Chapter 1

Design Process Management

After the installation is complete, a new program group called ACTIVE-CAD will appear (Figure 1-1). It may contain some or all of the following ACTIVE-CAD icons:

❒ *Project Manager*; allows to control the entire project environment, including ACTIVE-CAD programs and other vendor's EDA tools.
❒ *Schematic Editor;* starts hierarchical schematic editor.
❒ *Simulator;* starts functional and timing simulators which work on-line (real-time) with the schematic editor.
❒ *Library Manager;* allows to select additional libraries and create new models, including clones of existing models.
❒ *Test Vector Editor*; it is an independent off-line application. Several test vector editors are also available from the simulator program.
❒ *Symbol Editor;* allows creating symbols for schematic and FSM editors.
❒ *Key Utilities;* lists authorized software products, options and libraries.
❒ *Uninstall*; removes all ACTIVE-CAD files from the hard disk.
❒ *Release Notes;* lists software upgrades and newly discovered bugs.
❒ *VHDL*; it is a 1076-IEEE VHDL editor and compiler
❒ *MOBIC;* compiles Boolean equations into fast functional models.
❒ *State Machine Editor;* used for entry of designs as "bubble" diagrams.
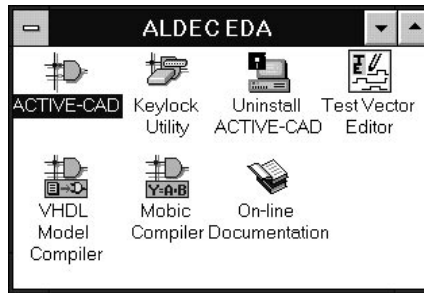❒ *Static Timing Analyzer*; calculates all possible design timing violations.

Figure 1-1. ACTIVE-CAD Program Group.

Project Manager

Project Manager controls the entire design process. It keeps track of the design revision levels, libraries, macros, test vector files and other design resources used in the design process. It is automatically activated by all ACTIVE-CAD applications. For example, starting schematic editor will automatically open Project Manager, which will be listed in the taskbar of Windows 95 or displayed as a minimized icon in Windows 3.1 and NT.

Project Manager can also be opened from the **File** menu of several ACTIVE-CAD applications such as schematic editor and logic simulator.

Double clicking on the **ACTIVE-CAD** icon in Figure 1-1 starts the Project Manager and displays a screen with menus, icons, design hierarchy and a flow diagram with buttons for invoking various design applications (Figure 1-2). The last design will be automatically loaded and the **Project Manager** will be configured for that application. This means that all ACTIVE-CAD files and applications and other programs such as XACT, X-BLOX, ACTMAP, EXEMPLAR, etc., may be loaded and displayed as control buttons in the design flow diagram.

The Project Manager performs the following functions:

❒ Controls the design revision level
❒ Automatically loads all design resources when a project is opened
❒ Checks if all project resources are current and available
❒ Shows the design process flow
❒ Provides buttons for launching various application
❒ Places all error and status messages in one window
❒ Allows setups of all tools from one window
❒ Automatically translates data while it is moved between tools
❒ Provides the design status information
❒ Brings your DOS-based tools into Windows environment

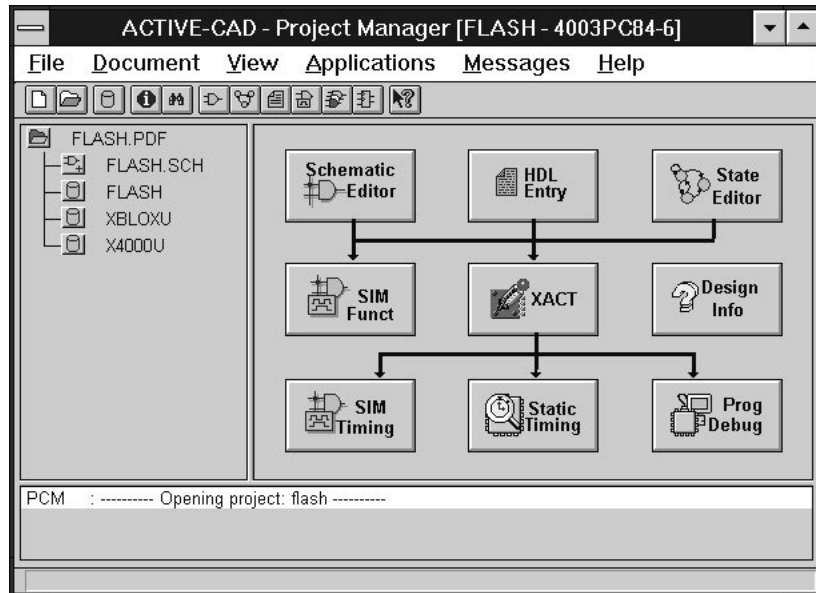❐ Works with network-based workstation EDA tools

Figure 1-2. Project Manager.

The **Project Manager** window is comprised of six (6) screen areas:

1. The title bar indicates if the window is active. It also displays the current project's name and part being designed.

2. The main menu at the top of the screen allows access to all ACTIVE-CAD tools and provides for editing, loading, saving and performing other project management operations

3. The icons below the main menu duplicate some of the main menu functions to allow quick execution of the most popular operations. The first three (3) icons are used for project management, and the remaining seven icons are used for activating various ACTIVE-CAD tools.

4. The Hierarchy Browser field shows the hierarchical project structure, including schematic sheets and macros.

5. The Design Process Flowchart includes buttons for direct control of EDA tools involved in the current project. This flowchart may look different for different projects. For example, the XILINX Project Manager will look completely different from the one for Cypress.

6. The field at the bottom of the screen lists messages from tools involved in the design process . The left-most column of the message identifies the EDA tool that has generated the associated message.

You can open a design by starting any ACTIVE-CAD application. Each of them will automatically invoke the **Project Manager** widow in Figure 1-2.

Note: To double-click means to press on the left mouse button twice in rapid succession.

The **Project Manager** shown in Figure 1-2 is the design process manager. It displays all available software tools that may be used for the selected design type or application. Clicking on the **Design Info** button displays a window with information on design status and presence/validity of the design files.

