PC Board Layout Tools 386+

Reference Guide

Printed on recycled paper

OrCAD®

Electronic Design Automation Tools

PC Board Layout Tools 386+

Reference Guide

Third Edition 1 Aug 1994

OrCAD®

9300 SW Nimbus Avenue
Beaverton, Oregon 97008-7137
U.S.A.

General Product Information &	
Non-Technical Customer Support	(503) 671-9500
Fax	(503) 671-9501
Technical Support Hotline	(503) 671-9400
24-Hour Bulletin Board System	(503) 671-9401

CONTENTS

Chapter 2: Edit Layout (continued)

Chapter 2: Edit Layout (continued)

Chapter 2: Edit Layout (continued)

Chapter 2: Edit Layout (continued)

Chapter 2: Edit Layout (continued)

Chapter 2: Edit Layout (continued)

OrCAD's **PCB 386+ 2.00** operates within the ESP design environment, which provides many features that make it easier to access and use OrCAD's electronic design automation (EDA) tool sets.

This book is a reference guide to **PCB 386+ 2.00**, the tool set used to lay out printed circuit boards. See the *ESP Design Environment User's Guide* for detailed information about how to use OrCAD tools and tool sets in the ESP design environment.

Finding the information you need

These documents accompany **PCB 386+ 2.00**:

- *Installation Instructions*
- *ESP Design Environment User's Guide*
- *PC Board Layout Tools 386+ User's Guide*
- *PC Board Layout Tools 386+ Reference Guide*
- *Stony Brook M2EDIT Text Editor User's Guide*

Users upgrading from **OrCAD/PCB II** to **PCB 386+ 2.00** also receive *Fast Track*.

Installation

Before you begin to explore the software, take a few minutes to install the tool set and register for technical support. Just follow the installation instructions that accompany **PCB 386+ 2.00**.

Be sure that you read the "readme" file for other installation information.

About this guide

This reference guide helps you learn how to use **PCB 386+ 2.00** with the ESP design environment to lay out printed circuit boards.

The first six parts of this guide are organized according to function. Each tool is described in one of these parts:

- *Part I: Configuration*
- *Part II: Editors*
- *Part III: Processors*
- *Part IV: Reporters*
- *Part V: Librarians*
- *Part VI: Transfers*

For example, to find information about **Edit Layout,** look in *Part II: Editors*.

At the end of the guide, Appendix A provides additional information about running the tools from the command line.

A glossary and index follow Appendix A.

Other OrCAD publications

Some types of information change more rapidly than the manuals do, so OrCAD publishes frequently changing information separately in technical notes and other documents.

You can call, write, send a fax, or post a message on the bulletin board system (BBS) to get copies of OrCAD publications.

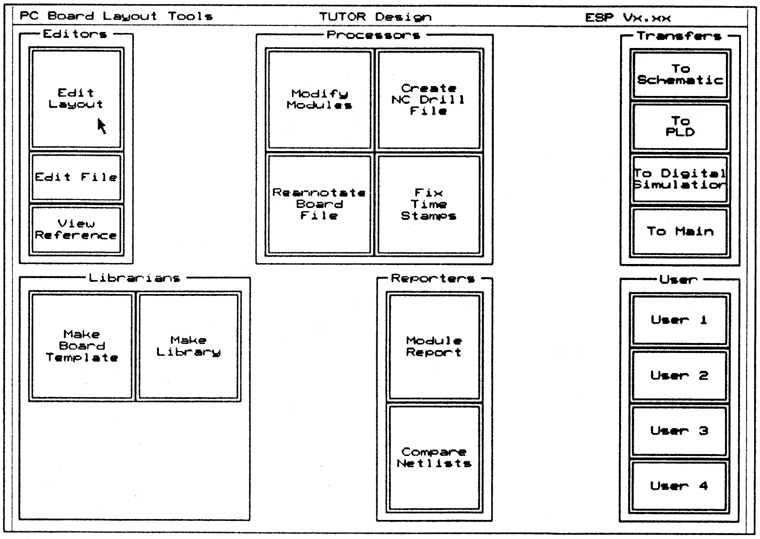
Tool sets and tools

A *tool set* is a collection of tools designed to perform a set of EDA tasks. Buttons that provide access to all four OrCAD tool sets display on the main screen, even if you only have one tool set installed on your computer. OrCAD's tool sets are:

- **Schematic Design Tools**
- **Digital Simulation Tools**
- **Programmable Logic Design Tools**
- **PC Board Layout Tools**

The tool sets manipulate the same design in different ways.

To select the **PCB 386+ 2.00** tool set from the main design environment screen, move the pointer to the **PC Board Layout Tools** button and double-click. After a moment, you see the **PC Board Layout Tools** screen shown in the following figure.



PC Board Layout Tools screen.

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

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

These categories are described on the following pages. The descriptions assume that you are already familiar with common electronic terms and concepts. If you are just learning about board layout, some terms we use to describe the tools may not be familiar to you. Don't worry: basic, essential concepts and skills are thoroughly covered in the *PC Board Layout Tools 386+ User's Guide.* Advanced concepts are fully explained in the reference chapters in parts I through VI of this guide.

Editors

Editors modify or create some part of the design database. An example of an editor is the board editor, **Edit Layout.** Another editor is **Edit File**, which uses a text editor to view reports and enter text. **PCB 386+ 2.00** contains several editors, some contained within **Edit Layout.**

❖ **Edit Layout** is the board editor at the heart of **PCB 386+ 2.00**. In **Edit Layout**, you create and modify printed circuit boards. **Edit Layout** also includes the following editors:

- **Library editor.** Creates, loads, and modifies modules and module libraries.
- **Pad stack editor.** Dialog box in which you define the shape, rotation, layer, and other aspects of the pad stacks used in the layout.
- **Via stack editor.** Dialog box in which you define the layer, drill width, and other aspects of the via stacks used in the layout.
- **Copper tool editor.** Dialog box in which you define all aspects of the copper tools used to define the layout's attributes.
- **Drill list editor.** Dialog box in which you define the drill diameters used in the layout.
- **Net property editor.** Dialog box in which you define the copper tool, name, and other aspects of the nets used in the layout.

❖ **Edit File** runs a text editor, with which you create and edit text files.

❖ **View Reference** runs the configured text editor to display reference material supplied with **PCB 386+ 2.00**. You can view files about drivers, libraries, and other topics of interest.

Processors

Processors read, modify, then rewrite the design database. For example, **Reannotate Board File** is a processor. Processors generally create or modify database information, and may also create reports. Processors may create data that will be used by tools outside the design environment.

PCB 386+ 2.00 includes the following processors:

- **Modify Modules** modifies pad shape, pad size, and drill size for modules in a board file.
- **Create NC Drill File** generates a file containing drilling information, including location and drill size, for a board file.
- **Reannotate Board File** reannotates your board file so the modules are numbered sequentially, according to their position on the board.
- **Fix Time Stamps** sets the time stamps that identify modules in your board file to match the corresponding time stamps in a netlist file.

Reporters

Reporters create reports, but do not modify design data in any way. Reporters may create reports that will be used by tools outside the design environment. PCB 386+ 2.00 has two reporters:

- **Module Report** reports module locations in your board file.
- **Compare Netlists** reports differences between an EDIF netlist file and a board file.

Librarians

Librarians are tools for managing and creating library objects that can be used by all designs, not just the current design. PCB 386+ 2.00 includes two librarians:

- **Make Board Template** creates a template (a "starting point" for new board layouts or libraries) from a PCB 386+ 2.00 board file.
- **Make Library** creates a library file from a **PCB 386+ 2.00** board file.

Transfers

Transfer tools manage the steps needed to move design information from one tool set to another. A transfer may modify the design database or simply transfer control to another tool set.

PCB 386+ 2.00 has four transfers: **To Schematic, To PLD, To Digital Simulation**, and **To Main.** Each of these transfers control to the specified tool set. For example, the **To PLD** transfer tool transfers control to the **Programmable Logic Design Tools** tool set.

User buttons

A user button can be set up to run any system command or any .EXE, .COM, or .BAT file. A user button is the simplest way in which the ESP design environment can be extended to fit your particular requirements and make your work easier and more convenient.

For example, you can set up a user button to run a spreadsheet program, which you can then use to analyze design information. If you depend on a particular set of operating system utility programs, you can assign a user button to call them up. See *Chapter 4: Defining a user button* in the *ESP Design Environment User's Guide* for detailed instructions.

Conventions

The conventions used in this guide are as follows:

Bold	Bold indicates a command.
`Courier bold`	Bold monospace indicates text you enter exactly as shown.
Italics	Italics indicate a reference to another section or chapter of this guide or to another publication.
<B>	Angle brackets enclose a key that you press. For example, <Esc> indicates the escape key.
"Prompt"	Quotation marks indicate program prompts and messages.

"Enter" and "type"

In OrCAD manuals, the terms "enter" and "type" mean two different things. When the instructions tell you to *enter* something, press the appropriate keys and end by pressing <Enter>. When the instructions tell you to *type* something, press the appropriate keys but do not press <Enter>.

Boxes

The box shown below represents a system prompt. Any bold type following the prompt indicates text that you enter.

```
C:> orcad
```

A box like the one shown at right represents an OrCAD menu.

```
Execute
Local Configuration
Assign Hot Key
Configure ESP
Help
```

A box like the one shown below represents a text entry box. Entry boxes appear on configuration screens, and can be empty or contain information you can edit.

```
Wildcard  *.*
```

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

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

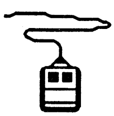
About entry boxes

You use all the entry boxes in the ESP design environment in the same way:

- ❖ Place the pointer inside the box and press <Enter> to enter insert mode. The pointer changes shape to become an underline cursor (_). In insert mode, the characters you type are inserted in any existing text at the point the cursor marks.
- ❖ To change to overtype mode, press <Insert>. The cursor becomes a square (■). In overtype mode, the characters you type replace any characters already there. You can toggle between insert and overtype modes as needed.
- ❖ Press <Enter> again to leave the entry box. The cursor is replaced by the pointer.

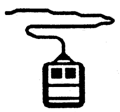
You can also use the editing keys on your keyboard to move around the entry box and edit its contents:

- ❖ <Home> moves the cursor to the beginning of the entry box.
- ❖ <End> moves the cursor to the end of the entry box.
- ❖ The arrow keys <→> and <←> move right and left one character at a time, without erasing what you typed.
- ❖ <Backspace> backs up one character and deletes it.
- ❖ <Del> erases the character at the cursor's position without moving the cursor.
- ❖ The <Ctrl><Del> combination deletes the entire contents of the entry box.
- ❖ <Esc> aborts any changes to the entry box and changes the cursor back to a pointer.



You can use the mouse like the <←> and <→> arrow keys to move the cursor inside the entry box.

Mouse techniques



You can do all your work in the ESP 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 select an object by pointing to it and *clicking* (pressing and then releasing) the left mouse button once. When you select a button, it becomes highlighted and a menu pops up in the upper-left corner of the screen.

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 either press the <Enter> key or click the left mouse button.
- 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 either press the <Esc> key or click the right mouse button.

Keyboard equivalents

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

- Press <Tab> to move the pointer to the first button in the next area on a tool set screen or, on configuration screens, to the next entry box.
- Press <Shift><Tab> to move the pointer backwards to the first button in the previous area.
- Press <Space bar> to move the pointer from button to button within a group of tools, a set of radio buttons, or the scroll buttons associated with a list box.
- Press <Enter> to select the item the pointer rests on.
- Press <Home> to move the pointer to the first button in the area nearest the upper-left corner of the screen or, on configuration screens, to the OK button.
- Press <End> to move the pointer to the first user button or, on configuration screens, to the last button in the last area.
- Press <Esc> to close a menu without selecting any of the commands or to cancel any changes to a text entry box.
- Press <Page Up> and <Page Down> to pan up and down on configuration screens.

You can also assign keys or key combinations, called *hot keys,* to tools so you can select tools from the keyboard. For information about assigning hot keys, see *Chapter 3: Customizing the ESP design environment* in the *ESP Design Environment User's Guide.*

Configuration screens

PCB 386+ 2.00 and the ESP design environment have many configuration screens. Some configuration screens apply only to a specific tool. These are called *local configuration* screens. Other configuration screens—such as the **Configure PC Board Layout** screen—are global in nature.

About ".\" in pathnames

Many configuration screens have entry boxes that specify path and filenames. Labels for these entry boxes include **Prefix/Wildcard**, **Source**, and **Destination**.

When you specify a pathname, you can use a period and a backslash (.\) as a convenient shortcut to specify the current design directory. For example, if the current design is TEMPLATE, then ".*.MLB" means all files in the \ORCAD\TEMPLATE directory that have a .MLB extension.

Prefix/Wildcard entry boxes

Many configuration screens have a **Prefix/Wildcard** entry box. These entry boxes contain a pathname and possibly a filename with a wildcard to indicate which files to display in a list box. The asterisk can be used as a wildcard in a filename. This example lists all files in the C:\ORCADESP\PCB\LIBRARY path that have a .MLB extension:

```
Prefix/Wildcard  C:\ORCADESP\PCB\LIBRARY\*.MLB
```

List boxes

Many configuration screens have *list boxes* containing lists of items from which to choose. Be sure you know how to select an item from a list box and how to use the scroll bars to scroll the item lists. Items with ".\" are found in the current design directory. Items without ".\" are found in the path given in the **Prefix/Wildcard** entry box. When you place the pointer on a filename in a list box and select <Enter> or click the left mouse button on a filename in a list box, the item automatically displays in the related entry box.

Filename entry boxes

Most local configuration screens have a **Source** entry box. Many have other filename entry boxes as well.

The first time you display a local configuration screen, its **Source** and **Destination** entry boxes contain—where appropriate—the name of the root sheet (specified in **Design Management Tools**) followed by a default extension. You can, however, change this to suit your needs.

If you change the filename extension in the **Source** entry box, when you select **OK** to leave the configuration screen and save the changes, the extension in the **Prefix/Wildcard** entry box also automatically changes to the same extension.

On many configuration screens, you can use a question mark (?) as a shorthand notation for the name of the root sheet. For example, if the current root sheet is TUTOR and you enter `?.MLB`, the ESP design environment interprets the "?" as "TUTOR" when you select **OK** to leave the configuration screen and save your changes. See the section *Using Design View* in *Chapter 2: Using Design Management Tools* of the *ESP Design Environment User's Guide* for a description of the root sheet and how it controls filenames in configuration screens.

△ *NOTE: This description applies only to ESP design environment configuration screens. In* ***Edit Layout****, the question mark (?) and asterisk (*) have the same function as standard DOS wildcards. That is, a question mark stands for any single character, and an asterisk stands for any number of characters.*

PART I: CONFIGURATION

When you install **PCB 386+ 2.00** on your system's hard disk, it is configured and ready to run.

Part I: Configuration explains how to customize your **PCB 386+ 2.00** configuration.

Chapter 1: *Configure Layout Tools* describes how to modify:

- ❖ Driver options
- ❖ Library options
- ❖ Prefix options
- ❖ Filter options
- ❖ Virtual memory options
- ❖ Miscellaneous options

Configure Layout Tools

The ESP design environment has three types of configuration, all of which customize and save information used to run OrCAD tools and tool sets.

- *ESP design environment configuration* defines driver options, the text editor, the startup design, and monitor display colors. Although the ESP design environment is already configured when installed, you can change the ESP design environment parameters whenever you want.

  The *ESP Design Environment User's Guide* provides detailed instructions for customizing the ESP design environment.

- *Tool set configuration* defines library, filename, and other tool set–specific options. Tool set configuration applies to all tools in a tool set and can be changed from every tool in the tool set except transfers and user buttons. It has a default configuration when installed but can also be changed anytime you want to change the tool set parameters.

  This chapter provides detailed instructions for customizing the **PCB 386+ 2.00** configuration.

- *Local configuration* determines input and output files and special processing options for a particular tool. If a tool runs several processes, each process can be locally configured.

  Local configuration is set up with input and output filenames defaulting to the design name in most cases. You usually configure a tool when you begin work on a design, or anytime you want to change the tool's parameters.

  The chapter that describes a tool also provides instructions for customizing its local configuration.

Display the Configure Layout Tools screen

With the **PC Board Layout Tools** screen displayed, select any of the editors, processors, librarians, or reporters. For example, select **Edit Layout.**

The menu shown at right displays at the top of the screen. Select **Configure Layout Tools.** Each area on the **Configure PC Board Layout** screen is shown in the sections that follow.

```
Execute
Local Configuration
Assign Hot Key
Show Version
Configure Layout Tools
Help
```

The **Configure PC Board Layout** screen contains more information than can fit on the display screen at one time. You can think of your display screen as a "window" onto the **Configure PC Board Layout** screen. Move the pointer down until it touches the lower edge of the display, and the display pans, moving the window to show more options.

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 screen and <Home> to return to the top again.

In various places within the configuration screen, there are boxes in which lists (usually of files) display. Using the scroll buttons to the right of each list box, you can move these lists up and down in a manner similar to the scrolling process used for the **Configure PC Board Layout** screen.

When you finish making changes, select **OK** to save your changes and return to the **PC Board Layout Tools** screen. If you do not want to save your changes, select **Cancel** to return to the **PC Board Layout Tools** screen.

Driver Options

The **Driver Options** area (figure 1-1) defines the driver prefix and the display and printer drivers. These are described in this section.

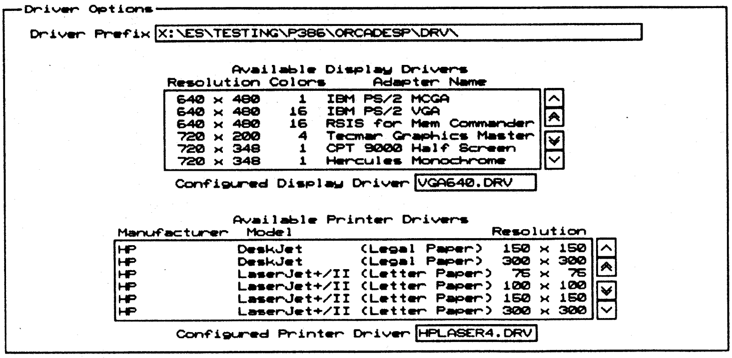


Figure 1-1. ***Driver Options*** *area of the* ***Configure PC Board Layout*** *screen.*

Driver Prefix

The **Driver Prefix** is the directory path or disk drive where **PCB 386+ 2.00** finds and loads the display and printer drivers.

The driver prefix is set during the installation process and does not need to change unless you move drivers to a different directory or create custom drivers in another directory.

To define the driver prefix, enter the pathname of the directory containing your device drivers.

Once you enter a driver prefix, all OrCAD-supplied drivers in that directory display in the appropriate list boxes: **Available Display Drivers, Available Printer Drivers,** and **Available Plotter Drivers**. Each of these list boxes is described in the sections that follow.

△ *NOTE: Only the drivers that are recognized by name appear in the list boxes. Custom drivers do not appear, and their names need to be typed into the entry boxes.*

Example

The default **Driver Prefix** is defined during the installation process. If you installed **PCB 386+ 2.00** on your C drive, the prefix is:

```
Driver Prefix  C:\ORCADESP\DRV\
```

This tells **PCB 386+ 2.00** to look for the drivers in the directory \ORCADESP\DRV on the C drive.

Available Display Drivers

The **Available Display Drivers** area is where you choose which graphics display driver to load. A list box (figure 1-2) lists the display drivers it recognizes in the directory path specified in the **Driver Prefix** entry box.

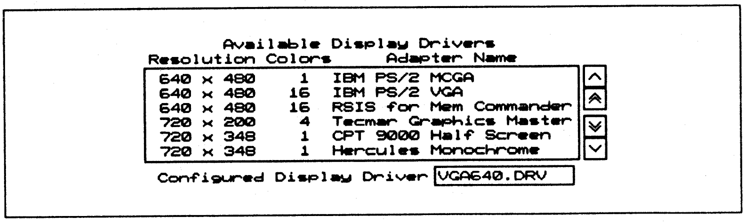


*Figure 1-2. **Available Display Drivers** list box.*

Use the scroll buttons at the right of the list box to scroll the list of drivers up and down. Select the driver appropriate for your system by clicking on it. The driver's filename displays in the **Configured Display Driver** entry box.

△ *NOTE: Drivers with a resolution lower than 640×480 are not supported for use with **PCB 386+ 2.00**.*

You do not have to select a display driver from the **Available Display Drivers** list box. Instead, simply click in the **Configured Display Driver** entry box and enter the driver name. Be sure, however, that the driver is in the directory displayed in the **Driver Prefix** entry box.

△ *NOTE: Only the drivers that are recognized by name appear in the list box. Custom drivers do not appear, and their names need to be typed into the **Configured Display Driver** entry box.*

Example

If you select **IBM PS/2 VGA** from the drivers displayed in figure 1-2, the following displays:

```
Configured Display Driver  VGA640.DRV
```

△ *NOTE: If a driver is not configured here,* ***PCB 386+ 2.00*** *uses the one selected during installation.*

Available Printer Drivers

The **Available Printer Drivers** area of the screen is where you choose which printer driver to load. A list box (figure 1-3) lists the printer drivers it recognizes in the directory path specified in the **Driver Prefix** entry box.

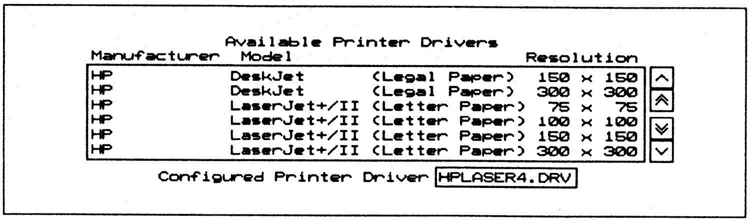


Figure 1-3. ***Available Printer Drivers*** *list box.*

Select the driver appropriate for your printer. Its filename displays in the **Configured Printer Driver** entry box.

You can also enter the driver name in the **Configured Printer Driver** entry box. Be sure, however, that the driver is in the directory displayed in the **Driver Prefix** entry box.

△ *NOTE: Only the drivers that are recognized by name appear in the list box. Custom drivers do not appear, and their names need to be typed into the* ***Configured Printer Driver*** *entry box.*

Example

If you select **LaserJet+/II (Letter Paper) 300 x 300** from the drivers displayed in figure 1-3, the following displays:

```
Configured Printer Driver  HPLASER4.DRV
```

Library Options

The **Library Options** area (figure 1-5) defines the prefix **PCB 386+ 2.00** uses to find libraries and filters the files displayed in the **Available Libraries** list box.

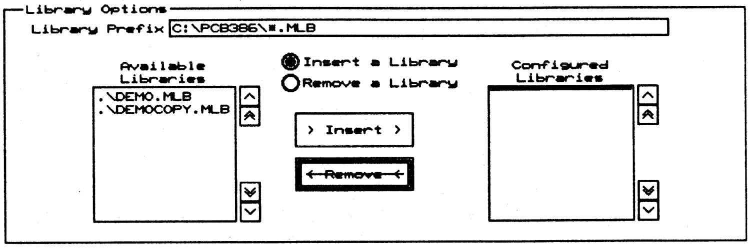


Figure 1-5. ***Library Options*** *area of the* ***Configure PC Board Layout*** *screen.*

Library Prefix

To define the **Library Prefix**, enter the pathname of the directory containing your module libraries followed by a filename or wildcard, such as *.MLB.

This example tells **PCB 386+ 2.00** to display the names of all files with a .MLB extension in the \ORCADESP\PCB\LIBRARY directory on the C drive:

```
Library Prefix  C:\ORCADESP\PCB\LIBRARY\*.MLB
```

Prefix Options

The **Prefix Options** area (figure 1-6) defines where **PCB 386+ 2.00** finds board files and netlist files and where it creates temporary files.

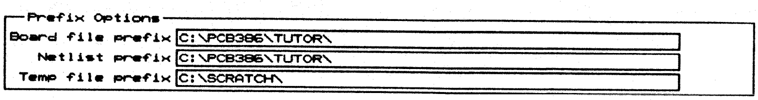


*Figure 1-6. **Prefix Options** area of the **Configure PC Board Layout** screen.*

Board file prefix

If you are using the standard OrCAD directory structure, this entry box should be blank:

```
Board file prefix [                              ]
```

If you are not using the standard OrCAD directory structure, enter the path to the directory containing your board file. This example tells **PCB 386+ 2.00** to look for the file in the \ORCADESP\PCB\BOARDS directory:

```
Board file prefix [\ORCADESP\PCB\BOARDS\         ]
```

Netlist prefix

If you are using the standard OrCAD directory structure, this entry box should be blank:

```
Netlist prefix [                                 ]
```

If you are not using the standard OrCAD directory structure, enter the path to the directory containing your netlist files. This example tells **PCB 386+ 2.00** to look for netlists in the \ORCADESP\PCB\NETLIST directory:

```
Netlist prefix [\ORCADESP\PCB\NETLIST\           ]
```

Temp file prefix

Enter the path to a drive and directory where **PCB 386+ 2.00** can create temporary files. This example tells **PCB 386+ 2.00** to create temporary files in the \SCRATCH directory on drive C:

```
Temp file prefix  C:\SCRATCH\
```

As you work, **PCB 386+ 2.00** may create many different temporary files. To be sure you have enough room for these files, you should have about five times as much available disk space as the size of your largest board file. For example, if your largest board file is 2 MB, you should have at least 10 MB free disk space.

△ *NOTE: The temp file prefix does not specify where the Phar Lap memory extender creates a swap file. See the section* Virtual Memory Options *in this chapter for more information.*

Filter Options

The **Filter Options** area (figure 1-7) defines what files **Edit Layout** lists in various dialog boxes.

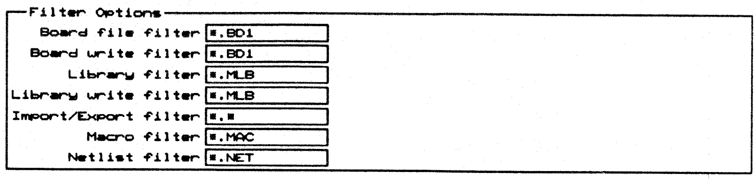


*Figure 1-7. **Filter Options** area of the **Configure PC Board Layout** screen.*

Board file filter

Enter the files to list in the dialog box that displays when you load a board file. This example tells **Edit Layout** to list all files with a .BD1 extension:

```
Board file filter  [*.BD1        ]
```

Board write filter

Enter the files to list in the dialog box that displays when you write out a board file. This example tells **Edit Layout** to list all files with a .BD1 extension:

```
Board write filter  [*.BD1        ]
```

Library filter

Enter the files to list in the dialog box that displays when you load a library file. This example tells **Edit Layout** to list all files with a .MLB extension:

```
Library filter  [*.MLB        ]
```

Library write filter

Enter the files to list in the dialog box that displays when you write out a library file. This example tells **Edit Layout** to list all files with a .MLB extension:

```
Library write filter  [*.MLB        ]
```

Import/Export filter

Enter the files to list in the dialog box that displays when you import or export a pad stack, via stack, copper tool, or drill list. This example tells **Edit Layout** to list all files:

```
Import/Export filter  *.*
```

Macro filter

Enter the files to list in the dialog box that displays when you load or save a macro. This example tells **Edit Layout** to list all files with a .MAC extension:

```
Macro filter  *.MAC
```

Netlist filter

Enter the files to list in the dialog box that displays when you load a netlist. This example tells **Edit Layout** to list all files with a .NET extension:

```
Netlist filter  *.NET
```

Virtual Memory Options

The **Virtual Memory Options** area (figure 1-8) is where you specify the directory where the Phar Lap memory extender can create a swap file and the filename it should use.

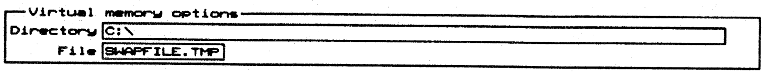


Figure 1-8. Virtual Memory Options area of the ***Configure PC Board Layout*** *screen.*

If you do not have enough contiguous space available for Phar Lap to successfully run, a warning will display indicating how much additional space is needed. Should you encounter this warning, run a defrag utility on your disk.

See *Technical Note #46: Memory considerations for 386+ programs* for a description of the Phar Lap™ memory extender.

Directory

Enter the path to a drive and directory where the Phar Lap memory extender can create a swap file. This example tells **PCB 386+ 2.00** to create the file in the SWAPDIR directory on drive C:

```
Directory  C:\SWAPDIR
```

File

Enter the name of the swap file to be created by the Phar Lap memory extender. The file will be created in the directory specified in the **Directory** entry box:

```
File  SWAPFILE.TMP
```

Note that the swap file is deleted when **PCB 386+ 2.00** exits normally. If for some reason the specified swap file is not deleted, however, the memory extender will fail when you next run **PCB 386+ 2.00**.

You can make sure the swap file is deleted by adding a line to your AUTOEXEC.BAT file. The line should look something like this:

```
DEL C:\SWAPFILE\SWAPFILE.TMP
```

Miscellaneous Options

The **Miscellaneous Options** area (figure 1-9) is where you specify the template file to load.

```
Miscellaneous Options
Template C:\PCB386\ORCADPCB._T_
```

Figure 1-9. ***Miscellaneous Options*** *area of the* ***Configure PC Board Layout*** *screen.*

ORCADPCB._T_ is the template file provided with **PCB 386+ 2.00.** ORCADPCB._T_ serves as the template board file and the template library file. You can create as many template files as you like to meet your needs.

Template

Enter the name of the template file. You can also specify a relative or absolute path to the file. See *Chapter 2: Edit Layout* for information about absolute and relative pathnames as they pertain to template files.

In this example, the absolute pathname tells **Edit Layout** exactly where to look for the template file:

```
Template  C:\ORCAD\TEMPLATE\ORCADPCB._T_
```

Part II: Editors

PCB 386+ 2.00 includes editors that you use to create and modify board files and library files, edit text files, and view files containing reference information.

Part II: Editors describes editors and provides instructions for their use.

Chapter 2: *Edit Layout* describes how to configure **Edit Layout** and provides an alphabetical reference to the procedures, concepts, commands, menus, and dialog boxes of **Edit Layout**.

Chapter 3: *Edit File* describes how to use **Edit File** to run the text editor of your choice.

Chapter 4: *View Reference* describes how to use **View Reference** to read supplemental reference material supplied by OrCAD.

Edit Layout

This chapter contains information needed to use **Edit Layout**, the board editor at the heart of **PCB 386+ 2.00**.

In this chapter, information on execution and local configuration is followed by descriptions of **Edit Layout** commands and concepts. These entries are listed in alphabetical order.

Execution

With the **PC Board Layout Tools** screen displayed, select **Edit Layout**. Select **Execute** from the menu that displays.

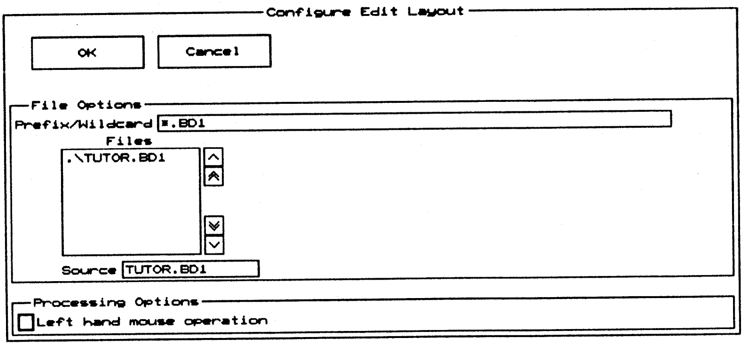
If you have not specified a board file, **Edit Layout** loads the template named in the **Miscellaneous Options** area of the **Configure Layout Tools** screen.

See the next section, *Local configuration*, and *Configuring template files* in the reference section of this chapter for more information about template files.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Edit Layout**. Select **Local Configuration** from the menu that displays.

