

OrCAD[®]

ESP Design Environment User's Guide

OrCAD/ESP
Design Environment
User's Guide



Electronic Design Automation Tools

OrCAD/ESP

Design Environment

User's Guide

Copyright © 1991 OrCAD L.P. All rights reserved.

No part of this publication may be reproduced, translated into another language, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise without the prior written consent of OrCAD L.P.

Every precaution has been taken in the preparation of this publication. OrCAD assumes no responsibility for errors or omissions. Neither is any liability assumed for damages resulting from the use of the information contained herein.

OrCAD® is registered trademark of OrCAD L.P.

IBM® is a registered trademark of International Business Machines Corporation.

All other brand and product names mentioned herein are used for identification purposes only, and are trademarks or registered trademarks of their respective holders.

Document Number: OR9061B 4-7-91

OrCAD® 

3175 NW Aloclek Drive

Hillsboro, Oregon 97124-7135

Sales & Administration	(503) 690-9881
Technical Support	(503) 690-9722
24-Hour Bulletin Board System	(503) 690-9791
FAX	(503) 690-9891

C O N T E N T S

Preface	<i>ix</i>
Chapter 1: Introducing the ESP design environment	1
Introducing the design environment	1
About this guide	3
Conventions	3
Terminology	4
Starting ESP	4
Mouse techniques	5
Keyboard equivalents	6
Tools and tool sets	7
Editors	8
Processors	8
Transfers	9
Librarians	9
Reporters	9
User buttons	10
About TEMPLATE	10
Chapter 2: Design management commands	13
Organized by design	13
Design management tool set	13
Design view	14
Create Design	16
About text entry boxes	17
Backup Design	18
Copy Design	20
Complex to Simple	21
OK	23
Delete Design	23
Restore Design	24
Update ESP Data	25
Suspend to System	25
Cancel	26
File view	27
Copy File	28

Rename File	29
Suspend to System	30
Delete File.....	31
Edit File.....	32
Filter.....	33
About non-text files.....	34
OK	34
Chapter 3: Customizing the environment	35
Display the configuration.....	36
Driver Options	38
Driver Prefix.....	38
Available Display Drivers	39
Configured Display Driver.....	39
Editor Options	40
Design Options	40
Color tables	41
Text.....	41
Configuration.....	41
Tool sets	41
Editors.....	41
Processors.....	41
Transfers.....	42
Librarians.....	42
Reporters	42
User buttons.....	42
Chapter 4: Defining a user button	43
Execute	43
Modify	43
Example	44
Define a command with parameters	44
Define more than one command in a user button	45
Specify the Help File Name.....	45
Delete	45
Help.....	45

Chapter 5: Design environment technical information.....	47
Directory structure.....	47
Contents.....	49
Executable software.....	49
ESP.EXE.....	49
ESP_C.EXE.....	49
ESPMAIN.EXE.....	49
PCB_C.EXE.....	49
PCB_CLC.EXE.....	49
PLD_C.EXE.....	49
PLD_CLC.EXE.....	49
SDT_C.EXE.....	49
SDT_CLC.EXE.....	49
VST_C.EXE.....	50
VST_CLC.EXE.....	50
Other files.....	50
#ESP_OUT.TXT.....	50
About .BCF and .CFG files.....	50
ESP.CFG.....	51
DESIGNS.LST.....	51
FILES.LST.....	51
ORCADESP.DAT.....	51
ORCADESP.HLP.....	51
PCB.BCF.....	51
PCB.CFG.....	51
PLD.BCF.....	51
PLD.CFG.....	51
SDT.BCF.....	51
SDT.CFG.....	51
VST.BCF.....	52
VST.CFG.....	52
DOS environment variables.....	53
Glossary.....	55
Index.....	59

ONE OF THE major goals of ESP is to change the way electronic design automation (EDA) tools are used to design. Traditionally, the EDA tools were the constraint: everything revolved around the tools. The tools did not work together well nor did there exist an environment within which to use them.

A primitive solution to the interaction of the tools with the design would be to build a command control shell. A shell allows you to select tools easily and usually supports a common look-and-feel user interface. A shell, though, does not directly support chaining the programs together to build processes.

What makes a design environment more powerful than a shell is that the environment is capable of doing more than just launching programs: it provides design processes.

ESP is a graphical user interface designed specifically for the electronic designer. Tasks that used to be non-intuitive and time-consuming can now be done by clicking a button. Nearly every button can be *locally configured*, a feature that eliminates the need to remember strings of command line switches and their arguments. Local configuration also allows subprocesses to be used as part of larger processes, with each process having a set of operating parameters to accomplish the action of the larger process.

Local configurations are particular to a design, so that you can switch between designs without having to change configuration every time you switch.

One important change to the design process is that all the files that are the database of a design are collected into a single work area. ESP provides design management commands to create, delete, archive and transfer the database of a design. Project management is quicker and easier than ever before since a design is not scattered across many directories or mixed in with other design files in the tool areas.

ESP groups tools used in different aspects of design into tool sets. An example of a tool set is **Schematic Design Tools**, all of the tools used to develop schematics. Another example is **Digital Simulation Tools**, all of the tools used for digital design verification.

Within tool sets, tools are grouped by function, making it easier for a designer to identify and select an appropriate tool. The groups are described in detail in this guide.

The transfer tools are what make ESP so powerful. Transfers signal the database and the environment that the view of the design is ready to be changed. Think of the design as a unified entity. The tool sets can then be thought of as simply providing the designer with different views of the whole. When the designer has completed the schematics for a design and is ready for simulation, the **To Digital Simulation** transfer button handles all of the process required to update the schematic and connectivity databases, as well as transferring any simulation directives to the simulator.

OrCAD continues to invest in advanced EDA design tools with the same spirit of enthusiasm and innovation as in the early days. Our mission is to be the world standard in EDA tools by designing, producing, and supporting high performance, accessible EDA tools. ESP is the beginning of a new generation. In this new generation, designers can focus on what's really important: the design, not the tools. ❖

*John Durbetaki
November 1990*



Introducing the ESP design environment

As the pace of technological development quickens, the art of design becomes more complex. The competitive pressure is on to bring a product to market in the shortest possible time, at the lowest development cost, with zero defects. Increasingly, designers cope by using more and better electronic design automation (EDA) tools to maximize productivity. As the number of tools and their power and complexity increase, the danger is that a designer has to spend proportionally more time learning how to use the tools and making them work together—time that could otherwise be spent designing.

Designing is the art of choosing among different design solutions. The process of designing, testing, building, and bringing a product to market is extraordinarily complicated. What this usually means to a designer is that the art of design must be as automated as possible and that design information must be made to flow as smoothly as possible from one phase of design to another.

Introducing the design environment

OrCAD's ESP environment is a graphical design environment designed specially for electronics designers. The environment links the software tools appropriate for different stages of the design together, providing a common and intuitive human interface for the tools. It handles such matters as file management and transferring information between tool sets.

Information management under the environment is simple and straight-forward: the design is the focus. Under the environment, all the files that belong to a design—schematics, netlists, PLD logic equations, custom libraries, and so on—are stored in one directory.

Design management commands built into the environment let you make new design directories, delete old ones, archive designs, and transfer the database of a design.

The environment is very flexible. Nearly every aspect of the environment can be tailored to meet your requirements using **Local Configuration** options. Configuration information is stored with your design files, so you can have different configurations for different designs.

The environment makes memorizing complex character-based command line sequences unnecessary: simply make the appropriate selections from the graphic configuration screens.

The environment is set up so that you can easily extend it to include your own (non-OrCAD) design tools. For example, a basic text editor is included with the environment, but you can easily substitute any other editor you prefer.

The environment increases your productivity by providing the following features:

- ❖ An intuitive and easy-to-learn interface
- ❖ Full mouse and keyboard support
- ❖ Support for expanded memory
- ❖ Full color control

About this guide This guide provides step-by-step instructions for all the tasks you are likely to perform using ESP.

You should use this book with your computer turned on and the OrCAD design environment running. Before you begin to explore the design environment, take a few minutes to install the software and register for technical support. Just follow the instructions in the *Installation & Technical Support Guide*.

Conventions The notation conventions used in this guide are as follows:

BOLD CAPS	Used for main menu commands.
Bold	Used for other commands.
Courier bold	Used for text you enter.
Bold Italics	Used for references to other manuals.
<i>Italics</i>	Used for references to other sections, chapters, or parts of this guide.
	Brackets <> are used to show a key that you press. For example, <Esc> means to press the escape key.
"Prompt"	Quotation marks indicate program prompts.

The shadow box shown below indicates a program or system prompt. Any bold type following the prompt indicates text that you enter. For example:

Abandon edits?

This kind of shadow box indicates an OrCAD menu.

Hardcopy
Destination
File

△ **NOTE:** Notes contain important reminders or hints.

▲ **CAUTION:** Cautions contain information about preventing damage to equipment, software, or data.

Terminology The instructions in this guide use the terms “enter” and “type” to mean two different things. When the instructions tell you to enter something, you press the appropriate keys and end by pressing <Enter>. When the instructions tell you to type something, you press the appropriate keys, but do *not* press <Enter>.

Starting ESP Start the OrCAD design environment after installation by entering the command shown below in bold at the system prompt:

```
C:\> orcad
```

In a moment, the design environment main screen displays.

- Title bar -----
Shows the tool set name, the currently selected design, and the software version number.

- Buttons-----
Pushbutton-like images that you click on to initiate an action.