The Project Manager provides automated data transfer between tools involved in processing your designs. Several examples of the **Project Manager** windows are shown and described in reference to the specific design applications (see *Xilinx Applications, Actel Applications,* etc.).

NOTE: Clicking on a button in the **Project Manager** flowchart, automatically launches the selected application and feeds the input design data.

Clicking on the **ACTIVE-CAD** icon displays Project Manager screen in Figure 1-2. Activating any other application will produce a minimized Project Manager in the taskbar of Windows 95 and a minimized application icon in Windows 3.1.

Basic Design Operations

To speed the most popular design operations the **New Project** , **Open Project** and **Libraries** operations have their own icons, located below the **File** menu.
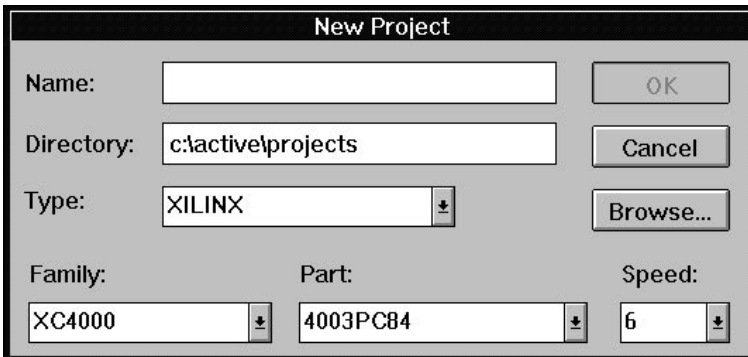

Figure 1-3. Design Manager Icons.

Starting A New Project

To start a new project click on the the **New Project** button ⬜ , which is located in the top left-hand corner of the **Project Manager** window. In response, ACTIVE-CAD opens the **New Project** window. To set up the new project environment, select the following items:

1. Enter the project's name into the **Name** field.

2. Select the project directory. Typically, you should use the *Projects* directory that is listed by the prompt. If a different directory is needed, enter into the **Directory** field the new project's directory or click on the **Browse** option to select the desired directory from another disk.

3. Click on the **Type** field and select the project type from a list provided by ACTIVE-CAD. The design **Type** will determine the **Project Manager**'s flowchart and its control button arrangement.

4. Select the **Family**, **Part** and **Speed** options when available for the the selected ASIC, FPGA and CPLD family.

5. Click on the **OK** button to complete the new project selection. All your entries and selections will appear in the **Project Manager**'s title bar.

Figure 1-4. New Project Window.

NOTE: If you are using ACTIVE-CAD for the first time, you are given the choice of creating a new project or opening the existing one.

ACTIVE-CAD is designed to work with one project at a time. The project is a set of resources like schematics, netlists, libraries, test vectors, etc., selected for the specific design.

The following is a typical design flow:

❐ Open a new project in Project Manager

❐   Create a schematic using the Schematic Editor
❐   Start the Simulator
❐   Select test points for applying test vectors
❐   Design test vectors
❐   Exercise the schematic design with the test vectors
❐   Save and print design analysis results

After each design session, save the project so that it can be used the next time around. When you open a project, ACTIVE-CAD will automatically load the previously saved schematic to the Schematic Editor.

Project contents are stored in a file with a *.PDF extension. This file is located in the ACTIVE\PROJECTS directory. The actual project files are stored in separate project directories, called the Project Working directories.

Once the new project type is selected, the Project Manager will automatically display a new flowchart with control buttons for handling all available programs for the selected design type.

NOTE: The **Project Manager** creates all necessary directories and files to start a new project.

Opening An Existing Design

To open an existing design, click on the 📂 icon. In response, ACTIVE-CAD displays the **Open Project** window (Figure 1-5).
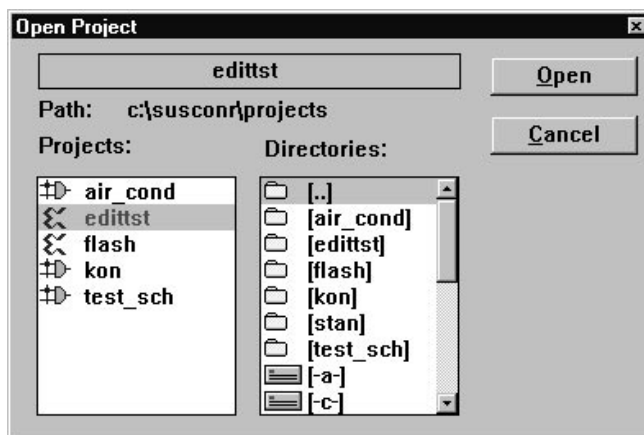

Figure 1-5. Open Project Window.

The last project's name is displayed at the top of the window. To select another project from the **Open Project** window, double-click on the selected project in the **Projects** field or select the project with the mouse pointer and then click on the **Open** button.

The **Directories** field lists all hard drives in the systems and all directories of the selected drive. You can access any project residing in these directories, using the standard Windows procedures.

NOTE: Invoking any ACTIVE-CAD application always starts the Project Manager which completely restores the last project you have been working on.

Selecting Project's Libraries

To select project's libraries, click on the **Project Libraries** icon ⬛ , or click on the **Project Libraries** option in the **File** menu of the simulator or schematic editor and the window in Figure 1-6 will appear. The **Project libraries** window (on the right) lists all libraries assigned to the current project. The **Attached libraries** window (on the left) lists all libraries that are available in the ACTIVE-CAD system and which can be added to the current project. The term *attached* library means that ACTIVE-CAD knows where the library is located.
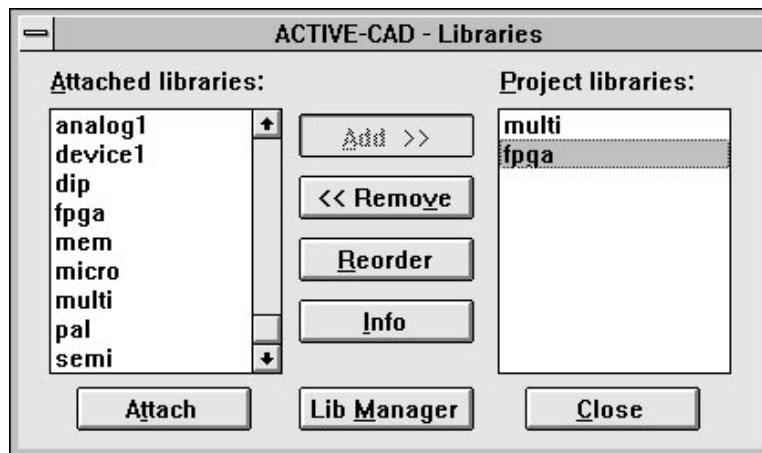

Figure 1-6. Selecting Project Libraries.