Select **Configure PCB386. Edit Layout's** local configuration screen displays.



File Options

The **File Options** area defines the source board file.

Prefix/Wildcard

Prefix/Wildcard contains a path to the directory that contains the board file you want to edit and a filter that controls the files displayed in the **Files** list box. For example, the following entry displays all files with a .BD1 extension in the \ORCAD\TUTOR directory on drive C:

```
Prefix/Wildcard  C:\ORCAD\TUTOR\*.BD1
```

Prefix/Wildcard entry boxes are described in the *ESP Design Environment User's Guide.*

△ *NOTE: If you change the extension in the* ***Source*** *entry box and select* ***OK****, the extension in the* ***Prefix/Wildcard*** *entry box displays the new extension when you display* ***Edit Layout's*** *local configuration screen again.*

Files

This box contains a list of files that match the path and filter specified in the **Prefix/Wildcard** entry box and files in the current design directory that match the wildcard. Files in the current design directory have ".\" before their names. The filename you select in this list box displays in the **Source** entry box.

Source

Source is the name of the **PCB 386+ 2.00** board file to load. It may have any valid pathname. The source is originally set to *rootSheet*.BD1.

If you do not specify a source file, **Edit Layout** displays a notice and loads the template file. For more information see *Chapter 1: Configure Layout Tools* in this manual and the entries *Template files* and *Configuring template files* in this chapter.

Processing Options

❑ `Left hand mouse operation`

Tells **Edit Layout** to reverse the functions (<Enter> and <Esc>) of the mouse buttons. Select **Left hand mouse operation** again to disable this option and return the mouse buttons to their standard functions.

Reference

The remainder of this chapter is a reference for **Edit Layout**, the board and library editor. Procedures, concepts, commands, menus, and dialog boxes are described in alphabetical order.

Some commands on the main menu also appear on other menus. These commands (such as **FIND, JUMP,** and **ZOOM**) are described under the main menu level entry.

Abandon Program command

Appears on the board editor **QUIT** menu.

Exits **Edit Layout** and displays the **PC Board Layout Tools** screen.

△ *NOTE:* ***Abandon Program*** *does not save edits to the board file currently loaded. However, it asks if you want to save edits before exiting* ***Edit Layout****. Make sure you update your board file with* ***QUIT Update Board File*** *before you select* ***QUIT Abandon Program****.*

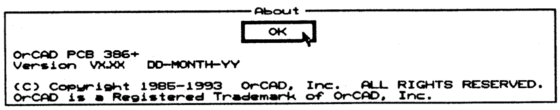
About button

Appears on the **Global Options** and **Conditions** dialog boxes.

Displays the **About** dialog box.

About dialog box

Shows the program name, version number, release date, and copyright information.



Add button

Appears on a number of dialog boxes.

In most cases, selecting **Add** adds an item shown in an entry box to the corresponding list.

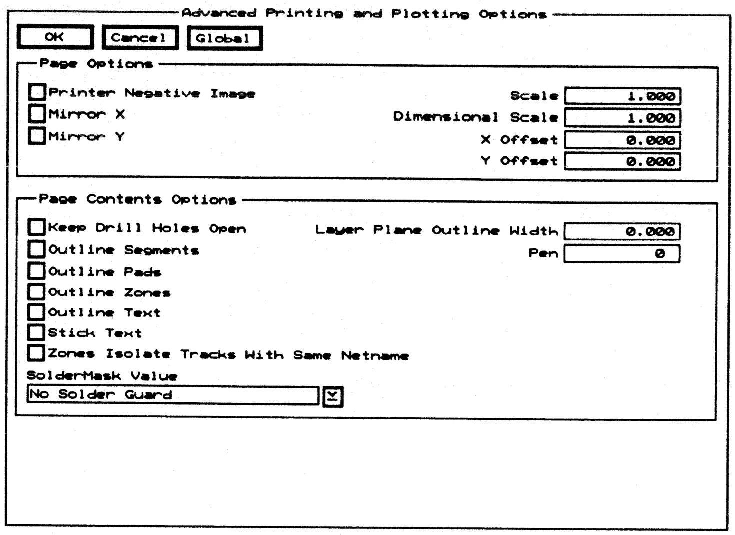
Advanced Options button

Appears on the **Printing and Plotting** dialog box.

Displays the **Advanced Printing and Plotting Options** dialog box.

Advanced Printing and Plotting Options dialog box

Use this dialog box to access infrequently used printing and plotting options.



Page Options area

Printer Negative Image Enable to print all selected pages in reverse color; that is, black for white and white for black. This option only applies to raster printers.

Mirror X Enable to flip all selected objects over the X axis.

Mirror Y Enable to flip all selected objects over the Y axis.

Scale Use to proportionally size a print or plot.

Dimensional Scale Use to scale the size of every object while keeping its position constant. This option can be used to circumvent the problems associated with smearing inner layers while laminating.

Most board houses will dimensionally scale the inner layers of a multi-layer board by adjusting the Gerber (274-D) apertures in your aperture file. When you use Gerber (274-X) or Fire 9xxx plotter output, the aperture list is embedded in the plot file and the board house may ask you to dimensionally shrink one or more inner layers before sending them your plot files. To shrink an inner layer by one half of one percent, enter 0.995 in the **Dimensional Scale** entry box.

Most board houses will dimensionally scale the inner layers of a multi-layer board by adjusting the Gerber (274-D) apertures in your aperture file. When you use Gerber (274-X) or Fire 9xxx plotter output, the aperture list is embedded in the plot file and the board house may ask you to dimensionally shrink one or more inner layers before sending them your plot files. To shrink an inner layer by one half of one percent, enter 0.995 in the **Dimensional Scale** entry box.

X Offset Use to specify the offset of the plot in the X direction. Negative values offset left; positive values offset right. This is primarily used for HPGL plotters.

Y Offset Use to specify the offset of the plot in the Y direction. Negative values offset down; positive values offset up. This is primarily used for HPGL plotters.

Note that when the paper is wider or longer than 16.787 inches, HPGL plotters move the origin to the center of the paper. When plotting on large paper using HPGL, you must offset the plot by the negative of half the paper width and height.

Page Contents Options area

Keep Drill Holes Open Enable to keep pads or routes from filling drill holes. Use this option for prototype boards which are manually drilled. Do not enable this option for art work used in large scale production.

Outline Segments Enable to prevent segments from being filled in. Typically, this option and the next three are used by designers using a pen plot on an HPGL , HPGL2, or HI29 who want to conserve ink.

Outline Pads Enable to prevent pads from being filled in.

Outline Zones Enable to plot the outline of every zone.

△ *NOTE: The enabling of* ***Copper Pour*** *on the* ***Printing and Plotting*** *dialog box, and the enabling of* ***Outline Zones*** *would cause dead shorts. Therefore, if* ***Copper Pour*** *is enabled, PCB 386+ 2.00 automatically disables* ***Outline Zones****.*

Outline Text Enable to prevent text from being filled in.

Stick Text Enable to print or plot text as single lines.

If **Stick Text** is enabled, text characters are printed or plotted with single lines, regardless of the setting of **Outline Text**.

Zones Isolate Tracks With Same Netname Enable to prevent copper pour zones from overlaying tracks on the same net.

If you have a very wide track which ties into a pad, the track may act as a heat wick during component soldering, transferring too much of the heat to the copper pour zone. This option will cause the track to be isolated from the zone, preventing this problem.

SolderMask Value Use to specify the SolderMask guard value.

No Solder Guard causes pads, vias, and test points on the solder mask plot to be plotted at their actual size.

Use Only Global Solder Guard refers to the value displayed in the **Solder Mask Guard** entry box in the **Global Options** dialog box. Uses that value for the solder guard when plotting.

Use Only Object Solder Guard refers to the values displayed in the **Solder Mask Guard** entry box in the **Edit Pad Stack** and **Edit Via Stack** dialog boxes. Uses those values for the solder guard when plotting.

Add Both Solder Guards refers to the value displayed in the **Solder Mask Guard** entry box in the **Global Options** dialog box and the values displayed in the **Solder Mask Guard** entry box in the **Edit Pad Stack** and **Edit Via Stack** dialog boxes. Adds these values for the solder guard when plotting.

This last selection can be particularly useful if you wish to remove a particular pad from the soldermask plot. To do this, edit the pad's pad stack and enter a large negative value, that falls within the allowed range, in the **Solder Mask Guard** entry box. By doing this, you don't merely zero out the clearance, but you actually zero out the whole pad.

Layer Plane Outline Width Use to specify how close the copper will come to the edge of the board. The copper will be recessed from the edge by a width equal to one-half the entered value. For example, if you enter 0.10 in the entry box, the copper will begin 0.05 from the edge of the board.

Pen Use to specify which pen will draw the layer plane outline. The allowed range is 0 to 32.

Note that

△ *NOTE: Pen number 0 has a special meaning on certain plotters. Unless you intend to invoke that particular meaning on your particular plotter, use pen number 1 as your first pen number.*

Alignment Target command

Appears on the **PLACE** menu.

Loads the current alignment target settings.

See also *Placing alignment targets.*

All command

Appears on the **Delete Block** menu.

Deletes all of the objects within or intersected by the block boundary.

All Filtered Nets button

Appears on the **Edit Net Properties** dialog box.

For each **Apply To All Filtered Nets** or **Apply** check box that is enabled, **All Filtered Nets** applies the corresponding property as marked (selected, entered, or enabled/disabled) to all nets.

All Off button

Appears on the **Set Block Parameters** dialog box.

Disables all the **OBJECTS AFFECTED** check boxes.

All On button

Appears on the **Set Block Parameters** dialog box.

Enables all the **OBJECTS AFFECTED** check boxes.

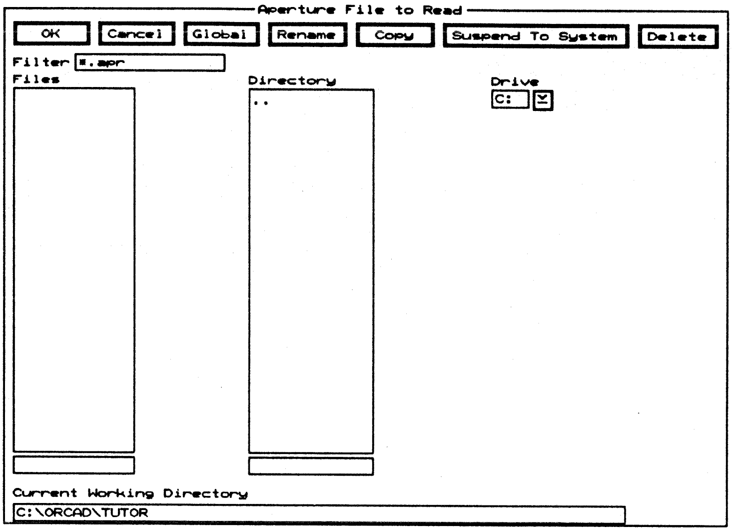
Append button

Appears on a number of dialog boxes.

Adds an item to the bottom of a list box.

Aperture File to Read dialog box

Use this dialog box to specify the input aperture filename associated with your Gerber 274-D plot.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to enter a new filename.

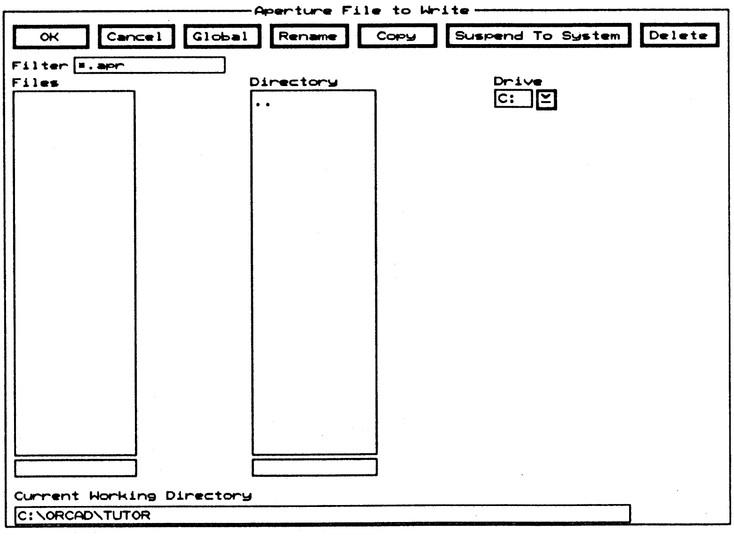
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Aperture File to Write dialog box

Use this dialog box to specify the output aperture filename associated with your Gerber 274-D plot.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to enter a new filename.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Area Autoroute command

Appears on the **Block End** menu that displays when you define the lower right corner of the autoroute block boundary.

Automatically routes the block; however, if an autoroute zone is defined which does not encompass the entire block, the autorouter routes only the autoroute zone.

Note that you can interrupt the autorouter at any time by pressing <Esc> or <Ctrl-C>.

Assigning netnames to plane layers

1. Display the **Copper Colors/Enables/ ...** dialog box.
2. Enable the **Layer Enabled** and **Layer is Plane** check boxes for the desired layer. A drop list box button displays next to the layer name's entry box.
3. Open the drop list box.
4. Select the desired netname. The netname appears in the layer name entry box.
5. Select **OK** to close the **Copper Colors/Enables/ ...** dialog box.

Assigning nets to fill zones

1. Place the pointer on the fill zone outline and select **EDIT**. The **Edit Zone** dialog box displays.
2. Select **Zone Properties**. The **Edit Zone Properties** dialog box displays.
3. Select a net from the **Netnames** list box, and then select **OK**.
4. Select **OK** to close the **Edit Zone** dialog box.

Assigning nets to pads

There are two methods of assigning nets to pads. To assign a specific net to a pad, perform the following:

1. Make sure that **Allow Move/Edit/Delete of Module Elements** is enabled in the **Global Options** dialog box.
2. Place the pointer on the desired pad and select **EDIT**. The **Edit Pad** dialog box displays.
3. Select an existing netname in the **Netnames** list box, or enter a new netname in the entry box directly beneath the list box.
4. Select **OK**. The netname in the entry box is assigned to the pad and, if necessary, added to the list of netnames.

You can also assign a net to a pad while manually routing. Routing to pads with no nets gives the pad the netname of the route.

1. Place the pointer in the center of a pad that has a net and select **ROUTE Begin**.
2. Move the pointer to the desired pad and select **End**.
3. The message "NET ASSIGNED TO <No Net> PAD" displays at the bottom of the screen indicating that a net has been assigned to the pad. Note that in this second method **Allow Move/Edit/Delete of Module Elements** does not have to be enabled.

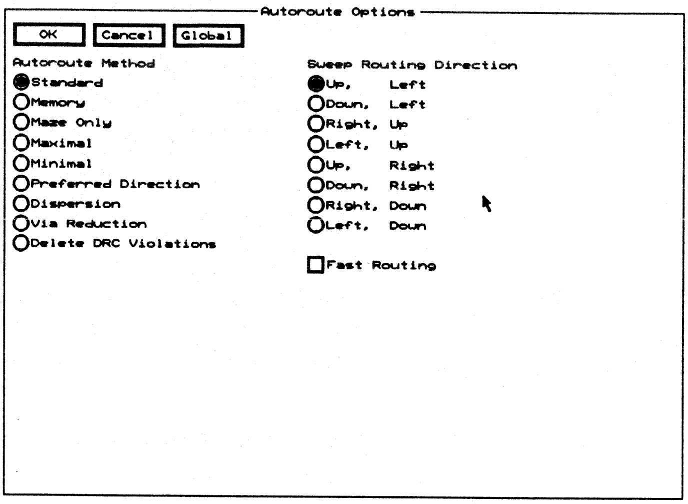
Attach command

Appears on the **ROUTE** menu.

Place your pointer on a wire or a ratsnest, select **ROUTE Attach**, and the wire or ratsnest becomes a rubberbanded trace.

Autoroute Options dialog box

Use this dialog box to set the autoroute methods.



Fast Routing Enable to run the autorouter in fast routing mode. This turns off full shoving and allows shoves to occur for vias only.

Note that autorouting is much faster when **Fast Routing** is enabled, but the autorouter may complete fewer connections and use more vias and greater wire length.

Autoroute Method

Standard Autoroutes in two passes: first by the memory method, then by the maze method.

Memory Autoroutes from pad to pad in the preferred direction, if there is nothing between the pads or if there is only one pad (on another net) between the pads.

Maze only Use to autoroute all connections.

Maximal Use to autoroute the remaining connections not completed by other methods. This method has more freedom to insert vias and find meandering paths, but takes longer.

Minimal Use to determine if the board is essentially routable. Automatically enables **Fast Routing**.

Preferred Direction Only performs maze routing in the preferred direction. If necessary, it allows routing up to two grid spaces in the non-preferred direction.

Dispersion Enables any surface mounted pad to get to internal layers by attaching a stub and via to the pad. Primarily used to connect to power and ground planes.

Via Reduction Reroutes connections with vias in an attempt to decrease the number of vias. Best used after routing is complete.

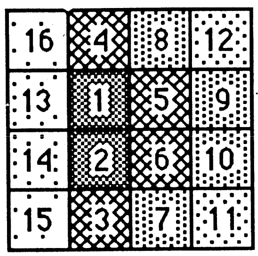
Delete DRC Violations Goes through the board and deletes one (chosen arbitrarily) of the two objects that together create a spacing violation.

Sweep Routing Direction

Use to specify the primary and secondary directions for the sweep routing window. The default selection is **Up, Left.** This causes the sweep window to move up from the starting location to the top of the board, then down from the starting location to the bottom of the board; then shift left and repeat the process (starting location to top, starting location to bottom), shifting left until it reaches the left edge of the board; then shift to the right of the starting location and repeat the process (starting location to top, starting location to bottom), shifting right until it reaches the right edge.

To take advantage of this feature, draw the sweep window boundary around the densest area on the board so it is routed first, and select the routing direction that moves the sweep window to the next densest area.

In the example at right, density is represented by the darkness of the shading. Draw the starting sweep window around the densest area, above and to the left of center, so this area will be routed first. Select **Down, Right** so the sweep window will move in the direction indicated by the numbers.



Note that, for whole board sweeps, the sweep windows overlap by 25%.

Autoroute Whole Board command

Appears on the **Whole Board** and **Sweep Window End** menus.

Automatically routes the whole board; however, if an autoroute zone is defined which does not encompass the entire board, the autorouter routes only the autoroute zone.

Note that you can interrupt the autorouter at any time by pressing <Esc> or <Ctrl><C>.

Autoroute Zone command

Appears on the board editor **PLACE** menu.

Loads the current zone properties.

See also *Autoroute zones* and *Placing zones*.

Autoroute zones

An *autoroute zone* defines the only autoroutable portion of a layer, and every layer must have at least one. If you do not specify an autoroute zone, the autorouter automatically creates one for you. It consists of a rectangular area large enough to include every object on every copper layer.

This may not be the best choice, though, if the board is not rectangular or if copper layers have objects that lie far outside what should be the routable area. For example, routes may be created outside the outline of a non-rectangular board.

You can create *no-autoroute zones* inside the autoroute zone to make parts of that area unroutable, but the reverse is not true. An autoroute zone inside a no-autoroute zone is still the only routable portion of the layer, but the no-autoroute zone surrounding it makes it unreachable—in other words, the layer has no routable area.

Similarly, each layer can have only one autoroute zone. If you define more than one autoroute zone on a given layer, the results are unpredictable.

Autorouter command

Appears on the board editor **GO TO FUNCTION** menu.

Begins the process of running the autorouter.

During autorouting, up to three numbers may display near the bottom of the screen:

- **Completed** is the number of successful routes in all windows processed.
- **Failed** is the number of failed routes in all windows processed.
- **Remaining** is the number of point-to-point connections in the current window left to be evaluated.

△ *NOTE: As the autorouter works, it stores information in a file called __WORK__._A_, which is normally deleted when you quit* ***Edit Layout****. In the unlikely event of a power failure or other abnormal exit, this file will remain on your hard disk. It will be deleted, however, when you next quit the editor.*

Autorouter Error dialog boxes

During autorouting, the following error messages may display. Autorouting cannot proceed until the problem is corrected. Select **OK** to dismiss the dialog box.

The following autorouter errors are defined.

No layers enabled for autorouting

You need to define which layers are to be autorouted. Enable some layers for autorouting.

Not enough memory

There is not enough available RAM or virtual space to load the database into the autorouter.

Number of net copper tools > limit of 500

There are more than 500 copper tools defined. Reduce the number of copper tools.

Routing area too large for available memory

There is not enough RAM or virtual space for the autorouter to route a window this large and on this many layers.

Via stack *n* is not square or round

Only square or round vias are allowed for autorouting. Change the shape of the via.

Note that via arrays are allowed, but they must be symmetrical and the connection point must be in the center of the array.

Error Code *n*

An unexpected error has occurred. Write down the error code, and call OrCAD technical support.

▲ *CAUTION: If this message displays, it is best to save the design and quit **Edit Layout** before attempting to either autoroute the board or run a DRC check again.*

Begin button

Appears on the **Printing and Plotting** dialog box.

Select to print or plot all of the items in the **Page Contents** list box.

Begin command

Appears on a number of menus.

Sets the starting point of the current action, such as placing an object.

On the ROUTE menu

Use **Begin** to draw net segments on the current layer, which must be a copper layer. The net segment must begin on a pad, via, or another net segment.

Begin All button

Appears on the **Printing and Plotting** dialog box.

Select to print or plot all the pages in the **Pages** list box.

BLOCK command

Appears on a number of menus.

Uses the current pointer position as the upper-left corner of a block. As you move the pointer, **Edit Layout** displays the block boundary.

Use a block boundary to delete, move, and drag objects, show ratsnests for pads, snap the first pad of each module to the grid, and define the area to be autorouted.

△ *NOTE: To move a module with a block, it must be completely enclosed in the block boundary.*

Block End command

Appears on the **BLOCK** menu.

Completes the block boundary and displays the menu shown at right.

```
Delete Block
Move Block
Drag Block
RatsNest Block
Module Snap Block
```

Board Editor command

Appears on the library editor **GO TO FUNCTION** menu.

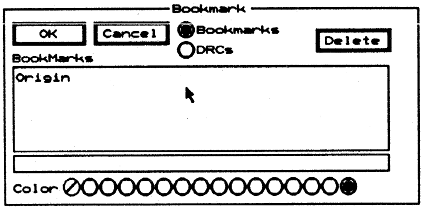
Displays the board editor. In the board editor, press <Enter> to display the board editor main menu shown at right.

Each of the commands shown is described in this chapter.

```
Block
Cut
Delete
Edit
Find
Go To Function
Highlight
Inquire
Jump
Track Delete
Layer
Move
Drag
Origin
Place
Quit
Route
Set
Selective
Undelete
Verbose Inquire
Window Zoom
X Show RatsNest
Zoom
= bookmark
+ layer
- layer
* layer
/ other
? conditions
% macro
```

Bookmark dialog box

Use this dialog box to select, create, edit, and delete bookmarks and to select DRCs.



See also *Creating bookmarks* and *Deleting bookmarks.*

Bookmarks Displays existing bookmarks in the list box.

DRCs Displays existing DRCs in the list box.

Delete Removes the selected bookmark or DRC from the list box.

△ *NOTE: You cannot delete the **Origin** bookmark. To reset it to the upper-left corner of the work space, move the pointer to that location and select **ORIGIN**.*

The list box contains a list of bookmarks or DRCs, as specified by the selected radio buttons.

Color Use to select one of sixteen colors for a bookmark.

△ *NOTE: Use **Show Bookmarks** in the **Global Options** dialog box to show and hide bookmarks.*

Browse button

Appears on a number of dialog boxes.

On the Load Setup from File dialog box

Displays the **Load Print/Plot Setup from File** dialog box.

On the Output Configuration dialog box

Displays the **Output Filename** dialog box.

On the Driver Configuration dialog box

The **Browse** buttons on the **Driver Configuration** dialog box only display when **Gerber (274-D)** is selected.

The **Read Aperture Browse** button displays the **Aperture File to Read** dialog box.

The **Write Aperture Browse** button displays the **Aperture File to Write** dialog box.

Build Name button

Appears on the **Edit Pad Stack** and **Edit Via Stack** dialog boxes.

Creates a name from the characteristics of pad stack element 1 (shown at the top of the **Elements** list box), and loads the name into the entry box directly beneath the **Pad Stack** or **Via Stack** list box.

Cancel button

Appears on a number of dialog boxes.

Closes the current dialog box without incorporating any changes.

Center command

Appears on the **ZOOM** menu.

Retains the current zoom scale, but shifts the view so the current pointer location becomes the center of the screen.

Changing copper tool names

1. Display the **Edit Copper Tool** dialog box.
2. Select the copper tool to be renamed in the **Copper Tool** list box. The copper tool name is loaded into the entry box directly beneath the **Copper Tool** list box.
3. Change the name of the copper tool by using one of these methods:
   - ❖ Edit the name in the entry box.
   - ❖ Select **Build Name**. A name formed from the value in the **Width** entry box is loaded into the entry box.
4. Select **Add**. The name in the entry box is added to the **Copper Tool** list box.

   In the **Copper Tool** list box there are now two copper tools with the same widths. To delete the original copper tool, follow the steps in *Deleting a copper tool*.

Changing filenames

To rename a file from within **Edit Layout**, follow these steps:

1. Select **QUIT Write Board File, QUIT Write Library File, QUIT Initialize Board File,** or **QUIT Initialize To Library**. The corresponding dialog box displays.
2. Select the file to rename from the **Files** list box or enter the filename in the entry box.
3. Select **Rename**. The **Rename File** dialog box displays.
4. Enter the new filename in the **Rename To** entry box, then select **OK**. The file is renamed in the current working directory.

Changing module names

Each module on a board or in a library must have a unique name.

In the board editor

1. Position the pointer on the desired module name and select **EDIT**. The **Edit Module Properties** dialog box displays.
2. Enter the new name in the **Name** entry box and select **OK**. The new name appears on the module.

   Note that if you enter a module name that is being used by another module on the board, the message "Module Name is currently in use by another Module" displays at the top of the screen and **OK** is disabled. Enter a different module name to clear the message.

△ *NOTE: The new module name exists in memory only. The module is not renamed in the board file until you select* ***QUIT Update Board File****.*

In the library editor

1. Display the **Initialize to Library File** dialog box.
2. Select the library that contains the module to be renamed from the **Files** list box, and then select **OK**. The **Get Module** dialog box displays.
3. Select the module to be copied from the **Module Name** list box, and then select **Rename**. The **Rename Module** dialog box displays.
4. Enter a new name for the module in the **Rename To** entry box, and then select **OK**. The new name replaces the old name in the **Module Name** list box.

△ *NOTE: The new module name exists in memory only. The module is not renamed in the library file until you select QUIT Update Library File.*

Changing pad stack names

1. Display the **Edit Pad Stack** dialog box.
2. Select the pad stack to be renamed in the **Pad Stack** list box. The pad stack name is loaded into the entry box directly beneath the **Pad Stack** list box.
3. Change the name of the pad stack by using one of these methods:
   - ❖ Edit the name in the entry box.
   - ❖ Select **Build Name**. A name formed from the characteristics of pad stack element 1 is loaded into the entry box.
4. Select **Add**. The name in the entry box is added to the **Pad Stack** list box.

   In the **Pad Stack** list box there are now two pad stacks with the same pad stack characteristics. To delete the original pad stack, follow the steps in *Deleting a pad stack.*

Changing via stack names

1. Display the **Edit Via Stack** dialog box.
2. Select the via stack to be renamed in the **Via Stack** list box. The via stack name is loaded into the entry box directly beneath the **Via Stack** list box.
3. Change the name of the via stack by using one of these methods:
   - ❖ Edit the name in the entry box.
   - ❖ Select **Build Name**. A name formed from the characteristics of pad stack element 1 is loaded into the entry box.
4. Select **Add**. The name in the entry box is added to the **Via Stack** list box.

   In the **Via Stack** list box there are now two via stacks with the same pad stack characteristics. To delete the original via stack, follow the steps in *Deleting a via stack.*

Changing the order of pad stack elements

1. Display the **Edit Pad Stack** dialog box.
2. In the **Elements** list box, select the pad stack element you want to move.
3. Select **Delete** directly above the **Elements** list box. Note that the pad stack element's pad parameters remain in the entry boxes.
4. Select **Insert** or **Append** to recreate the deleted pad stack element and add it to the list. **Insert** adds it above the currently highlighted pad stack element. **Append** adds it at the bottom of the list.

Changing the order of via stack elements

1. Display the **Edit Via Stack** dialog box.
2. In the **Elements** list box, select the via stack element you want to move.
3. Select **Delete** directly above the **Elements** list box. Note that the via stack element's via parameters remain in the entry boxes.
4. Select **Insert** or **Append** to recreate the deleted via stack element and add it to the list. **Insert** adds it above the currently highlighted via stack element. **Append** adds it at the bottom of the list.

Circle command

Appears on the **PLACE** menu.

Loads the current circle settings.

See also *Placing circles*.

Cleanup Stubs command

Appears on the board editor **QUIT** menu.

A *stub* is either a net segment or a chain of segments, arcs, and vias that has only one end attached to a test point, a pad, or another segment.

Selecting **Cleanup Stubs** places all stubs and unconnected wires in the selective undelete buffer. Select **SELECTIVE** to view the stubs and determine which should be permanently deleted.

△ *NOTE: **Cleanup Stubs** does not remove stubs on nets that have **Do Not Route Net** enabled. See **Edit Net Properties dialog box**.*

Clear Aperture List Now button

Appears on the **Driver Configuration** dialog box when either **Gerber (274-X)** or **Fire 9xxx** is selected in the **Vector Device** droplist box.

Clears the current aperture list from memory.

If you do not load an aperture list from a file after selecting **Clear, Edit Layout** builds an aperture list from the copper tools, pad stacks, and via stacks defined in the design.

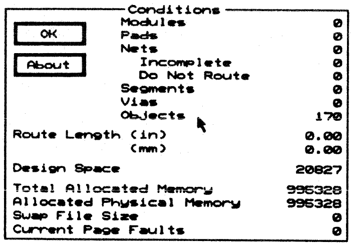
Close button

Appears on a number of dialog boxes.

If changes have been recorded in the database or system and the changes cannot be undone, the **Cancel** button becomes a **Close** button.

Conditions dialog box

Use this dialog box to view information about the status of the board.



About Displays the **About** dialog box.

Modules The number of modules placed on the board.

Pads The number of pads placed on the board.

Nets The number of existing nets.

Incomplete The number of nets that are not completely routed on the board.

Do Not Route The number of nets marked as excluded from autorouting, unconnected display, and stub cleanup.

Segments The number of net, outline, and zone segments placed on the board.

Vias The number of vias placed on the board.

Objects The number of objects placed on the board.

Route Length The combined length, in inches and millimeters, of every routed net on the board.

Design Space The amount of memory, in bytes, used by the board file.

Total Allocated Memory The amount of extended and virtual memory, in bytes, used by the program.

Allocated Physical Memory The amount of extended memory, in bytes, used by the program.

Swap File Size The current size, in bytes, of the swap file.

Current Page Faults The number of page faults which have occurred since the last selection of **? Conditions**. A few page faults are fine, but if the number is large and increases each time you select **? Conditions**, either allocate more physical memory or reduce the magnitude of the task.

Configuring pages

You produce a print or plot in **Edit Layout** by selecting layers and objects, then assigning those layers and objects to a page. You can specify how objects print on each page, and you can print selected pages, or you can print all pages.

