4. Click the **Destination Library** button to display the Library File Listing dialog, and type a name for your destination library. (In our example, we could type PROPCB1.LIB.)

   Your dialog now appears as follows:



5. Click **Translate** to begin the translation process.
6. Repeat the process for each pattern library you want to convert. (In our example, we would now translate PCB2.LIB, and name it PROPCB2.LIB.)
7. Click **Close** to exit the dialog.

The translated libraries are now ready to use with ACCEL PCB, but not with ACCEL Schematic, because they don`t have symbol information.

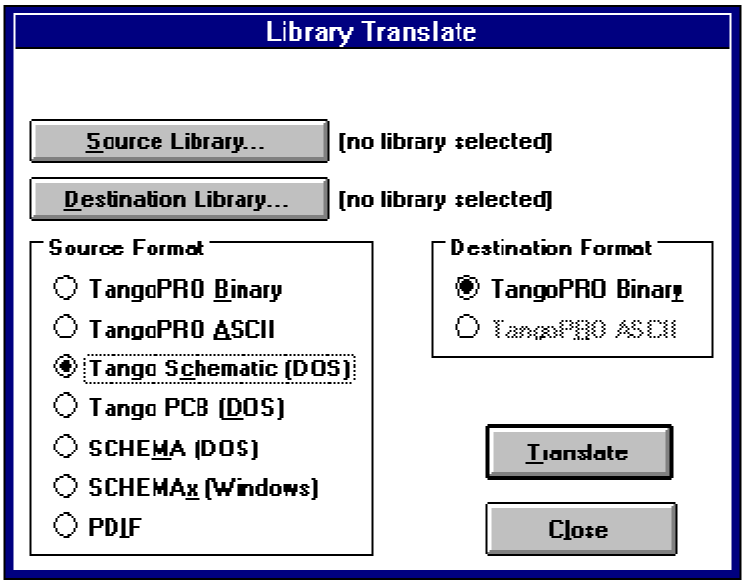
Tango-Schematic Only (Converting Libraries)

To translate your Tango-Schematic symbol libraries, run the Library Translate command.   The Library Translate dialog appears.

1. For our example, we would translate SCH1.LIB.
2. Click the **Source Library** button to display the Library File Listing dialog, where you can choose your libraries.   (In our example, we would choose SCH1.LIB.)
3. Click the **Tango-Schematic (DOS)** radio button in the **Source Format** area.

   The only available destination format is **ACCEL Binary**.
4. Click the **Destination Library** button to display the Library File Listing dialog, and type a name for your destination library.   (In our example, we could type PROSCH1.LIB.)

   Your dialog now appears as follows:



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 ACCEL Schematic, but not with ACCEL PCB because it doesn`t have pattern information.

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

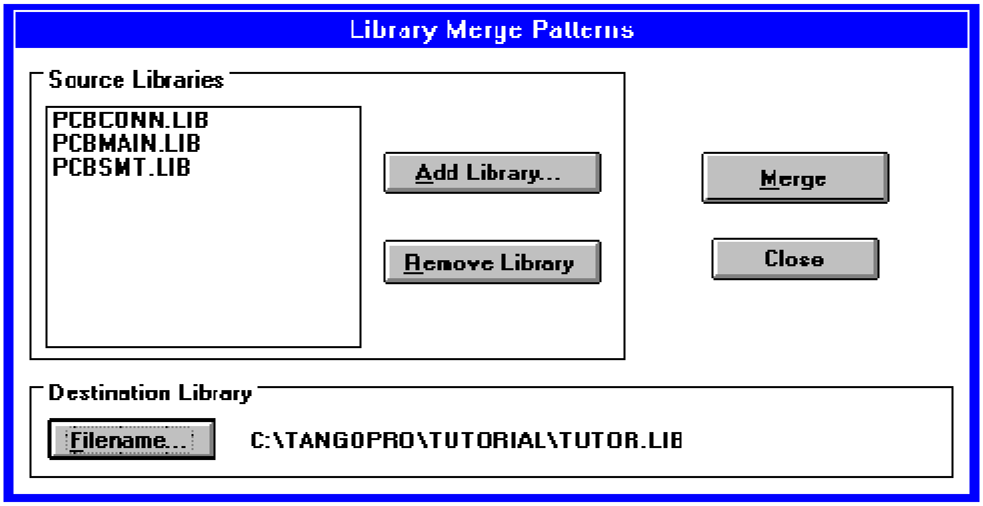
where DESTLIB is the name of the destination library, XX is a unique number, and A equals N(ormal), I(EEE), or D(eMorgan).