Only the project libraries are used when you draw a schematic or load your netlist to the simulator. If some IC models are missing, verify which libraries are currently selected and add additional libraries if necessary.

Add only those libraries that will actually be used. Removing libraries that are not used will speed up the component search.

To add a new library, select it from the left-hand window and click on the **Add** button. To remove a library from the current project, select it from the right-hand window and click the **Remove** button.

The order of the libraries in the **Project Libraries** window determines the sequence in which ACTIVE-CAD searches libraries for the components in your design. To reorder libraries, click on the library to be moved and then press on the **Reorder** button. Next, click over the location where the selected library should be inserted.

NOTE: The current project name is also included in the library listing. This library contains all the components that you have created with the help of VHDL modeling tools or by defining a new part or macro as a hierarchical symbol netlist.

Attaching Libraries

The libraries displayed in the left-hand window in Figure 1-6 are the ones that are currently *attached* to the system. ACTIVE-CAD keeps a record of where the libraries are located on the hard disk or on the network, and the term *attached* means that ACTIVE-CAD knows exactly where the libraries are located.

There are two types of libraries:

❑ The *system libraries*, which are provided on the installation diskettes or CD ROM. These libraries are typically located in the ACTIVE\SYSLIB directory. Models in these libraries cannot be modified by the user.
❑ The *project libraries*, which are created by the user. They store hierarchical macros and IC models created by copying and modifying the system library models. The modified components from system libraries may have changed pins, part name or timing but their functional behavior is the same as the model from which they were created.

The models created by modifying components from system libraries can be used only in conjunction with the keylock under which control they were generated. For large company installations, you should use the network ACTIVE-CAD keylocks because they will make the models available to all designers in the company.

The system libraries are automatically attached by the installation program. However, if you change the directory where the libraries are lo-

cated or receive the library without the installation program, you may need to use the **Attach** option to add the library to the system. Press the **Attach** button in Figure 1-6 and the **Attach Library** window (Figure 1-7) will appear.

Select in the **Attach Library** window the drive where new library is located and press the **Update** button. A list of available libraries on the selected drive is displayed in the **Libraries Found** window. Only the libraries that are not yet attached are displayed. Choose the desired libraries with the cursor and press the **Attach** button. The selected libraries will be added to the libraries listed in the **Attached Libraries field** of the Libraries window (Figure 1-6).

You can attach project libraries from other ACTIVE-CAD installations. For example, if one ACTIVE-CAD user creates a library with custom simulation models, the library can be copied to a diskette and installed on another ACTIVE-CAD system.

For more information on library operations, please refer to the **File Menu** section of the current chapter, or see the *Library Manager* chapter of this manual.
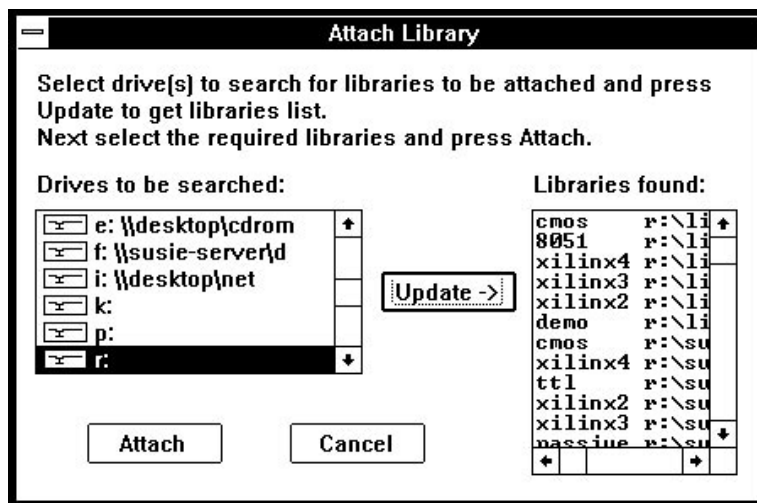

Figure 1-7. Attach Library Window.

Saving a Project

The editing of the project resources or libraries, automatically updates the current project configuration. This configuration is saved upon clos-

ing the Project Manager. The designs developed in various applications controlled by the Project Manager can only be saved by the Save options of these applications. They cannot be saved from the Project Manager.

NOTE: You are prompted for saving changed documents before quitting the Project Manager.

Title Bar

The Project Manager's title bar performs the following functions:

❐ Color indicates if the window is active. Gray indicates a deactivated window.
❐ Lists the name of the ALDEC software. This name may change depending upon the OEM version of the software. However, the overall operation generally will remain the same.
❐ Lists the current project's name in square brackets "[ ]".
❐ Lists the selected device and its packaging for a device-level design.
❐ It is used for dragging the application window to various screen locations.

The title bar is shown in detail in the Listing Of The Main Menu Options (Figure 1-8).

Main Menu

The main menu is comprised of six (6) sub-menus, which group operations on applications and project files. The most popular operations can be activated by clicking on the applications icons, provided below the main menu listing.

ACTIVE-CAD - Project Manager [flash - 4002APC84-6]
File   Document   View   Applications   Messages   Help

Figure 1-8. Main Menu Options.

File Menu

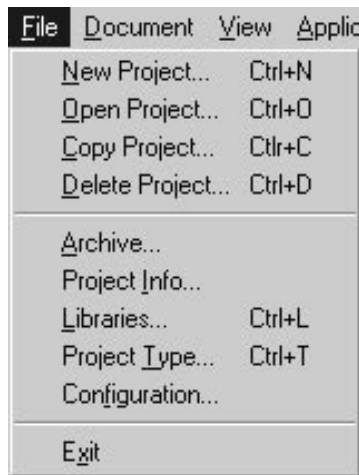The **File** menu is listed in the figure below.