For related information, see *Printing and Plotting dialog box, Save Print/Plot Setup to File dialog box,* and *Load Print/Plot Setup from File dialog box.*

1. In the board editor, select **GO TO FUNCTION Printing and Plotting**. The **Printing and Plotting** dialog box displays.
2. Select a layer from the **Layer** droplist box.
3. Select the objects you want to include on the printed page, and then select **Insert** or **Append** above the **Page Contents** list box.
4. Enter a page name in the entry box below the **Pages** list box.
5. Select **Insert** or **Append** above the **Pages** list box to associate the items listed in the **Page Contents** list box with the page name.

Configuring template files

From the **PC Board Layout Tools** screen, select **Edit Layout**, and then select **Configure PC Board Tools**. Enter the name of the template file you want to use in the **Template** entry box in the **Miscellaneous Options** area. You can specify a full path as well as a filename. This is called an *absolute* filename, and it tells **Edit Layout** exactly where to find the file.

If you specify a *relative* filename—just the filename or a partial path and filename—**Edit Layout** evaluates the relative pathname from one of two places specified on the **Configure PC Board Tools** screen:

- For a board template, relative to the directory named in the **Board file prefix** entry box in the **Prefix Options** area.
- For a library template, relative to the directory named in the **Library Prefix** entry box in the **Library Options** area.

Continue button

Appears on the **Netlist Load Error** dialog box.

Tells **Edit Layout** to continue parsing the netlist until another error is encountered.

Continue, Do Not Pause on Errors button

Appears on the **Netlist Load Error** dialog box.

Tells **Edit Layout** to continue parsing the netlist, record any errors it encounters, and display the number of errors found after the netlist is parsed.

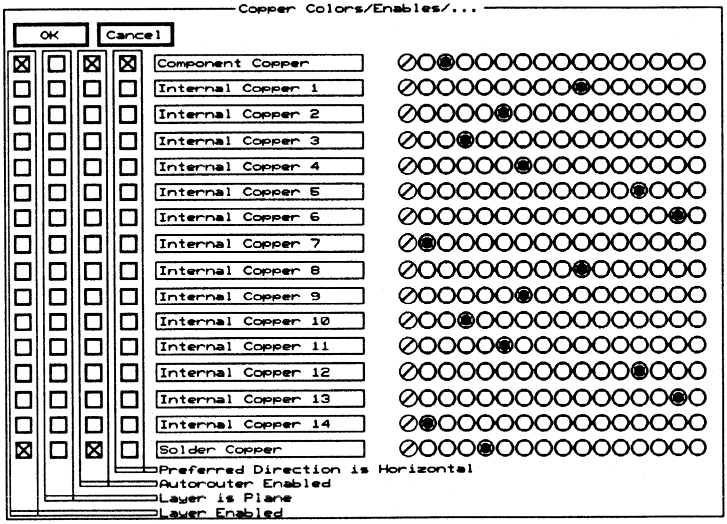
Copper Colors/Enables/... button

Appears on the **Layer** dialog box.

Displays the **Copper Colors/Enables/...** dialog box.

Copper Colors/Enables/... dialog box

Use this dialog box to set the colors and other attributes of the 16 copper layers.



The layers are listed in the order in which they appear on a 16-layer board. **Component Copper** is the top of the board, where the components are mounted. Internal layers 1 through 14 are used for internal routing and for power and ground planes. **Solder Copper** is the bottom of the board, which is routable and can also have components.

Check boxes

Four columns of check boxes, labeled near the bottom of the dialog box, are associated with each copper layer.

Layer Enabled Enables or disables the copper layer for manual routing and editing. Any segments, arcs, or text on a disabled layer will appear gray. Note that you must enable a copper layer before you can select **ROUTE Begin** on that layer.

Layer is Plane Designates the copper layer as a plane layer for connectivity. When this check box is enabled, a drop list box button displays next to the layer name's entry box. Select this button to display a list of netnames.

See *Assigning netnames to plane layers.*

Autorouter Enabled Enables or disables the copper layer for autorouting.

Preferred Direction is Horizontal Causes the autorouter to be biased in the horizontal direction (displayed from left to right in **Edit Layout**).

Entry boxes

Use the entry boxes to change the name of a copper layer, but you should choose names that reflect the order shown in the dialog box. For example, internal layer 1 is nearest the top (component) layer, and layer 14 is nearest the bottom (solder) layer.

Color radio buttons

Use the color radio buttons to change the copper layer colors. Copper layer colors help you distinguish one copper layer from another when viewing the board. There are 16 colors available.

△ *NOTE: Setting a copper layer color to black (the leftmost color radio button) prevents edits on that layer.*

Copper Tool Editor button

Appears on a number of dialog boxes.

Displays the **Edit Copper Tool** dialog box.

Copper Tool Editor command

Appears on the **GO TO FUNCTION** menu.

Displays the **Edit Copper Tool** dialog box.

Copy button

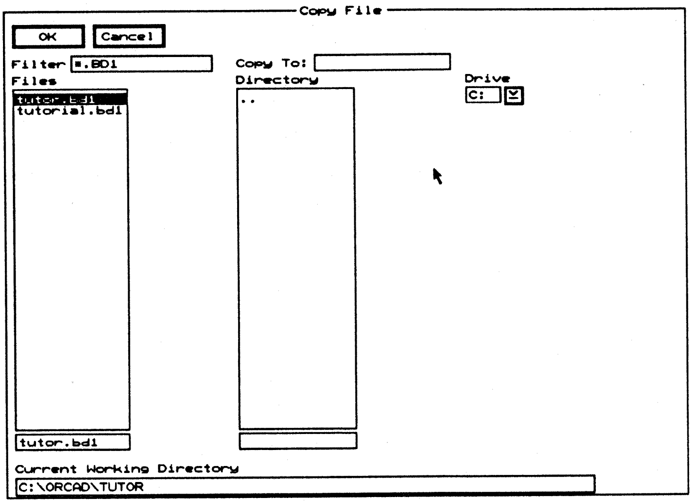
Appears on a number of dialog boxes.

Copies an item in a list box.

Copy File dialog box

Use this dialog box to duplicate the contents of an existing file.

See also *Copying files*.



Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Copy to Use to enter the new filename.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

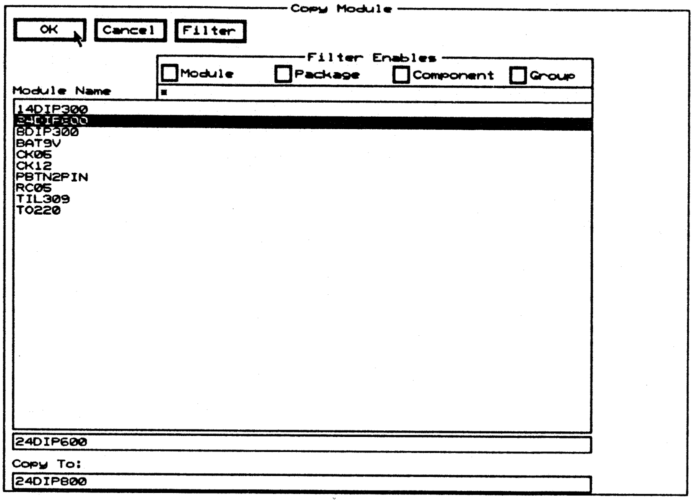
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Copy Module dialog box

Use this dialog box to duplicate or rename a module.



Filter Displays the **Edit Filter** dialog box.

Filter Enables Use this area to restrict the modules shown in the **Module Name** list box to those which match the filter shown in the corresponding droplist box on the **Edit Filter** dialog box.

Module Name Contains a list of modules.

Use any combination of wildcards (* or ?) and other characters in the entry box directly above the **Module Name** list box to further restrict the list of modules shown.

Copy To Use this entry box to duplicate and rename the module selected in the **Module Name** list box.

Copying files

To make a copy of any file within the same directory as the original, follow these steps:

1. Display the **Write Board File, Write Library File, Initialize to Board File,** or **Initialize to Library File** dialog box.
2. Select the file to be copied from the **Files** list box, and then select **Copy**. The **Copy File** dialog box displays.
3. Enter a name for the new copy in the **Copy To** entry box. You can also select a different directory or drive from the corresponding list box.
4. Select **OK**. The new file is created in the current working directory, and the dialog box in which you selected **Copy** displays again.

Copying modules

It is a good idea to make a copy of a module before you edit it, and then edit the copy. Copying is also a fast way to create modules that are very similar.

See also *Copying modules to another library.*

To make a copy of a module within the same library as the original, follow these steps:

1. Display the **Initialize to Library File** dialog box.
2. Select the library that contains the module to be copied from the **Files** list box, and then select **OK**. The **Get Module** dialog box displays.
3. Select the module to be copied from the **Module Name** list box. The module name highlights, and then select **Copy**. The **Copy Module** dialog box displays.
4. Enter a name for the new copy in the **Copy To** entry box, and then select **OK**. The name displays in the **Module Name** list box.

△ *NOTE: The new copy of the module exists in memory only. The copy is not stored on the disk until you select QUIT Update Library File.*

Copying modules to another library

1. If necessary, load the library file containing the module you want to copy, and select the module.
2. Export the module to a file, such as MODULE.EXP.
3. Load the library file you want to contain the module copy.
4. Specify a name for the new module.
5. Import the file MODULE.EXP.
6. Save the library with the new module.

See *Export* and *Import* for information on these processes.

Use the **Copy Module** dialog box to duplicate a module within the same library.

Creating a board module

Once you have created a mechanical object or simple electrical object in one layout, you can save the object as a module and reuse it in other layouts. For example, you might create a library of outlines with standard connectors, alignment targets, mounting holes, dimensions, and text. You might also make a module of a power supply with only GND and VCC connections.

Follow these steps to prepare the layout:

1. Make a temporary copy of the board file containing the objects you want to save as a module.
2. In **Edit Layout**, load the temporary board file.
3. Edit the layout as needed, deleting anything you don't want included in the new module.
4. Select **QUIT Update Board File** to save any changes to the layout.

Follow these steps to import the board file:

1. Select **GO TO FUNCTION Library Editor**. The **Initialize to Library File** dialog box displays.
2. Select an existing module library from the **Files** list box, or enter a new library name in the entry box below it.
3. Select **OK**. The **Get Module** dialog box displays.
4. Enter a new module name in the entry box below the **Module Name** list box.
5. Select **Import**. The **Import Module from File** dialog box displays.
6. Enter `*.BD1` in the **Filter** entry box.
7. Select the desired board file from the **Files** list box, or enter a filename in the entry box below it. You can also select a different directory or drive from the corresponding list box.
8. Select **OK** to load the selected board file. The **Pad Name Disposition** dialog box displays.
9. Make sure **Keep all Pad Names** is selected, and select **OK**. The message "Import Module" displays near the lower right corner of the screen, then the **Get Module** dialog box displays.
10. Select **OK** to accept the selected module. The new module, with place holders for reference, value, and module text, displays in the library editor.

△ *NOTE: Netnames are not retained in board modules created by this process. If your layout contains any zones or layer planes, you must re-establish electrical connections for every board module you place. To do so, you must delete and replace any vias; you must also edit any pads and reassign netnames before you can include them in a net outside the board module.*

Creating boards

The tutorial in the *PC Board Layout Tools 386+ User's Guide* is the best introduction to the process of taking a design from netlist to photoplot with **PCB 386+ 2.00**. These steps describe the general sequence:

1. Configure **PCB 386+ 2.00**, as described in *Chapter 1: Configure Layout Tools* in this manual.
2. Configure **Edit Layout**, as described in the section *Local configuration*, near the beginning of this chapter.
3. Run **Edit Layout**, as described in the section *Execution*, at the beginning of this chapter.
4. If you want to change the settings for many aspects of the working environment, select **SET** to display the **Global Options** dialog box. See *Global Options dialog box* in this chapter for a complete description.
5. If you want to change the current layer or define other layer characteristics, select **LAYER** to display the **Layer** dialog box. See *Layer dialog box* in this chapter for a complete description.
6. In the board editor, create the design by drawing the board's outline, loading netlists, placing modules, and routing.

See also *Template files* and *Configuring template* files.

Creating bookmarks

1. Point to where you want the bookmark.
2. Select = **BOOKMARK**. The **Bookmark** dialog box displays.
3. If necessary, select **Bookmarks** to display existing bookmarks in the list box.
4. Enter the name of the new bookmark in the entry box directly beneath the list box.
5. Select **OK. Edit Layout** creates the bookmark and places it on the board at the pointer's location.

△ *NOTE: Use* ***Show Bookmarks*** *in the* ***Global Options*** *dialog box to show and hide bookmarks.*

Creating copper tools

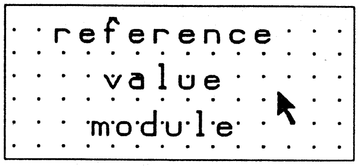
1. Display the **Edit Copper Tool** dialog box.
2. In the **Width** entry box, enter the width of the copper tool.
3. Select **Build Name**. A name is created from the value in the **Width** entry box and loaded into the entry box directly below the **Copper Tool** list box.
4. If desired, edit the name in the entry box.
5. Select **Add**. The name in the entry box is added to the **Copper Tool** list box.

Creating drill diameters

1. Display the **Edit Drill List** dialog box.
2. Edit the drill diameter shown in the entry box beneath the **Drill Diameter** list box. Drill diameters have a range of 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm).
3. Select **Add**. The drill diameter is added to the list in the **Drill Diameter** list box.

Creating modules

1. Display the **Initialize to Library File** dialog box.
2. Select a library from the **Files** list box or enter a name in the entry box below it. If you enter the name of a file that doesn't exist, **Edit Library** loads the template and uses the filename later when you save the library.
3. Select **OK**. The **Get Module** dialog box displays.
4. Enter a name for the new module in the entry box below the **Module Name** list box, and then select **OK**. The library editor displays three text strings, as shown at right.

   These text strings are placeholders for values that are assigned to each module when it is loaded from a netlist and placed in **Edit Layout**.

   The "reference" place holder receives the reference designator that is assigned to the schematic symbol for the module in **Draft**. The "value" place holder receives the schematic part value, such as "10K" for a resistor. The "module" place holder receives the module name you specified in the **Get Module** dialog box.
5. Select **LAYER**. The **Layer** dialog box displays.
6. Select **SilkScreen Component** from the **Current Layer** list, and then select **OK**. The library editor displays. Note that "SilkScreen Component" displays at the bottom of the screen, indicating the current working layer.
7. If you want non copper graphic objects (such as the module outline and special reference symbols) silkscreened to the component copper layer during fabrication, place them on the silkscreen component layer.

8. Draw the module's outline, add mounting holes, lay out pads, and arrange the placeholders as desired.
9. Select **QUIT Update Library File** to save the module in the specified library file.
10. Select **QUIT Leave Library Editor** or **GO TO FUNCTION Board Editor** to return to **Edit Layout.**

Creating pad stack elements

1. Display the **Edit Pad Stack** dialog box.
2. In the **Pad Stack** list box, select the pad stack you want the pad stack element to be associated with.
3. Using the entry boxes below the **Elements** list box, enter the pad parameters.
4. Select **Insert** or **Append. Insert** adds the new pad stack element above the highlighted pad stack element in the **Pad Stack** list box. **Append** adds the new pad stack element to the bottom of the list in the **Elements** list box.
5. Select **Add.** The new pad stack element becomes part of the highlighted pad stack.

Creating pad stacks

1. Display the **Edit Pad Stack** dialog box.
2. Create a pad stack element by following the steps outlined in the section *Creating pad stack elements.*
3. Select **Build Name.** A name formed from the characteristics of pad stack element 1 is loaded into the entry box directly below the **Pad Stack** list box.
4. Select **Add.** The contents of the entry box are added to the **Pad Stack** list box.

Creating template files

Use **Make Board Template** on the **PC Board Layout Tools** screen to create a template file from a **PCB 386+ 2.00** board file. See *Chapter 11: Make Board Template* for more information.

Creating via stack elements

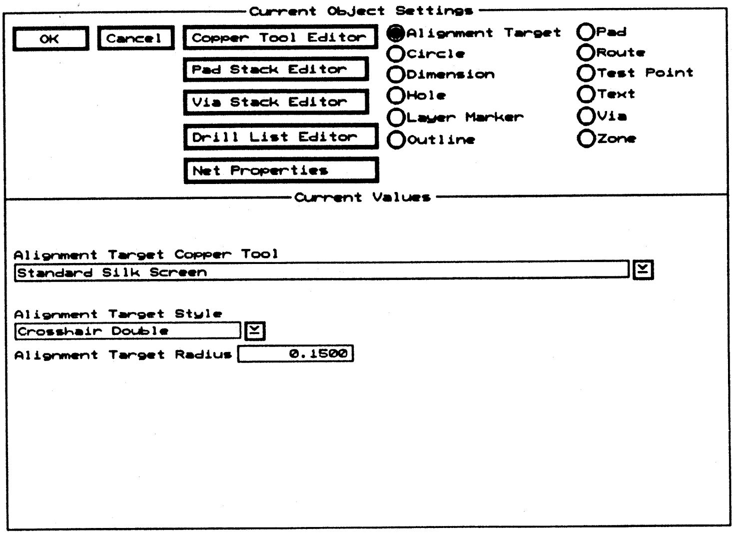
1. Display the **Edit Via Stack** dialog box.
2. In the **Via Stack** list box, select the via stack you want the via stack element to be associated with.
3. Using the entry boxes below the **Elements** list box, enter the via parameters.
4. Select **Insert** or **Append. Insert** adds the new via stack element above the highlighted via stack element in the **Via Stack** list box. **Append** adds the new via stack element to the bottom of the list in the **Elements** list box.
5. Select **Add.** The new via stack element becomes part of the highlighted via stack.

Creating via stacks

1. Display the **Edit Via Stack** dialog box.
2. Create a via stack element by following the steps outlined in the section *Creating via stack elements.*
3. Select **Build Name.** A name formed from the characteristics of via stack element 1 is loaded into the entry box directly below the **Via Stack** list box.
4. Select **Add.** The contents of the entry box are added to the **Via Stack** list box.

Current Object Settings dialog box

Use this dialog box to establish the default settings for all new objects placed on the board.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Pad Stack Editor Displays the **Edit Pad Stack** dialog box.

Via Stack Editor Displays the **Edit Via Stack** dialog box.

Drill List Editor Displays the **Edit Drill List** dialog box.

Net Properties Displays the **Edit Net Properties** dialog box.

Radio buttons

Use the following radio buttons to establish the default settings for the objects shown below. The **Current Values** area changes with each object, showing what default settings are available.

Alignment Target Establishes the copper tool, style, and radius.

Circle Sets the copper tool.

Dimension Establishes the copper tool, text (including format), character and end bar height, dimension text placement, use of metric notation, use of tick marks instead of arrows, and placement of arrows outside of end bars.

Hole Sets the drill diameter.

Layer Marker Establishes the copper tool, angle, and character height.

Outline Sets the copper tool.

Pad Sets the pad stack.

Route Sets the copper tool.

Test Point Sets the test point.

Text Establishes the copper tool, angle, and character height.

Via Sets the via stack.

Zone Establishes the fill copper tool, boundary copper tool, stripe width copper tool, and tool spacing.

Current Settings button

Appears on the **Global Options** dialog box.

Displays the **Current Object Settings** dialog box.

CUT command

Appears on a number of menus.

Cuts a net, or outline into two segments.

Places a "cut" mark on a zone, or polygon boundary which determines the portion of the boundary that can be dragged.

See also *Dragging zones.*

Place the pointer on the segment where you want to cut it, and select **CUT**. Select **Outline Tracks** in the **Global Options** dialog box to see the cut.

Note that you cannot cut arcs.

Defining zoom windows

1. Position the pointer at the location you want to be the upper-left corner of the window zoom boundary.
2. Select **WINDOW ZOOM**.
3. Move the mouse to the location you want to be the lower-right corner of the window zoom boundary, or select **Jump** and specify the location by X and Y coordinates. The window zoom boundary shows the outer edges of the new view.
4. Select **Window Zoom End.** The screen changes to show the area enclosed by the window zoom boundary.

While drawing the window zoom boundary, you can use **ORIGIN** and **= BOOKMARK** for convenience and control.

DELETE command

Appears on a number of menus.

Deletes an object.

The undelete buffer can store up to 254 individual deletions. A deletion can contain any number of objects. The top of the buffer contains your latest deletion. If the buffer is full when you delete an object, **Edit Layout** permanently discards the object at the bottom of the buffer to free space at the top of the buffer.

To delete module elements (parts of modules, as opposed to the entire module), you must enable **Allow Move/Edit/Delete of Module Elements** in the **Global Options** dialog box.

Delete button

Appears on a number of dialog boxes.

Deletes the selected item from a list box.

See also *Deleting files* and *Deleting modules*.

Delete ALL button

Appears on the **Macro Maintenance** dialog box.

Deletes every macro in the **Defined Macros** list box.

Delete Block command

Appears on the **Block End** menu.

Displays the menu shown at right. From this menu, select which objects you want deleted from the block.

```
Module
Route
Text
All
```

Select **Delete Block** to delete some or all of the objects enclosed or intersected by the block boundary and store them in the undelete buffer. Note that to delete a module, the module must be completely enclosed by the block boundary.

See also *Deleting modules*.

Delete Details button

Appears on the **Edit Net Properties** dialog box.

Deletes all objects related to a net, except the modules.

The deleted objects are stored in the undelete buffer.

Select **UNDELETE** to restore the deleted objects.

Deleting

Depending on your color configuration, portions of deleted objects may remain on the screen until you select **ZOOM Refresh** to clear the display.

Deleting bookmarks

1. Select = **BOOKMARK**. The **Bookmark** dialog box displays.
2. If necessary, select **Bookmarks** in the list box to display existing bookmarks.
3. Select the bookmark to be deleted.
4. Select **Delete**. The bookmark is deleted.

△ *NOTE: You cannot delete the* ***Origin*** *bookmark. To reset it to the upper-left corner of the work space, move the pointer to that location and select* ***ORIGIN****.*

Deleting copper tools

1. Display the **Edit Copper Tool** dialog box.
2. Select the copper tool to be deleted in the **Copper Tool** list box. The copper tool name is loaded into the entry box directly beneath the **Copper Tool** list box.

   If the selected copper tool is not being used, the copper tool is deleted from the **Copper Tool** list box. If the selected copper tool is being used, it cannot be deleted, and a **Notice** dialog box displays.
3. If necessary, select **OK** to dismiss the **Notice** dialog box. The **Edit Copper Tool** dialog box displays.

△ *NOTES: To delete a copper tool that is in use, you must disassociate it from all objects that use it. To do so, either delete all the objects or associate them with a different copper tool.*

The Standard defaults for all copper tools can be edited, but they cannot be deleted.

Deleting drill diameters

1. Display the **Edit Drill List** dialog box.
2. Select the drill diameter to be deleted in the **Drill Diameter** list box. The drill diameter is loaded into the entry box directly beneath the **Drill Diameter** list box.

   If the selected drill diameter is not being used, the drill diameter is deleted from the **Drill Diameter** list box. If the selected drill diameter is being used, it cannot be deleted, and a **Notice** dialog box displays.
3. If necessary, select **OK** to dismiss the **Notice** dialog box. The **Edit Drill List** dialog box displays.

△ *NOTE: To delete a drill diameter that is in use, you must disassociate it from all objects that use it. To do so, either delete all the objects or associate them with a different drill diameter.*

Deleting files

To delete any file from within **Edit Layout**, follow these steps:

1. Select **QUIT Write Board File** or **QUIT Initialize Board File**. The **Write Board File** or **Initialize to Board File** dialog box displays.
2. Select the file to delete from the **Files** list box or enter the filename in the entry box. You can also select a different directory or drive from the corresponding list box.
3. Select **Delete**. The file is deleted from the disk and removed from the list box.

See also *Deleting macro files*.

Deleting macro files

Like the **Write Board File** and **Initialize to Board File** dialog boxes, the **Load ALL Macros from File** and **Save ALL Macros to File** dialog boxes include a **Delete** button. If one of these dialog boxes is already open, it may be more convenient to use the following method to delete a macro file from the disk:

1. Select a filename from the **Files** list box or enter a name in the entry box below it. The **Delete** button becomes active. You can also select a different directory or drive from the corresponding list box.
2. Select **Delete**. The macro file is deleted from the disk and its filename disappears from the list box. The **Delete** button also returns to its inactive state.
3. Select **Close**. The dialog box closes and the **Macro Maintenance** dialog box displays.

See also *Deleting macros* and *Deleting files*.

Deleting macros

In **Edit Layout**, you can delete one macro or all macros from memory.

See also *Deleting macro files*.

Deleting one macro

1. Select **GO TO FUNCTION Macro Maintenance**. The **Macro Maintenance** dialog box displays. All defined macros display in the **Defined Macros** list box. The macros listed are those now stored in memory.
2. Select the macro to delete from the **Defined Macros** list box, and then select **Delete**. The macro is deleted from memory and its name is removed from the list box.

Deleting all macros

1. Select **GO TO FUNCTION Macro Maintenance**. The **Macro Maintenance** dialog box displays. All defined macros display in the **Defined Macros** list box. The macros listed are those now stored in memory.
2. Select **Delete ALL**. All macros in **Edit Layout** memory are deleted and no macro names display in the list box.

Deleting modules

In the board editor, you delete a module from the board; in the library editor, you delete a module from the library.

In the board editor

1. Select * **LAYER** to enable all layers.
2. Select **BLOCK** and enclose all of the module's elements within the block boundary.
3. Select **Block End Delete Block Module**. The module is deleted from the board.

   Note that several modules can be deleted at once by enclosing them within a single block boundary.

   Or

1. Select * **LAYER** to enable all layers.
2. Place your pointer on a module's element.
3. Select **Delete**. The entire module is deleted.

In the library editor

1. Display the **Get Module** dialog box.
2. Select the desired module from the **Module Name** list box.
3. Select **Delete**. The module is deleted from the library.

△ *NOTE: The module still exists in the library file on disk. To delete the module from the library file, select* ***QUIT Update Library File*** *after completing the steps above.*

Deleting pad stack elements

1. Display the **Edit Pad Stack** dialog box.
2. Select the pad stack element, and then select **Delete** directly above the **Elements** list box. The selected pad stack element is deleted from the **Elements** list box.

The deleted pad stack element is no longer a part of the highlighted pad stack.

Deleting pad stacks

1. Display the **Edit Pad Stack** dialog box.
2. Select the pad stack, and then select **Delete** directly above the **Pad Stack** list box.

   If the selected pad stack is not being used, it is deleted from the **Pad Stack** list box. If the pad stack is being used, it cannot be deleted, and a **Notice** dialog box displays.
3. If necessary, select **OK** to dismiss the **Notice** dialog box. The **Edit Pad Stack** dialog box displays.

To delete a pad stack that is in use, you must disassociate it from all the pads that use it. To do so, either delete all the pads, or associate them with a different pad stack, as described in the following steps:

1. Display the **Edit Pad Stack** dialog box.
2. Select the pad stack you wish to delete.
3. Select **Build Name** or enter a name in the entry box below the **Pad Stack** list box.
4. Select **Add**, and then select **OK**.
5. Place the pointer on any pad associated with the pad stack to be deleted. (Select **INQUIRE** to determine the pad stack.)
6. Select **Edit**. The **Edit Pad** dialog box displays.
7. In the **Pad Stack** list box, select the new pad stack.
8. Enable **Apply Pad Stack to All Module Pads, Apply Pad Stack to Like Module Pads,** and **Apply Pad Stack to Like Library Module Pads.**
9. Select **OK**.

Now you can delete the original pad stack, as described in the preceding set of steps.

△ *NOTE: The Standard Through Pad stack can be edited, but it cannot be deleted.*

Deleting via stacks

1. Display the **Edit Via Stack** dialog box.
2. Select the via stack to be deleted in the **Via Stack** list box and then select **Delete** directly above the **Via Stack** list box.

   If the selected via stack is not being used, it is deleted from the **Via Stack** list box. If the via stack is being used, it cannot be deleted, and a **Notice** dialog box displays.
3. If necessary, select **OK** to dismiss the **Notice** dialog box. The **Edit Via Stack** dialog box displays.

To delete a via stack that is in use, you must disassociate it from all the vias that use it. To do so, either delete all the vias or associate them with a different via stack, as described in the following steps:

1. Display the **Edit Via Stack** dialog box.
2. Select the via stack you wish to delete.
3. Select **Build Name** or enter a name in the entry box below the **Via Stack** list box.
4. Select **Add**, and then select **OK.**
5. Place the pointer on any via associated with the via stack to be deleted. (Select **INQUIRE** to determine the via stack.)
6. Select **Edit**. The **Edit Via** dialog box displays.
7. In the **Via Stack** list box, select the new via stack.
8. Enable **Apply Via Stack to All Net Vias, Apply Via Stack to Like Net Vias, Apply Via Stack to All Board Net Vias,** and **Apply Via Stack to Like Board Net Vias.**
9. Select **OK.**

Now you can delete the original via stack, as described in the preceding set of steps.

△ *NOTE: The Standard Through Via stack can be edited, but it cannot be deleted.*

Dimension command

Appears on the **PLACE** menu.

Loads the current dimension object settings.

See also *Placing dimension objects.*

Down button

Appears on the **Driver Configuration** dialog box when **HPGL, HPGL2,** or **HI29** are the selected vector devices.

Use to scroll the list of pen properties down.

DRAG command

Appears on a number of menus.

Use **DRAG** to drag objects. **DRAG** maintains connections to zone, net, and outline segments.

When **Allow Move/Edit/Delete of Module Elements** in the **Global Options** dialog box is enabled, you can drag individual module elements (for example, just pads).

When **Allow Move/Edit/Delete of Module Elements** is not enabled, you drag the entire module. Select *** LAYER** to enable all layers.

Drag Block command

Appears on the **Block End** menu.

Use **Drag Block** to drag the objects enclosed or intersected by the block boundary. **Drag Block** maintains connections to net segments and arcs outside of the block.

When **Allow Move/Edit/Delete of Module Elements** in the **Global Options** dialog box is enabled, you can drag certain module elements (for example, just pads) the same way you would drag other objects.

When **Allow Move/Edit/Delete of Module Elements** is not enabled, you can drag a module by enclosing all of the module's elements within the block boundary. Select *** LAYER** to enable all layers.

Drawing board outlines

You define the shape and size of the board by drawing an outline. It's good practice to enable **Stay On Grid** in the **Global Options** dialog box before you begin. Use **PLACE Outline** to draw the board outline.

Drill List Editor button

Appears on a number of dialog boxes.

Displays the **Edit Drill List** dialog box.

Drill List Editor command

Appears on the **GO TO FUNCTION** menu.

Displays the **Edit Drill List** dialog box.

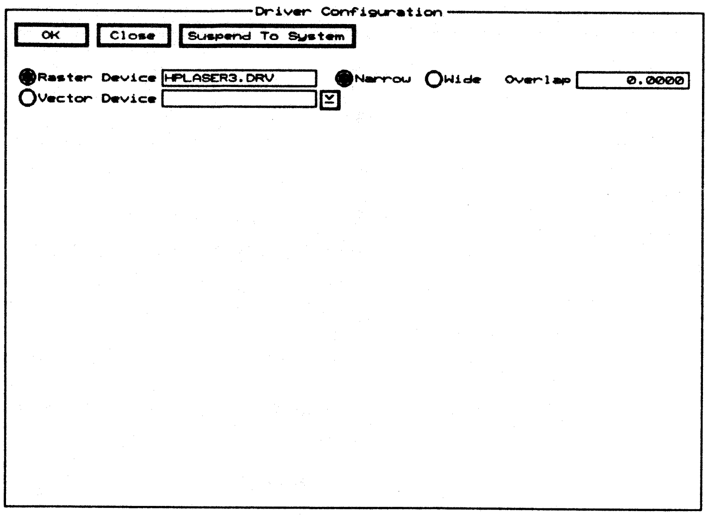
Driver button

Appears on the **Printing and Plotting** dialog box.

Displays the **Driver Configuration** dialog box.