- Logo-----

- Pointer-----
An arrow on the screen that moves as you move the mouse.

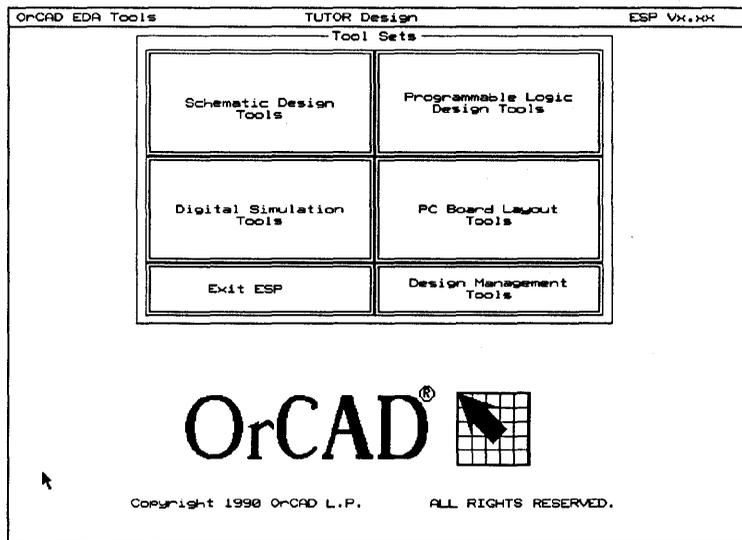


Figure 1-1. Design environment main screen.

△ **NOTE:** To start running the design environment on a particular design, just type ORCAD followed by a blank space and the name of a particular design. When you press <Enter>, the ESP design environment runs and the design name you entered will be the current design.

Mouse techniques



You can do all your work in the OrCAD design environment (except typing text and numbers) using the mouse.

You *point* to an object by moving the pointer until the tip of the arrow touches the object. Do this by moving the mouse.

You *click* by pointing to an object and then pressing and releasing the left mouse button once. When you click on a button, it becomes *highlighted* and a menu pops up in the upper left corner of the screen.

In this guide, the words “click,” “highlight,” and “select” all mean the same thing. In every case the action you take is the same: position the pointer, press the left mouse button, and quickly release it.

You *double-click* by first pointing to an object and then clicking the left mouse button twice. Don’t move the mouse while you double-click.

Left and right mouse buttons

- ❖ Clicking the left mouse button is the same as pressing the <Enter> key. In OrCAD guides, when you are instructed to “press <Enter>,” you can use either the keyboard or the mouse, whichever you prefer.
- ❖ Clicking the right mouse button is the same as pressing the <Esc> key. In OrCAD guides, when you are instructed to “press <Esc>,” you can use either the keyboard or the mouse, whichever you prefer.

Keyboard equivalents



Many of the explanations and instructions in this book use the mouse terminology explained above. If you prefer to use the keyboard, however, there are keyboard equivalents to nearly every mouse-performed operation. Instead of moving the mouse to move the pointer from button to button, you can:

- ❖ Press <Tab> to move from one category of tools to the next.
- ❖ Press the spacebar to move from button to button within a category.
- ❖ Press <Shift><Tab> to move the pointer backwards to the next category.
- ❖ Press <Enter> to select the button the pointer rests on.

Tools and tool sets

A *tool set* is a collection of tools designed to perform a set of electronic design automation tasks. The grouping is designed to make selecting an action easier. There are currently four OrCAD tool sets. They are:

- ❖ Schematic Design Tools
- ❖ Programmable Logic Design Tools
- ❖ Digital Simulation Tools
- ❖ Printed Circuit (PC) Board Layout Tools

The tool sets allow you to access the same design in different ways.

Buttons for the OrCAD design tool sets appear on the design environment screen, even if you have only one tool set installed on your system.

Before going on to the next section, select a tool set. Point to any one of the four tool set buttons and double-click. In a moment, you'll see the screen for that tool set.

For example, the tools that make up the **Schematic Design Tool set** are shown in the following figure.

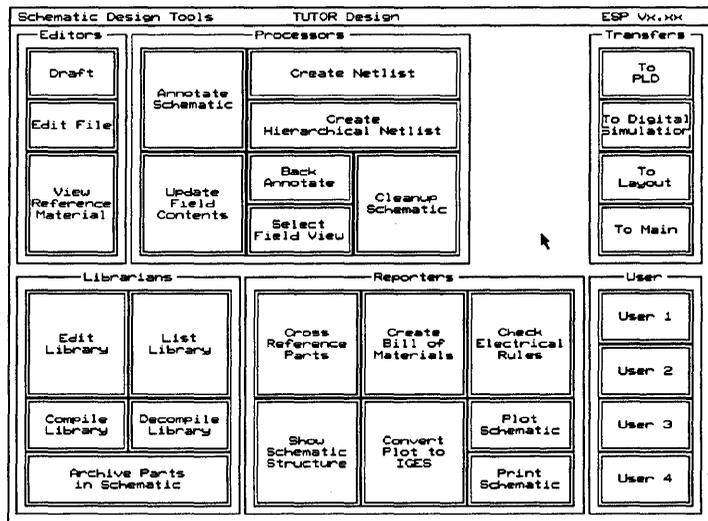


Figure 1-2. Schematic Design Tools screen.

In tool sets, tools are grouped according to function. The six categories are:

- ❖ Editors
- ❖ Processors
- ❖ Librarians
- ❖ Reporters
- ❖ Transfers
- ❖ User buttons

For information about the tools within each of the four tool sets, see the manuals that accompany that product.

Editors Editors modify or create some part of the design database. An example of an editor is the schematic editor, **Draft**. Another editor is the text editor, used to view reports and enter text.

Processors Processors read, modify, then rewrite the design database. For example, on the **Schematic Design Tools** screen, **Annotate Schematic** is a processor. Processors generally do not create human-readable reports, but rather create or modify database information.

Transfers Transfer tools manage the steps needed to move design information from one tool set to another. Transfers have two parts. The first updates the database used by the current tool set so that it is current and up-to-date in every respect. The second part changes to the new tool set used to view the design. The transfer tools take care of intermediate steps so that you don't have to.

For example, on the **Schematic Design Tools** screen, the **To Digital Simulation** transfer tool performs these steps:

- ❖ Annotates the reference designators in the design
- ❖ Updates part fields for components in the design
- ❖ Builds the connectivity database
- ❖ Builds the link between the schematic and the simulator, so that the simulation directives inserted in the schematic can be accessed by the simulator
- ❖ Transfers control to the **Digital Simulation Tool** set

Librarians Librarians are tools for managing and creating library objects that can be used by all designs, not just the current design. An example of a librarian is a tool used to create a new schematic symbol for a component. This component should be available in all future design work, so it is stored in the library database.

Reporters Reporters create human-readable reports, but do not modify design data in any way. For example, a reporter produces the **Create Bill of Materials** report, a list of all the components used in the design. The tools for printing and plotting are also reporters. Reporters may create reports that will be used by tools outside the ESP environment.

User buttons A user button can be set up to run any system command. A user button is the most basic way in which the ESP environment can be extended to fit your particular requirements. As an example, you can set up a user button to call up a spreadsheet program, which can then be used to analyze design information. Or, you may depend on a particular set of operating system utility programs, so you can assign another user button to call them up.

User buttons can also be programmed to run batch files.

How to define a user button is described in chapter 4.

About TEMPLATE

Before you begin using the ESP design environment, there's one unassuming but very powerful feature you should know about.

It's the *template*. You'll see it for the first time you start ESP after installation. The title "TEMPLATE Design" displays in the top center of your screen.

The template design is a set of patterns ESP uses to create new designs. Anything added to or taken away from the template will be added to or withheld from designs you make in the future using ESP's **Create Design** tool. Changes you make to configurations in the template become part of all new designs, too.

When you install **Schematic Design Tools**, eight files are placed in the template. To see the list, select **Edit file** from the **Schematic Design Tools** screen. Other files are added when you install other OrCAD products. To find out more about these files and how ESP uses them, see *Chapter 5: Design environment technical information* in this guide.

Here are some tips for making the best use of the template.

- ❖ Remember the TEMPLATE design is just that—a template for new designs. To save disk space, only files that you want to include in *every* design you create should be copied into TEMPLATE. If hard disk space is at a premium on your system, keep the number of files in TEMPLATE to a minimum.

- ❖ If you use certain configurations routinely, make these settings in **TEMPLATE**. They will automatically become part of every new design, eliminating the need to configure tools again and again. For example, if you always want **Create Netlist** to produce one particular netlist format, set this up in the **TEMPLATE**.
- ❖ As soon as the configurations in **TEMPLATE** are set to match your requirements, create a new design. Then, select **Configure ESP** from the **Design Management Tools** menu. Under **Design Options**, change **Startup Design** to be the design you just created. From now on, running ESP automatically takes you to your design directory. Work in **TEMPLATE** only if you need to change configuration settings for all future designs.
- ❖ **INSTALL** automatically places the macro files **MACRO1.MAC** and **MACRO2.MAC** in **TEMPLATE**. You may remove these files or replace them with your own custom macro files, if you like. *Do not remove* any of the other files **INSTALL** places in **TEMPLATE**.