Figure 1-9. File Menu Options.

File | New Project option

The **New Project** option invokes the **New Project** window which has been described in the *Starting A New Project* section of this chapter.

File | Open Project option

The **Open Project** option invokes the **Open Project** window which has been described in the *Opening An Existing Design* section of this chapter.

File | Copy Project

Copying a Project

The project configuration files contain information about the location of various project resources and libraries. Therefore, do not manually copy these files to a different location, because they will not be found by ACTIVE-CAD. Instead, select the **Copy Project** option from the Project Manager's **File** menu. Enter the new design name and its directory in the **Copy Project** window and press the **OK** button.
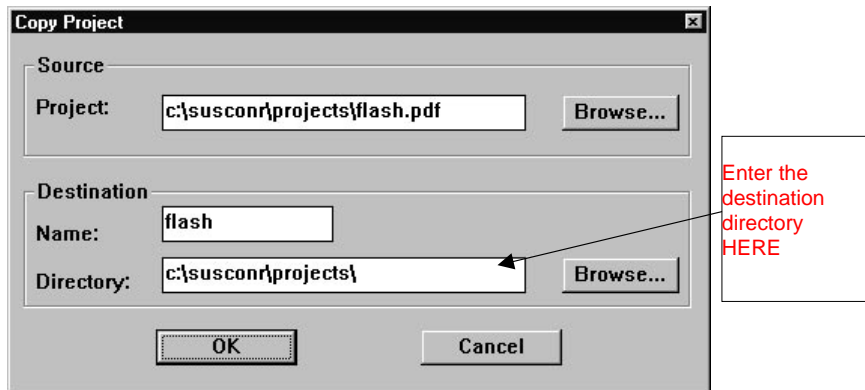
Figure 1-10. Copy Project Window.

File | Delete Project

Deleting a Project

The project files may be located in many directories. Therefore, do not manually delete project files because you may miss quite a few of them. Instead, select a project from the project list and press the **Delete Project** button. This deletes all files associated with the selected project.

NOTE: You cannot delete the current project. First, open another project, and then delete the previous one

---

File | Archive

One of the most important functions is archiving design files. The **File | Archive** option displays the **Save Current State** window (Figure 1-11), which allows saving each file with detailed manually entered comments (**Comment** field). The associated comment files, such as Release Notice, Application Notice, Engineering Change Order, and similar documents that control the design progress and its quality may be entered in the **Additional files** field.
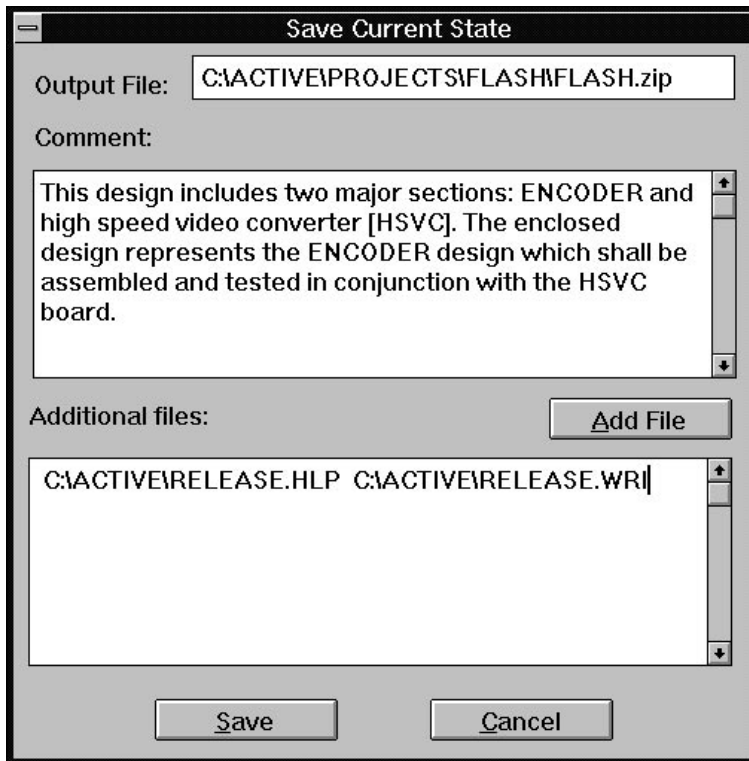
Figure 1-11. Save Current State Window.

The associated comment files, can be entered by hand by specifying their directory and file names in the **Additional files** field. Clicking on the **Add File** button allows browsing of network directories and automatic assignment of other related documents.
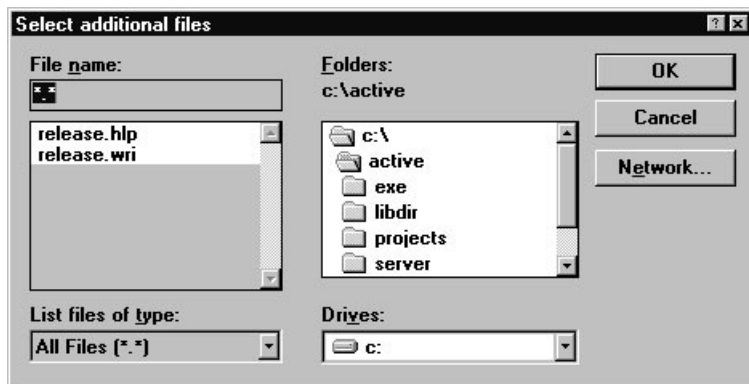

Figure 1-12. Add Archive Files Window.

File | Project Info

Project Info

This option displays key information about the current project (Figure 1-13). To exit this option, press the **OK** button.

Figure 1-13. Project Information Window.

File | Libraries

Selecting Project Libraries

Clicking o this option displays the **Libraries** window, which lists in the **Attached Libraries** field all available libraries. The libraries that actually have been used in the current project are listed in the **Project Libraries** field.

Since ACTIVE-CAD searches for components starting from the top-most library in the **Project Library** field, the order of the libraries in the **Project Libraries** field may be very important, particularly if you have two devices with the same name but different characteristics. Also, to speed the software operation, the most often used components should have their library listed at the top of the **Project Libraries** field.