Driver Configuration dialog box

Use this dialog box to configure the driver for printing and plotting.



Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Printers

Raster Device Use to select the printer which was previously configured using **PC Board Layout Tools**. See *Chapter 1: Configuring Layout Tools.*

Narrow (raster devices) The paper size and type used by printers varies. Typically, narrow paper is 8.5 in. (21.59 mm) wide.

Wide (raster devices) The paper size and type used by printers varies. Typically, wide paper is 13 in. (33.02 mm) wide.

Overlap (raster devices) Overlap determines how much of the image consecutive pages share. The allowed range is 0.0000 in. (0.0000 mm) to 2.0000 in. (5.0800 mm).

Plotters

Vector Device Use to select the plotter device. See also *Gerber formats.*

PostScript Select to choose PostScript as your vector device.

HPGL Select to choose HPGL as your vector device.

Up Use to scroll the pen properties up.

Down Use to scroll the pen properties down.

Pen A list of available pens.

Width Use to specify pen width. Note that if **Display Metric Dimensions** is enabled in the **Global Options** dialog box, then the width is in millimeters. Otherwise, it is in inches.

Force Use to specify pen force in plotter specific units.

Velocity Use to specify pen velocity in plotter specific units.

Acceleration Use to specify pen acceleration in plotter specific units.

Gerber (274-X) Select to choose Gerber (274-X) as your vector device.

Format Select **2.3** to have two digits to the left and three to the right of the assumed decimal point. Select **3.4** to have to have three digits to the left and four to the right of the assumed decimal point.

No Leading Zeros Select to remove leading zeros from the X and Y coordinates in the output.

△ *NOTE: If* ***No Leading Zeros*** *is enabled, leading zeros are stripped from 2.3- and 3.4-format coordinates before they are output.*

No Trailing Zeros Select to remove trailing zeros from the X and Y coordinates in the output.

Decimal Select to keep all zeros in the output.

Draw Tracks With A Square Aperture Enable to draw square tracks for multi-chip modules (silicon, large-scale integrated circuits).

Clear Aperture List On Every Begin All Enable to clear the aperture list each time **Begin All** is selected.

Clear Aperture List Now Select to clear the current aperture list from memory.

Gerber (274-X) and Fire 9xxx formats embed the aperture list within the plot file. Many gerber viewers display Gerber (274-X) and Fire 9xxx files by transparently converting them to Gerber (274-D) format as they are loaded.

If you display several plots overlaid at one time in such a viewer, it is important that the aperture list embedded in your plot files is consistent across all the plots made from your board file. **PCB 386+ 2.00** provides this consistency by keeping an aperture list on disk for every Gerber (274-X) and Fire 9xxx plot made from your board. When you switch to a new board file, the aperture list is automatically cleared. You can manually clear it by selecting **Clear Aperture List Now.** The **Clear Aperture List On Every Begin All** is a recommended option. Not using this option may increase the size of your plot file by one percent and may slightly slow the plotting of your files

Gerber (274-D)

Select to choose Gerber (274-D) as your vector device.

Format Select **2.3** to have two digits to the left and three to the right of the assumed decimal point. Select **3.4** to have to have three digits to the left and four to the right of the assumed decimal point.

No Leading Zeros Select to remove leading zeros from the X and Y coordinates in the output.

△ *NOTE: If **No Leading Zeros** is enabled, leading zeros are stripped from 2.3- and 3.4-format coordinates before they are output.*

No Trailing Zeros Select to remove trailing zeros from the X and Y coordinates in the output.

Decimal Select to keep all zeros in the output.

Draw Tracks With A Square Aperture Enable to

Read Aperture Browse Displays the **Aperture File to Read** dialog box.

Filename Use to enter the desired read aperture filename, which may include the drive and path. If you specify the drive and path of the read aperture filename, and then select its associated **Browse** button, the **Aperture File to Read** dialog box will display that drive, in that directory, and a list of filenames.

Write Aperture Browse Displays the **Aperture File to Write** dialog box.

Filename Use to enter the desired write aperture filename, which may include the drive and path. If you specify the drive and path of the read aperture filename, and then select its associated **Browse** button, the **Aperture File to Write** dialog box will display that drive, in that directory, and a list of filenames.

Output Aperture Format Use to select an output aperture format.

Fire 9xxx Select to choose Fire 9xxx as your vector device. Fire 9xxx has the same driver configuration options as Gerber (274-X).

HPGL2 Select to choose HPGL2 as your vector device. HPGL2 has the same driver configuration options as HPGL.

HI29 Select to choose HI29 as your vector device. HI29 has the same driver configuration options as HPGL.

DXF Select to choose DXF as your vector device.

DXF2D Select to choose DXF2D as your vector device.

Edit button

Appears on a number of dialog boxes.

Loads the characteristics of the selected item into the appropriate list boxes and entry boxes for editing.

EDIT command

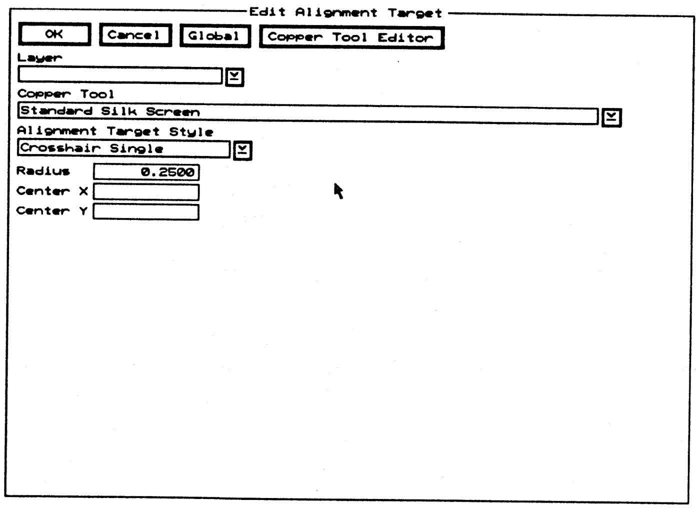
Appears on a number of menus.

Displays a context sensitive dialog box for each type of object. To edit an object, position the pointer on the object, select **EDIT**, and then change any of the entries in the displayed dialog box.

To edit objects that are part of a module, you must enable **Allow Move/Edit/Delete of Module Elements** in the **Global Options** dialog box.

Edit Alignment Target dialog box

Use this dialog box to select a different layer, copper tool, and alignment target style; change the radius; and position the alignment target by coordinates.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

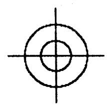
Droplist boxes

Layer Use to place the alignment target on the selected layer.

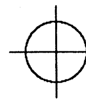
Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Alignment Target Style Use to select the style of the alignment target. The available styles are:

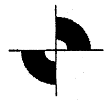
- **Crosshair Double.** A crosshair with two rings in all four quadrants.



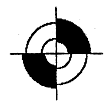
- **Crosshair Single.** A crosshair with one ring in all four quadrants.



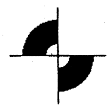
- **Quadrant 1 3.** A crosshair with filled arcs in the first and third quadrants.



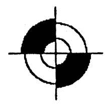
- **Quadrant 1 3 Ring.** Quadrant 1 3, with ring segments in the second and fourth quadrants.



- **Quadrant 2 4.** A crosshair with filled arcs in the second and fourth quadrants.



- **Quadrant 2 4 Ring.** Quadrant 2 4, with two rings in the first and third quadrants.



NOTE: Due to limits imposed by some vector devices, the size of the alignment targets is limited. For Fire9XXX and Gerber (274-D) output, the filled wide arcs are not drawn at all. For Gerber (274-X) output, the arcs may not exceed an outer radius of 1 inch. Postscript and HPGL2 and all raster devices have no limits on the size of the alignment targets.

Entry boxes

Radius Use to edit the radius of the alignment target. The radius is measured from the center of the alignment target to the end of any of the crosshair lines.

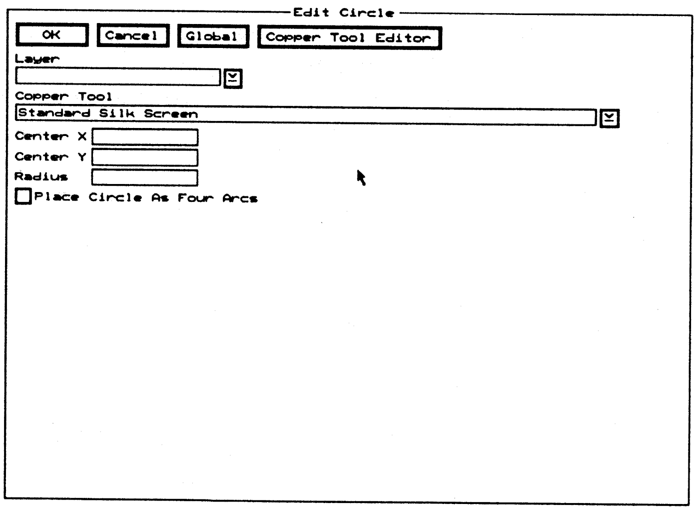
Center X Use to move the alignment target to the left or right of its current location.

Center Y Use to move the alignment target up or down from its current location.

△ *NOTE: In the Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). The center of the alignment target is the reference point. Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Edit Circle dialog box

Use this dialog box to select a different layer or copper tool, position the circle, change the radius, and allow the circle to be placed as four arcs.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

List boxes

Layer Use to place the circle on a different layer.

Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Entry boxes

Center X Use to move the circle to the left or right of its current location.

Center Y Use to move the circle up or down from its current location.

△ *NOTE: In the Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). The center of the circle is the reference point. Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Radius Use to edit the radius of the circle. The allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm).

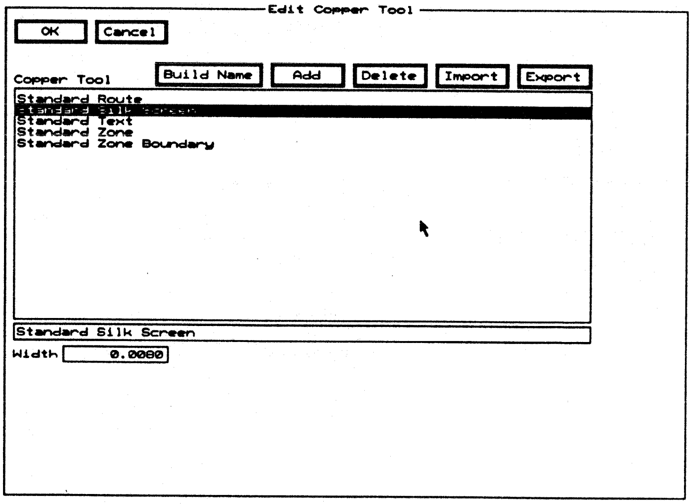
Check box

Place Circle As Four Arcs Typically, this check box is enabled to allow the placing of a semicircle. Place the circle as four arcs and then delete two of the arcs.

Edit Copper Tool dialog box

Use this dialog box to create, edit, and delete copper tools.

See also *Creating copper tools, Editing copper tools, Changing the name of copper tools,* and *Deleting copper tools.*



Copper Tool Contains a list of copper tools.

Use the entry box directly beneath the **Copper Tool** list box to change the name of a copper tool.

Build Name Creates a name from the value in the **Width** entry box, and loads the name into the entry box directly below the **Copper Tool** list box.

Add Adds the copper tool name shown in the entry box to the **Copper Tool** list box. If the copper tool name in the entry box matches an existing copper tool name, **Add** updates that copper tool with the value shown in the **Width** entry box.

Delete Removes the highlighted copper tool from the **Copper Tool** list box. If the highlighted copper tool is being used, it cannot be deleted, and a **Notice** dialog box displays. Select **OK** to dismiss the dialog box.

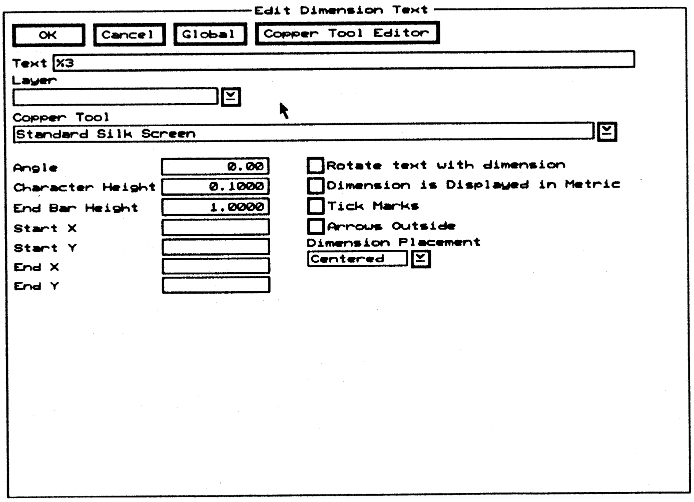
Import Displays the **Import Copper Tool from File** dialog box, where you specify the file from which a copper tool is to be imported.

Export Displays the **Export Copper Tool to File** dialog box, where you specify the file to which the highlighted copper tool is to be exported.

Width Use to change the width of the copper tool.

Edit Dimension Text dialog box

Use this dialog box to set the dimension object's text, layer, copper tool, starting and ending positions, and various other attributes.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Entry box

Text Use to set the dimension object's text. The syntax of the text string is:

```
[text] %{0-9} [text] [%s] [text]
```

This means you must enter a percent sign (%), followed by a single digit between 0 and 9, with no intervening spaces, to specify the number of digits shown to the right of the decimal point in the dimension text. Zero (0) removes the decimal point. You can also display the current unit of measurement (inches or millimeters) by entering `%s` anywhere to the right of the *%n*. Any other characters are simply reproduced.

For example, if you enter `Size: %4 %s`, **Edit Layout** displays "Size: *n.nnnn* mm" or "Size: *n.nnnn* in" for the dimension object.

If you enter an invalid string, **Edit Layout** displays a syntax statement similar to the one shown above.

List boxes

Layer Use to place the dimension object on a different layer.

Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more objects to this list.

Entry boxes

Angle Use to rotate the dimension object's text. The allowed range is 0.00 to 359.99 degrees.

Character Height Use to change the height of the dimension object's text. The allowed range is 0.0001 in. (0.0025 mm) to 10.0000 in. (254.0000 mm).

End Bar Height Use to change the height of the dimension object's end bars. The allowed range is 0.0001 in. (0.0025 mm) to 10.0000 in. (254.0000 mm).

Start X Use to move the dimension object's starting end bar to the left or right of its current location.

Start Y Use to move the dimension object's starting end bar up or down from its current location.

End X Use to move the dimension object's ending end bar to the left or right of its current location.

End Y Use to move the dimension object's ending end bar up or down from its current location.

△ *NOTE: In the Start and End entry boxes, the center of the end bar is the reference point. The allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Check boxes

Rotate text with dimension Rotate the dimension object's text with the end bars. This option is used during placement only.

Dimension is Displayed in Metric Display the dimension object's text in metric.

Tick Marks Use tick marks instead of arrows.

Arrows Outside Always place the arrows outside of the end bars.

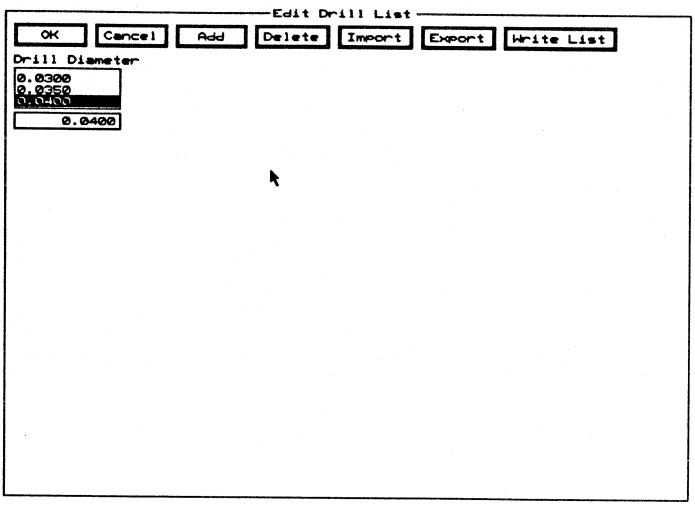
List box

Dimension Placement Use to place the dimension object's text in another location. Note that the orientation (north, south, and so on) is relative to the angle at which you placed the dimension object, which is assumed to be from west to east.

Edit Drill List dialog box

Use this dialog box to select, add, delete, import, and export drill diameters.

See also *Creating drill diameters* and *Deleting drill diameters.*



Add Adds the drill diameter shown in the entry box to the **Drill Diameter** list box.

Delete Removes the selected drill diameter from the **Drill Diameter** list box. If the selected drill diameter is being used, it cannot be deleted, and a **Notice** dialog box displays. Select **OK** to dismiss the **Notice** dialog box.

Import Displays the **Import Drill List from File** dialog box.

Export Displays the **Export Drill List to File** dialog box.

Write List Displays the **Write Drill List to Text File** dialog box.

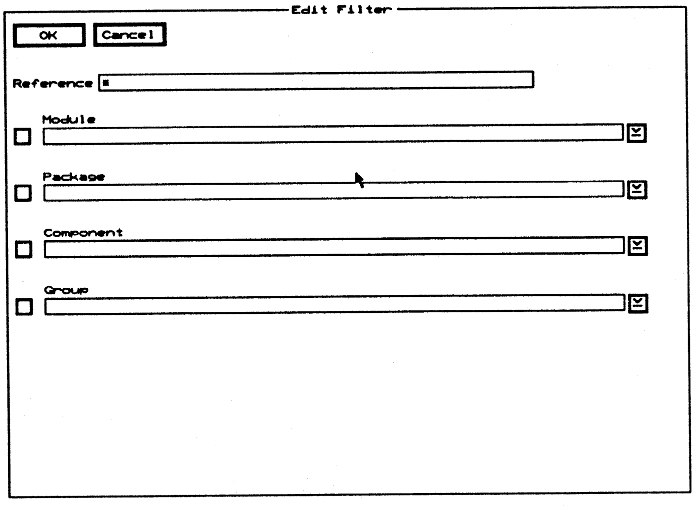
Drill Diameter Contains a list of drill diameters.

Use the entry box directly beneath the **Drill Diameter** list box to create a new drill diameter.

Edit Filter dialog box

Use this dialog box to set the wildcard and select filters for module names. The filters shown on this screen are used by the **Filter Enables** check boxes on the **Place Module** and **Get Module** dialog boxes to restrict the number of modules shown in the **Module Name** list box.

See also *Netlist Load Options dialog box* and *Edit Other Module Properties dialog box.*

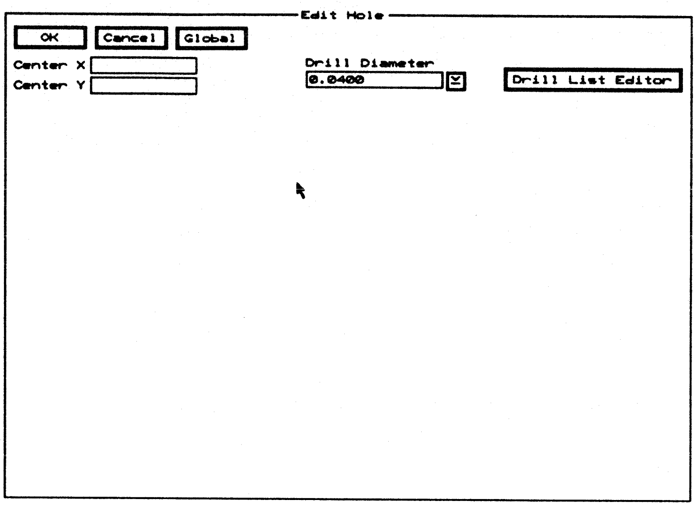


Reference Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of modules shown.

Module, Package, Component, and **Group** Enable to select one of the filters in the droplist box.

Edit Hole dialog box

Use this dialog box to select a drill diameter and position a hole.



Drill List Editor Displays the **Edit Drill List** dialog box.

Center X Use to move the hole to the left or right of its current location.

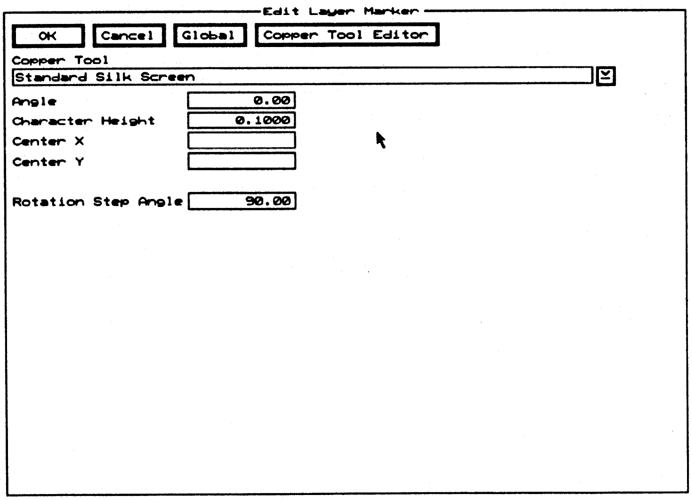
Center Y Use to move the hole to up or down from its current location.

△ *NOTE: In the Center entry boxes, the center of the hole is the reference point. The allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Drill Diameter Use to select another drill diameter. Select **Drill List Editor** to add drill diameters to this list.

Edit Layer Marker dialog box

Use this dialog box to select a copper tool, position the layer marker on the screen, and specify various layer marker attributes.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Copper Tool Use to select another copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Angle Use to rotate the layer marker. The allowed range is 0.00 to 359.99 degrees.

Character Height Use to edit the height of the marker numbers. The allowed range is 0.0001 in. (0.0025 mm) to 10.0000 in. (254.0000 mm).

Center X Use to move the layer marker to the left or right of its current location.

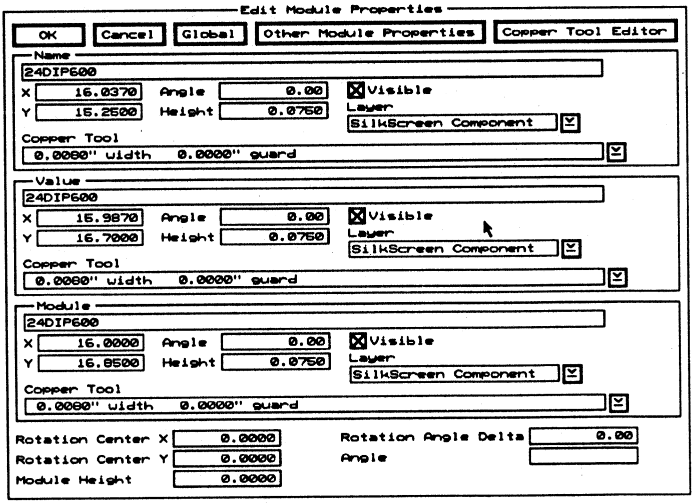
Center Y Use to move the layer marker up or down from its current location.

NOTE: In the Center entry boxes, the center of the layer marker is the reference point. The allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.

Rotation Step Angle Use to set how many degrees **> Rotate Clockwise** and **< Rotate Counter Clockwise** rotate the layer marker enclosed or intersected by the block boundary. The allowed range is 0.00 to 359.99 degrees and the default is 90.00 degrees.

Edit Module Properties dialog box

Use this dialog box to edit and position a module's reference designator, module value, and module type, and to set various values used by pick-and-place machines in board manufacturing.



If the **Edit Module Properties** dialog box is displayed from the library editor, or if the dialog box is displayed from the board editor and the module you are editing is not loaded from a netlist, the **Name, Value,** and **Module** entry boxes show only the module's reference designator.

Also, if the **Edit Module Properties** dialog box is displayed from the library editor, all edits pertain to the placeholders for the reference designator, module value, and module type.

Other Module Properties Displays the **Edit Other Module Properties** dialog box.

Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Name area Use the **Name** entry box to edit the reference designator's name.

X Use to move the reference designator to the left or right of its current location.

Y Use to move the reference designator up or down from its current location.

△ *NOTE: In the **X** and **Y** entry boxes, the center of the reference designator is the reference point. The allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Angle Use to rotate the reference designator. The allowed range is 0.00 to 359.99 degrees.

Height Use to edit the height of the reference designator. The allowed range is 0.0001 in. (0.0025 mm) to 10.0000 in. (254.0000 mm).

Visible Enable to make the reference designator visible. Note that **Hide Reference Designator Text** in the **Global Options** dialog box must be disabled.

Layer Use to place the reference designator on another layer.

Copper Tool Use to select another copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Value area

Use the **Value** entry box to edit the module value's name.

X Use to move the module value to the left or right of its current location.

Y Use to move the module value up or down from its current location.

△ *NOTE: In the X and Y entry boxes, the center of the module value is the reference point. The allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Angle Use to rotate the module value. The allowed range is 0.00 to 359.99 degrees.

Height Use to edit the height of the module value. The allowed range is 0.0001 in. (0.0025 mm) to 10.0000 in. (254.0000 mm).

Visible Enable to make the module value visible. Note that **Hide Module Value Text** in the **Global Options** dialog box must be disabled.

Layer Use to place the module value on another layer.

Copper Tool Use to select another copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Module area

Use the **Module** entry box to edit the module type's name.

X Use to move the module type to the left or right of its current location.

Y Use to move the module type up or down from its current location.

NOTE: In the X and Y entry boxes, the center of the module type is the reference point. The allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.

Angle Use to rotate the module type. The allowed range is 0.00 to 359.99 degrees.

Height Use to edit the height of the module type. The allowed range is 0.0001 in. (0.0025 mm) to 10.0000 in. (254.0000 mm).

Visible Enable to make the module type visible. Note that **Hide Module Type Text** in the **Global Options** dialog box must be disabled.

Layer Use to place the module type on another layer.

Copper Tool Use to select another copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Entry boxes

Rotation Center X Use to enter the X coordinate of pin 1. Used by board manufacturers to compute the assembly origin.

Rotation Center Y Use to enter the Y coordinate of pin 1. Used by board manufacturers to compute the assembly origin.

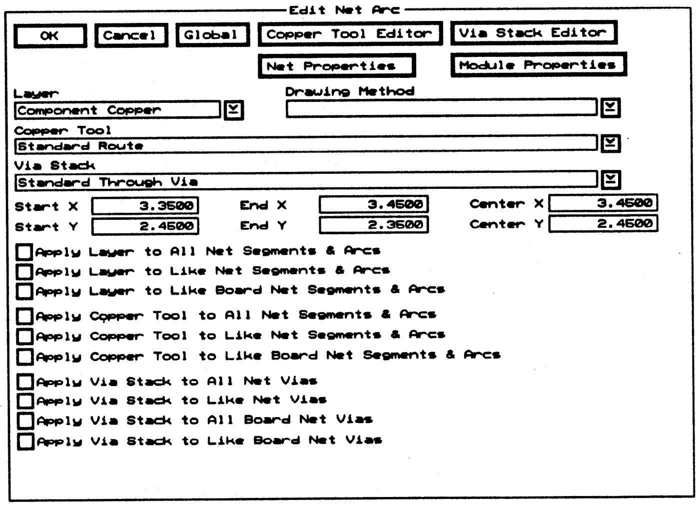
Module Height Use to enter the physical height of the module.

Rotation Angle Delta After being dropped onto a board, some modules are no longer at the angle specified by the designer. Use this entry box to compensate for this occurrence.

Angle Use to change the angle of placement for the module.

Edit Net Arc dialog box

Use this dialog box to place net arcs on different layers, select a different copper tool, position a net arc, and apply attributes to other net arcs.



△ *NOTE: Edit Layout supports 90° arcs that are contained in a single quadrant (0°–90°, 90°–180°, 180°–270°, 270°–360°). If you rotate an arc so that either condition no longer applies, Edit Layout breaks the arc into four segments. Note that you cannot recreate the arc from these four segments.*

Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Via Stack Editor Displays the **Edit Via Stack** dialog box.

Net Properties Displays the **Edit Net Properties** dialog box.

Module Properties Displays the **Edit Module Properties** dialog box.

List boxes

Layer Use to place the net arc on a different layer.

Drawing Method Use to select another drawing method.

Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Via Stack Use to select a different via stack. Select **Via Stack Editor** to add more via stacks to this list.

Entry boxes

Start X Use to move the net arc's starting point to the left or right of its current location.

Start Y Use to move the net arc's starting point up or down from its current location.

End X Use to move the net arc's ending point to the left or right of its current location.

End Y Use to move the net arc's ending point up or down from its current location.

Center X Use to move the net arc to the left or right of its current location. The center of the net arc is the reference point.

Center Y Use to move the net arc up or down from its current location. The center of the net arc is the reference point.

△ *NOTE: In the Start, End, and Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Layer check boxes

Apply Layer to All Net Segments & Arcs Enable to apply the selected layer to all segments and arcs in this net.

Apply Layer to Like Net Segments & Arcs Enable to apply the selected layer to all segments and arcs in this net, for which the segment's or arc's layer matches the edited object's original layer.

Apply Layer to Like Board Net Segments & Arcs Enable to apply the selected layer to all segments and arcs in all nets, for which the segment's or arc's layer matches the edited object's original layer.

Copper tool check boxes

Apply Copper Tool to All Net Segments & Arcs Enable to apply the selected copper tool to all segments and arcs in this net.

Apply Copper Tool to Like Net Segments & Arcs Enable to apply the selected copper tool to all segments and arcs in this net, for which the segment's or arc's copper tool matches the edited object's original copper tool.

Apply Copper Tool to Like Board Net Segments & Arcs Enable to apply the selected copper tool to all segments and arcs in all nets, for which the segment's or arc's copper tool matches the edited object's original copper tool.

Via stack check boxes

Apply Via Stack to All Net Vias Enable to apply the selected via stack to all vias in this net.

Apply Via Stack to Like Net Vias Enable to apply the selected via stack to all vias in this net, for which the via's via stack matches the edited object's original via stack.

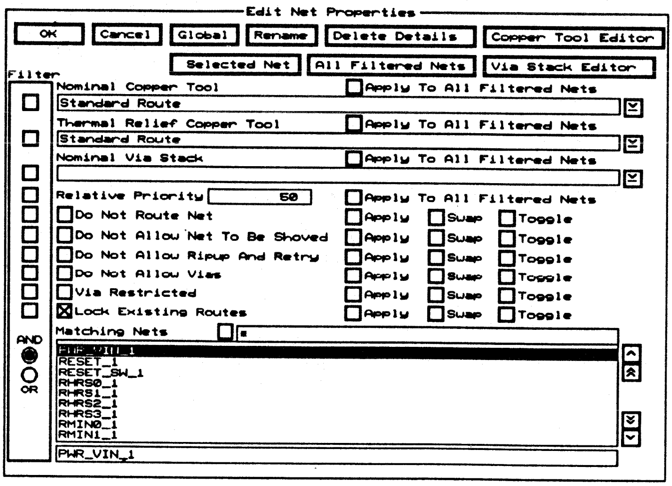
Apply Via Stack to All Board Net Vias Enable to apply the selected via stack to all vias in all nets.

Apply Via Stack to Like Board Net Vias Enable to apply the selected via stack to all vias in all nets, for which the via's via stack matches the edited object's original via stack.

Edit Net Properties dialog box

Use this dialog box to set a number of attributes and conditions for the nets on a board.

See also *Editing Net Properties.*



Buttons

Rename Displays the **Rename Net Objects** dialog box.

Delete Details Deletes all of the segments, arcs, and vias of the net that is highlighted in the **Matching Nets/Not Matching Nets** list box.

Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Selected Net This button applies the current status of all properties, regardless of the state of the corresponding **Apply To All Filtered Nets** check box, to the net whose name appears in the entry box below the **Matching Nets/Not Matching Nets** list box.