Design management commands

Organized by design

As you work with OrCAD EDA tools and create more designs, you'll want to organize your work to make it easier to find what you need. As with most computer programs, OrCAD EDA tools use files to create, save and retrieve your work on a hard disk. Within the design environment, you can:

- ❖ Organize files belonging to a design
- ❖ Add descriptions to designs and files to make locating a particular design or file easy
- ❖ Protect your work with backup copies

The design itself is the foundation of your organization.

In the OrCAD design environment, designs work the same way as a folder in a file cabinet. You decide how many folders you need and what they are called. Each folder contains all the documents, or files, for a design.

Design management tool set

The design environment provides two groups of tools for managing design files.

- ❖ **Design View**
- ❖ **File View**

In **Design View**, you perform tasks that affect the entire design.

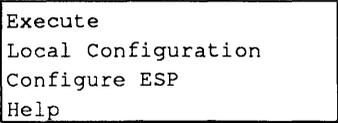
In **File View**, you perform tasks that affect individual files.

These two groups of tools and how to use them are described in the remainder of this chapter.

Design view

The next sections assume you have the design environment running. If you don't, start it now by entering the command **orcad** at the system prompt:

```
C:\> orcad
```

1. Click the **Design Management Tools** button on the design environment screen. A menu (shown below) displays in the top left corner of your screen.
 - ❖ **Execute** opens the tool set. This is always the first command on the design environment menus. When you are more familiar with the environment, you can open tool sets quickly by double-clicking the buttons on the environment screen.A screenshot of a menu box with a black border. It contains four text items: "Execute", "Local Configuration", "Configure ESP", and "Help", listed vertically from top to bottom.

Execute
Local Configuration
Configure ESP
Help
 - ❖ **Local Configuration** lets you control the action the environment will take when you select the button. When the button's active behavior is always the same (that is, there are no configurable options: it always does the same thing), the message, "Nothing to Configure" appears when this command is selected. Local configuration will be described in more detail later in this guide.
 - ❖ **Configure ESP** lets you tailor certain characteristics of the design environment (such as the colors used to display items) to suit your personal preferences. Chapter 3 contains configuration information.
 - ❖ **Help** tells you about the commands found on a menu and where you can find more information. All design environment menus have **Help** commands.
2. Select **Execute**. The **Design Management Tools** screen displays.

Radio buttons-----

Small round buttons used to show selections. Only one radio button in a group can be selected at a time.

List boxes with scroll buttons--

Scroll buttons move a directory in its window so that other names are visible.

The four scroll buttons are:

- Line Up
- Page Up
- Page Down
- Line Down

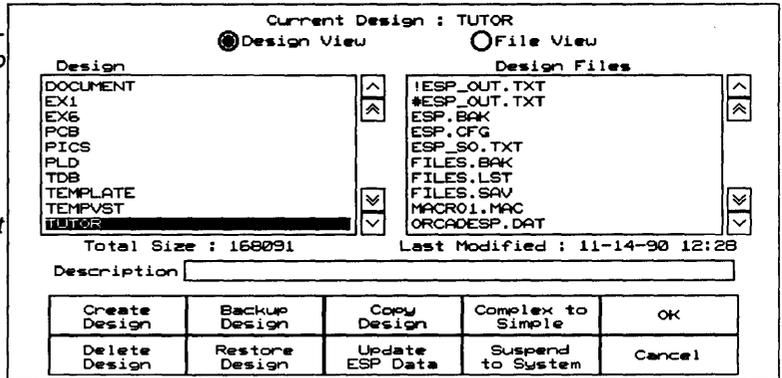


Figure 2-1. Design management tools screen.

The left list box displays the names of designs. If you are using the OrCAD design environment for the first time, only two names, that of the factory-built design TEMPLATE with a brief description, and the TUTOR design appear in the list.

The TUTOR design contains all of the files you need to complete the tutorials in the User's guides. As you add designs, this list grows. The Design Files list box on the right displays the names of files found within the design directory selected in the Design list box on the left.

▲ **CAUTION:** The files in the TEMPLATE design are the master copies the design environment uses to create new projects. Changes you make to the TEMPLATE design become part of any new designs. Any files you add to the TEMPLATE design are copied into your new designs. Therefore be very careful when working in the TEMPLATE design.

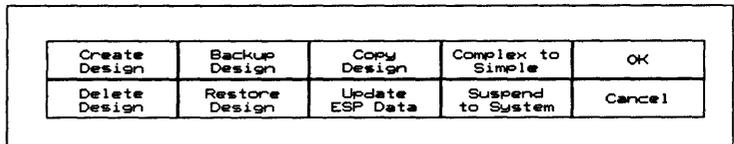


Figure 2-2. Design management tools.

The ten Design Management Tools are described below in the order they appear across the screen.

Create Design You can create as many as 150 separate designs in the design environment. Here's how.

1. Select **Create Design**. The screen shown below displays.

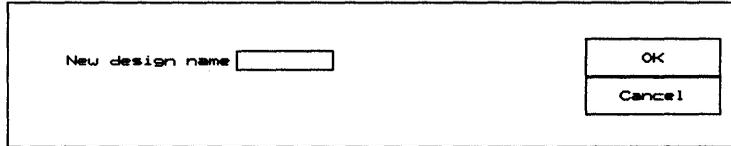


Figure 2-3. New design name screen.

2. Place the cursor in the **New design name** entry box.
3. Click once inside the box to enter the insert mode for the entry field. The pointer changes shape to become a square cursor, ■.
4. Enter a name for the design. The name may not be longer than eight characters and cannot contain periods, colons, or other special characters.

When you are done, press <Enter> or click the left mouse button again to leave the entry box.

For more information about text entry boxes, see the next section, *About text entry boxes*.

The design name must be unique. The environment will not create two designs with the same name, nor will it create a design with the same name as an existing directory.

5. Press <Enter> when you are done with the name or click the left mouse button.
6. Click the **OK** button. While the message "Working ..." is displayed at the top of the screen, the tool creates the new design you have named. In a moment, you'll see the new design listed in the Design list box.

If you enter a design name that already exists or that duplicates the name of a directory, ESP displays the message "A design with this name already exists" and does not create a new design.

About text entry boxes



All text entry boxes in the environment work the same way. You follow these steps:

- ❖ Click once inside the box to enter the insert mode for the entry field. The pointer changes shape to become a square cursor: ■.
- ❖ Type the text.
- ❖ Press <Enter> or click the left mouse button again to leave the entry box. The square cursor is replaced by the pointer.

In addition, you can use the editing keys on your keyboard to move around the entry box and edit its contents.

- ❖ <Home> moves the cursor to the beginning of the field
- ❖ <End> moves the cursor to the end of the field
- ❖ The arrow keys → and ← move right and left one character at a time, without erasing what you've typed
- ❖ <Backspace> backs up one character and deletes it
- ❖ erases the character at the current cursor position without moving the cursor



You can also use the mouse to move the cursor inside the text entry box.

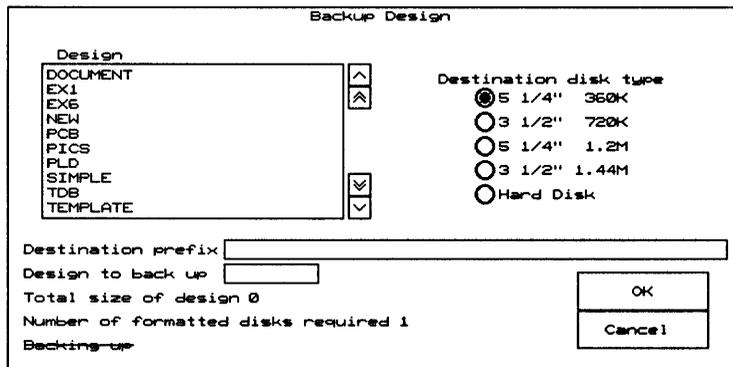
Backup Design

If you have a regular procedure for backing up your hard disk that adequately protects your data, you don't need a separate procedure for backing up your design files. If you do need to back up your design data, use this tool.

Backup Design lets you back up all the files belonging to a design onto floppy disks or to another part of your hard disk. To conserve disk space, backup files are stored in a condensed format. To restore the files to their normal format, you use the **Restore Design** tool, which is described later in this chapter.

To back up a design, follow these steps:

1. Select the **Backup Design** tool from the **Design Management Tools** screen. The screen shown below displays.



Unavailable options -----

On monochrome screens and in OrCAD manuals, options that are not available are shown with a line through them. On color monitors, the options are dimmed.

Figure 2-4. Backup design screen.

2. Select the design to back up by choosing its name in the **Design** list box.
3. Move the pointer to the **Destination prefix** entry box and press <Enter> or click the left mouse button.
4. Enter the path to use for the backup. For example, to back up the design on a floppy disk, you type the destination prefix **A:** and press <Enter>.

The environment looks at the disk or directory specified as the destination. If the disk specified is not available (for example, if you have forgotten to format the disk in the drive), the design environment displays this message:

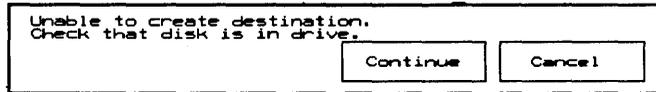


Figure 2-5. Disk not properly formatted message.

If you have forgotten to insert a disk in the drive, the design environment displays this message:

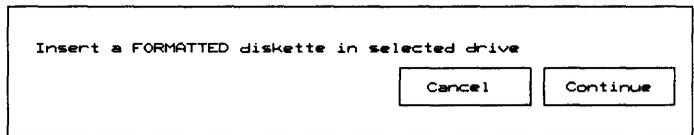


Figure 2-6. Empty drive message.