To reorder libraries, click on the library in the **Project Libraries** field, which will highlight or enable the **Reorder** button. Clicking on the **Reorder** button and then on the desired location within the **Project Libraries** field, will move the selected library to its new position.

To add a library to the current project, click on the selected library in the **Attached Libraries** field, and then click on the **Add** button. The library will be listed at the bottom of the **Project Libraries** field. You can use the reorder option to place the newly added library at the desired priority level.

To add multiple libraries with one **Add** button operation, first select them by pressing the **Shift** key and clicking on the libraries in the **At-**

tached Libraries** field. The selected libraries will turn blue. Clicking on the **Add** button moves the selected libraries to the **Project Libraries** field.

To remove libraries from the project, select them in the **Project Libraries** field and then click on the **Remove** button.

NOTE: Removing a library may destroy the design if its components have been used in the project. You can safely remove only these libraries which components have not been used in the project.
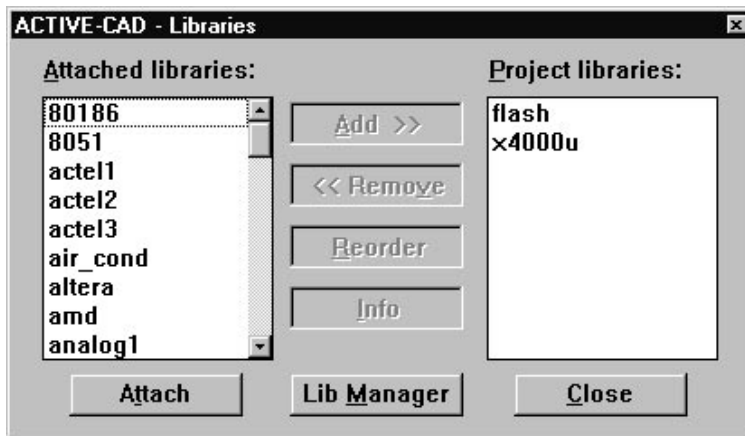

Figure 1-14. Operations On Project Libraries.

The **Attached Libraries** field list all libraries that can be used by a project. The **Attach** button allows searching network directories for additional libraries which have not been yet attached to the ACTIVE-CAD software. The process of attaching libraries is comprised of three (3) steps:

1. Click on the **Attach** button in the **Libraries** window (Figure 1-14). A listing of all network hard drives will be displayed in the **Attached Library** window (Figure 1-15).
2. Click on the drive that has the desired additional libraries, and then click on the **Update** button. The additional libraries will be listed in the **Libraries Found** field.
3. Select the library or libraries from the **Libraries Found** field and click on the **Attach** button.

The selected libraries will be automatically added to the ACTIVE-CAD resources and will be immediately available to all current and future projects.
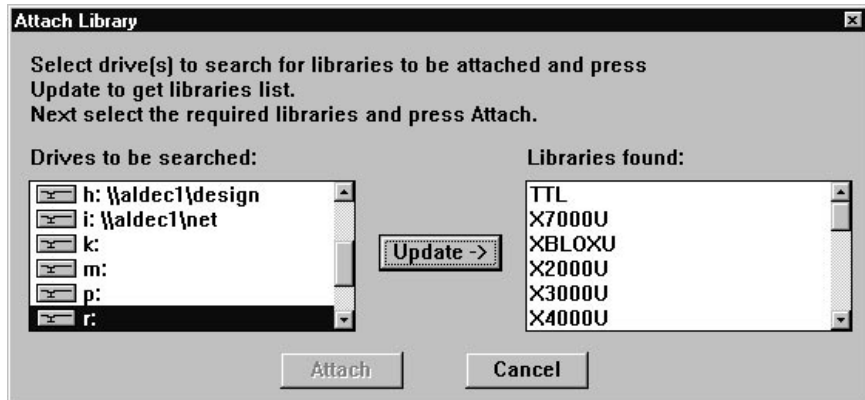
Figure 1-15. Attaching Additional Libraries.

The **Lib Manager** button activates the **Library Manager** application which is used for operations on parts and libraries. A detailed description of this application is provided in the *Library Manager* chapter.

File | Project Type

Redefining Project Type

Each time a specific technology such as XILINX, ALTERA or ACTEL is selected for your project, the Project Manager produces a specific design process flowchart for that technology. To switch to another type of the design environment, e.g. from XILINX 3000 to XILINX 5200, or from AL-TERA to XILINX, use the **File | Type** option which activates the **Change Project Configuration** window (Figure 1-16). Clicking in the **Available Types**, **Family**, **Part**, and other fields allows selection of new options.
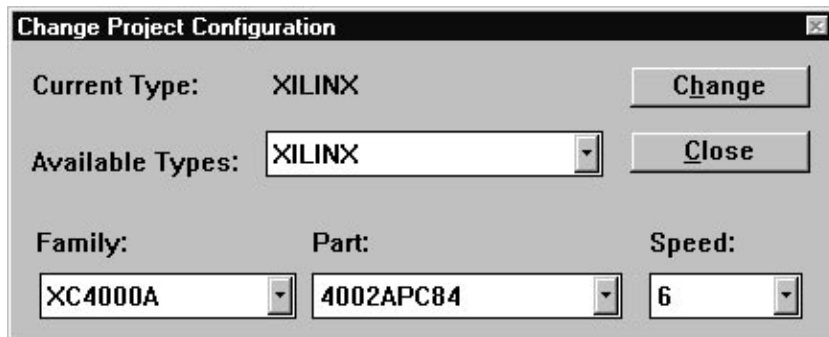

Figure 1-16. Changing Project Configuration.

Document Menu

The **Document** menu (Figure 1-17) provides for such operations on documents as adding schematic or "bubble" diagram sheets from other projects, etc. It also allows adding files generated in other projects such as JEDEC or hex files, VHDL source code files, test vector files and similar.
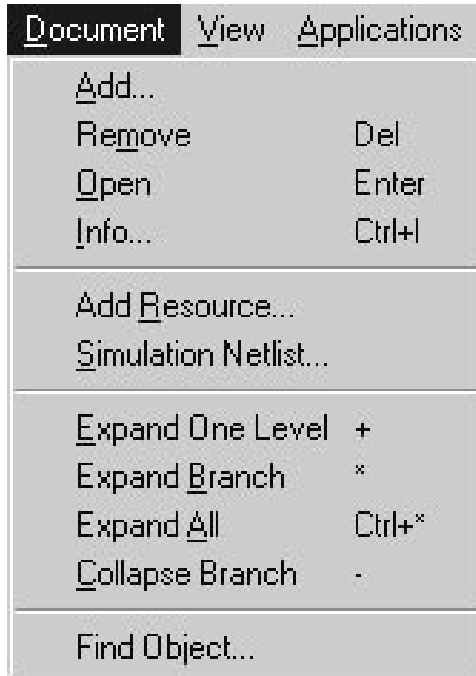

Figure 1-17. Document Menu Options.