Tango-PCB and Schematic (Converting Libraries)

When all your pattern and symbol libraries have been converted to ACCEL format, use the Library Merge Pattern command to merge pattern and symbol libraries into integrated pattern/symbol/component libraries which can be used by all ACCEL applications.

In our example, we would merge PROPCB1.LIB and PROPCB2.LIB into PROSCH1.LIB.

1. Select the Library Merge Patterns command 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.

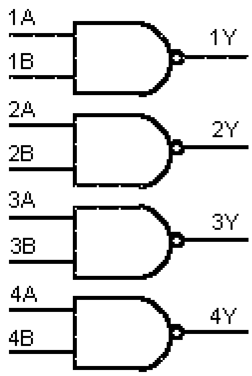
   You need to select all the pattern libraries that contain the patterns referenced in all components in the destination library. (In our example, you would choose PROPCB1.LIB and PROPCB2.LIB.)

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 our 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.

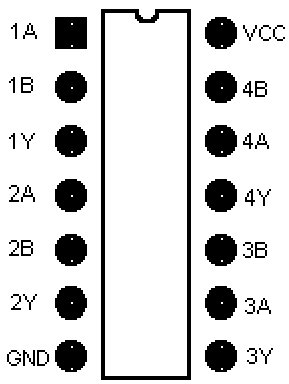
Your destination libraries should now be fully integrated and ready to use with both ACCEL Schematic and ACCEL PCB. (Following our example, PROSCH1.LIB is a fully integrated library

Target Component Specifications (tutorial)

A 7400 is a component consisting of four independent 2-input NAND gates. The logic diagram for our 7400 appears as follows:



This component is packaged in a DIP package as follows:



We`ll use these specifications throughout this tutorial to build our hypothetical 7400 component.

Common Pins

You can use the spreadsheet in the Pins View dialog to create common pins by defining a pin with a gate number of CMN.   Only one entry in the spreadsheet is used to define a common pin for a component or set of gates.

The scope of a common pin is defined by the pins gate equivalence value.   If the gate equivalence value is set to a specific gate equivalence value then the common pin is only applied to gates with that gate equivalence value.   If the gate equivalence value is 0 then the common pin applies to all gates in the component.

Pin Equivalence

Common pins are not pin swappable and, therefore, have a pin equivalence value of 0.

The Pin Eq value is required to be 0 for common pins.   As a component is saved, if the Pin Eq is not 0, then they will be automatically changed to 0.   No error will be given.

Heterogeneous Components

A common pin may be defined for all gates in a component by setting the pin gate equivalence value to 0, or a common pin may be defined for all gates in the component having the same gate equivalence value.   A common pin may not be defined for multiple gates within a component having different gate equivalence values unless the pin is defined for all gates as described above.

Since pin numbers must start at 1 and be continuous, you must be careful when defining a common pin that spans multiple gates.   You must give the common pin a pin number that is valid for *all* gates for which it is defined.   For example, a common pin defined in a component where one of the gates has 3 pins, including the common pin, may not have a pin number 4.   For heterogeneous components, the easiest way to guarantee a correct result is to begin common pin numbering at 1.

common pins

A common pin is a pin that exists in more than one gate within a component and maps to a single PCB component pad.

Utils Query

The Query command allows you to retrieve information from the library database.