Replace the disk in the drive with a properly formatted disk and select **Continue**. Select **Cancel** if you want to cancel the backup for the time being.

If the disk in the specified drive already contains a backup of the design you selected, the design environment displays this message:

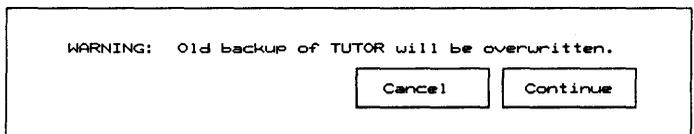


Figure 2-7. Warning to prevent accidentally overwriting design data.

Select **Cancel** if you want to remove the disk and insert another. Select **Continue** to proceed with the backup.

5. Select **OK** from the **Backup design** screen. The environment makes a backup copy of the selected design in the disk or directory specified.

The file is now backed up. To restore it, use the **Restore Design** tool.

Copy Design This tool copies all the files belonging to a design.

To copy a design, follow these steps:

1. Select the **Copy Design** tool from the design management tools screen. The screen shown below displays.

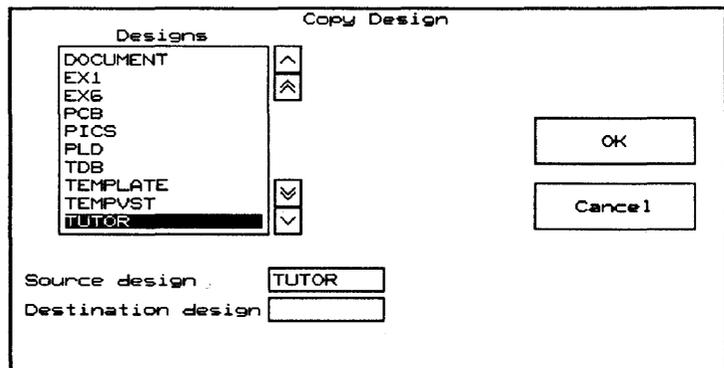


Figure 2-8. Copy design screen.

2. Move the pointer to the **Source design** entry box and press <Enter>.
3. Enter the name of the design to copy.
4. Move the pointer to the **Destination design** entry box and press <Enter>.
5. Enter the new name for the design.
6. Select **OK** to make the copy. Select **Cancel** to cancel the copy.

If the name you enter for the destination is the same as a design or directory that already exists, this message displays:

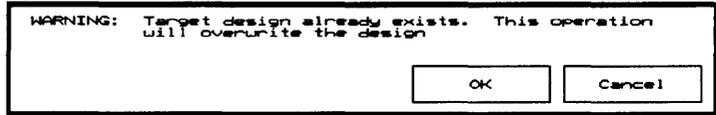


Figure 2-9. Design already exists message.

If it is your intention to copy the files to an existing design, click the OK button to proceed with the copy. This will copy the source over an existing design.

If it is NOT your intention to copy the files over an existing design, click the Cancel button to abandon the copy. Enter a unique name in the destination text entry box and select OK again.

Complex to Simple

This tool converts a complex hierarchical design structure to a simple one. Complex hierarchical designs are more efficient for the designer to create because repetitive circuitry is drawn only once. However, a simple hierarchy is required whenever the design is to be used to map directly to physical components. When this is the case, a one-to-one schematic symbol to physical component relationship is required so that all symbols can be assigned unique reference designators.

For more information about hierarchical design structures, see the *Schematic Design Tools Reference Guide*.

Complex to Simple creates a new project, scans the complex design files, creates the appropriate simplified design files, and places the new files in the new project. The original complex design files are not changed in any way.

To convert a design structured as a complex hierarchy to a new design with a simple hierarchical structure, follow these steps:

1. Select the **Complex to Simple** tool from the design management tools screen. The screen shown below displays.

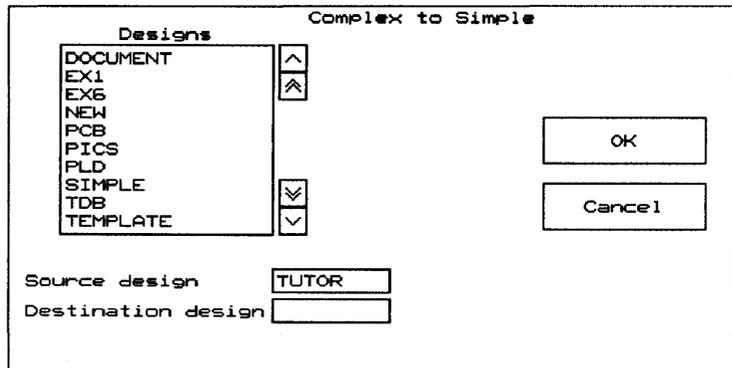


Figure 2-10. Complex to simple screen.

2. Move the pointer to the **Source design** entry box and press <Enter> or click the left mouse button.
3. Enter the name of the complex hierarchical design.
4. Move the pointer to the **Destination design** entry box and press <Enter> or click the left mouse button.
5. Enter the name for the simplified design. This need not be a new name, but may be the name of an already existing design.
6. Select **OK** to convert the design. Select **Cancel** to cancel the copy.

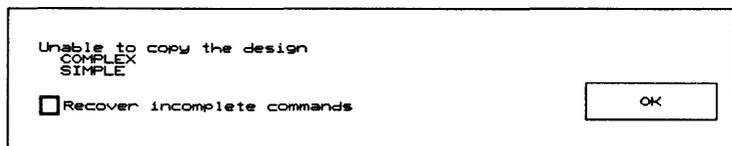


Figure 2-11. Unable to copy the design message.

In case **Complex to Simple** fails (for example, if it runs out of disk space before all the design files are placed in the new design), you can select **Recover incomplete commands**. This button deletes any files **Complex to Simple** placed in the new design before the process failed.

OK This command exits the **Design Management Tools** set and returns to the main design environment screen. If you changed the current design, **OK** saves the change.

Delete Design This tool deletes a design and all the files it contains.

▲ **CAUTION:** *This command cannot be undone.*

To delete a design, follow these steps:

1. Select the **Delete Design** tool from the **Design Management Tools** screen. The screen shown below displays.

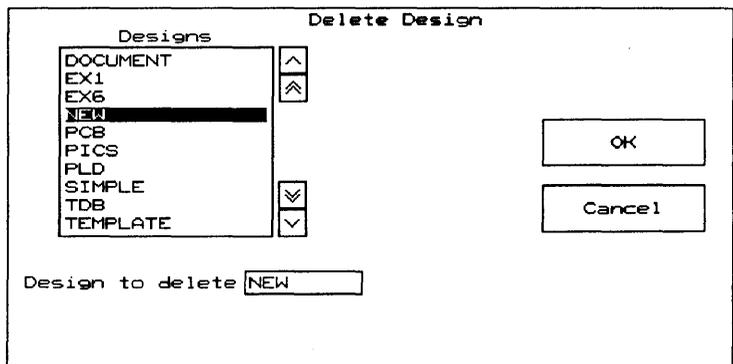


Figure 2-12. Delete design screen.

2. Move the pointer to the **Design to delete** text entry box and press <Enter>.
3. Enter the name of the design you want to delete.
4. Select **OK** to delete the design. Select **Cancel** to return to the **Design Management Tools** screen.

Deleted designs are permanently removed from the disk.

Restore Design

This tool restores a design that was backed up using the **Backup Design** tool.

To restore a design from a backup copy, follow these steps:

1. Select the **Restore Design** tool from the **Design Management Tools** screen. The screen shown below displays.

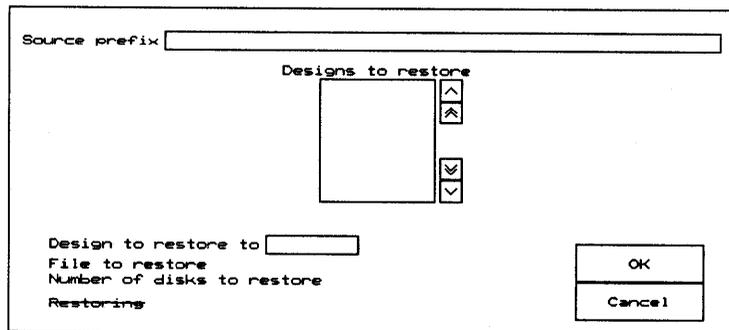


Figure 2-13. Restore design screen.

2. Move the pointer to the **Source prefix** text entry box and press <Enter>.
3. Enter the path for the backup file in the entry box. For example, if you backed up the design on a floppy disk, you would type **A:** and then press <Enter>.

The software looks for backup files in the location you specified and lists them in the **Designs to restore** list box.

If you don't see the design you wish to restore listed, try entering a different **Source prefix**.

4. Select the design to restore by highlighting its name in the **Designs to restore** list box. Use the scroll buttons if necessary to display the design name.
5. Move the pointer to the **Design to restore to** text entry box and press <Enter>.
6. Enter a name for the design to restore. You can either enter its old name or enter a new name.
7. Select **Restore** to restore the design. Select **Cancel** to return to the **Design Management Tools** screen.

Update ESP Data

The tool saves the current environment settings and selections. The design environment saves the information automatically when you open another tool set.

Suspend to System

This command temporarily suspends operation of the environment and returns to the operating system. At the system prompt, you can enter any system commands you like.