Document | Add

Adding Documents To The Project

This option allows adding additional document files to the current project. Clicking on this option displays the **Select Document** window (Figure 1-18). Select from the **Folders** field the directory which holds the documents that you want to add to the current project. Double clicking on a directory in the **Folders** field displays all documents in that directory that have the file name extension selected in the **List file types** window.

Click (select) in the **File name** field on the document that you want to add to the current project, followed by pressing of the **OK** button.
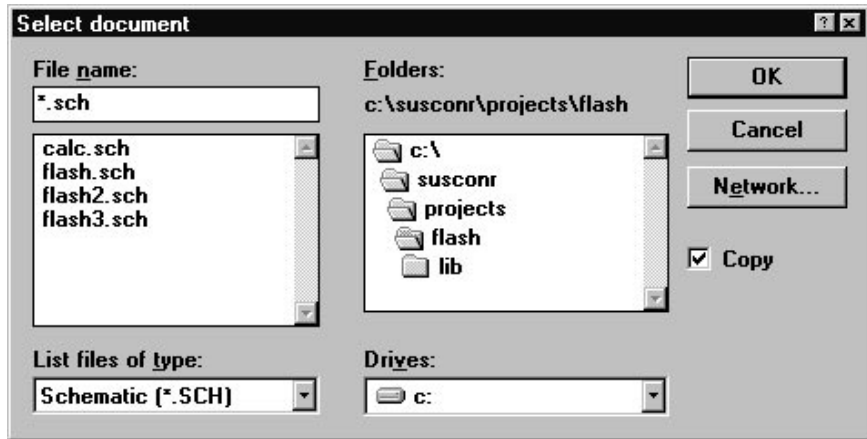
Figure 1-18. Selecting Additional Documents.

Document|Remove

Only an entire document file can be removed from a project. You cannot remove a section or subsection of any document.

To remove a document from the current project, select it from the hierarchical project tree and click on the **Document|Remove** option. If the item is an independent design file, Project Manager will display an advisory message (Figure 1-19). If the selected item is not an independent file, an advisory message will be displayed at the bottom of the Project Manager.

Figure 1-19. Document Removal Prompt Message.

Document|Open

Selecting a document from the hierarchical project tree (Hierarchy Browser field) and clicking on the **Document|Open** option will open the document and the associated application. For example, clicking on a

schematic sheet will open both the selected schematic sheet and the schematic editor.

Document | Info...

Basic information about the selected document (source file name, last changes date, library, document type) can be displayed by invoking this option.

Document | Add resource

Because adding project resources is of great importance, a special section entitled *Managing Project Resources* has been added at the end of this chapter.

Document | Simulation Netlist

This option prepares the simulator environment. Clicking on the **Document | Simulation Netlist** option displays the **Project Netlist Configuration** window (Figure 1-20). Selecting the **Schematic Netlist** option and clicking on the **OK** button will automatically generate a netlist for the on-line simulator. However, the simulator will not be activated.

Figure 1-20. Selecting A Netlist For Simulation.

To simulate externally generated netlists, e.g. by other schematic editors or logic synthesis tools, select the **External Netlist** option. Next:

1.   Select the desired netlist format from the **External Netlist Format** field.

2.   Click on the **Select Netlist** button to display the **Select Netlist** window.

Select from that window the desired file name to be simulated. You may need to select the appropriate Drive, Folders, Type of file (file name extension) and the file name. These are typical Windows operations used during file access.
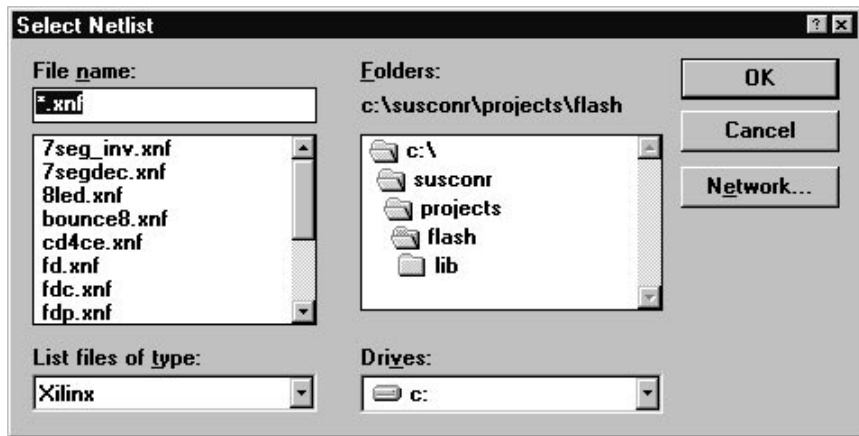
Figure 1-21. Selecting An External Simulation Netlist.

Hierarchy Tree Operations

Four options from the **Document** menu control the display of the hierarchical project structure. Highlighting a hierarchical level and clicking on the option allows selective display of the hierarchical project structure.

❒ The **Expand One Level** [+] option, displays the next hierarchical level within the selected hierarchical macro.

❒ The **Expand Branch** [*] option displays all hierarchical levels within the selected hierarchical macro

❒ The **Expand All** [Ctrl + *]option displays all hierarchical levels within the entire project.

❒ The **Collapse Branch** [-] option hides all hierarchical levels below the selected macro.
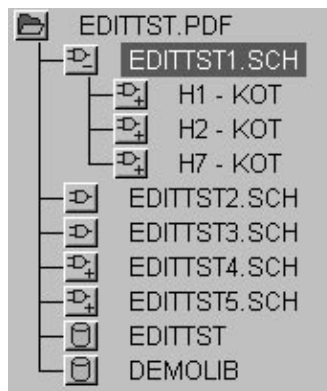

Figure 1-22. Hierarchical Project Tree Example.