All Filtered Nets For each **Apply To All Filtered Nets** check box that is enabled, **All Filtered Nets** applies the corresponding property as marked (selected, entered, or enabled/disabled) to all nets which appear in the **Matching Nets/Not Matching Nets** list box.

Via Stack Editor Displays the **Edit Via Stack** dialog box.

Properties The following net properties can be edited:

List boxes

Nominal Copper Tool Use this list box to select a nominal copper tool. The default is Standard Route.

Thermal Relief Copper Tool Use this list box to select a thermal relief copper tool. This tool is used to draw the thermal relief connection from the pad to the plane or zone. The default is Standard Route.

Nominal Via Stack Use this list box to select a nominal via stack. Your selection is used only if **Via Restricted** is enabled.

Entry box

Relative Priority Normally, the autorouter first tries to route the widest wires first. To override that behavior—to route critical nets first, for example—use this entry box to assign specific priorities to individual nets. The higher the priority, the sooner the autorouter tries to route the net. The range is 0 to 100. Note that these priorities are relative, not absolute.

Check boxes

Do Not Route Net Prevents the selected net from being autorouted.

Do Not Allow Net To Be Shoved Prevents the selected net from being shoved in an attempt to make another connection.

Do Not Allow Ripup and Retry Prevents the autorouter from breaking an existing route in an attempt to make another connection.

Do Not Allow Vias Prevents the autorouter from inserting vias for the net.

Via Restricted Prevents the autorouter from placing any via type, other than the one shown in the **Nominal Via Stack** list box, on the net.

Lock Existing Routes Prevents any autorouter changes being made to completed routes.

Apply To All Filtered Nets

Each property in the dialog box has an associated Apply To All Filtered Nets or check box. See the preceding descriptions of the Selected Net and All Filtered Nets buttons for information on how these check boxes affect the values and conditions applied to the nets in the Matching Nets/Not Matching Nets list box and the net displayed in the entry box below the Matching Nets/Not Matching Nets list box.

△ *NOTE: Due to limited space, the last six **Apply To All Filtered Nets** check boxes are simply labeled **Apply**.*

Swap

Use these check boxes to swap the properties applied to the nets that match the filter with those nets that do not.

For example, enable **Do Not Route Net**, and its corresponding **Filter** check box. All the **Do Not Route Net** nets display in the **Matching Nets/Not Matching Nets** list box and the **Swap** check box becomes active. Enable the **Swap** check box, and then select **All Filtered Nets**. All the routable nets become **Do Not Route Net** nets, and all the **Do Not Route Net** nets become routable nets.

These check boxes become active when a corresponding **Filter** check box is enabled.

△ *NOTE: Swap only swaps within the set of nets matching the alphabetic filter.*

Toggle Use these check boxes to toggle the properties applied to specific nets that match the filter.

For example, enable **Do Not Route Net**, and its corresponding **Filter** check box. All the **Do Not Route Net** nets display in the **Matching Nets/Not Matching Nets** list box and the **Toggle** check box becomes active. Enable the **Toggle** check box, and then select **All Filtered Nets**. The **Do Not Route Net** nets become routable nets.

These check boxes become active when a corresponding **Filter** check box is enabled.

Filter Use these check boxes to filter the netnames in the **Matching Nets/Not Matching Nets** list box.

For example, if the **Do Not Allow Vias** check box is enabled and you select its corresponding check box in the **Filter** area, only **Do Not Allow Vias** nets will display in the **Matching Nets/Not Matching Nets** list box.

AND Use this radio button to conditionally filter the netnames in the **Matching Nets/Not Matching Nets** list box.

For example, if the **Do Not Allow Net To Be Shoved** and **Do Not Allow Vias** check boxes are enabled and you select their corresponding check boxes in the **Filter** area, only nets which are both **Do Not Allow Net To Be Shoved** and **Do Not Allow Vias** nets will display in the **Matching Nets/Not Matching Nets** list box.

OR Use this radio button to conditionally filter the netnames in the **Matching Nets/Not Matching Nets** list box.

For example, if the **Do Not Allow Net To Be Shoved** and **Do Not Allow Vias** check boxes are enabled and you select their corresponding check boxes in the **Filter** area, nets which are either **Do Not Allow Net To Be Shoved** or **Do Not Allow Vias** nets (or both) will display in the **Matching Nets/Not Matching Nets** list box.

List box

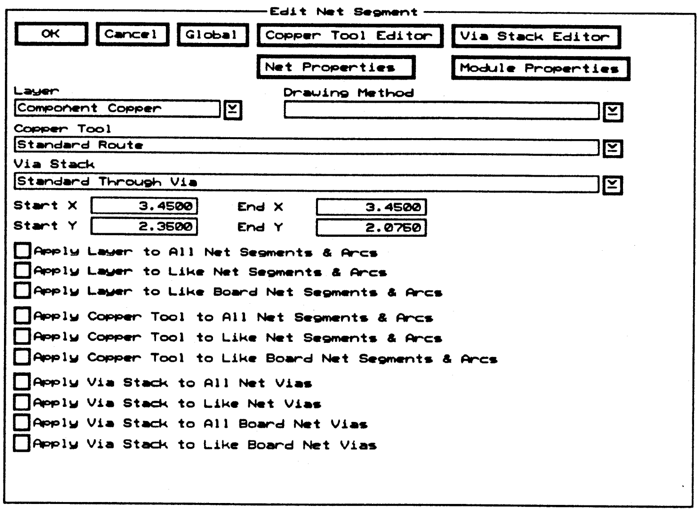
Matching Nets/Not Matching Nets Contains a list of netnames. Use any combination of wildcards (* or ?) and other characters in the entry box above the **Matching Nets/Not Matching Nets** list box to restrict the list of netnames shown.

The check box toggles the filter to only display those netnames which match the filter value (**Matching Nets**) or only those netnames which do not (**Not Matching Nets**).

△ *NOTE: It is easier to use the **Edit Net Properties** dialog box than read about it.*

Edit Net Segment dialog box

Use this dialog box to place net segments on different layers, select a different copper tool, position a net segment, and apply edited attributes to other net segments.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Via Stack Editor Displays the **Edit Via Stack** dialog box.

Net Properties Displays the **Edit Net Properties** dialog box.

Module Properties Displays the **Edit Module Properties** dialog box.

List boxes

Layer Use to place the net segment on a different layer.

Drawing Method Use to select another drawing method.

Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Via Stack Use to select a different via stack. Select **Via Stack Editor** to add more via stacks to this list.

Entry boxes

Start X Use to move the net segment's starting point to the left or right of its current location.

Start Y Use to move the net segment's starting point up or down from its current location.

End X Use to move the net segment's ending point to the left or right of its current location.

End Y Use to move the net segment's ending point up or down from its current location.

Center X Use to move the net segment to the left or right of its current location. The center of the net segment is the reference point.

Center Y Use to move the net segment up or down from its current location. The center of the net segment is the reference point.

△ *NOTE: In the Start, End, and Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Layer check boxes

Apply Layer to All Net Segments & Arcs Enable to apply the selected layer to all segments and arcs in this net.

Apply Layer to Like Net Segments & Arcs Enable to apply the selected layer to all segments and arcs in this net, for which the segment's or arc's layer matches the edited object's original layer.

Apply Layer to Like Board Net Segments & Arcs Enable to apply the selected layer to all segments and arcs in all nets, for which the segment's or arc's layer matches the edited object's original layer.

Copper tool check boxes

Apply Copper Tool to All Net Segments & Arcs Enable to apply the selected copper tool to all segments and arcs in this net.

Apply Copper Tool to Like Net Segments & Arcs Enable to apply the selected copper tool to all segments and arcs in this net, for which the segment's or arc's copper tool matches the edited object's original copper tool.

Apply Copper Tool to Like Board Net Segments & Arcs Enable to apply the selected copper tool to all segments and arcs in all nets, for which the segment's or arc's copper tool matches the edited object's original copper tool.

Via stack check boxes

Apply Via Stack to All Net Vias Enable to apply the selected via stack to all vias in this net.

Apply Via Stack to Like Net Vias Enable to apply the selected via stack to all vias in this net, for which the via's via stack matches the edited object's original via stack.

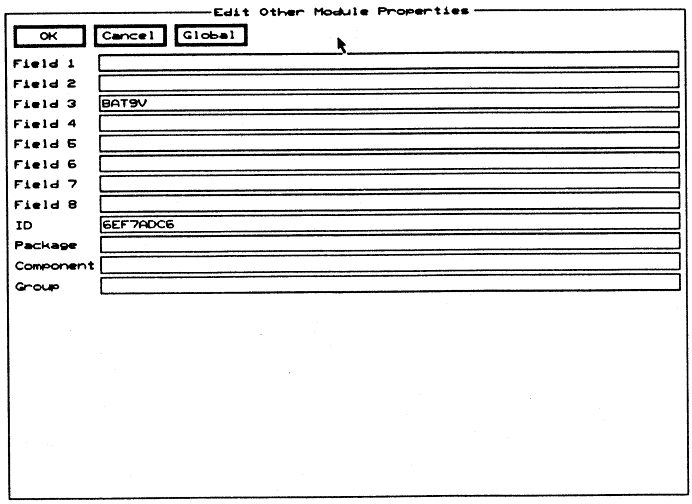
Apply Via Stack to All Board Net Vias Enable to apply the selected via stack to all vias in all nets.

Apply Via Stack to Like Board Net Vias Enable to apply the selected via stack to all vias in all nets, for which the via's via stack matches the edited object's original via stack.

Edit Other Module Properties dialog box

Use this dialog box to view the contents of fields brought in from the netlist, edit those fields, and create filters for module name lists.

See also *Edit Filter dialog box* and *Netlist Load Options dialog box*.



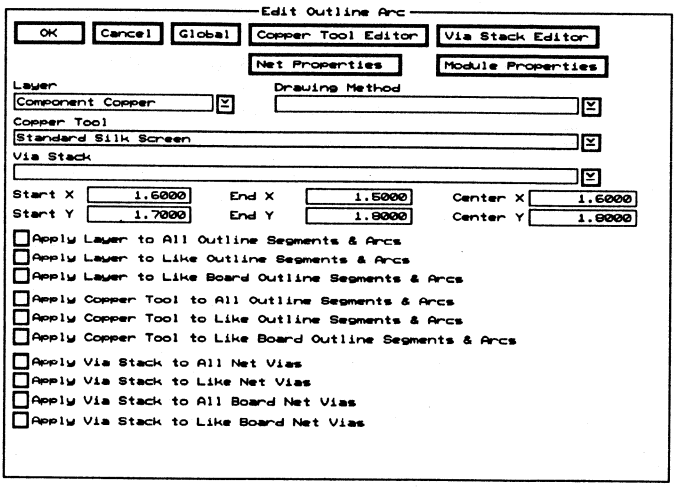
Field 1 through **Field 8** Displays values read into **Edit Layout** by the netlist loader. The values are stored here for reference use by the board designer.

ID Shows the module's time stamp. You can edit this value, but it is not advisable.

Package, Component, and **Group** Use these three entry boxes to create filters for module name lists. You can also place values in these entry boxes using the **Netlist Load Options** dialog box.

Edit Outline Arc dialog box

Use this dialog box to place outline arcs on different layers, select a different copper tool, position an outline arc, and apply edited attributes to other outline arcs.



△ *NOTE: Edit Layout supports 90° arcs that are contained in a single quadrant (0°–90°, 90°–180°, 180°–270°, 270°–360°). If you rotate an arc so that either condition no longer applies, Edit Layout breaks the arc into four segments. Note that you cannot recreate the arc from these four segments.*

Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Via Stack Editor Displays the **Edit Via Stack** dialog box.

Net Properties Displays the **Edit Net Properties** dialog box.

Module Properties Displays the **Edit Module Properties** dialog box.

Layer Use to place the outline arc on a different layer.

Drawing Method Use to select a different drawing method.

Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Via Stack This option does not apply to outline arcs.

Start X Use to move the outline arc's starting point to the left or right of its current location.

Start Y Use to move the outline arc's starting point up or down from its current location.

End X Use to move the outline arc's ending point to the left or right of its current location.

End Y Use to move the outline arc's ending point up or down from its current location.

Center X Use to move the outline arc to the left or right of its current location.

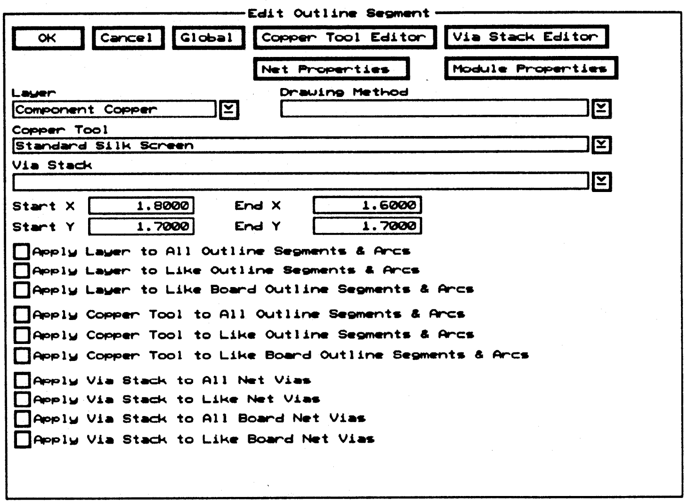
Center Y Use to move the outline arc up or down from its current location.

△ *NOTE: In the Start, End, and Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Layer check boxes	**Apply Layer to All Outline Segments & Arcs** Enable to apply the selected layer to all segments and arcs in this outline. **Apply Layer to Like Outline Segments & Arcs** Enable to apply the selected layer to all segments and arcs in this outline, for which the segment's or arc's layer matches the edited object's original layer. **Apply Layer to Like Board Outline Segments & Arcs** Enable to apply the selected layer to all segments and arcs in all outlines, for which the segment's or arc's layer matches the edited object's original layer.
Copper tool check boxes	**Apply Copper Tool to All Outline Segments & Arcs** Enable to apply the selected copper tool to all segments and arcs in this outline. **Apply Copper Tool to Like Outline Segments & Arcs** Enable to apply the selected copper tool to all segments and arcs in this outline, for which the segment's or arc's copper tool matches the edited object's original copper tool. **Apply Copper Tool to Like Board Outline Segments & Arcs** Enable to apply the selected copper tool to all segments and arcs in all outlines, for which the segment's or arc's copper tool matches the edited object's original copper tool.
Via stack check boxes	The options in this area do not apply to outline arcs.

Edit Outline Segment dialog box

Use this dialog box to place outline segments on different layers, select a different copper tool, position an outline segment, and apply edited attributes to other outline segments.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Via Stack Editor Displays the **Edit Via Stack** dialog box.

Net Properties Displays the **Edit Net Properties** dialog box.

Module Properties Displays the **Edit Module Properties** dialog box.

Layer Use to place the outline segment on a different layer.

Drawing Method Use to select a different drawing method.

Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Via Stack This option does not apply to outline segments.

Start X Use to move the outline segment's starting point to the left or right of its current location.

Start Y Use to move the outline segment's starting point up or down from its current location.

End X Use to move the outline segment's ending point to the left or right of its current location.

End Y Use to move the outline segment's ending point up or down from its current location.

Center X This option does not apply to outline segments.

Center Y This option does not apply to outline segments.

△ *NOTE: In the Start, End, and Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Layer check boxes

Apply Layer to All Outline Segments & Arcs Enable to apply the selected layer to all segments and arcs in this outline.

Apply Layer to Like Outline Segments & Arcs Enable to apply the selected layer to all segments and arcs in this outline, for which the segment's or arc's layer matches the edited object's original layer.

Apply Layer to Like Board Outline Segments & Arcs Enable to apply the selected layer to all segments and arcs in all outlines, for which the segment's or arc's layer matches the edited object's original layer.

Copper tool check boxes

Apply Copper Tool to All Outline Segments & Arcs
Enable to apply the selected copper tool to all segments and arcs in this outline.

Apply Copper Tool to Like Outline Segments & Arcs
Enable to apply the selected copper tool to all segments and arcs in this outline, for which the segment's or arc's copper tool matches the edited object's original copper tool.

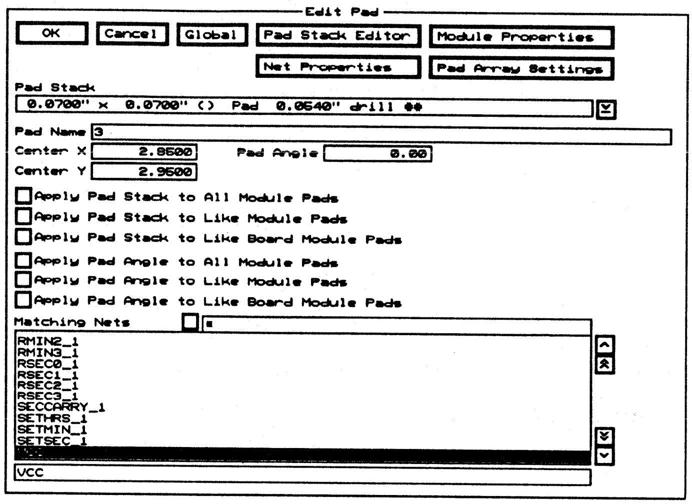
Apply Copper Tool to Like Board Outline Segments & Arcs Enable to apply the selected copper tool to all segments and arcs in all outlines, for which the segment's or arc's copper tool matches the edited object's original copper tool.

Via stack check boxes

The options in this area do not apply to outline segments.

Edit Pad dialog box

Use this dialog box to select, position, and apply pad stacks. There are two versions of this dialog box, one for the board editor and one for the library editor.



Pad Stack Editor Displays the **Edit Pad Stack** dialog box.

Module Properties Displays the **Edit Module Properties** dialog box.

Net Properties Displays the **Edit Net Properties** dialog box.

Pad Array Settings Displays the **Edit Pad Array Settings** dialog box.

Pad Stack Use to select a different pad stack. Select **Pad Stack Editor** to add more pad stacks to this list.

Pad Name Displays the name of the pad you are editing. Change the name of the pad by entering a new name in the entry box.

Center X Use to move the pad to the left or right of its current location.

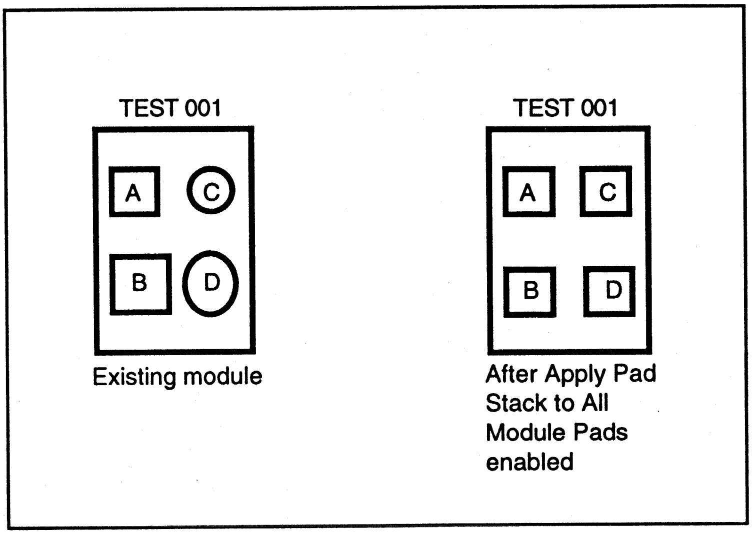
Center Y Use to move the pad up or down from its current location.

△ *NOTE: In the Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). The center of the pad is the reference point. Note that the location shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Pad Angle Use to rotate the pad. The allowed range is 0.00 to 359.99 degrees.

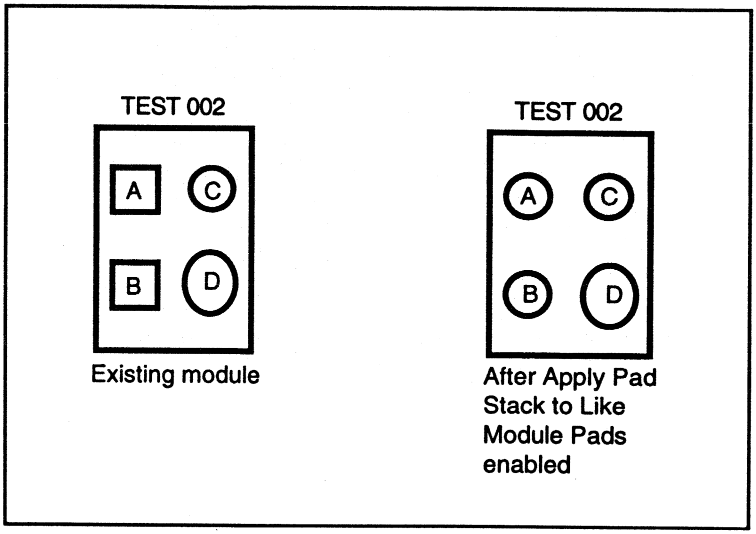
Pad stack check boxes

Apply Pad Stack to All Module Pads Enable to apply the selected pad stack to all pads in this module.



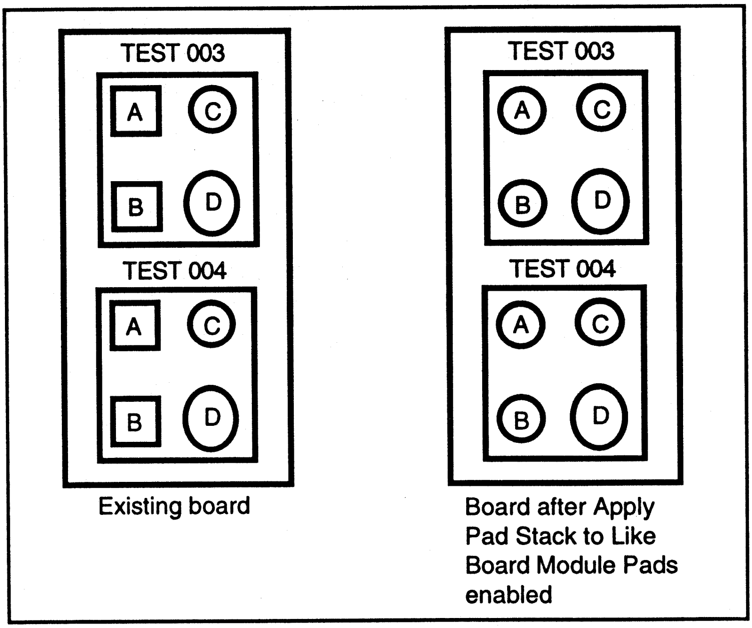
Pads A, B, C, and D all belong to module TEST 001. They all have different pad stacks. If you place your pointer on pad A, select **Edit**, and enable **Apply Pad Stack to All Module Pads**, all the pads of module TEST 001 will have the same pad stack as pad A.

Apply Pad Stack to Like Module Pads Enable to apply the selected pad stack to all pads in this module, for which the pad's pad stack matches the edited object's original pad stack.



Pads A, B, C, and D all belong to module TEST 002. Pads A and B have the same pad stack, pads C and D each have their own unique pad stack. If you place your pointer on pad A, select **Edit**, select pad C's pad stack from the **Pad Stack** list box, and enable **Apply Pad Stack to Like Module Pads**, pads A, B, and C will have the same pad stack. Pad D will remain unchanged.

Apply Pad Stack to Like Board Module Pads Board editor only. Enable to apply the selected pad stack to all pads in all modules, for which the pad's pad stack matches the edited object's original pad stack.



The board consists of two similar modules, TEST 003 and TEST 004. Pads A and B have the same pad stack, pads C and D each have their own unique pad stack. If you place your pointer on pad A of module TEST 003, select **Edit**, select pad C's pad stack from the **Pad Stack** list box, and enable **Apply Pad Stack to Like Board Module Pads**, pads A, B, and C on both modules will have the same pad stack. Pad D on both modules will remain unchanged.

Apply Pad Stack to Like Library Module Pads Library editor only. Enable to apply the selected pad stack to all pads in all modules, for which the pad's pad stack matches the edited object's original pad stack.

This check box will perform the same function as **Apply Pad Stack to Like Board Module Pads** except that it will affect pads throughout the selected library.

Pad angle check boxes

Apply Pad Angle to All Module Pads Enable to apply the selected pad angle to all pads in this module.

Apply Pad Angle to Like Module Pads Enable to apply the selected pad angle to all pads in this module, for which the pad's pad stack matches the edited object's original pad stack.

Apply Pad Angle to Like Board Module Pads Board editor only. Enable to apply the selected pad angle to all pads in all modules, for which the pad's pad stack matches the edited object's original pad stack.

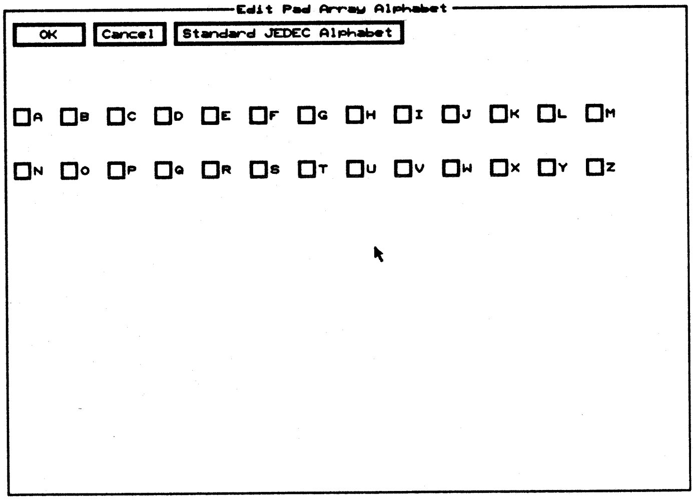
Apply Pad Angle to Like Library Module Pads Library editor only. Enable to apply the selected pad angle to all pads in all modules, for which the pad's pad stack matches the edited object's original pad stack.

List box

Netnames Board editor only. Use to select the pad's net. Use any combination of wildcards (* or ?) and other characters in the entry box above the **Netnames** list box to restrict the list of netnames shown. Use the entry box below the **Netnames** list box to edit the netname.

Edit Pad Array Alphabet dialog box

Use this dialog box to set the alphabetic characters to be used for pad names.

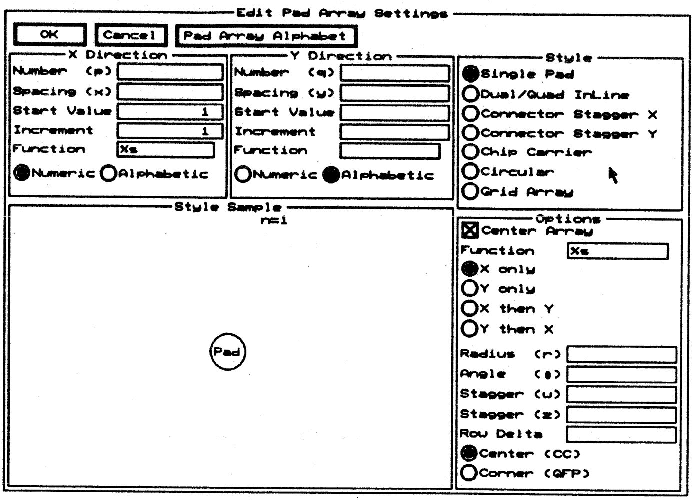


Standard JEDEC Alphabet Enables the appropriate check boxes to create a standard JEDEC alphabet.

Letters Enable the check boxes needed for pad names.

Edit Pad Array Settings dialog box

Use this dialog box to create and edit pad arrays.



Use the radio buttons in the **Style** area to select the desired pad array style.

The selected pad array style determines the options available in the **X Direction, Y Direction,** and **Options** areas.

Style

Seven styles are available: **Single Pad, Dual/Quad InLine, Connector Stagger X, Connector Stagger Y, Chip Carrier, Circular,** and **Grid Array.**

X Direction

Number (p) Use this entry box to set the number of pad columns in the array.

Spacing (x) Use this entry box to set the spacing between the center of the pad columns. The letter "x" shown in the **Style Sample** area represents this spacing. The allowed range is 0.0010 in. (0.0254 mm) to 10.0000 in. (254.0000 mm).

Start Value Use this entry box to set the value that **Edit Layout** starts with when assigning pad names. The start value can be numeric or alphabetic; however, the start value must be entered in the entry box as a number. Zero corresponds to the letter "A."

Increment Use this entry box to set the amount each pad name is incremented from the preceding pad name.

Function Use this entry box to format a text string for the X portion of the pad name.

Numeric Select to specify that the start value is numeric; thus, the pad names are numeric.

Alphabetic Select to specify that the start value is alphabetic; thus, the pad names are alphabetic.

Y Direction

Number (q) Use this entry box to set the number of pad rows in the array.

Spacing (y) Use this entry box to set the spacing between the center of the pad rows. The letter "y" shown in the **Style Sample** area denotes this spacing. The allowed range is 0.0010 in. (0.0254 mm) to 10.0000 in. (254.0000 mm).

Start Value Use this entry box to set the value that **Edit Layout** starts with when assigning pad names. The start value can be numeric or alphabetic; however, the start value must be entered in the entry box as a number. Zero corresponds to the letter "A."

Increment Use this entry box to set the amount each pad name is incremented from the preceding pad name.

Function Use this entry box to format a text string for the Y portion of the pad name.

Numeric Select to specify that the start value is numeric; thus, the pad names are numeric.

Alphabetic Select to specify that the start value is alphabetic; thus, the pad names are alphabetic.

Options

Center Array Enable to center the array on the pointer; otherwise, the pad assigned the start value is centered on the pointer.

Function Use this entry box to format a text string for the X and Y portion of the pad name.

X only Select to assign pad names with only one value. The numeric and alphabetic radio buttons in the **X Direction** area determine the value type.

Y only Select to assign pad names with only one value. The numeric and alphabetic radio buttons in the **Y Direction** area determine the value type.

X then Y Select to assign pad names with two values. The numeric and alphabetic radio buttons in the **X Direction** and **Y Direction** areas determine the first and second value types.

Y then X Select to assign pad names with two values. The numeric and alphabetic radio buttons in the **Y Direction** and **X Direction** areas determine the first and second value types.

Radius (r) Use this entry box to set the radius of the circular pad array. The letter "r" shown in the **Style Sample** area denotes this radius.

Angle (θ) Use this entry box to set the angle between the center of the pads in a circular pad array. The symbol "θ" shown in the **Style Sample** area denotes this angle.

Stagger (w) Use this entry box to set the amount every other pad in the left column is staggered from the column line. The letter "w" shown in the **Style Sample** area denotes this staggering. The allowed range is –10.0000 in. (–254.0000 mm) to 10.0000 in. (254.0000 mm).

Stagger (z) Use this entry box to set the amount every other pad in the right column is staggered from the column line. The letter "z" shown in the **Style Sample** area denotes this staggering. The allowed range is –10.0000 in. (–254.0000 mm) to 10.0000 in. (254.0000 mm).

Row Delta Use this entry box to specify whether or not a pad is added or subtracted from the right side of a staggered row. The allowed range is –1 to 1.

Center (CC) Select to give the center top pad in a chip carrier style array the start value.

Corner (QFP) Select to give the top left corner pad in a chip carrier style array the start value.

Style Sample

This area shows a formula and a graphic representation of the pad array based on the selected style and the specified values.