▲ **CAUTION:** *As a precaution, always use the **Update ESP Data** command before suspending to the operating system. While the environment is suspended, it remains in system memory. Inconsiderate application programs may not check to see if memory is being used before writing information to memory. If this happens, the environment may not run properly when you exit back to it.*

To suspend the environment and return to the operating system, select **Suspend to System**. After a moment, messages similar to the ones shown below display.

```
Type EXIT to return to ESP  
MS-DOS (R) Version x.xx  
C:\ORCAD\DESIGN>>
```

Figure 2-14. Suspend to System messages.

To remind you that the design environment is suspended in the background, the environment adds a right angle bracket to the system prompt.

You can now enter and run any system commands.

To return to the design environment, type the **exit** command and press <Enter>.

```
C:\ORCAD\DESIGN>> exit
```

In a moment, the environment displays the **Design View** screen.

- Cancel** This command exits the **Design Management Tool** set and returns to the main design environment screen without changing the current design if you made a change to the **Current Design**.

File view

You can have as many as 1600 files in a design. The design environment offers you complete flexibility in managing your design files.

To use the **File Management Tools**, follow these steps:

1. On the main design environment screen, double-click on the **Design Management Tools** button.
2. Select the design you wish to work with on the **Design** list box.
3. Click the **File View** radio button at the top of the screen. In a moment, the **File Management Tools** screen appears.

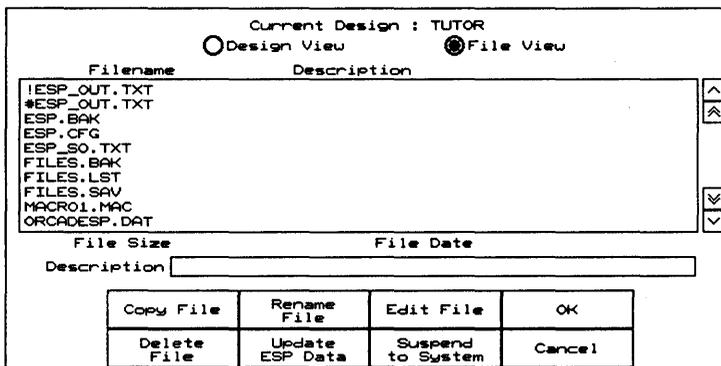


Figure 2-15. File management tools screen.

Copy File This tool copies a file.

To copy a file, follow these steps:

1. Select the **Copy File** tool from the **File View** screen. The screen shown below displays.

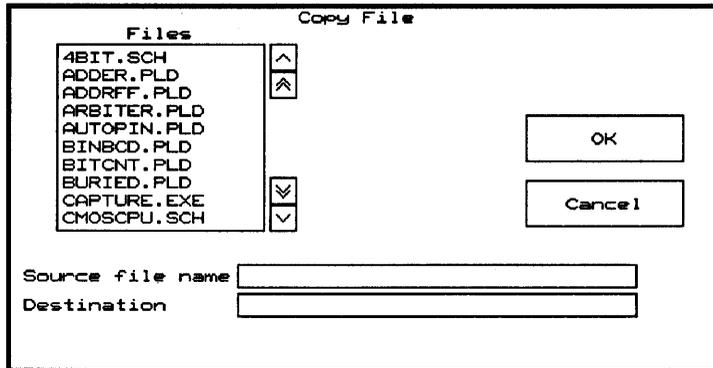


Figure 2-16. Copy file screen.

2. Move the pointer to the **Files** list box and select a file to copy. Its name then appears in the source file name text entry box.
3. Move the pointer to the **Destination file name** entry box and press <Enter> or click the left mouse button.
4. Enter the name of the destination file.
5. Click **OK** to make the copy. Click **Cancel** to cancel the copy.

If a file already exists with the name you have specified as the destination, the environment displays this message:

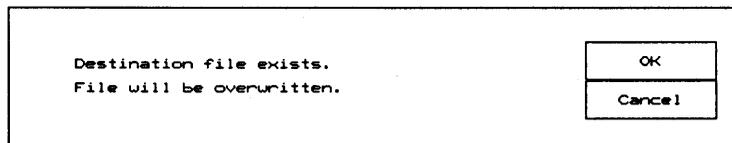


Figure 2-17. Destination file exists message.

Select **OK** to write over the existing file. Select **Cancel** to return to the **Copy File** screen, where you can enter a new destination filename.

Rename File This tool renames a file.

To rename a file, follow these steps:

1. Select the **Rename File** tool from the **File View** screen. The screen shown below displays.

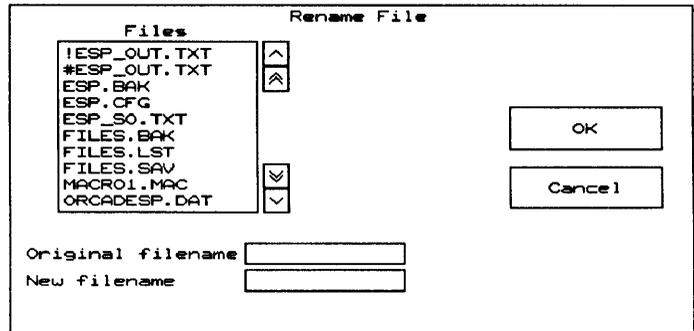


Figure 2-18. Rename file screen.

2. Move the pointer to the **Original filename** entry box and press <Enter> or click the left mouse button.
3. Enter the name of the design to rename.
4. Move the pointer to the **New filename** entry box and press <Enter> or click the left mouse button.
5. Enter the new name for the file.
6. Select **OK** to rename the file. Select **Cancel** to cancel the rename process.

Suspend to System This command temporarily suspends operation of the environment and returns to the operating system. At the system prompt, you can enter any system commands you like.

▲ **CAUTION:** *As a precaution, always use the Update ESP Data command before suspending to the operating system. While the environment is suspended, it remains in system memory. Inconsiderate application programs may not check to see if memory is being used before writing information to memory. If this happens, the environment may not run properly when you exit back to it.*

To suspend the environment and return to the operating system, select **Suspend to System**. After a moment, messages similar to the ones shown below display.

```
Type EXIT to return to ESP  
  
MS-DOS (R) Version x.xx  
  
C:\ORCAD\DESIGN>>
```

Figure 2-19. Suspend to System message.

To remind you that the design environment is suspended in the background, the environment adds a right angle bracket to the system prompt.

You can now enter and run any system commands.

To return to the design environment, type the **exit** command and press <Enter>.

```
C:\ORCAD\DESIGN>> exit
```

In a moment, the environment displays the File View screen.

Delete File This tool deletes a file.

▲ **CAUTION:** This action cannot be undone.

To delete a file, follow these steps:

1. Select the **Delete File** tool from the **File View** screen. The screen shown below displays.

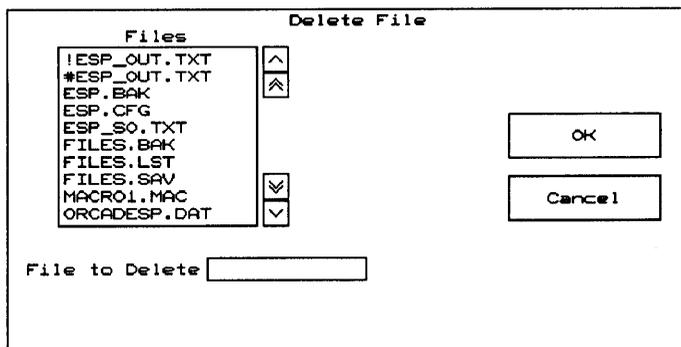


Figure 2-20. Delete file screen.

2. Move the pointer to the **File to delete** entry box and press <Enter> or click the left mouse button.
3. Enter the name of the file to delete.
4. Select **OK** to delete the file. Select **Cancel** to cancel the deletion.

Edit File This tool allows you to view and edit a text file. The OrCAD design environment includes a text editor. Its features and commands are described in the *Stony Brook M2EDIT Text Editor User's Guide*. You can also configure the environment to use another text editor if you prefer. The steps for configuring the editor and other environment characteristics are found in chapter 3.

To edit a file, follow these steps:

1. Select the **Edit File** tool from the **File View** screen. The screen shown below displays.

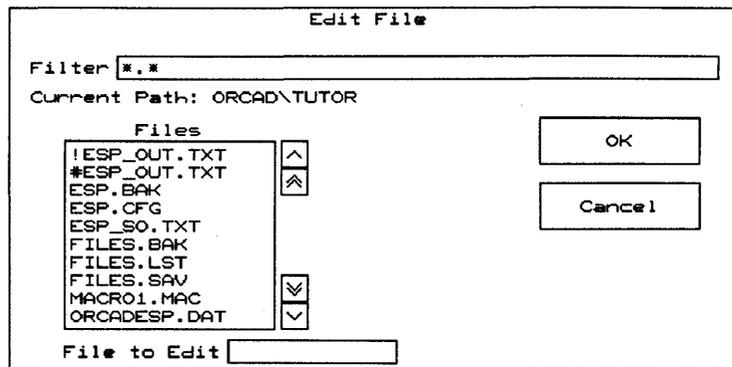


Figure 2-21. Edit File screen.

Filter **Filter** determines which files are displayed in the **Files** list box. A filter looks for and shows filenames that match the string you specify. The files that match the search filter entered in the **Prefix/wildcard** entry box are listed in this box.

Wildcards can be used to indicate which files to display in the scroll window. The asterisk character (*) is used as a wildcard.