Document | Find Object

This options allows searching for a symbol, pin or net name in the whole hierarchy. When the object id found, Schematic Editor is opened and the object is highlighted.

View Menu

The View menu controls display of options within the Project Manager.

- ☐ **Toolbar** - turns on/off the icons located below the main menu
- ☐ **Status Bar** - turns on/off the message bar at the bottom of the screen
- ☐ **Lines** - turns on/off the lines within the hierarchical tree
- ☐ **Sort by References** - sorts all hierarchical macros by references
- ☐ **Sort by Names** - sorts hierarchical macros by names

Figure 1-23. Project Manager Display Control.

Applications Menu

The Applications menu (Figure 1-25) allows launching ACTIVE-CAD applications. The applications can also be started from the icons located below the main menu (Figure 1-24).

Figure 1-24. Application Icons.

- ☐ **Schematic Editor** - allows schematic designs
- ☐ **State Editor** - provides for design entry in the form of graphical bubble diagrams
- ☐ **HDL Editor** - allows ABEL and VHDL design entries
- ☐ **Logic Simulator** - starts functional and timing logic simulators
- ☐ **Library Manager** - allows management of libraries, macros and models

❐   **Symbol Editor** - used for creating new cell, macro and device symbols

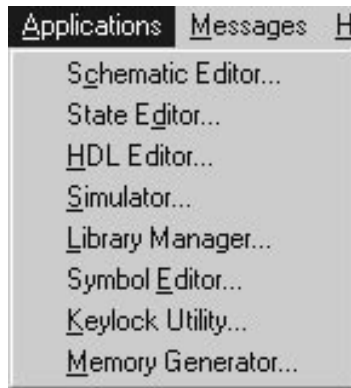*Figure 1-25. Applications Menu Options.*

Messages Menu

This menu includes two (2) options:

❐   **Preferences** - controls which messages will be displayed in the Messages window, located at the bottom of the Project Manager window.
❐   **Clear** - clears all messages

Selecting the **Preferences** option displays the **Message Preferences** window (Figure 1-26), which has two (2) fields:
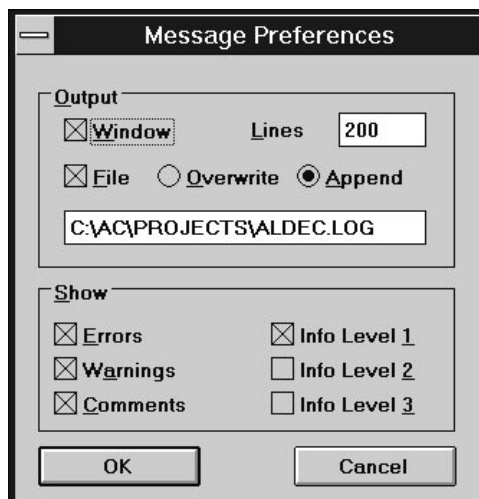

Figure 1-26. Selecting Messages For Display.

1. *Output* field controls how and where the messages will be displayed and stored

❒ **Window** displays the messages in the Project Manager window
❒ **File** saves the messages on file
❒ **Overwrite** overwrites the old messages with new ones
❒ **Append** keeps track of all messages in one file
❒ **Lines** sets the limit on the design file size

2. *Show* field controls what will be displayed

❒ **Errors** - displays all errors in red
❒ **Warnings** - displays all warnings in blue
❒ **Comments** - displays design comments in black
❒ **Info Levels** (**1**-**3**) - displays comments about the progress of software processing; checking only the Level 1 box displays the most important messages - checking all boxes displays all messages.

Help menu

Displays the current version of the Project Manager. An example of such Help menu is shown below (Figure 1-27).
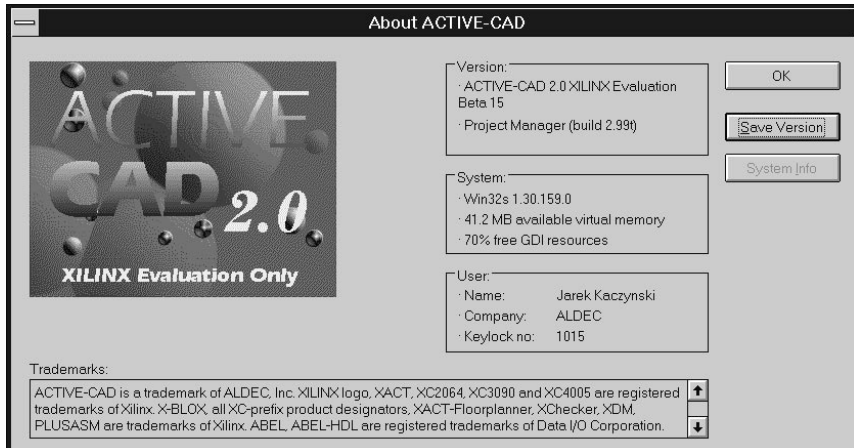

Figure 1-27. Project Manager Help Window.

Project Management

General

A project is a set of resources and libraries configured for a specific design.

The project resources (schematics, netlists, VHDL source code files, JEDEC files, etc.) associated with the design are stored in the Project Working Directory. They are created by ACTIVE-CAD applications or imported from external programs. All project resources that are created during the design sessions are automatically attached to the current project. You can also manually add, browse or delete project resources.

Project libraries should be configured to match your design needs. To speed the design process, select only those libraries that you will actually use. If several libraries list the same part name, you can specify which library will be used, by arranging the order of libraries.

NOTE: If there are duplicate names in the libraries, the project libraries will be searched for these parts starting from the top-most listed library.

Each project has a configuration file that stores the information about the location of the project resources and libraries. This file has the *.PDF extension and is typically stored in the ACTIVE\PROJECTS directory. The location of this directory is set in the SUSIE.INI file as the PROJECTS=C:\ACTIVE\PROJECTS setting.

Managing Project Resources

ACTIVE-CAD is a design database management system. This database is comprised of various design files called project resources. The Table 1-1 lists all resources that can be used with ACTIVE-CAD programs.