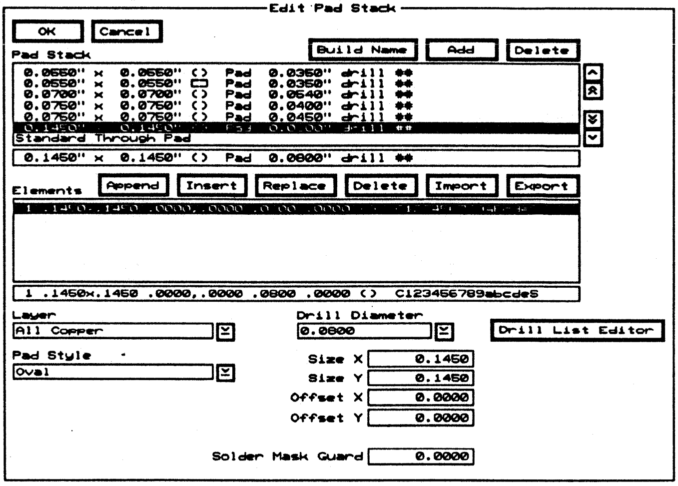
The formula which displays at the top of the area describes how many pads the selected pad array style will yield. For example, if you select the style **Dual/Quad InLine** and enter 4 in the **Number (q)** entry box, the formula "n=8=2*q" displays. This indicates that 8 pads will be created: two (columns in a dual/quad inline pad array) times four (the specified number of pads).

The pad labels in the graphic representation show the order of the pad naming sequence. As in the formula, **n** represents the number of pads in the array. The letters **x**, **y**, **w**, and **z** correspond to the letters shown in parentheses before the **Spacing** and **Stagger** entry boxes in the **Options** area. The letters indicate which aspect of the pad array is affected by the corresponding value.

Edit Pad Stack dialog box

Use this dialog box to create, edit, and delete pad stacks and pad stack elements.

See also *Creating pad stacks, Changing the name of pad stacks, Deleting pad stacks, Creating pad stack elements, Editing pad stack elements, Deleting pad stack elements,* and *Changing the order of pad stack elements.*



Pad stacks

Pad Stack Contains a list of pad stacks. A pad stack represents all its associated pad stack elements.

Use the entry box directly beneath the **Pad Stack** list box to change the name of a pad stack.

Build Name Creates a name from the characteristics of pad stack element 1 (shown at the top of the **Elements** list box), and loads the name into the entry box directly beneath the **Pad Stack** list box.

Add Adds the pad stack name shown in the entry box to the **Pad Stack** list box. If the pad stack name in the entry box matches an existing pad stack, **Add** updates that pad stack with the pad stack elements shown in the **Elements** list box.

Delete Removes the selected pad stack from the **Pad Stack** list box. If the selected pad stack is being used, it cannot be deleted, and a **Notice** dialog box displays. Select **OK** to dismiss the **Notice** dialog box.

Pad stack elements

Elements Contains a numbered list of pad stack elements. A pad stack element consists of the following pad parameters: layer, pad style, drill diameter, pad size, pad offset, and solder mask guard width.

Append Adds a pad stack element at the bottom of the list in the **Elements** list box. The pad stack element consists of the pad parameters shown in the entry boxes below the **Elements** list box.

Insert Adds a pad stack element above the highlighted pad stack element in the **Elements** list box. The pad stack element consists of the pad parameters shown in the entry boxes below the **Elements** list box.

Replace Loads the edited pad parameters into the highlighted pad stack element. Edit the pad parameters by changing the values in the entry boxes.

Delete Removes the selected pad stack element from the **Elements** list box.

Import Displays the **Import Pad Stack Elements from File** dialog box, where you specify the file from which a pad stack element is to be imported.

Export Displays the **Export Pad Stack Elements to File** dialog box, where you specify the file to which the highlighted pad stack element is to be exported.

Pad parameters

Layer Designates the layer on which a pad is located.

Pad Style Designates the pad shape: rectangle or oval.

Drill Diameter Designates the diameter of the drill hole. Select **Drill List Editor** to add drill diameters to the drill diameter list.

Drill List Editor Displays the **Edit Drill List** dialog box.

Size X Designates the size of the pad in the X direction.

Size Y Designates the size of the pad in the Y direction.

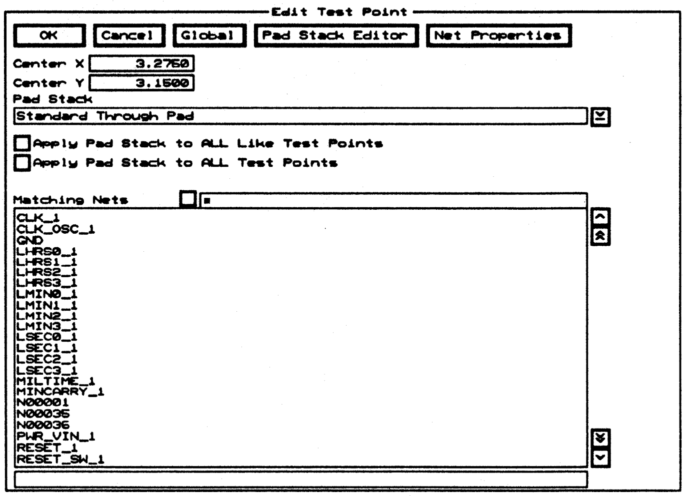
Offset X Specifies the horizontal distance by which the connection point is offset from the center of the pad.

Offset Y Specifies the vertical distance by which the connection point is offset from the center of the pad.

Solder Mask Guard Designates the width of the solder mask guard around the pad. This width is added to the solder mask guard width designated in the **Global Options** dialog box.

Edit Test Point dialog box

Use this dialog box to edit and position test points, and to apply the selected pad stack to other test points.



Pad Stack Editor Displays the **Edit Pad Stack** dialog box.

Net Properties Displays the **Edit Net Properties** dialog box.

Center X Use to move the test point to the left or right of its current location.

Center Y Use to move the test point up or down from its current location.

△ *NOTE: In the Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). The center of the test point is the reference point. Note that the location shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Pad Stack Use to select a different pad stack. Select **Pad Stack Editor** to add more pad stacks to this list.

Apply Pad Stack to ALL Like Test Points Enable to apply the selected pad stack to all test points, for which the test point's pad stack matches the edited object's original pad stack.

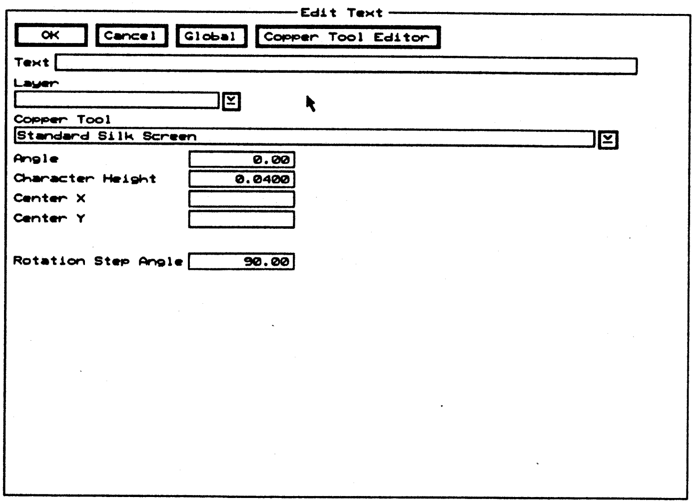
Apply Pad Stack to ALL Test Points Enable to apply the selected pad stack to all test points.

Netnames Use to assign a netname to a via. Use any combination of wildcards (* or ?) and other characters in the entry box above the **Netnames** list box to restrict the list of netnames shown. Use the entry box below the **Netnames** list box to edit the netname.

Edit Text dialog box

Use this dialog box to:

- ❖ Edit text
- ❖ Change the layer on which the text is placed
- ❖ Select a different copper tool
- ❖ Change the angle and the rotation step angle
- ❖ Change the text height
- ❖ Reposition the text



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Entry box

Text Use to edit the text.

Droplist boxes

Layer Use to place the text on a different layer.

Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Entry boxes

Angle Use to rotate the text. The allowed range is 0.00 to 359.99 degrees.

Character Height Use to edit the height of the text. The allowed range is 0.0001 in. (0.0025 mm) to 10.0000 in. (254.0000 mm).

Center X Use to move the text to the left or right of its current location.

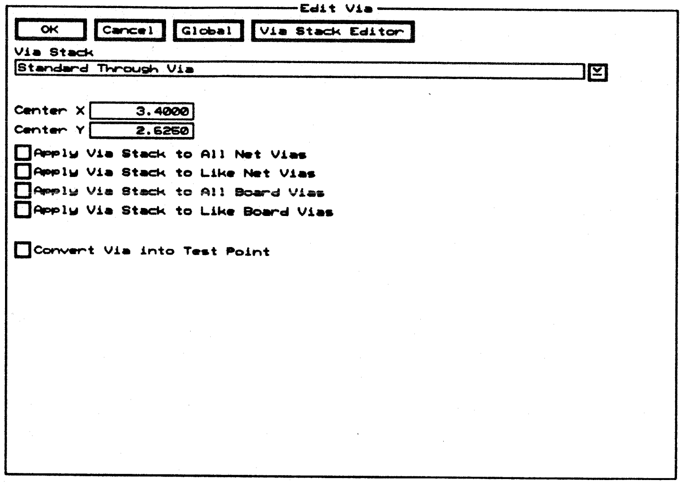
Center Y Use to move the text up or down from its current location.

△ *NOTE: In the Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). The center of the text is the reference point. Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Rotation Step Angle Use to set how many degrees **> Rotate Clockwise** and **< Rotate Counter Clockwise** rotate the text enclosed or intersected by the block boundary. The allowed range is 0.00 to 359.99 degrees and the default is 90.00 degrees.

Edit Via dialog box

Use this dialog box to select, position, apply, and convert vias.



Via Stack Editor Displays the **Edit Via Stack** dialog box.

Via Stack Use to select a different via stack. Select **Via Stack Editor** to add more via stacks to this list.

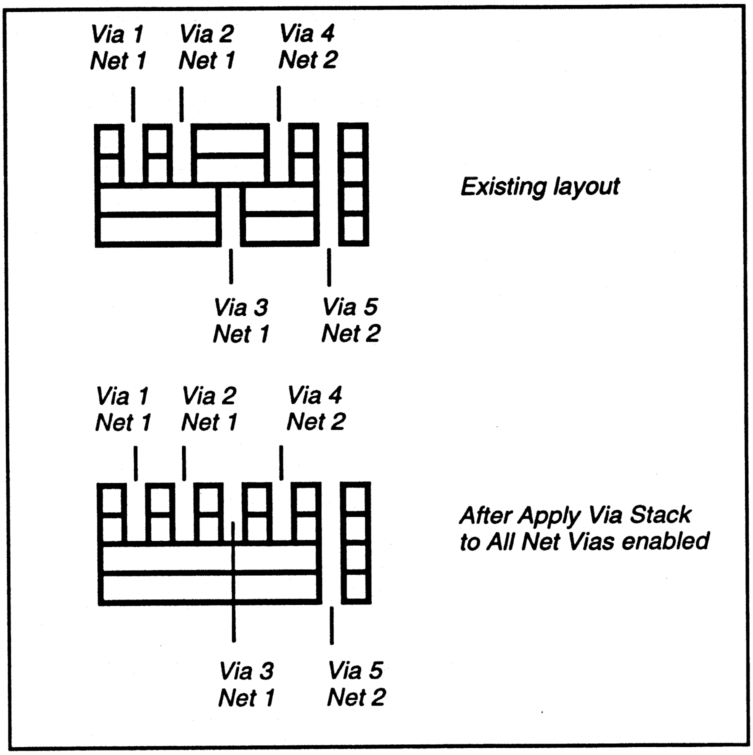
Center X Use to move the via to the left or right of its current location.

Center Y Use to move the via up or down from its current location.

△ *NOTE: In the Center entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). The center of the via is the reference point. Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Check boxes

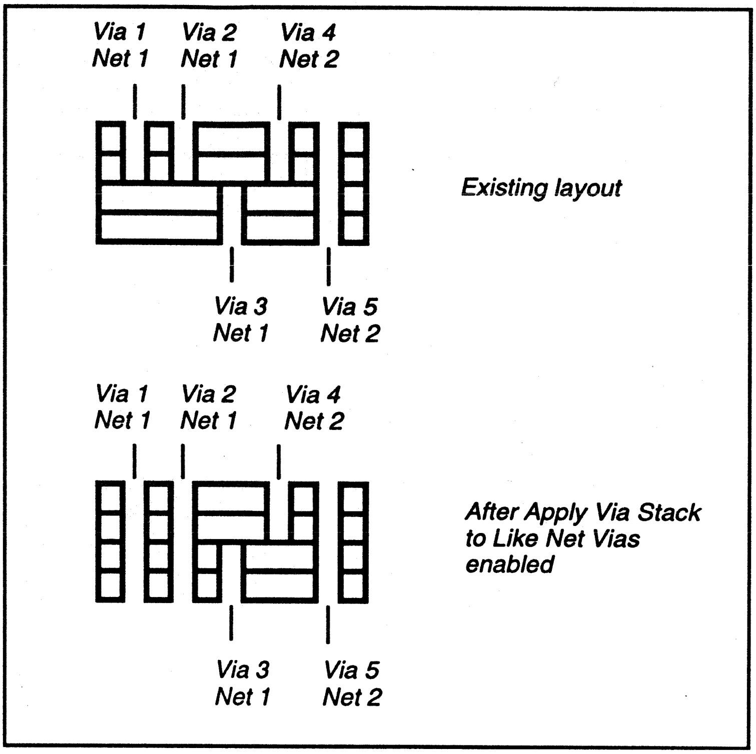
Apply Via Stack to All Net Vias Enable to apply the selected via stack to all vias in this net.



Side view of four layer board.

Vias 1, 2, and 3 belong to the same net. If you place your pointer on via 1, select **Edit**, and enable **Apply Via Stack to All Net Vias**, vias 1, 2, and 3 will have the same via stack properties. Vias 4 and 5 remain unchanged.

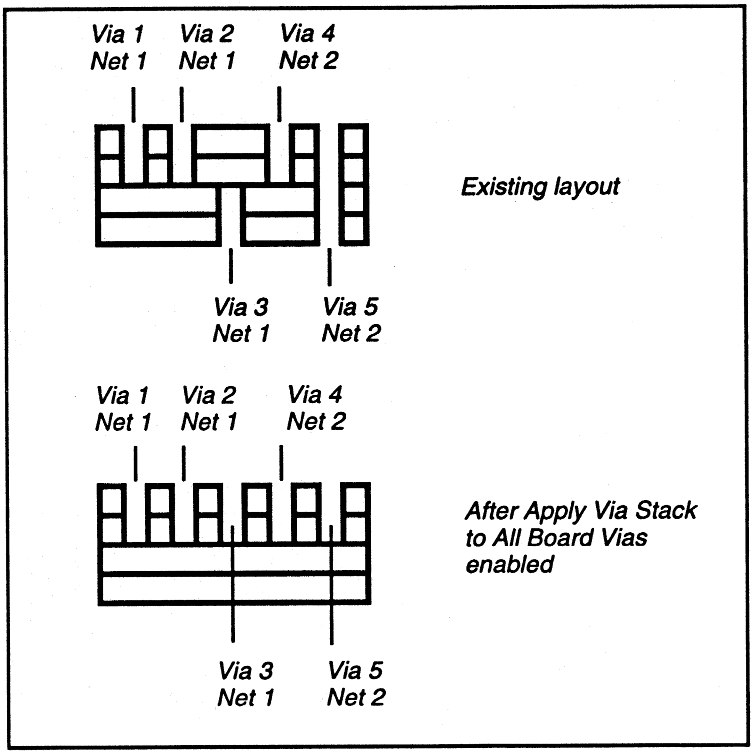
Apply Via Stack to Like Net Vias Enable to apply the selected via stack to all vias in this net, for which the via's via stack matches the edited object's original via stack.



Side view of four layer board.

Vias 1, 2, and 3 belong to same net. If you place your pointer on via 1, select **Edit**, select the via stack that created via 5 from the **Via Stack** list box, and enable **Apply Via Stack to Like Net Vias**, vias 1 and 2 will have the same via stack properties as via 5. Vias 3, 4, and 5 remain unchanged.

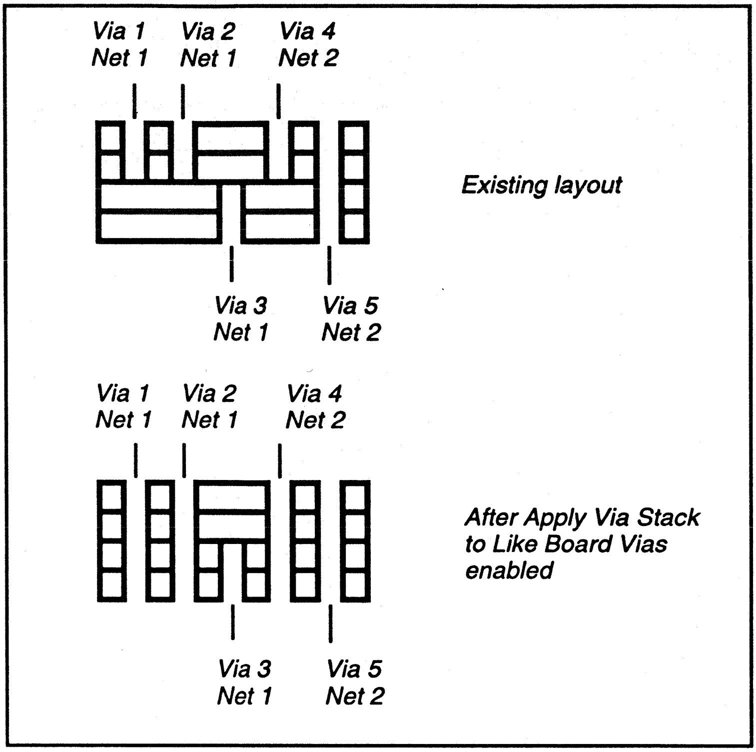
Apply Via Stack to All Board Vias Enable to apply the selected via stack to all vias on the board.



Side view of four layer board.

If you place your pointer on via 1, select **Edit,** and enable **Apply Via Stack to All Board Vias,** all the vias on the board will all have the same via stack properties as via 1.

Apply Via Stack to Like Board Vias Enable to apply the selected via stack to all vias on the board, for which the via's via stack matches the edited object's original via stack.



Side view of four layer board.

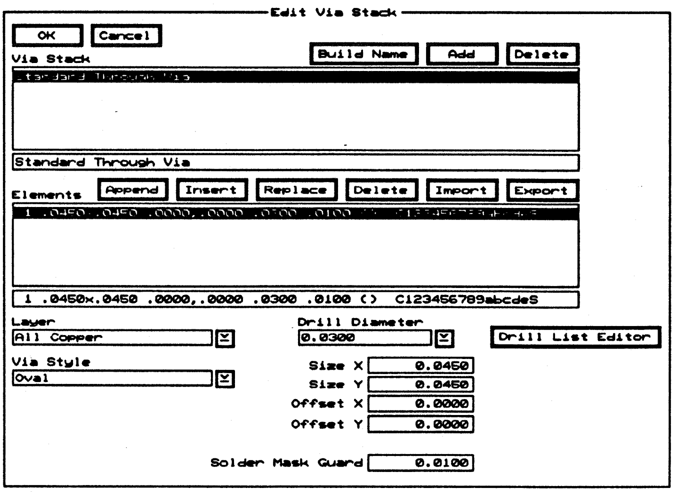
If you place your pointer on via 1, select **Edit**, select the via stack that created via 5 from the **Via Stack** list box, and enable **Apply Via Stack to Like Board Vias**, vias 1, 2, and 4 will have the same via stack properties as via 5. Vias 3 and 5 remain unchanged.

Convert Via into Test Point Enable to change the selected via into a test point.

Edit Via Stack dialog box

Use this dialog box to create, edit, and delete via stacks and via stack elements.

See also *Creating via stacks, Changing the name of via stacks, Deleting via stacks, Creating via stack elements, Editing via stack elements, Deleting via stack elements,* and *Changing the order of via stack elements.*



Via stacks

Via Stack Contains a list of via stacks. A via stack represents all its associated via stack elements.

Use the entry box directly beneath the **Via Stack** list box to change the name of a via stack.

Build Name Creates a name from the characteristics of via stack element 1 (shown at the top of the **Elements** list box), and loads the name into the entry box directly beneath the **Via Stack** list box.

Add Adds the via stack name shown in the entry box to the **Via Stack** list box. If the via stack name in the entry box matches an existing via stack, **Add** updates that via stack with the via stack elements shown in the **Elements** list box.

Delete Removes the selected via stack from the **Via Stack** list box. If the selected via stack is being used, it cannot be deleted, and a **Notice** dialog box displays. Select **OK** to dismiss the **Notice** dialog box.

Via stack elements

Elements Contains a numbered list of via stack elements. A via stack element consists of the following via parameters: layer, via style, drill diameter, via size, via offset, and solder mask guard width.

Append Adds a via stack element at the bottom of the list in the **Elements** list box. The via stack element consists of the via parameters shown in the entry boxes below the **Elements** list box.

Insert Adds a via stack element above the highlighted via stack element in the **Elements** list box. The via stack element consists of the via parameters shown in the entry boxes below the **Elements** list box.

Replace Loads the edited via parameters into the highlighted via stack element. Edit the via parameters by changing the values in the entry boxes.

Delete Removes the selected via stack element from the **Elements** list box.

Import Displays the **Import Via Stack Elements from File** dialog box, where you specify the file from which a via stack element is to be imported.

Export Displays the **Export Via Stack Elements to File** dialog box, where you specify the file to which the highlighted via stack element is to be exported.

Via parameters

Layer Designates the layer on which a via is located.

Via Style Designates the via shape: Rectangle or Oval.

Drill Diameter Designates the diameter of the drill hole. Select **Drill List Editor** to add drill diameters to the drill diameter list.

Drill List Editor Displays the **Edit Drill List** dialog box.

Size X Designates the size of the via in the X direction.

Size Y Designates the size of the via in the Y direction.

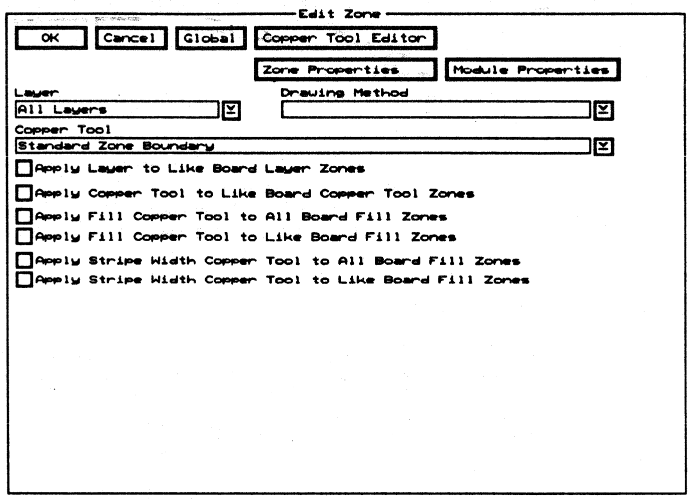
Offset X Specifies the horizontal distance by which the connection point is offset from the center of the via.

Offset Y Specifies the vertical distance by which the connection point is offset from the center of the via.

Solder Mask Guard Designates the width of the solder mask guard around the via. This width is added to the solder mask guard width designated in the **Global Options** dialog box.

Edit Zone dialog box

Use this dialog box to place zones on different layers, select a different copper tool, and apply edited attributes to other zones.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Zone Properties Displays the **Edit Zone Properties** dialog box.

Module Properties Displays the **Edit Module Properties** dialog box.

Layer Use to place the zone on a different layer.

Drawing Method Use to select a different drawing method.

Copper Tool Use to select a different copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Layer check box

Apply Layer to Like Board Layer Zones Enable to apply the selected layer to all board layer zones, for which the zone's layer matches the edited object's original layer.

Copper tool check box

Apply Copper Tool to Like Board Copper Tool Zones Enable to apply the selected copper tool to all zones, for which the zone's copper tool matches the edited object's original copper tool.

Fill copper tool check boxes

Apply Fill Copper Tool to All Board Zones Enable to apply the selected fill copper tool to all zones.

Apply Fill Copper Tool to Like Board Zones Enable to apply the selected fill copper tool to all zones, for which the zone's fill copper tool matches the edited object's original fill copper tool.

Stripe width copper tool check boxes

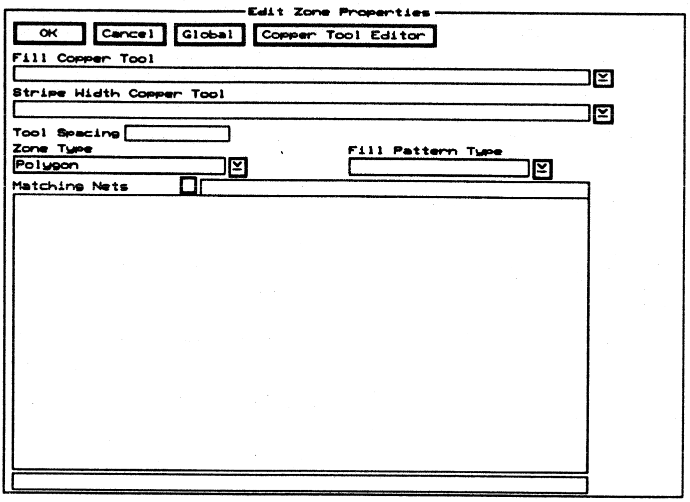
Apply Stripe Width Copper Tool to All Board Zones Enable to apply the selected fill copper tool to all zones.

Apply Stripe Width Copper Tool to Like Board Zones Enable to apply the selected stripe width copper tool to all zones, for which the zone's stripe width copper tool matches the edited object's original stripe width copper tool.

See also *Stripe Width Copper Tool.*

Edit Zone Properties dialog box

Use this dialog box to select fill copper tools, tools spacing, zone types, and fill pattern types.



Copper Tool Editor Displays the **Edit Copper Tool** dialog box.

Fill Copper Tool Use to select a fill copper tool. Select **Copper Tool Editor** to add more copper tools to this list.

Tool Spacing Use to specify the center to center spacing of the fill copper tool when filling the zone. To assure complete coverage of solid zones, specify the tool spacing to be less than the fill copper tool width. The allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm).

Zone Type Use to select a different type of zone.

Fill Pattern Type Use to select a fill pattern.

Netnames Use to select the zone's netname. Use any combination of wildcards (* or ?) and other characters in the entry box above the **Netnames** list box to restrict the list of netnames shown. Use the entry box below the **Netnames** list box to edit the zone's netname.

Editing copper tools

1. Display the **Edit Copper Tool** dialog box.
2. Select the copper tool in the **Copper Tool** list box. The copper tool name is loaded into the entry box directly beneath the **Copper Tool** list box.
3. Edit the value in the **Width** entry box.
4. Select **Add**. The copper tool is updated with the new width.

Editing modules

1. Make sure that Allow Move/Edit/Delete of Module Elements is enabled in the Global Options dialog box.
2. Place the pointer on the module object you wish to edit and select **EDIT**. The appropriate edit dialog box displays.

See also *Changing module names* and *Deleting modules*.

Editing net properties

The following describes how the **Edit Net Properties** dialog box can be used when routing a board.

The board in this example has regular signal nets, and power and ground nets. You will dispersion route the power and ground nets, lock all of their routes, and then route the signal nets.

1. Display the **Edit Net Properties** dialog box.
2. Enable **Do Not Route Net**.
3. Enable its associated **Apply To All Filtered Nets** check box (due to space constraints it is simply labled **Apply**).
4. Select **All Filtered Nets**. All the nets displayed in the **Matching Nets** list box are now **Do Not Route Net** nets.

Perform the following to make the power and ground nets autoroutable.

1. From the **Matching Nets** list box, select the power net.
2. Disable **Do Not Route Net.**
3. Select **Selected Net.**
4. From the **Matching Nets** list box, select the ground net.
5. Disable **Do Not Route Net.**
6. Select **Selected Net.** The power and ground nets are now routable.
7. Select **OK.**
8. Dispersion route the power and ground nets.

After the board has been successfully routed, perform the following.

1. Display the **Edit Net Properties** dialog box.
2. Enable **Do Not Route Net.**
3. Enable its associated **Filter** check box. All the nets displayed in the **Matching Nets** list box are now **Do Not Route Net** nets.
4. Enable its associated **Swap** check box.
5. Select **All Filtered Nets.** The power and ground nets display in the **Matching Nets** list box.
6. Enable **Lock Existing Routes.**
7. Enable its associated **Apply To All Filtered Nets** check box (due to space constraints it is simply labled **Apply**).
8. Select **All Filtered Nets.** The ground and power nets are now not autoroutable.
9. Select **OK** and route the rest of the board.

Editing pad stack elements

1. Display the **Edit Pad Stack** dialog box.
2. Select the pad stack element, and then edit the pad stack element's pad parameters.
3. Select **Replace** directly above the **Elements** list box. The new pad parameters appear in the pad stack element.

Editing pad stacks

1. Display the **Edit Pad Stack** dialog box.
2. Select the pad stack in the **Pad Stack** list box. The pad stack name is loaded into the entry box directly beneath the **Pad Stack** list box.
3. Modify the pad stack as described in *Creating a pad stack, Deleting a pad stack element, Editing a pad stack element,* and *Changing the order of pad stack elements.*
4. Make sure the pad stack whose pad stack elements you just edited is still highlighted in the **Pad Stack** list box.
5. Select **Add**. The pad stack is updated.

△ *NOTE: When you change the details of a pad stack, every pad and test point that uses that pad stack reflects the change.*

See also *Deleting pad stacks* and *Changing the name of pad stacks.*

Editing via stacks

1. Display the **Edit Via Stack** dialog box.
2. Select the via stack in the **Via Stack** list box. The via stack name is loaded into the entry box directly beneath the **Via Stack** list box.
3. Modify the via stack as described in *Creating a via stack, Deleting a via stack element, Editing a via stack element,* and *Changing the order of via stack elements.*
4. Make sure the via stack whose via stack element you just edited is still highlighted in the **Via Stack** list box.
5. Select **Add**. The via stack is updated.

△ *NOTE: When you change the details of a via stack, every via that uses that via stack reflects the change.*

See also *Deleting via stacks* and *Changing the name of via stacks.*

End command

Appears on a number of menus.

Sets the ending point of the current action, such as placing an object.

Erase All Routes command

Appears on the board editor **QUIT** menu.

Removes all routes and saves them in the undelete buffer. **Edit Layout** prompts you to verify your selection of this command.

Select **UNDELETE** to restore all routes or **SELECTIVE** to restore certain routes.

Exiting Edit Layout

1. If necessary, select **QUIT Update Board File** to save the file in the current working directory.
2. Select **QUIT Abandon Program.** The **PC Board Layout Tools** screen displays.

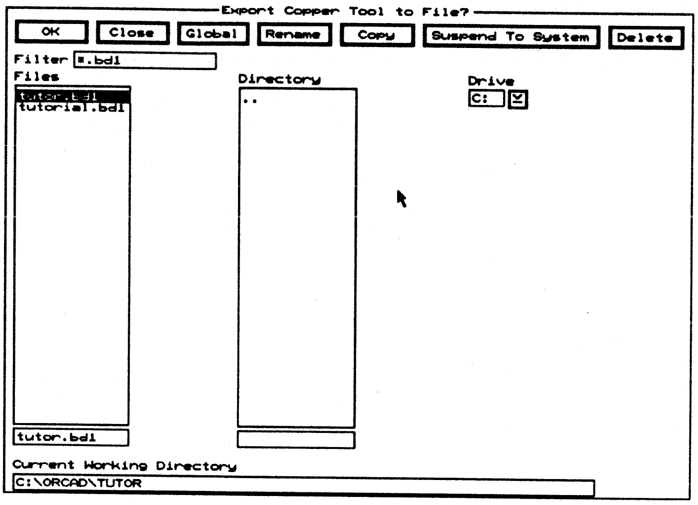
Export button

Appears on a number of dialog boxes.

Displays the appropriate **Export... to File** dialog box.