- ❖ If you enter *.XYZ in this field, all files in the design that have an extension of .XYZ are shown in the display box.
- ❖ If you enter T*.XYZ , all files in the design that start with the letter "T" and have an extension of .XYZ are shown in the display box.
- ❖ If you enter T*.X*, all files in the design that start with "T" and that have an extension that starts with "X" are shown in the display box.

The default **Prefix/wildcard** is *.* which means all the files in the directory specified by the **Current path** are listed (no files are filtered). You can reduce the number of files displayed by making the **Prefix/wildcard** more demanding and letting the **Design Environment** screen out files that don't match the criteria you specify. For example, to list only the PLD source files in the list box, use this filter:

Filter:

More than one filter can be specified at a time. To see library source and PLD files, use this filter:

Filter:

The editor opens the file you specified. You can now view and edit the file using the editor's commands. See the *Stony Brook M2EDIT Text Editor User's Guide* if you are using the text editor OrCAD provides.

About non-text files

OrCAD graphic design files—such as schematics and layouts—display their file types at the top of the screen when they are opened by the **Edit File** text editor, and nothing else. If you open a non-OrCAD file containing something other than text, you may see some rather interesting gobbledeygook on the screen inside the editor. (Exactly what you see depends on the file's type.) Just exit from the text editor without saving any changes and no harm is done.

OK Selecting **OK** exits the **File View** screen and returns to the Design Environment screen.



Customizing the environment

The design environment is configured and ready to run when installed on your personal computer's hard disk. However, you may wish to customize this configuration. The **Configure ESP** screen allows you to configure:

- ❖ Driver options (prefix and display drivers)
- ❖ The selection of a text editor
- ❖ The project selected on startup
- ❖ Colors used to display and highlight tools

When you are done making changes, click the **OK** button to save changes and return to the environment screen.

If you do not want to implement your changes, click the **Cancel** button to return to the environment screen without saving the changes.

Display the configuration

To display the design environment's configuration options, follow these steps.

1. Click the **Design Management Tools** button. The menu shown at right displays at the top of the screen.

Execute
Local Configuration
Configure ESP
Help

2. Select **Configure ESP**. The screen shown below displays.

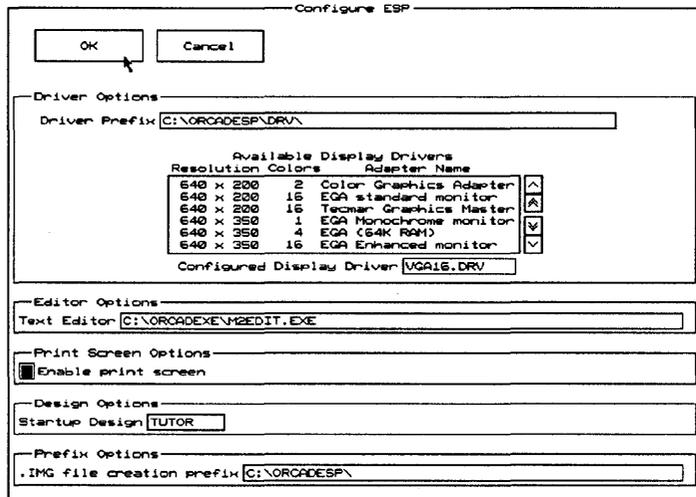
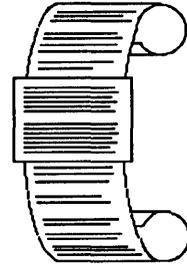


Figure 3-1. Configure ESP screen.

The **Configure ESP** screen contains more information than can fit on the screen at one time. Configure information displays in a long, continuous format, much like a scroll. Your display shows one portion, or "window full," at a time.



Move the pointer down until it touches the lower edge of the display, and the display pans to show more of the options. When you get to the bottom, the display will only pan up.

If you prefer to use keyboard commands, press <Page Down> to move the window down part of a screen at a time, and <Page Up> to go up again. Press <End> to go to the bottom of the configuration screen, and <Home> to return to the top again.

Driver Options

The **Driver Options** area defines the driver prefix and display driver the design environment uses.

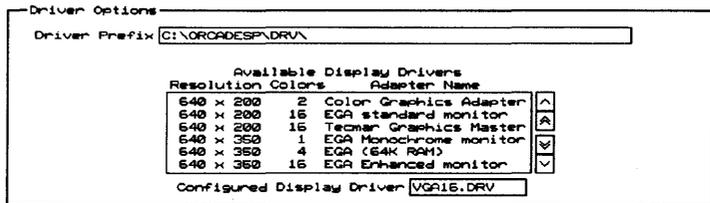


Figure 3-2. Driver Options area.



NOTE: Each OrCAD tool set can be individually configured to use a display driver. For example, you may want to use 640 x 480 resolution display driver for the environment, and a higher resolution 800 x 600 display driver for PC board layout.

Driver Prefix

The **Driver Prefix** defines the directory path or disk drive where the environment finds and loads the graphics driver programs.

The default **Driver Prefix** is created during the installation process. If you installed the environment on your C: drive, the prefix is:

Driver Prefix

All of the drivers in that directory are listed in the **Available Display Drivers** list box.

To change the driver prefix, place the cursor in the **Driver Prefix** entry box and enter the pathname of the directory containing your device drivers. After you press <Enter>, the drivers in the specified directory are listed in the **Available Display Drivers** list box.

Available Display Drivers

The **Available Display Drivers** list box lists the display drivers available in the subdirectory path specified in the **Driver Prefix** entry box.

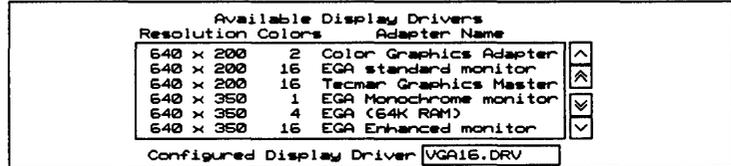


Figure 3-3. Available Display Drivers list box.

Select the driver appropriate for your computer by clicking on it. If you want to see drivers not visible in the window, use the arrow keys at the right of the window to scroll the list of drivers up and down.

- Scrolls the list of drivers up, one line at a time.
- Scrolls the list up, one "window full" at a time.
- Scrolls the list down, one "window full" at a time.
- Scrolls the list down, one line at a time.

Configured Display Driver

Once you select a display driver, its filename displays in the **Configured Display Driver** entry box. For example, if you select the Enhanced monitor from the drivers displayed in figure 3-3, the following displays:

Configured Display Driver

If you want to use a display driver that is not listed, enter the driver name directly in the **Configured Display Driver** entry box by clicking on the entry box and entering the driver name. This driver must be found in the same directory as the other drivers.



NOTE: If you are updating from an older version of OrCAD software, do not mix the old display, printer, and plotter drivers with the new ones. The new drivers are incompatible with the old versions of OrCAD software; the old drivers are incompatible with the new.

Editor Options

The **Editor Options** area defines the text editor to use when any **Edit File** command is used. You can use any editor or word processor on your system. The default selection is the OrCAD text editor.

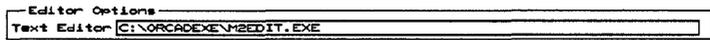


Figure 3-4. Editor Options area.

Enter a complete path and editor name in the **Text Editor** entry box.

Design Options

The **Design Options** area is for selecting the project you wish to work on when you start running OrCAD tools. The default selection is the TEMPLATE design.



Figure 3-5. Design Options area.

Enter a design name in the **Startup Design** entry box.

Color tables

The remaining configuration options allow you to tailor the colors the environment displays to match your personal preferences. In the figures shown in this section, the default selections are shown.

To change the color of an item, simply click on the desired color.

Text

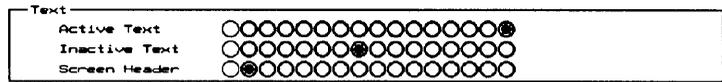


Figure 3-6. Color Table for text options.

Configuration

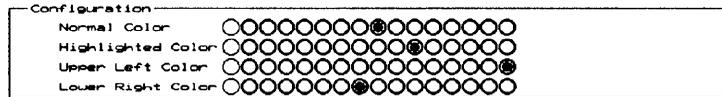


Figure 3-7. Color table for configuration options.

Tool sets

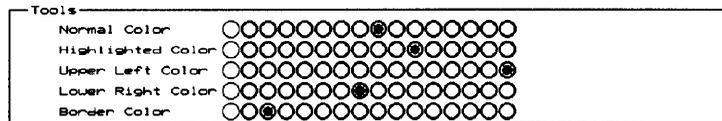


Figure 3-8. Color table for tool sets.

Editors

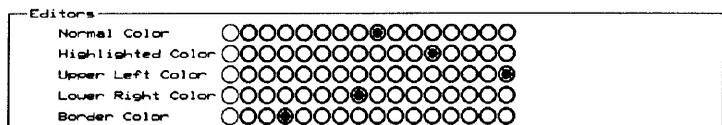


Figure 3-9. Color table for editors.

Processors

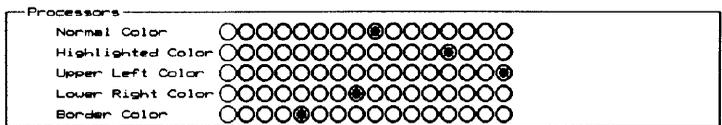


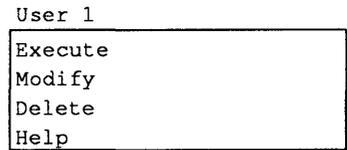
Figure 3-10. Color table for processors.