Any existing ACTIVE-CAD resources can be added to or removed from the current project configuration by using the **Document|Add Resources** menu option. When the **Project Resources** window (Figure 1-28) appears, click on the arrow button in the **Resource Type** listbox and select the desired resource type, e.g. test vectors. A list of currently attached test vector files will be displayed in the **Existing Project Resources** field.

Table 1-1. ACTIVE-CAD Project Resources.

Resource Name	Filename Extension
Schematic Sheets*	*.SCH
Schematic Macros	*.MAC
Simulator Macros	*.MAC
VHDL Source Codes	*.HDL
Simulation Session Files	*.DES
PLD JEDEC Fuse Maps	*.JED
ASCII Test Vectors	*.ASC
Binary Test Vectors	*.TIM
Memory Contents	*.HEX, *.BIN
Internal State Editor Screens	*.SCR

The Additional resources window displays all files located in the current directory that have the same file name extension as the selected project resource. To list files in another directory, change directories using the **Directories** window on the right-hand side. Double-clicking on the selected directory will display all files with the selected file name extension, which reside in that directory. These files will instantly appear in the **Additional Resources Available In Directory** field.

Select the desired file name and then press the **Copy** or **Link** button to add the resource to the current project. **Copy** will import the resource and **Link** will attach the resource.
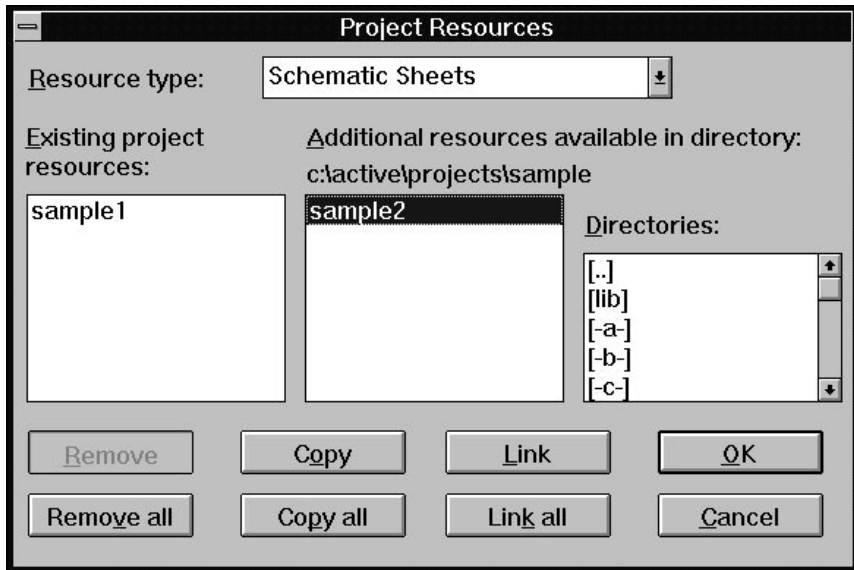
Figure 1-28. Selecting Project Resources.

The **Copy** option makes a copy of the imported file in the project work-
ing directory. For example, if you import an ASCII timing file using the
Copy button, the file will be copied to the project working directory. If
the original file is ever changed or deleted, it will not affect the current
project. The **Copy all** button copies all resources displayed in the se-
lected directory to the project directory.

The **Link** button attaches the selected resource to the current project, but
does not make a copy of the resource file. It will be used directly from
the directory where it is located at the time you import it. This means
that every time the original file is changed, your project will be using a
new version automatically. For example, if you link a JEDEC file,
ACTIVE-CAD will load the latest version of the file automatically. If the
file is moved or deleted, then ACTIVE-CAD will no longer be able to
find a resource file and it has to be linked again. The **Link all** button
links all resource files in the selected directory.

To remove existing resources from the current project, use the **Remove**
or **Remove all** buttons. If a resource file was copied, ACTIVE-CAD will
delete the file from the disk. If it was linked, the file will not be deleted.

Simulating External Netlist

The simulator is an independent application which can simulate designs entered in any schematic editor. To load such a design into simulator, use the **Document|Simulation Netlist** menu option in the **Project Manager**.

Select the **External** netlist option in the new **Project Netlist Configuration** window. Next, select the netlist format and then click on the **Select Netlist** button to locate the desired netlist file on the disk. All files with the selected file name extension will be automatically displayed.
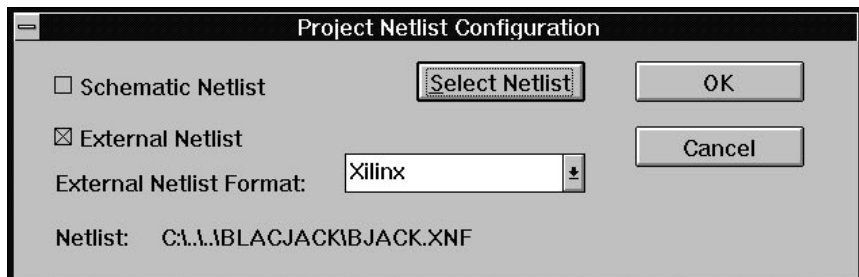

Figure 1-29. Selecting External Netlist Format.

To switch the project back to the schematic netlist, select the **Schematic Netlist** option in the **Project Netlist Configuration** window.

Post-Layout FPGA Netlists

A special case of the external netlist is a netlist that was originally generated from the ACTIVE-CAD schematic editor and then modified by some layout tools. For example, you may have a Xilinx FPGA design that has been entered on the schematic and then routed by Xilinx layout tools. The post-layout netlist is generally similar to the schematic netlist but contains some additional information about circuit timing delays. This information is required for the timing verification of the design and has to be loaded directly to the simulator.

To simulate the routed netlist and be able to view the schematic at the same time, select the **External netlist** in the **Project Netlist Configuration** window. This will allow you to load an external netlist to simulator and simulate the timing delays from the routed netlist. You can also place and view the simulation probes (LED lamps) directly on the schematic. However, the schematic changes will not show in the simulator.

NOTE: When simulating routed designs some probes from the pre-routing test vector files may not be displayed during the post-routing simulation. This is usually due to the differences between the schematic and the routed netlist. Generally, the net names are preserved by all layout tools and you can view their probes on the schematic.