Export Copper Tool to File dialog box

Use this dialog box to export a copper tool to a file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

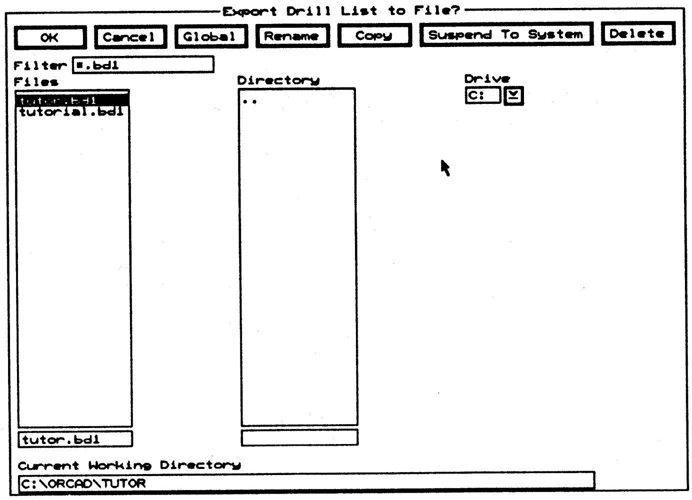
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Export Drill List to File dialog box

Use this dialog box to export a drill list to a file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

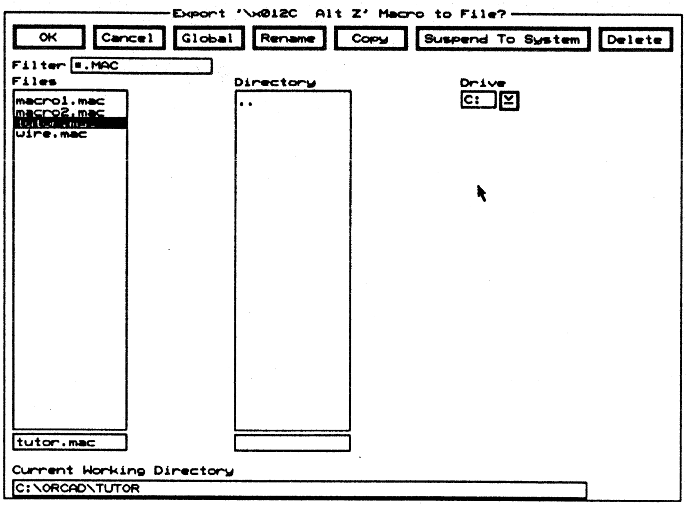
Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Export *'macroName'* Macro to File dialog box

Use this dialog box to export a selected macro to a file.

See also *Saving macros*.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

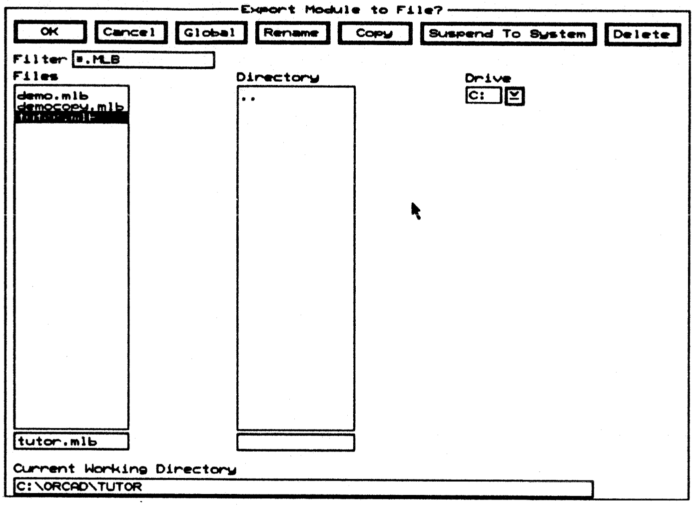
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Export Module to File dialog box

Use this dialog box to export a module to a file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

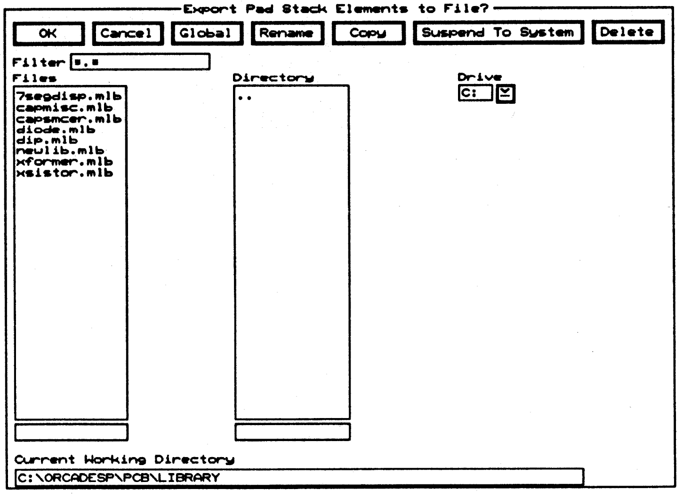
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Export Pad Stack Elements to File dialog box

Use this dialog box to export pad stack elements to a file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

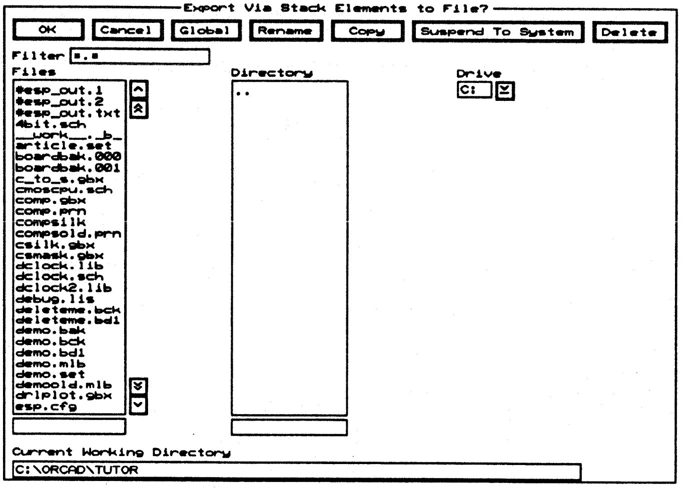
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Export Via Stack Elements to File dialog box

Use this dialog box to export via stack elements to a file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Fill Zone command

Appears on the **PLACE** menu.

Loads the current zone segment settings.

A *fill zone* defines an area to be filled in a copper pour. A *no-fill* zone defines an area within a fill zone that is not to be filled.

Copper filling does not prevent routing. In other words, routes can pass in and out of copper pour areas without exception—the copper flows around existing copper objects and those placed after the pour.

See also *Placing zones*.

Filter button

Appears on the **Get Module** and **Place Module** dialog boxes.

Displays the **Edit Filter** dialog box.

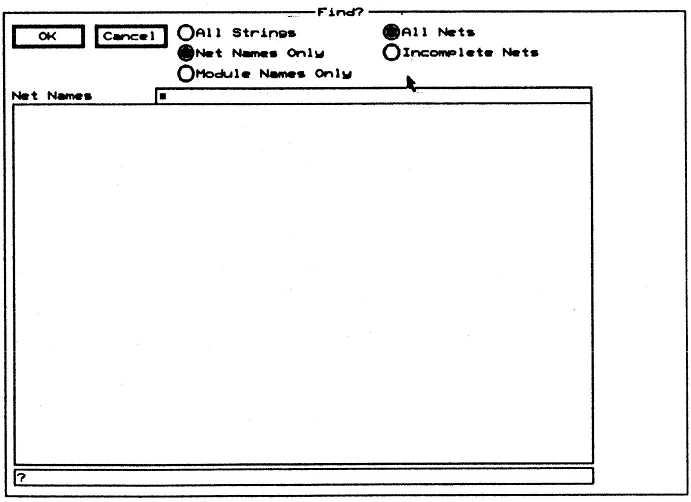
FIND command

Appears on a number of menus.

Displays the **Find** entry box.

Find dialog box

Use this dialog box to find a netname, module name, or any other text string in a layout.



Radio buttons

All Strings Select to display all strings in the list box.

Netnames Only Select to display only netnames in the list box. Enables the **All Nets** and **Incomplete Nets** radio buttons.

Module Names Only Select to display only module names in the list box.

All Nets Select to display the names of all existing nets in the **Netnames** list box.

Incomplete Nets Select to display the names of only the incomplete nets in the **Netnames** list box.

List boxes

In the entry box directly above the list box, use any combination of wildcards (* or ?) and other characters to restrict the list of strings, netnames, or module names shown.

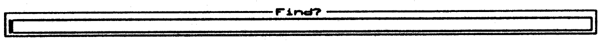
All Strings Contains a list of all strings in the board file or otherwise known to **Edit Layout**.

Netnames If **All Nets** is selected, contains a list of all the nets in the board file. If **Incomplete Nets** is selected, contains a list of all the incomplete nets in the board file.

Module Names Contains a list of all the module names in the board file.

Find entry box

Use this entry box to locate reference designators and netnames, and to display the **Find** dialog box.



Finding reference designators

Enter a reference designator in the **Find** entry box. **Edit Layout** moves the pointer to the specified reference designator and displays the message "Module Reference Designator" in the lower-right corner of the screen.

If **Find Highlights** in the **Global Options** dialog box is enabled, **FIND** highlights the module's reference designator, type, and value, and all the objects on the module.

Finding netnames

Board editor only. Enter a netname in the **Find** entry box. **Edit Layout** moves the pointer to the nearest pad connected to the specified net and displays the message "Netname" in the lower-right corner of the screen. If you select **FIND** again with the same netname, the pointer moves to the next nearest pad connected to the specified net.

If **Find Highlights** in the **Global Options** dialog box is enabled, **FIND** also highlights the pads, segments, arcs, and vias that make up the net.

Displaying the Find dialog box

Enter a question mark (?) in the **Find** entry box to display the **Find** dialog box.

Finished dialog box

Displays after the autorouter is finished routing a board and after the autorouter is finished with a spacing/DRC check.

After autorouting a board

Time Used The time in hours, minutes, and seconds that it took for the autorouter to perform the routing.

Total Number of Nets The number of nets on the board.

Number of Active Nets The number of nets on the board which did not have **Do Not Route Net** applied to them.

Number of Incomplete Active Nets The number of routable nets that are totally unrouted or partially routed.

Number of Completed Active Nets The number of routable nets that are completely routed.

After a spacing/DRC check

Time Used The time in hours, minutes, and seconds that it took the autorouter to perform the spacing/DRC check.

Number of Spacing/DRC Errors Found The total number of spacing/DRC errors found.

Flush Undelete Buffer command

Appears on the **QUIT** menu.

Permanently removes all objects from the undelete buffer. **Edit Layout** prompts you to verify your selection of this command.

Force vectors

A force vector is a single vector that represents the mathematical sum of all ratsnest vectors for the associated module. The force vector's length and direction indicate how far and in what direction to go. Your goal is to place the module so that the force vector is as short as possible.

To display force vectors, enable **Show Force Vectors** in the **Global Options** dialog box. All modules display their force vectors. Disable **Show Force Vectors** to prevent them from displaying.

Gerber formats

PCB 386+ 2.00 includes drivers for three Gerber formats: 274-X, 274-D, and Fire 9xxx. The Fire 9xxx and Gerber (274-X) drivers embed D-codes in the Gerber output file. The Gerber (274-D) driver produces Gerber output without embedded D-codes. The D-code aperture list is created separately and saved as a text file.

When you select the vector device **Fire 9xxx** in the **Driver Configuration** dialog box, **Edit Layout** produces Fire 9xxx output. **Fire 9xxx** can include up to 2,304 embedded D-codes (apertures) from 0.002 inches to 0.4 inches in diameter.

When you select the vector device **Gerber (274-X), Edit Layout** produces EIA RS-274-X output. Note that each orientation of an object requires a separate aperture, and RS-274-X is limited to 989 apertures, ranging from 0.002 inches to 2.048 inches in diameter.

When you select the vector device **Gerber (274-D),** you are given the opportunity to enter an input aperture filename, an output aperture filename, and an output aperture file format. The aperture file formats supported are WISE (same as previous versions of **PCB 386+ 2.00**), ECAM (CAD solutions) format, and **PCB II** format.

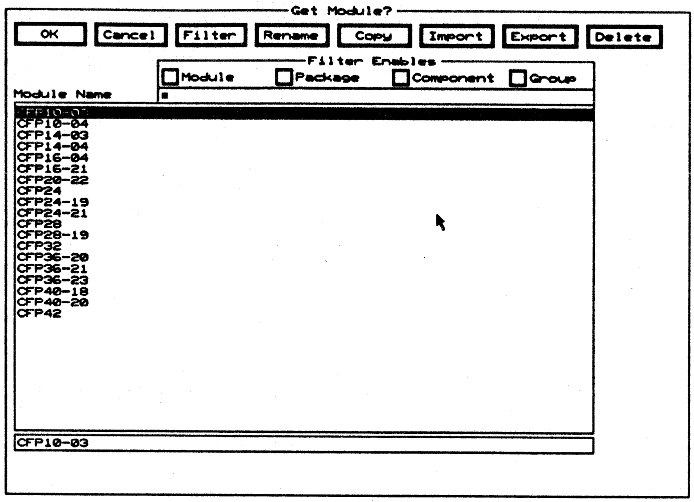
Before plotting a page, the input aperture file, if it exists, is read. If an output aperture file has been defined, any new apertures required are created as the page is plotted. After the page has been plotted, the combined apertures (old and new) are written to the output aperture file in the requested format. If no output aperture file has been defined, **PCB 386+ 2.00** will attempt to use one of the input apertures to draw an object in multiple strokes. If neither input nor output aperture files have been defined, **PCB 386+ 2.00** will refuse to plot. You can define up to 130 apertures.

See also *Driver Configuration dialog box, Aperture File to Read dialog box,* and *Aperture File to Write dialog box.*

△ *NOTE: Commercial and shareware Gerber viewers are available. Check OrCAD's 24-hour technical support bulletin board for more information about Gerber viewers.*

Get Module dialog box

Use this dialog box to select, rename, copy, import, export, and delete modules.



Filter Displays the **Edit Filter** dialog box.

Rename Displays the **Rename Module** dialog box.

Copy Displays the **Copy Module** dialog box.

Import Displays the **Import Module from File** dialog box.

Export Displays the **Export Module to File** dialog box.

Delete Removes the selected module from the **Module Name** list box.

If the module has been placed on a board and you delete it from the **Module Name** list box, **Edit Layout** displays an error message the next time the board is loaded.

Filter Enables Use this area to view the modules shown in the **Module Name** list box by type.

- ❖ **Module** Enable to list all modules which exactly match the value shown in the **Module** droplist box on the **Edit Filter** dialog box.
- ❖ **Package** Enable to list all modules which exactly match the value shown in the **Package** droplist box on the **Edit Filter** dialog box.
- ❖ **Component** Enable to list all modules which exactly match the value shown in the **Component** droplist box on the **Edit Filter** dialog box.
- ❖ **Group** Enable to list all modules which exactly match the value shown in the **Group** droplist box on the **Edit Filter** dialog box.

Module Name Contains a list of modules.

Use any combination of wildcards (* or ?) and other characters in the entry box directly above the **Module Name** list box to restrict the list of modules shown.

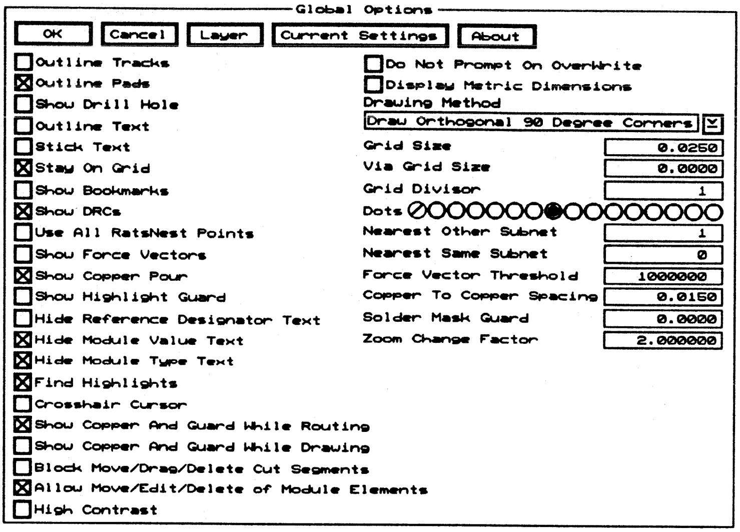
Global button

Appears on a number of dialog boxes.

Displays the **Global Options** dialog box.

Global Options dialog box

Use this dialog box to set the default preferences that determine how **Edit Layout** displays and maintains boards, modify object appearance and selectability, and designate cursor style and grid size.



Layer Displays the **Layer** dialog box.

Current Settings Displays the **Current Object Settings** dialog box.

About Displays the **About** dialog box.

Outline Tracks Enable to remove the color fill from all tracks on the board and display tracks as outlines.

Outline Pads Enable to remove the color fill from all module pads and test points on the board, and display module pads and test points as outlines.

Show Drill Hole Enable to display the drill holes for all pads. If the pad has a color fill, the drill hole appears as a black hole in the pad. If the pad is outlined, the drill hole appears as an outline with the same color as the pad outline.

Outline Text Enable to remove the color fill from all text on the board and display text as outlined segments.

Stick Text Enable to display all text as single lines.

If **Stick Text** is enabled, text characters are drawn with single lines, regardless of the setting of **Outline Text.**

Stay On Grid Determines whether or not object movement and placement is restricted to grid intersections. Enable **Stay On Grid** to constrain movement to grid spacing. Disable **Stay On Grid** to move or place objects off grid.

△ *NOTE: It is a good practice to enable **Stay On Grid**, especially when you capture or run a macro, unless you have a compelling reason to be off grid.*

Show Bookmarks Enable to display bookmarks.

Show DRCs Enable to display DRC markers. DRC markers are created by selecting **Spacing/DRC Check Whole Board.**

Use All RatsNest Points Enable to allow all net intersection points to be candidates for ratsnest endpoints. When this is disabled, the ratsnest will only display connections to pads, vias, and stub end points; the ratsnest will not display connections to wire intersections.

Show Force Vectors Enable to display module force vectors, which represent the mathematical sum of all the ratsnest vectors for a module. Force vectors are useful when you want to determine optimum module placement—use force vectors to help you identify long vectors and vectors that intersect.

A force vector has two components: a circle that shows the center of gravity of the module's pads, and a line that indicates the general direction the module should move to equalize the distance between connection points.

Use **Force Vector Threshold** to set the threshold for displaying module force vectors.

Show Copper Pour Enable to display the copper pour area of a fill zone. Disable **Show Copper Pour** to display copper zones as outlines.

Show Highlight Guard Enable to display the copper to copper spacing area surrounding highlighted tracks, pads, copper text, and copper fills when these objects are highlighted.

Hide Reference Designator Text Enable to make all module reference designator text invisible.

Hide Module Value Text Enable to make all module value text invisible.

Hide Module Type Text Enable to make all module type text invisible.

Find Highlights Enable to highlight a module or a net that is found with FIND.

Crosshair Cursor Enable to display the pointer as a cross or plus sign that extends the full length and width of the screen. Note that the pointer still displays as an arrow in a dialog box.

Show Copper And Guard While Routing Enable to display the copper-to-copper spacing area for a route while it is being manually routed on a copper layer.

Show Copper And Guard While Drawing Enable to display the copper-to-copper spacing area for all objects placed on copper layers.

Block Move/Drag/Delete Cut Segments Enable to move, drag, and delete portions of tracks that are separated from the rest of the tracks.

Allow Move/Edit/Delete of Module Elements Enable to move, edit, and delete module elements. This option is initially disabled for board files newly converted from OrCAD/PCB II.

High Contrast Enable to display all layer colors (and thus all objects, except those that are highlighted) as dark gray. Highlighted objects are displayed in the color of the layer they are placed on, rather than outlined in white.

Do Not Prompt On OverWrite Enable to suppress the prompt that asks if you want to overwrite a file with the same filename.

Display Metric Dimensions Enable to display metric values for all functions.

Drawing Method Select to display a list of drawing method options. The options apply to any item that is drawn on the board, such as outlines, zones, and routes.

Grid Size Use this entry box to set the grid spacing (the space between points on the grid). The allowed range is from 0.000100 in. (0.002540 mm) to 33.000000 in. (838.200000 mm). The grid size is affected by the grid divisor.

Via Grid Size Use this entry box to set the grid spacing that the autorouter uses to place vias. The value can normally be left at its default of 0.000000 in which case the via grid used by the autorouter will be the same as the routing grid. If your design requires vias to be placed on a different grid than the routing grid, set this value to a non-zero value. The via grid size is affected by the grid divisor. The allowed range is from the grid size to 33.000000 in. (838.200000 mm).

Grid Divisor Use this entry box to divide the **Grid Size** and **Via Grid Size** into smaller partitions. The allowed range is from 1 to 100.

For example, to set the grid to 8.333 mils, the grid size would be set to 0.025 and the grid divisor would be set to 3.0.

Dots Select a color radio button to set the color of the grid dots.

Nearest Other Subnet Use this entry box to set the number of nearest valid pad targets or stub ends on another subnet that display as a ratsnest during manual routing. The allowed range is from 0 to 64.

Nearest Same Subnet Use this entry box to set the number of nearest valid pad targets or stub ends on the same subnet that display as a ratsnest during manual routing. The allowed range is from 0 to 64.

△ *NOTE: A subnet is a single object or a group of connected objects that is not yet connected to the rest of the net.*

Force Vector Threshold Use this entry box to set the sensitivity threshold for displaying net force vectors. Only those nets with a pad count below the threshold are used in computing the force vector for a module. Use a smaller value to show the effect of short nets and eliminate large nets from the calculation.

If a module has no nets considered for force vectors, only the pins are shown. If a module has no pins, nothing is shown. The valid threshold range is from 1 to 1000000.

Copper To Copper Spacing Use this entry box to set the minimum amount of space allowed between objects on copper layers.

Enable **Show Copper And Guard While Routing** and **Show Copper And Guard While Drawing** to see the spacing.

Solder Mask Guard Use this entry box to set the amount of clearance between the edge of a pad or via and the edge of the solder mask.

△ *NOTE: The solder mask guard set in the Pad Stack Editor for each pad stack element is added to this clearance.*

Zoom Change Factor Use this entry box to set the value that the present zoom scale is divided by when the **ZOOM In** command is used, or is multiplied by when the **ZOOM Out** command is used.

Go To Editor command

Appears on a number of menus.

Displays the menu shown at right.

```
Pad Stack Editor
Via Stack Editor
Copper Tool Editor
Drill List Editor
Net Property Editor
```

GOTO FUNCTION command

Appears on a number of menus.

On the board editor main menu

Displays the menu shown at right.

```
Pad Stack Editor
Via Stack Editor
Copper Tool Editor
Drill List Editor
Net Property Editor
Library Editor
Autorouter
Netlist Loader
Printing and Plotting
Macro Maintenance
```

On the library editor main menu

Displays the menu shown at right.

```
Pad Stack Editor
Via Stack Editor
Copper Tool Editor
Drill List Editor
Board Editor
Module Selection
Macro Maintenance
```

HIGHLIGHT command

Appears on a number of menus.

Highlights an object and displays the same information about the object as when you select **INQUIRE**. You can highlight the following kinds of objects:

- Module pads, text, and outline segments
- Net segments, arcs, pads, and vias
- Outlines
- Pads and the net they are connected to
- Vias and the net they are connected to
- Zone segments and arcs

HIGHLIGHT shows the copper-to-copper spacing when **Show Highlight Guard** is enabled in the **Global Options** dialog box.

HIGHLIGHT displays objects in their designated layer color when **High Contrast** is enabled in the **Global Options** dialog box. Note that, if the background color of high-contrast mode (normally light gray) is the same as the color of any highlighted object, that object will not be visible in high-contrast mode.

To highlight an object, place the pointer the on the object and select **HIGHLIGHT**. To clear all highlights, place the pointer in an open area on the screen and select **HIGHLIGHT** again.

△ *NOTE: Highlights are cumulative. You may highlight any combination of objects and all will be displayed as highlighted until the highlight is cleared.*

Hole command

Appears on the **PLACE** menu.

Loads the current hole settings.

See also *Placing holes.*

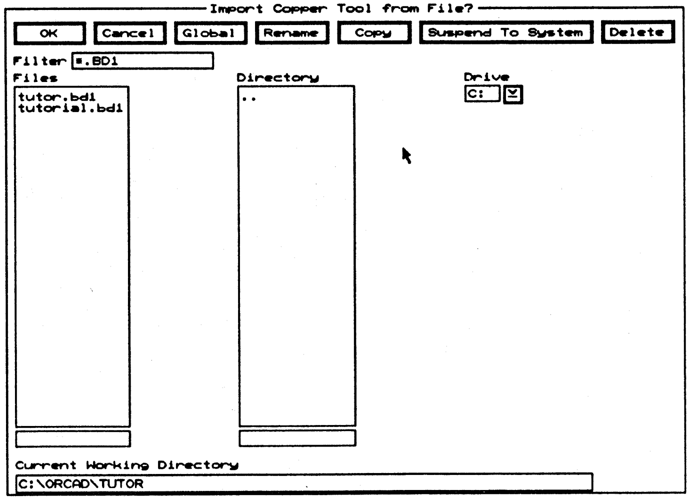
Import button

Appears on a number of dialog boxes.

Displays the appropriate **Import... from File** dialog box.

Import Copper Tool from File dialog box

Use this dialog box to import a copper tool from a file created in the **Export Copper Tool to File** dialog box.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

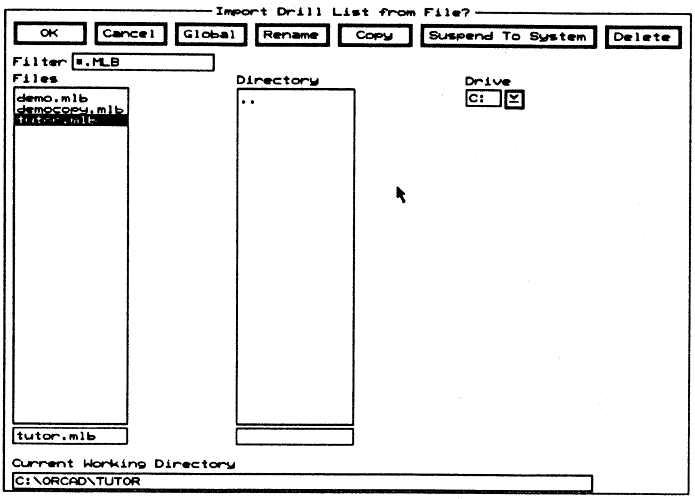
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Import Drill List from File dialog box

Use this dialog box to import a drill list from a file created in the **Export Drill List to File** dialog box.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

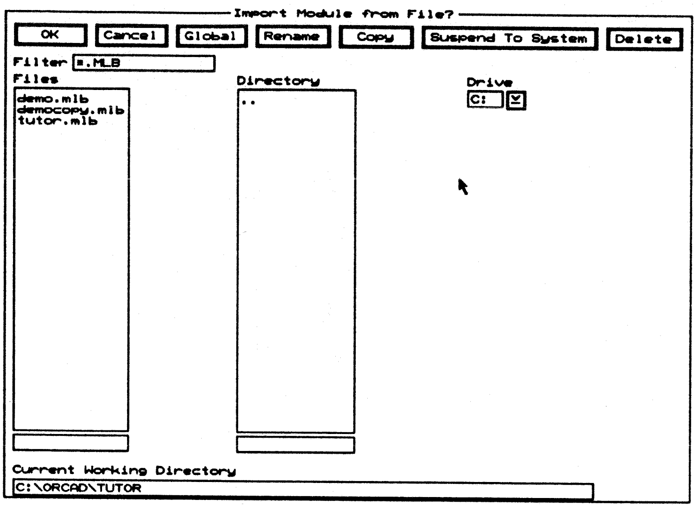
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Import Module from File dialog box

Use this dialog box to import a module from a file created in the **Export Module to File** dialog box.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

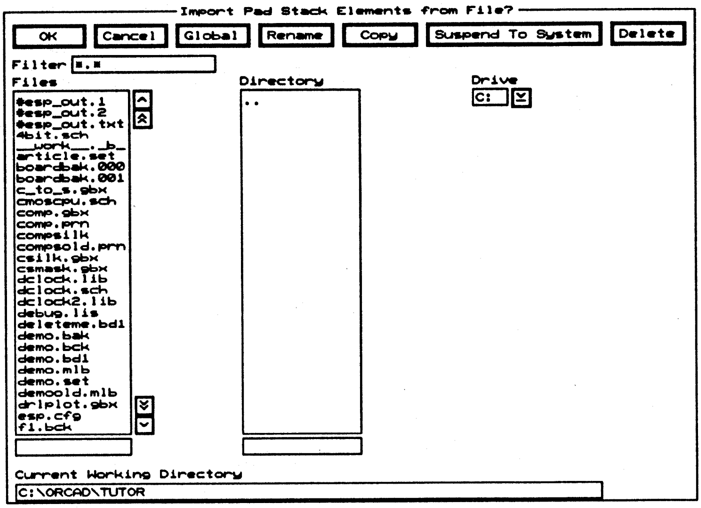
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Import Pad Stack Elements from File dialog box

Use this dialog box to import pad stack elements from a file created in the **Export Pad Stack Elements to File** dialog box.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

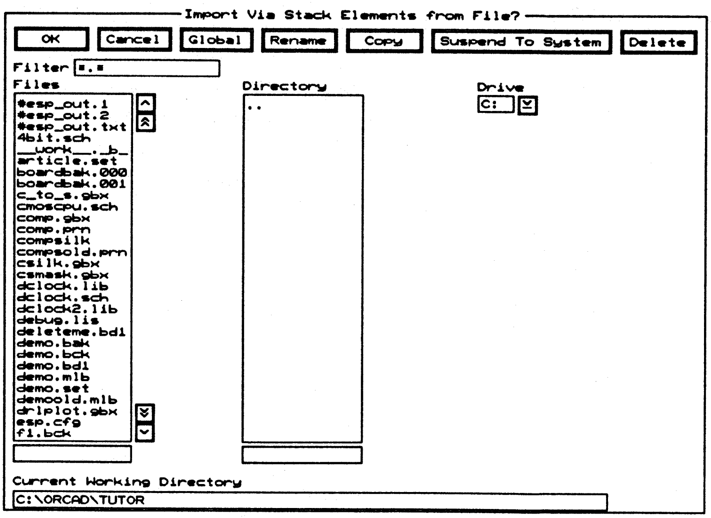
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Import Via Stack Elements from File dialog box

Use this dialog box to import via stack elements from a file created in the **Export Via Stack Elements to File** dialog box.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

In command

Appears on the **ZOOM** menu.

In increases the size of displayed objects and moves the current pointer position to the center of the screen. Use the **Zoom Change Factor** entry box in the **Global Options** dialog box to specify the factor by which the size of the displayed objects increases when you select **ZOOM In**.

Initialize Board File command

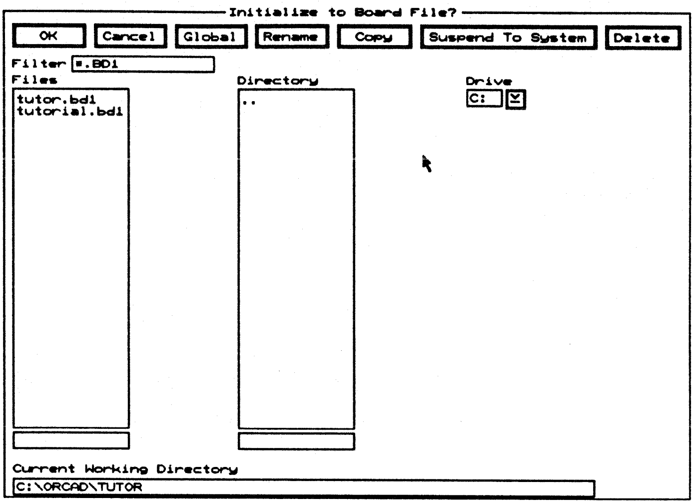
Appears on the board editor **QUIT** menu.