Defining a user button

Four buttons labeled **User 1**, **User 2**, **User 3**, and **User 4** are found inside every tool set. You can program these buttons to do what you want them to.

When you click on a user button, the menu shown at right displays. The commands on this menu and how to use them are explained in this chapter.



Execute

This command runs the command defined in the user button. Just select the user button and **Execute**.

Modify

Modify defines a new button or redefines a button that is already defined.

Example Here is an example of how you would define a user button to run a utility called PCSHELL:

1. Select **User 1** and **Modify**. The Modify user button screen displays.

The screenshot shows a dialog box titled 'Modify user button'. It has the following fields:
- Button Name (top line): User 1
- Button Name (bottom line):
- Help File Name:
- Command:
- Parameters:
At the bottom right, there are two buttons: 'OK' and 'Cancel'.

Figure 4-1. Modify user button screen.

2. To change the button name in the top or bottom line, click the **Button Name** entry box you want to change. Click the top **Button Name** entry box. Delete the default entry (User 1) and enter **pcshell**.

A **Button Name** may be up to ten characters long.

3. In the **Command** text entry box, enter **pcshell**.
4. Save the new definition of the button by clicking the **OK** button.

If you select the **Cancel** button, the design environment leaves the **Modify user button** screen without updating the definition.

Define a command with parameters

To define a user button to run a command that has parameters, you enter the parameters in the **Parameters** entry box. For example, to list the C: root directory to a file, you enter the following command and parameters:

The screenshot shows two text input fields:
- Command: dir
- Parameters: c:\ > directry.lst

This tells the design environment to list the directory named in the parameters entry box—in this case, the C: drive—and save the output in a file named **DIRECTRY.LST**.

**Define more than one
command in a user
button**

A user button runs only one command. To define a user button that runs more than one command, you create a batch file and define the user button to run the batch file. Do this by entering the batch file name in the **Command** entry box.

Batch files normally end with a .BAT extension. You can create the batch file using the OrCAD text editor. For information on creating batch files, refer to your operating system manual.

**Specify the Help File
Name**

The **Help File Name** entry box tells the design environment the name of a text file you create. The text file must be saved in \ORCADUSR in order for the environment to find it. This file displays when you select **Help** from the menu that displays when you press the user button.

A help file is a good way to store more information about the button's function than can fit in the **Button Name** fields.

To use a help file, enter the filename of the text file in the **Help File Name** entry box.

Delete

This command deletes the definition of the selected user button and renames the user button to its original name.

To delete the definition of the PCSHELL button you just defined, select **PCTOOLS** and then select **Delete**.

Help

This command searches in the directory \ORCADUSR for a text file created by you and displays the file on the screen. See *Specify the Help File Name* above for information on creating the help file.



Design environment technical information

The OrCAD design environment embodies the same human-friendly philosophy as the original OrCAD programs, while providing more sophisticated design management, faster performance, and support for expanded memory.

Directory structure

The design environment allows you to keep all files for a design—schematics, PLD logic, simulation files, and printed circuit board layouts—in one directory. This organization requires a directory structure that is different from the one used in previous versions of OrCAD software. The new structure is shown on the next page.

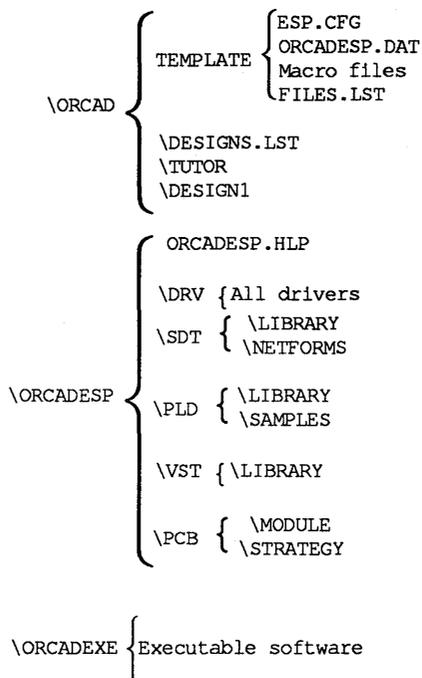


Figure 5-1. OrCAD directory structure.

Contents

The contents of the disks for the OrCAD design environment are described below.

Executable software

These files are found in the directory ROOT:\ORCADEXE. In addition to these files, the directory contains the executable software for the other installed OrCAD products.

ESP.EXE Startup program for the OrCAD design environment.

ESP_C.EXE Configuration program for the OrCAD design environment.

ESPMAIN.EXE Main program for the OrCAD design environment.

PCB_C.EXE General configuration program for **PC Board Layout Tools**.

PCB_CLC.EXE Local configuration for each process on each button on the **PC Board Layout Tools** screen.

PLD_C.EXE General configuration program for **Programmable Logic Design Tools**.

PLD_CLC.EXE Local configuration for each process on each button on the **Programmable Logic Design Tools** screen.

SDT_C.EXE General configuration program for **Schematic Design Tools**.

SDT_CLC.EXE Local configuration for each process on each button on the **Schematic Design Tools** screen.

VST_C.EXE	General configuration program for Digital Simulation Tools .
VST_CLC.EXE	Local configuration for each process on each button on the Digital Simulation Tools screen.
Other files	The following files are also used by the OrCAD design environment.
#ESP_OUT.TXT	This file contains the screen output from the last tool executed in the design. This filename is the default; you can configure the filename to be anything you want.
<i>About .BCF and .CFG files</i>	The configuration that was stored by previous versions of OrCAD software in .OVL files (or, in the case of OrCAD/VST, in .CNF files) is now stored in two files. *.BCF Configuration information in binary format. *.CFG Configuration information in ASCII format. This lets people who want to write their own utility programs bypass the normal method of entering ESP and change configuration there. This file also eliminates the need to transfer information from an older version of the .OVL file to the newer one when OrCAD updates the software. Each OrCAD program checks the date and time the .BCF and .CFG files were created. If the .CFG file is newer than the .BCF file (which would only happen if the .CFG file is changed manually), the software automatically converts the .CFG file to binary format and saves it as the .BCF file.

<i>ESP.CFG</i>	The design environment configuration information in ASCII format. This is the central configuration file for ESP; this file must be located in the \ORCAD\TEMPLATE directory.
<i>DESIGNS.LST</i>	Description file for designs.
<i>FILES.LST</i>	Description file for files.
<i>ORCADESP.DAT</i>	Data file that stores configuration information.
<i>ORCADESP.HLP</i>	A text file that stores information displayed when Help is requested.
<i>PCB.BCF</i>	PC Board Layout Tools (Release 4) configuration information in binary format.
<i>PCB.CFG</i>	PC Board Layout Tools (Release 4) configuration information in ASCII format.
<i>PLD.BCF</i>	Programmable Logic Design Tools (Release 4) configuration information in binary format.
<i>PLD.CFG</i>	Programmable Logic Design Tools (Release 4) configuration information in ASCII format.
<i>SDT.BCF</i>	Schematic Design Tools (Release 4) configuration information in binary format.
<i>SDT.CFG</i>	Schematic Design Tools (Release 4) configuration information in ASCII format.

VST.BCF **Digital Simulation Tools (Release 4)** configuration information in binary format.

VST.CFG **Digital Simulation Tools (Release 4)** configuration information in ASCII format.

DOS environment variables

There are four environment variables used by the design environment in conjunction with DOS. They are defined in the AUTOEXEC.BAT file for your computer, using DOS's SET (Set Environment) command, so they are loaded every time you turn on your computer. The OrCAD INSTALL program can add these definitions to your AUTOEXEC.BAT file automatically during installation.

```
SET ORCADEXE=C:\ORCADEXE\  
SET ORCADESP=C:\ORCADESP\  
SET ORCADPROJ=C:\ORCAD\  
SET ORCADUSER=C:\ORCADESP\
```

△ **NOTE:** *In the examples above, C: is shown as an example. During installation, you specify the drive on which to install the environment and other OrCAD software. The installation program places the appropriate drive designator in AUTOEXEC.BAT when it adds the environment variables.*

By default, DOS reserves a fixed amount of memory for environment variables. Occasionally, adding these four variables to your AUTOEXEC.BAT file means DOS needs more than the default amount of memory in order to load these variables. When this happens, DOS displays this message when you reboot your computer following installation:

```
Out of environment space
```

When this message displays, you need to increase the number of bytes DOS reserves for environment variables by using the DOS SHELL command. Using an editor, add this statement to your CONFIG.SYS file:

```
SHELL=COMMAND.COM /P /E:###
```

where:

/P causes COMMAND.COM to remain loaded and to execute AUTOEXEC.BAT when it is installed.

/E:### specifies a base 10 integer and indicates the number of bytes to set the environment size. This number must be in the range of 160 to 32768. Non-numeric characters are not allowed. A number between 700 and 800 is usually plenty. For more information about the SHELL command, refer to your DOS documentation.