When you run this command, the Utils Query dialog appears with the Search Filters tab selected.

Tabs

The dialog has these additional tabs:

- Search Filters Tab
- Output Fields Tab
- Sort Key Tab
- Report Format Tab

Search Filters Tab

The Search Filters tab allows you to specify filtering parameters used to filter the query output data. Filtering is an AND function. You are specifying that you want an output query which contains every record in every selected library which matches all of the specified filtering criteria.

- **Libraries:** These are the libraries to be queried. Use the **Add** and **Remove** buttons to add and remove libraries to the **Selected** list box.
- **Standard Filters:** This area lets you perform field filtering by selecting a field from the **Field Name** list, entering a value for the selected field in the Field Value edit control, and clicking the **Add** button.

  The **Field Value** edit box has a wildcard capability which allows you to enter a value containing the wildcard characters asterisk (*) and question mark (?). The asterisk is used to represent character strings and the question mark represents a single character. A value may contain any number of wildcard types.

  The **Remove** button allows you to remove highlighted single or multiple items.

  The **Selected** list is a two-column list displaying the selected filters in the format ***field name = field value***.
- **User Defined Filters:** Use the **Field Name** and **Field Values** boxes to enter a field name-field value pair which is not listed in the **Field Name** list, then click the **Add User Defined** button to append it to the **Selected** list.

  The **User Defined** edit control has the same wildcard capability as the **Standard Filters Field Name** edit box.

Output Fields Tab

The Output Fields tab allows you to build a list of fields to be included in a query by selecting (highlighting) the desired field in the **Available Fields** list and clicking the **Append** or the **Insert** button. The order of the **Selected Field** list is the order in which the fields are written to the query output, where the field names are the column headings.

- **Available Fields:** Lists available query fields in the database.
- **Selected Fields:** Lists selected query fields. The default Selected fields, in query order, are Type, Value, Pattern Name, Symbol, Library Name, and Description.
- **Append Button:** When you click this button, the new item is added after any highlighted item, or at the end of the list if no Selected item is highlighted.
- **Insert Button:** When you click this button, the new item is inserted before any highlighted item, or at the beginning of the list if no Selected item is highlighted.
- **Remove Button:** Reverses the **Append** and **Insert** button actions by removing a highlighted field(s) from the **Selected Fields** list.
- **Add User Defined Button:** This button allows you to enter a field name which is not listed in the **Available Field** list, then click the **Insert** or **Append** button to add it to the **Selected Field** list.

Sort Key Tab

The Sort Key tab allows you to specify sort keys for the query output.

- **Available Keys:** Lists fields found in the **Selected Fields** list of the Fields tab. To add a sort key, select a field from this list.
- **Selected Keys:** Lists selected sort keys.
- **Append Button:** When you click this button, the new item is added *after* any highlighted item, or at the end of the list if no Selected Keys field is highlighted.
- **Insert Button:** When you click this button, the new item is inserted *before* any highlighted item, or at the beginning of the list if no Selected Key field is highlighted.
- **Remove Button:** Reverses the **Append** and **Insert** button actions by removing a key from the **Selected Keys** list.

Report Format Tab

The Report Format page allows you to define the format of the query. You can specify the report format, the report style format, and output options.

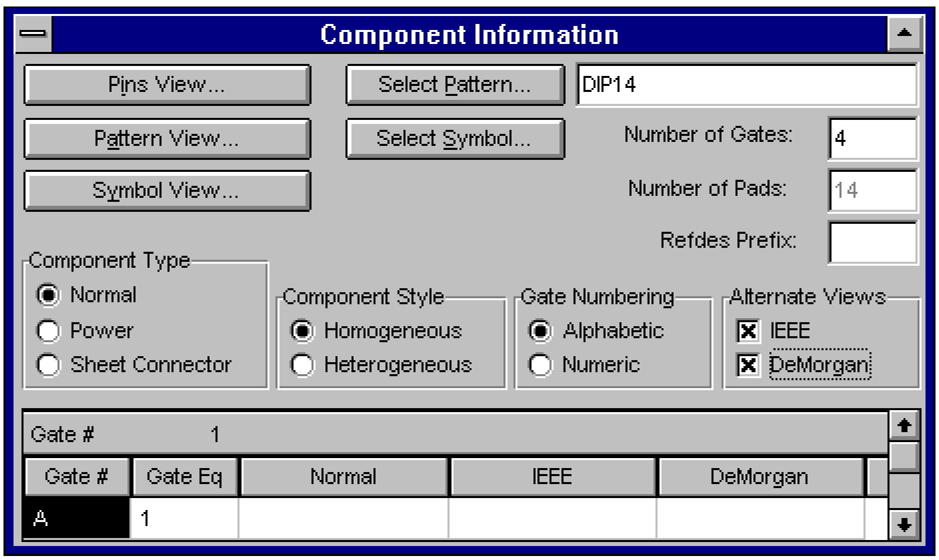
- **Header/Footer:** Enter any header or footer text you want to appear on the query output.
- **Page Format: Use Header** and **Use Footer** include the information you specified in the header and footer dialog fields. **Date/Page** includes the current date and the page number. **Pagination** allows you to create your own pagination (lines per page).
- **Style Format: Comma Separated** puts all data in comma-separated format. This format can be imported into other spreadsheet and database programs. **Report** produces a report format with columns and spaces, etc.
- **Output Destination:** These options determine the source of your output. **Screen** sends the output to a file and opens the file using the selected File Viewer. **Printer** sends output directly to the printer without creating files. **File** sends the output to a file.
- **Lines per Page:** This option allows you to specify the number of lines per page in your query report.
- **Column:** This option allows you to specify the number of columns per line.
- **Output Filename:** This box displays the name of the report if you choose to save it to a file. The default filename is library.rpt. Use the **Filename** button to change the default name.
- **File Save:** This button allows you to save the report to a name other than the default name. The filename appears in the **Output Filename** box.
- **Output File Viewer:** This box displays the name of the editor which will be used to view the output file. The default viewer is NOTEPAD.EXE. Use the **File Viewer** button to change the viewer.
- **File Viewer:** This button allows you to change the name of the Output File Viewer.

Adding Component Properties

From the Component Information dialog, you need to set the component type, set the number of gates in the component, set alternate representations of the component, and set the Refdes prefix.

1. To enable alternate representations of the component, select **IEEE** and **DeMorgan** from the **Alternate Views** box.
2. Choose the **Homogeneous** radio button in the **Component Style** box.