- | | |
|------------------|---|
| <i>ORCADESP</i> | This variable points to the directory where the design environment files are found, usually ROOT:\ORCADESP. |
| <i>ORCADEXE</i> | This points to the directory containing all OrCAD executable programs, usually ROOT:\ORCADEXE. |
| <i>ORCADPROJ</i> | This points to the directory containing design directories, usually ROOT:\ORCAD. |
| <i>ORCADUSER</i> | This points to the directory containing help files, usually ROOT:\ORCADESP. |

A

Analog ■ Circuitry where both voltage and frequency output vary continuously as a function of the input.

Annotation ■ Assigning reference designators to components in a schematic.

ASCII ■ An acronym for *American Standard Code for Information Interchange*; a seven-bit code used to represent letters of the alphabet, the ten decimal digits, and other instructions used to edit text on a computer, such as Backspace, Carriage Return, Line Feed, and so on.

B

Button ■ A pushbutton-like image you click to start an *action*. The *action* runs a single tool or a series of tools. Buttons can be configured, which is called *local configuration* to distinguish button configuration from configuration of the entire tool set.

Byte ■ A piece of computer data composed of eight contiguous bits that are grouped together as a single unit.

C

Check box ■ A small square button: . Check boxes are used in lists of options when more than one option can be active at a time.

Complex hierarchy ■ A design in which two or more sheet symbols reference a single worksheet. Compare with *simple hierarchy*.

Configuration ■ The information a button or tool set uses to operate. Configurations can be tailored to your needs. The configuration for a tool set applies to all tools in the set. Local configuration applies to the tool (or tools) the button runs.

Cursor ■ A square marker inside a text field showing where characters typed on the keyboard will appear: ■ See *pointer*.

D

Default ■ A preselected parameter.

Design ■ *noun*: A set of plans for electronic circuitry.

Design cycle ■ The process of conceiving, developing, testing, and producing a circuit.

Digital ■ Circuitry where data in the form of digits are produced by binary on and off or positive and negative electronic signals.

E

EDA ■ An acronym for *electronic design automation*.

Editor ■ A tool used to create or modify a design file.

Entry box ■ A box indicating that something (text or numbers) should be entered using the keyboard:



F

Flat file structure ■ A schematic diagram in which output lines of one sheet connect laterally to input lines of another sheet through graphical objects called *module ports*. Flat file structures are practical for small designs of three or fewer sheets. See *module port, sheet, hierarchical structure*.

H

Hierarchical structure ■ A schematic design structure in which sheets are interconnected in a tree-like pattern vertically and laterally. At least one sheet, the root sheet, contains symbols representing other sheets, called subsheets.

K

K ■ A unit of measurement. 1K byte is equal to 1024 bytes. The "K" is taken from the metric system, where it stands for "kilo," or 1000. 1024 is 2^{10} and is close to 1000.

L

Layout ■ A to-scale drawing of a printed circuit board, its components, and its electromechanical connections. Also called artwork.

Library ■ A collection of standard, often-used part symbols stored as templates to speed up design work on the system.

Librarian ■ A tool used to manage or create library parts.

Local configuration ■ Configuration settings for a particular button. If the button runs several tools, each tool can be locally configured. One tool can have different configurations in different buttons. For example, **Annotate** may be configured differently under the **To Layout** button and under the **To Digital Simulation** button.

M

MB ■ An abbreviation for *megabyte*. See *megabyte*.

Megabyte ■ Slightly more than one million bytes; 10 megabytes equals 10 million bytes. A megabyte is equal to 2^{20} bytes (1,048,576). "Mega" is taken from the metric system, where it is a prefix meaning one million.

Module port ■ Graphical objects that conduct signals between schematic worksheets. See *flat file structure*.

N

Netlist ■ An ASCII file that lists the interconnections of a schematic diagram by the names of the signals, modules, and pins connected together on a PCB. The nodes in a circuit.

P

PCB ■ An acronym for *printed circuit board*.

PLD ■ An acronym for *programmable logic device*. See *programmable logic device*.

Pointer ■ An arrow on the screen that moves as you move the mouse: 

Processor • A tool that subjects a design file to a specific process.

Programmable logic device ■ A type of integrated circuit that contains fuses that can be blown, eliminating certain logical operations in the device and leaving others intact, giving the device one of many possible logical architectures or logical configurations.

Prompt ■ A query from a program asking you to enter specific information.

R

Radio button ■ A small round button: . Radio buttons are used in lists of mutually exclusive options: only one button can be active at a time.

Reporter ■ A tool that creates a report, but does not modify design data.

Root directory ■ The main directory on your computer; the directory that the computer boots from.

Root sheet ■ The sheet at the top of a hierarchy that references all other sheets in a schematic diagram. See *sheet*, *hierarchical structure*, and *schematic*.

S

Schematic ■ A graphical representation of a circuit using a standard set of electronics symbols. See *sheet*, *hierarchical structure*, and *root sheet*.

Scroll buttons ■ Buttons used to move a directory in its window so that a different part is visible. The four scroll buttons are:

 Page Up

 Line Up

 Page Dn

 Line Down

Sheet ■ A page of a schematic diagram. Also called worksheet. Each schematic file contains one sheet. See *root sheet*, *hierarchical structure*, and *schematic*.

Simple hierarchy ■ A one-to-one correspondence between sheet symbols and the schematic diagrams they reference. Each sheet symbol represents a unique subsheet. See *hierarchical structure*.

Simulation ■ A computer-generated prediction of circuit behavior in response to an event or set of events.

Source code ■ The words, phrases, and logical expressions in a high-level computer language that define the logic that a compiler, such as the PLD compile processor, translates into instructions that machines can use.

Syntax ■ The formal structure of a language. Syntax includes the rules for making statements in the language, but excludes the meanings of the statements.

T

Tag ■ A marked or saved location on a schematic or layout. You can use the **JUMP** command to go to a tag.

Template ■ A set of patterns used to create new designs. The template is *not* a design itself. See *design*.

Text export ■ The process of copying text from a schematic worksheet to an ASCII file.

Text import ■ The process of copying text from an ASCII file to a schematic worksheet.

TTL ■ An acronym for *transistor to transistor logic*.

Tool ■ A tool is a computer program you can use to do some useful task. OrCAD tools are grouped into five categories: editors, processors, reporters, librarians, transfers.

Tool set ■ A collection of tools designed to perform a set of electronic design automation tasks. OrCAD tool sets include: **Schematic Design Tools**, **Programmable Logic Design Tools**, **Digital Simulation Tools**, and **PC Board Layout Tools**.

Transfer ■ A tool that transfers design information from one tool set to another tool set. Also runs whatever processes are necessary to go from one tool set to another.

U

Unlocked sequential logic ■ Logic whose behavior depends on its current inputs and its recent history but is not synchronized to a clock.

Upload ■ The process of sending a file to another computer.

User button ■ A button that you can program to perform whatever combination of functions you find useful (such as executables or batch files). User button programs are saved with the design files, so you can create design-specific buttons and not worry about overwriting user button programs for other designs.

W

Worksheet ■ See *sheet*.

Z

Zoom ■ The ability to change the view on the screen by making the objects appear larger or smaller.

<Enter> 5, 6
 <Esc> 5
 <Shift><Tab> 6
 <Tab> 6

A

Analog 55
 Annotation 55
 ASCII 55
 AUTOEXEC.BAT 53

B

Backup Design 18
 Brackets 3
 Button 4, 7, 55
 Byte 55

C

Check box 55
 Click 5
 Commands 3
 Complex hierarchy 21, 55
 Complex to Simple 21
 Configuration 11, 35, 55
 Configure ESP 14
 Conventions 3
 Copy Design 20
 Copy File 28
 Create Design 16
 Cursor 55
 Customizing the environment 35

D

Default 55
 Delete Design 23
 Delete File 31
 Design 13, 55
 Cycle 55
 Options 11
 View 13, 14
 DESIGNS.LST 51

Digital 55
 Directory structure 47
 Double-click 5

E

EDA 55
 Edit File 32
 Editors 8, 55
 Enter 4
 Entry box 17, 55
 Environment space 53
 Environment variables 53

F

File View 13, 27
 FILES.LST 51
 Filter 33
 Flat file structure 56

H

Help command 14
 Hierarchical structure 56
 Highlight 5

L

Layout 56
 Librarian 3, 9, 56
 Library 56
 List boxes 15
 Local Configuration 14, 56

M

Main menu commands 3
 MB 56
 Megabyte 56
 Module port 56
 Mouse 5

N

Netlist 56
 Notation conventions 3

O

ORCADESP 54
ORCADESP.DAT 51
ORCADESP.HLP 51
ORCADEXE 54
ORCADPROJ 54
ORCADUSER 54

P

PCB 56
PLD 56
Point 5
Pointer 4, 57
Prefix/wildcard 33
Processors 8, 57
Programmable logic device 57
Prompt 57

Q

Quotation marks 3

R

Radio button 15, 57
Rename File 29
Reporters 9, 57
Restore Design 24
Root directory 57
Root sheet 57

S

Schematic 57
Scroll buttons 15, 57
Sheet 57
Simple hierarchy 21, 57
Simulation 57
Source code 57
Spacebar 6
Startup Design 11
Suspend to System 25, 30
Syntax 57

T

Tag 58
Technical information 47
TEMPLATE 10, 58
Terminology 4
Text export 58
Text import 58
Title bar 4
Tool *vii*, 1, 58
Tool set *viii*, 7, 58
Transfers *viii*, 9, 58
TTL 58
Type 4

U

Unclocked sequential logic 58
Update ESP Data 25
Upload 58
User buttons 10, 43, 58

W

Worksheet 58

Z

Zoom 58

OrCAD®