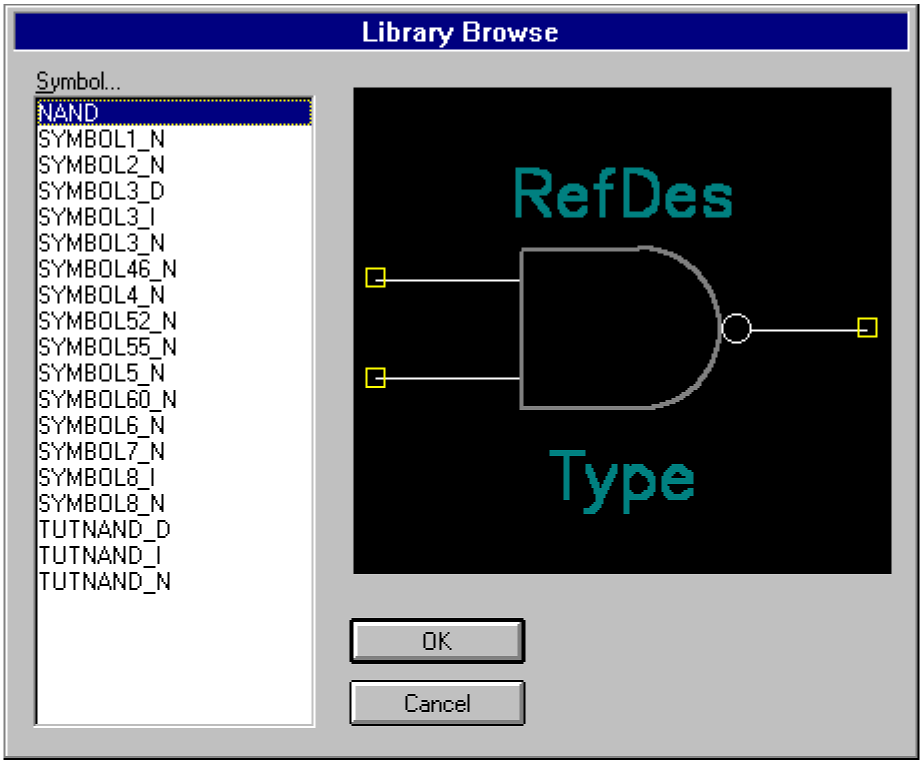
   This means that all four gates in this component use the same symbol. A heterogeneous component may use different symbols for each gate.
3. Choose the **Normal** radio button in the **Component Type** box.
4. Choose the **Alphabetic** radio button in the **Gate Numbering** box. This means that the four gates in this component will be numbered A, B, C, and D.
5. Type the number of gates for the component. Type 4 since the 7400 has four independent 2-input NAND gates.
6. Type U as the **Refdes Prefix**.



Attaching a Symbol

Next, we will attach a symbol. A symbol is the basic graphical structure that is used in the creation and display of a component in Schematic. To attach a symbol, it must reside in the same library as the component. We will use TUTNAND, the symbol created and saved to TUTOR.LIB earlier in this chapter. If you didn't create TUTNAND, you can use NAND, a symbol already in TUTOR.LIB.

1. Attach symbol TUTNAND for the normal representation of gate A by clicking the Select Symbol button or by placing the cursor in the corresponding cell and double clicking .



2. When the Library Browse dialog appears select TUTNAND_N from the list of symbols.

   Because this is a homogeneous component, the symbol automatically fills in for the remaining three gates.

3. Assign symbols for the alternate representations. You could have created these symbols as you did the symbol for the normal representation, but we did it for you.

4. Place the cursor in the **IEEE** cell for gate A, double-click and select TUTNAND_I from the list of symbols. The same symbol automatically appears in the columns of the remaining gates.

5. Place the cursor in the **DeMorgan** cell for gate A, double-click and select TUTNAND_D from the list of symbols. The same symbol automatically appears in the columns the remaining gates.

Notice that the pin number, gate number and gate equivalence columns are automatically placed in the Pins View dialog.

Gate Equivalence values indicate which gates within a component, or between components of the same type, are logically equivalent and may be swapped using the Utils Optimize Nets gate swap commands. For two gates to be considered logically equivalent, the **GateEq** values must be identical, and non-zero. Additionally, equivalent gates must have the same number of pins. A zero value (shown as a blank in the spreadsheet) indicates that the gate isnt swappable. See the Utils Optimize Nets command section in the PCB Users Guide for information on swapping gates.

Edit Slide Selection Up *(Ctrl+Up)*

This command slides the selected spreadsheet information up one row in the spreadsheet.   You can use this command to slide cells, partial or entire rows, and even more than one row.

Edit Slide Selection Down *(Ctrl+Down)*

This command slides the selected spreadsheet information down one row in the spreadsheet. You can use this command to slide cells, partial or entire rows, and even more than one row.

Master Designer Command Reference

Set Libs & Search Paths

Library > Setup (SCH & PCB)

Library Manager - Component > Open > Library button

Library Maintenance

Library Manager

Symbol Editor

Not Required (see SCH)

Part Editor

Not required (See PCB)

Component Editor

Library Manger

Report Editor

Not Necessary.   Reports can be directly output to the screen or a user preferred file viewer can be selected in Windows to view previously generated reports