Displays the **Initialize to Board File** dialog box.

△ *NOTE: **Initialize Board File** does not save edits to the board file currently loaded. However, it asks if you want to save edits before initializing another board file. Make sure you update the board file with **QUIT Update Board File** before you select **QUIT Initialize Board File**.*

Initialize to Board File dialog box

Use this dialog box to load an existing board file or the template into **Edit Layout**.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Select a filename from the **Files** list box or create a new board file by entering a filename in the entry box below it.

If you enter the name of a file that does not exist and select **OK, Edit Layout** displays a **Notice** dialog box. Select **OK** to dismiss the dialog box. **Edit Layout** loads the template board file and then displays the board editor.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Initialize to Library command

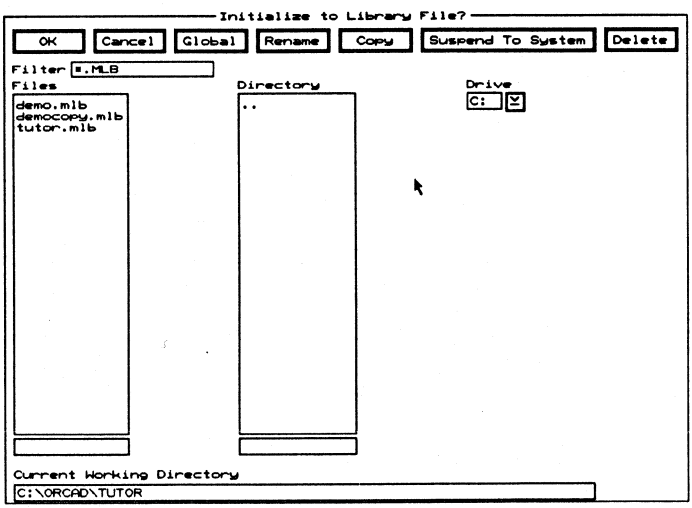
Appears on the library editor **QUIT** menu.

Displays the **Initialize to Library File** dialog box.

△ *NOTE: **Initialize to Library** does not save edits to the library file currently loaded, nor does it prompt you to save edits before initializing another library file. Make sure you update the library file with **QUIT Update Library File*** before *you select **QUIT Initialize to Library**.*

Initialize to Library File dialog box

Use this dialog box to load an existing library file or the template into **Edit layout**.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Select a filename from the **Files** list box or create a new library file by entering a filename in the entry box below it.

When you enter a new library file name in the entry box beneath the **Files** list box and select **OK, Edit Layout** displays a **Notice** dialog box. Select **OK**. The **Get Module** dialog box displays. Enter a module name in the entry box beneath the **Module Name** list box and select **OK**. **Edit Layout** displays the library editor. You can now create the library using the methods described in this manual.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

INQUIRE command

Appears on a number of menus.

Displays information about an object in the lower-right corner of the screen. Note that the information may take more room than is available on your screen.

Position the pointer on the desired object and select **INQUIRE**. The following table describes the information that displays for each object.

Object	*Information displayed*
Alignment target	Object type
Arc	Object type, copper tool name
Circle	Object type, radius
Comment text	Object type, text string
Dimension	Object type
Hole	Drill diameter, object type
Layer marker	Object type
Module text	Module name, value, type
Net	Netname, object type, copper tool name
Outline	Reference designator (if the outline is part of a module), object type, copper tool name
Pad	Reference designator, pad name, netname, object type, pad stack name
Test Point	Object type, netname, pad stack name
Via	Netname, object type, pad stack name
Zone	Zone type

Insert button

Appears on a number of dialog boxes.

Adds an item above the selected item in a list box.

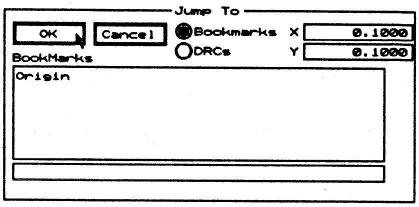
JUMP command

Appears on a number of menus.

Displays the **Jump To** dialog box.

Jump To dialog box

Use this dialog box to select a bookmark or DRC, or to specify board location by its X and Y coordinates.



See also *Jumping to bookmarks or DRCs* and *Jumping to specific board locations.*

Bookmarks Displays existing bookmarks in the list box.

DRCs Displays existing DRCs in the list box.

X Use to enter an X coordinate. When the **Jump To** dialog displays, the entry box shows the pointer's current X coordinate.

Y Use to enter a Y coordinate. When the **Jump To** dialog displays, the entry box shows the pointer's current Y coordinate.

The list box contains a list of bookmarks or DRCs, as specified by the selected radio buttons.

△ *NOTE: Use* ***Show Bookmarks*** *in the* ***Global Options*** *dialog box to show and hide bookmarks. Use* ***Show DRCs*** *in the* ***Global Options*** *dialog box to show and hide DRC markers.*

Jumping to bookmarks or DRCs

1. Display the **Jump To** dialog box.
2. Select a bookmark or DRC from the list box, or enter its name in the entry box directly beneath the list box.
3. Select **OK**. The pointer moves to the selected bookmark or DRC.

Jumping to specific board locations

1. Display the **Jump To** dialog box.
2. Enter the desired coordinates in the **X** and **Y** entry boxes.
3. Select **OK**. The pointer moves to the specified location.

Layer button

Appears on the **Global Options** dialog box.

Displays the **Layer** dialog box.

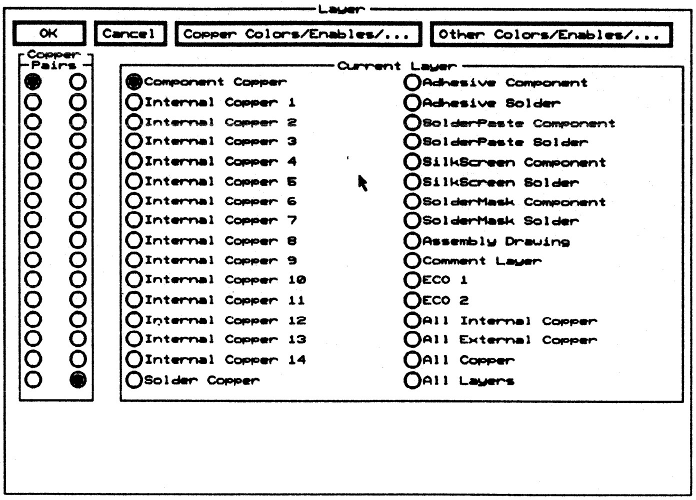
LAYER command

Appears on a number of menus.

Displays the **Layer** dialog box.

Layer dialog box

Use this dialog box to enable and disable copper layers for routing; and define layer names, layer colors, copper pairs, and the current layer.



Copper Colors/Enables/... Displays the **Copper Colors/Enables/...** dialog box.

Other Colors/Enables/... Displays the **Other Colors/Enables/...** dialog box.

Copper Pairs

Use to set which two copper layers the **/ OTHER** command switches between.

All routing is performed on these two copper layers. Placing a via while routing on one of the layers in **Copper Pairs** causes the router to automatically switch routing to the other layer.

Current Layer

Current Layer includes all the copper and unroutable layers. Use **Current Layer** to set which layer is the current layer. A copper layer must be enabled before you can use it as the current layer. **ECO 1** and **ECO 2**, which appear in the second column, are comment text layers.

Layer Marker command

Appears on the **PLACE** menu.

Loads the current layer marker settings.

See also *Placing layer markers.*

Leave Library Editor command

Appears on the library editor **QUIT** menu.

Displays the board editor.

Left hand mouse operation check box

In the **Processing Options** area of the **Configure Edit Layout** screen, select **Left hand mouse operation** to reverse the functions (<Enter> and <Esc>) of the mouse buttons. Select **Left hand mouse operation** again to return the mouse buttons to their default functions.

Library Editor command

Appears on the board editor **GO TO FUNCTION** menu.

Displays the library editor, if a library file has been initialized. Displays the **Initialize to Library File** dialog box, if a library file has not been initialized.

In the library editor, press <Enter> to display the library editor main menu shown at right.

Each of the commands shown is described in this chapter.

```
Block
Cut
Delete
Edit
Find
Go To Function
Highlight
Inquire
Jump
Track Delete
Layer
Move
Drag
Origin
Place
Quit
Route
Set
Selective
Undelete
Verbose Inquire
Window Zoom
X Show RatsNest
Zoom
= bookmark
+ layer
- layer
* layer
/ other
? conditions
% macro
```

Load button

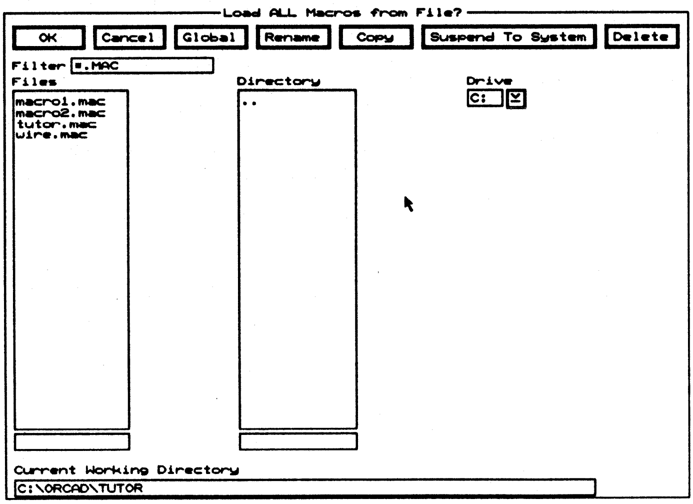
Appears on a number of dialog boxes.

Displays the appropriate **Load... from File** dialog box.

Load ALL Macros from File dialog box

Use this dialog box to load all the defined macros from a file.

See also *Loading macro files.*



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

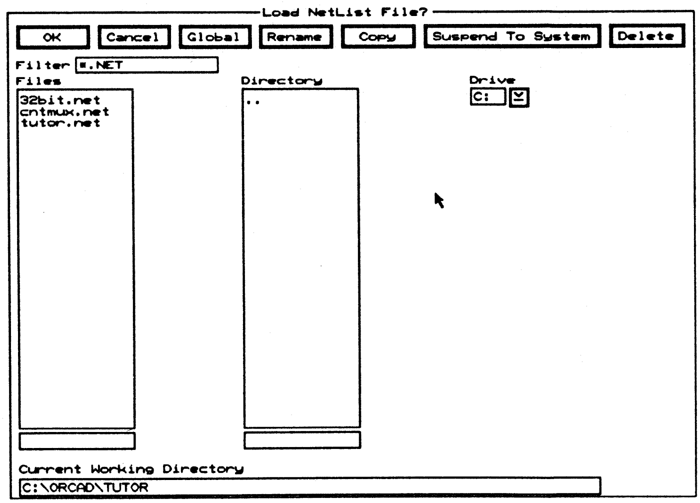
Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Load Netlist File dialog box

Use this dialog box to load an EDIF netlist file.

See also *Loading netlists*.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

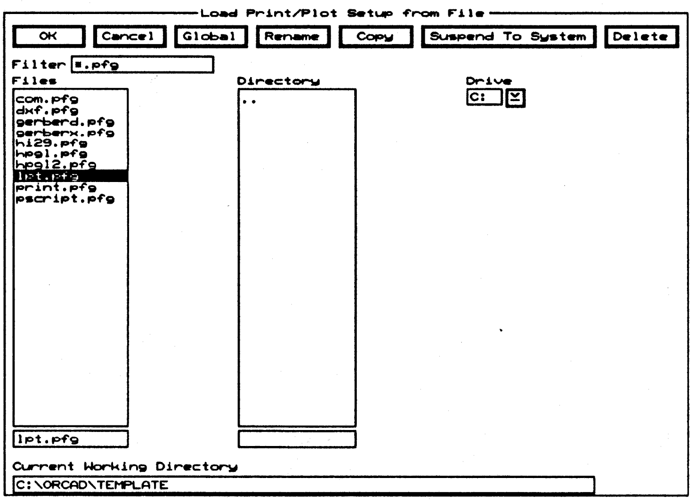
Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Load Print/Plot Setup from File dialog box

Use this dialog box to load a printing or plotting setup from a file.

See also *Printing and plotting*.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to create a new filename.

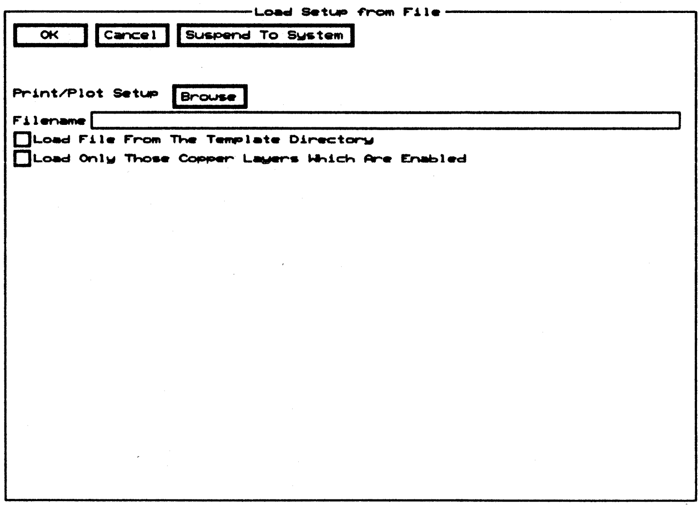
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Load Setup from File dialog box

Use this dialog box to specify the setup filename conditions.



Suspend To System Will only become active if your system includes an 80387 math coprocessor. This button clears the dialog box, and displays the system prompt.

Print/Plot Setup

Browse Displays the **Load Print/Plot Setup from File** dialog box.

Filename Use to enter the desired load setup filename, which may include the drive and path. If you specify the drive and path of the load setup file, and then select **Browse**, the **Output Filename** dialog box will display that drive, in that directory, and that filename.

Load File From The Template Directory Enable to load the setup file, specified in **Filename** entry box, from the TEMPLATE directory.

During the installation process, **PCB 386+ 2.00** places the following default printing and plotting configuration files in the TEMPLATE directory:

GERBERX.PFG	Gerber-274X configuration
GERBERD.PFG	Gerber-274D configuration
FIRE9XXX.PFG	Fire 9xxx configuration
HPGL.PFG	Hewlett-Packard HPGL configuration
HPGL2.PFG	Hewlett-Packard HPGL2 configuration
DXF.PFG	AutoCad DXF configuration
HI29.PFG	Houston Instruments HI29 configuration
PSCRIPT.PFG	Postscript configuration
PRINT.PFG	Print to file configuration
LPT.PFG	Print to LPT1 configuration
COM.PFG	Print to COM1 configuration

Load Only Those Copper Layers Which Are Enabled
Enable to map the file specified in **Filename** entry box directly into something that matches what you've got loaded in memory.

For example, you have a two-layer board in memory, a component copper layer and a solder copper layer. Your print/plot setup files have definitions for those two layers plus others. When **PCB 386+ 2.00** loads the copy in, it removes the definitions of the layers that are not applicable.

With this check box enabled, **PCB 386+ 2.00** will also automatically remove drill plots that are not required for the board in memory.

Finally, because layer planes are negative images, it is not a good idea to plot their pads, vias, test points, and nets. With this check box enabled, **PCB 386+ 2.00** will check to see if you have any layer planes defined, and if you do, it will then determine if the .PFG file just loaded maps to these layer planes. If this is the case, **PCB 386+ 2.00** will dynamically turn off pads, vias, test points, and nets on any plot of a layer plane.

△ *NOTE: Typically, you will enable both check boxes. However, you may wish to load the TEMPLATE directory files in order to make changes to them. In that case, enable only* ***Load File From The Template Directory.***

Loading macro files

1. Select **GO TO FUNCTION Macro Maintenance,** and then select **Load.** The **Load ALL Macros from File** dialog box displays.
2. Select a filename from the **Files** list box or enter a name in the entry box below it. You can also select a different directory or drive from the corresponding list box.
3. Select **OK** to load the selected file. The **Load ALL Macros from File** dialog box closes and the **Macro Maintenance** dialog box displays. Any macros in the selected file are loaded into memory, and their names appear in the **Defined Macros** entry box.

Loading netlists

Before you load a netlist, you define an area to load the modules into and specify the netlist filename. When a netlist is loaded, the modules assigned to the parts on the schematic are loaded into the board layout, and nets are assigned to the appropriate pads on the modules.

1. Select **GO TO FUNCTION Netlist Loader.**
2. Move the pointer to one corner of the area where you want the netlist to be loaded, and then select **Block.**
3. Move the pointer to the opposite corner of the area, and then select **Block End.** The **Load Netlist File** dialog box displays.
4. Select the netlist file to be loaded from the **Files** list box, and then select **OK.** The **Netlist Load Options** dialog box displays.
5. Set the netlist load options as desired, and then select **OK.** The netlist loads and the modules display in the area defined by the block.

The netlist loader spaces modules evenly inside the area defined by the block boundary. If the block is small, modules may overlap.

Any net which has no pads or test points is deleted when you load a netlist. Also, if the netlist loader encounters an error, any remaining netlists are ignored and the resulting design should be considered invalid.

See also *Netlist considerations*.

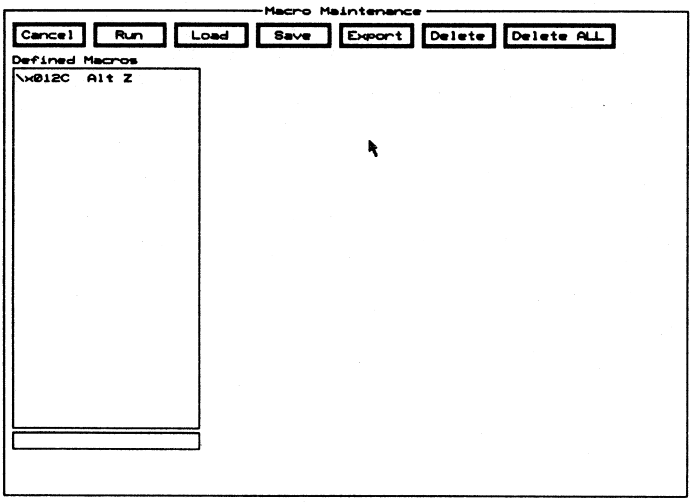
Macro Maintenance command

Appears on the **GO TO FUNCTION** menu.

Displays the **Macro Maintenance** dialog box.

Macro Maintenance dialog box

Use this dialog box to run, load, save, export, or delete a macro.



Run Executes the selected macro.

Load Displays the **Load ALL Macros from File** dialog box.

Save Displays the **Save ALL Macros to File** dialog box.

Export Displays the **Export 'macroName' Macro to File** dialog box.

Delete Removes the selected macro from the **Defined Macros** list box.

Delete ALL Removes all macros from the **Defined Macros** list box.

Defined Macros Contains a list of all existing macros.

Macros

Macros in **Edit Layout** capture data about *relative events*—any sequence of keystrokes and pointer movements. This means that a macro executes its commands relative to the pointer's current location, rather than from its location when the macro was created.

For example, a graphic object is placed on the board at a given distance from the pointer's starting location when you capture the macro. When you run the macro, though, the object is placed at the same distance from the pointer's *current* starting location.

Note that your macros run faster if you record commands as keystrokes, rather than as menu selections, because the macro doesn't have to display menus as it plays back.

See also *Loading macro files.*

△ *NOTE: It's good practice to enable **Stay On Grid** in the **Global Options** dialog box before you capture or run a macro.*

Assigning a key or key combination

When you select **% MACRO**, the **Press Macro Capture Key** dialog box displays. Press the key or key combination you want assigned to the macro, and then press <Enter>.

△ *NOTES: If you press an invalid key or combination, it does not display in the **Press Macro Capture Key** dialog box. Select a valid key or combination, and then press <Enter>.*

If the last entry before the end of the macro recording session is the macro capture key, the macro will automatically repeat when played.

When you enter a valid key or combination, the message "Macro Capture" displays in the lower-right corner of the screen, reminding you that you are defining a macro. Any commands you execute while "Macro Capture" displays, including pointer movements and selections, are added to the list of commands stored in the macro.

Valid keys and combinations

You can assign macros to many keys and key combinations. Single keys that run macros include the function keys (<F1> through <F12>) and the <Insert> key on the main keyboard (*not* the <Ins> on the numeric keypad).

The following key combinations use the keys on the main keyboard, *not* on the numeric keypad:

<Alt> + alphabet keys
<Alt> + number keys
<Alt> + function keys
<Alt> <Tab>
<Alt> <Esc>
<Alt> <Insert>
<Alt> <Delete>
<Alt> <Home>
<Alt> <End>
<Alt> <Page Up>
<Alt> <Page Down>
<Alt> <,> (*comma*)
<Alt> <.> (*period*)
<Alt> </> (*slash*)
<Alt> <`> (*back apostrophe*)

<Alt> <–> (*minus sign*)
<Alt> <=> (*equal sign*)
<Alt> <\> (*backslash*)

<Ctrl> + function keys
<Ctrl> <Tab>
<Ctrl> <Insert>
<Ctrl> <End>
<Ctrl> <Page Up>
<Ctrl> <Page Down>

<Shift> + function keys
<Shift> <Tab>

Manual routing

Some board designs specify that connections between pads have tracks of a specific length or shape. You must place these tracks using manual routing techniques.

PCB 386+ 2.00 provides you with the following new manual routing features:

- On-line check for proper net connectivity. You cannot inadvertently connect pads in different nets.
- Rubberbanding of nets.
- Vias can be moved with rubberbanded nets dragged on all levels.
- Automatic net segment clean up.
- Connection points are automatically created for nets drawn through pads with the same net assignment.
- Place pointer on a ratsnest, select **ROUTE Attach**, and the ratsnest becomes a rubberbanded trace.

Highlighting a net

Follow these steps to more easily identify all module pads associated with the net to be routed:

1. In the **Global Options** dialog box, make sure **Find Highlights** is enabled.
2. Select **FIND** and enter a question mark (?) in the **Find** entry box. The **Find** dialog box displays.
3. Select **Netnames Only**, and then select the desired net from the **Netnames** list box.
4. Select **OK**. All pads with the selected netname are highlighted.

Setting conditions

1. In the **Global Options** dialog box, make sure **Show Copper And Guard While Routing** is enabled, and select the desired drawing method.
2. Select **Layer**. The **Layer** dialog box displays.
3. Select the desired layer, and then select **Copper Colors/Enables/...** . The **Copper Colors/Enables/...** dialog box displays.
4. Enable **Layer Enabled** for the desired layer, and then select **OK**. The **Layer** dialog box displays.
5. Select **OK**. The **Global Options** dialog box displays.
6. Select **OK**. The board editor displays.

Routing the track

1. Place the pointer in the center of a highlighted pad.
2. Select **ROUTE Begin** and move the pointer to the desired highlighted pad.
3. Select **End**. The highlight guard disappears and the message "*netName*: *n* Unconnected" displays near the lower-right corner of the screen.
4. Repeat steps 1–3 until the message "*netName*: Complete" displays near the lower-right corner of the screen.
5. To remove the highlight from the pads, move the pointer to an empty part of the board editor and select **HIGHLIGHT**.

See also *Chapter 7: Routing the TUTOR board* in the *PC Board Layout Tools 386+ User's Guide*.

Manufacturing a board

1. Load your board into **PCB 386+ 2.00**.
2. Select **GO TO FUNCTION**, and then select **Printing and Plotting**. The **Printing and Plotting** dialog box displays.
3. Select **Load**. The **Load Setup from File** dialog box displays.
4. Enable **Load File From The Template Directory** and **Load Only Those Copper Layers Which Are Enabled**.
5. Select **Browse**. The **Load Print/Plot Setup from File** dialog box displays.
6. Select the desired file from the **Files** list box.
7. Select **OK**. The **Load Setup from File** dialog box displays.
8. Select **OK**. The **Printing and Plotting** dialog box displays.
9. Select **Begin All**. As the pages are being plotted, messages will display in the upper-left corner of the screen.
10. Select **Save**. The **Save Setup to File** dialog box displays.
11. Select **OK**. The **Printing and Plotting** dialog box displays.
12. Select **OK**. The board editor displays.
13. Select **QUIT**, and **Abandon Program**.

At the system prompt

1. Go to the design directory that contains your boardfile.
2. Type `makencd.bat` and press <Enter>.
3. Type `pkzip tofab.zip @pp_files.lst` and press <Enter>.

△ *NOTES: This assumes that you have PKZIP and that your path to PKZIP is in your path statement.*

PKZIP can be obtained from the OrCAD BBS by downloading the PKZ204G.EXE file.

4. Copy TOFAB.ZIP to a floppy.
5. Send the floppy to your fab house.

Mirroring and flipping objects

To *mirror* an object is to reverse its representation—the object stays on the same layer. To *flip* an object is to move it to the other side of the board—the object changes layers. You can mirror most objects along the X and Y axes, and you can flip any object to the other side.

When you mirror an object along the X axis, its image flips from left to right. When you mirror an object along the Y axis, its image flips from top to bottom.

When you mirror a block of objects, you reverse the representation of the entire block, rather than reversing in place the representation of each object in the block.

When you flip a module to the other side of the board, its associated text is not reversed in your view of the layout. This makes the text easier to read. When you print or plot the board, though, the flipped module's text can be mirrored.

Mirroring an object along both the X and Y axes has the same effect as rotating the object by 180 degrees, with one exception. Text associated with a rotated object rotates along with the object, whereas text associated with a mirrored object retains its normal orientation.

△ *NOTE: In the board editor, you cannot mirror a module or a block that contains a module. However, if **Allow Move/Edit/Delete of Module Elements** is enabled, then all selected module elements can be mirrored.*

Mirroring

1. Use **BLOCK** and **Block End** to draw a block boundary around the objects to be mirrored.
2. Select **Move Block**, and then select **Set**. The **Set Block Parameters** dialog box displays.
3. Enable **Mirror X**, **Mirror Y**, or both, and then select **OK**.
4. Move the image of the mirrored block to the desired location, and then select **Place**.

Flipping

1. In the **Global Options** dialog box, make sure **Allow Move/Edit/Delete of Module Elements** is disabled and **Stay On Grid** is enabled.
2. Select the objects to be flipped to the other side:
   - ❖ In the library editor, place the pointer on a module label or pad and select **MOVE Set**.
   - ❖ In the board editor:
     a. Use **BLOCK** and **Block End** to draw a block boundary around them.
     b. Select **Move Block**, and then select **Set**.

   The **Set Block Parameters** dialog box displays.
3. Enable **Flip to other side of board**, and then select **OK**.
4. Move the image of the flipped module or block to the desired location, and then select **Place**. The selected objects are placed on the Silkscreen Solder layer.

△ NOTE: *Labels and label placeholders flip to the other side of the board, but do not display as reversed text. When you print or plot the module, you specify whether you want the text printed or plotted as mirrored text.*

Module command

Appears on a number of menus.

See also *Placing modules.*

On the board editor PLACE menu

Displays the **Place Module** dialog box.

On the Delete Block menu

Deletes only the modules enclosed or intersected by the block boundary.

Module libraries

The file PCBLIB.LST contains a list of the libraries shipped with **PCB 386+ 2.00** and a brief description of the modules they contain. Use **View Reference** to display PCBLIB.LST

Module Properties button

Appears on the **Edit Pad** dialog box.

Displays the **Edit Module Properties** dialog box.

Module Selection command

Appears on the library editor **GO TO FUNCTION** menu.

Displays the **Get Module** dialog box.

Module Snap Block command

Appears on the **Block End** menu.

Snaps the first pad of each module that is enclosed by the block boundary to the grid selected in the **Global Options** dialog box. Note that all of the module's objects must be enclosed within the block boundary, and that the block layer must contain the module object layer.

MOVE command

Appears on a number of menus.

Moves an object to a different location. Because **MOVE** does not maintain net connections, objects can be mirrored or flipped to the other side of the board. **DRAG** and **Drag Block** do maintain net connections, and objects moved using these methods cannot be mirrored or flipped.

See also *Moving objects* and *Moving modules.*

Move Block command

Appears on the **Block End** menu.

Moves objects enclosed or intersected by the block boundary. Modules are an exception; see *Moving modules* for more information.

Select **Set** to display the **Set Block Parameters** dialog box. The check boxes beneath **Objects Affected** control which objects are selected: enabled objects on active layers are selected; disabled objects and enabled objects on inactive layers are not selected.

Use the pointer to move the bound block and then select **Place** to place the block.

Moving modules

Before you move a module, check the following conditions:

- If the module components are on more than one layer, all layers must be enabled.
- **Allow Move/Edit/Delete of Module Elements** in the **Global Options** dialog box must be disabled, so the objects that make up the module cannot be moved individually.
- **Stay On Grid** in the **Global Options** dialog box must be enabled, if you want your movements constrained to the grid.

See also *Moving objects* and *Moving objects within a module.*

△ *NOTE: If the module is being moved as part of a block, all of the module's objects must be enclosed within the block boundary. The block layer must contain the module object layer. Note that **Move Block** does not maintain connections to net segments and arcs outside of the block boundary. To maintain connections, use **Drag Block**.*

Selecting a module

1. Select **PLACE Module.** The **Place Module** dialog box displays.
2. Select the module to be placed from the **Module Name** list box, and then select **OK.** The pointer jumps to the module and the module is ready to move.

Positioning the module

You can position a module by specifying its new location and angle in the **Set Block Parameters** dialog box (coordinate placement) or by rotating and positioning the module manually (dynamic placement).

❖ Coordinate placement:

1. Select **Set.** The **Set Block Parameters** dialog box displays.
2. Enter the desired angle in the **Angle** entry box.
3. Enter the desired coordinates in the **Block X** and **Block Y** entry boxes, and select **OK.** The module moves to the specified coordinates.

❖ Dynamic placement:

1. Select **> Rotate Clockwise** or **< Rotate Counter Clockwise** to rotate the module to the desired angle. As you rotate the module, the rotation angle displays in the lower-right corner of the screen.
2. Using the mouse and arrow keys, move the module to the new location.

Placing the module

1. Select **Place.** The module is placed and you return to the **Place Module** dialog box.
2. Select **Cancel** to close the dialog box.

Moving objects

Position the pointer on the object to be moved and select **MOVE**. As you move the pointer, an outline of the object moves with it so you have a visual reference of the object's size. Note that the object does not actually move until you select **Place**.

When **Allow Move/Edit/Delete of Module Elements** is enabled in the **Global Options** dialog box, you can move specific elements of a module (for example, just pads) the same way you would move other objects.

When **Allow Move/Edit/Delete of Module Elements** is disabled, you can move a module by placing the pointer on any of its elements and selecting **MOVE**.

If the object to be moved is off grid, make sure **Stay On Grid** is disabled in the **Global Options** dialog box; otherwise, you cannot select the object.

See also *Moving objects within a module* and *Moving modules*.

Moving elements within a module

Follow these steps to move individual elements in a module.

1. In the **Global Options** dialog box, make sure **Allow Move/Edit/Delete of Module Elements** is enabled.
2. Select * **LAYER** to enable all layers.
3. Place the pointer on the element to be moved, and select **MOVE**.
4. Move the selected element to the desired location, and select **Place**.

See also *Moving objects* and *Moving modules*.

Net Properties button

Appears on the **Edit Net Segment** dialog box.

Displays the **Edit Net Properties** dialog box.

Net Property Editor command

Appears on the board editor **GO TO FUNCTION** menu.

Displays the **Edit Net Properties** dialog box.

Netlist considerations

When you create a netlist in **Schematic Design Tools 386+** for use with **PCB 386+ 2.00**, keep the following points in mind:

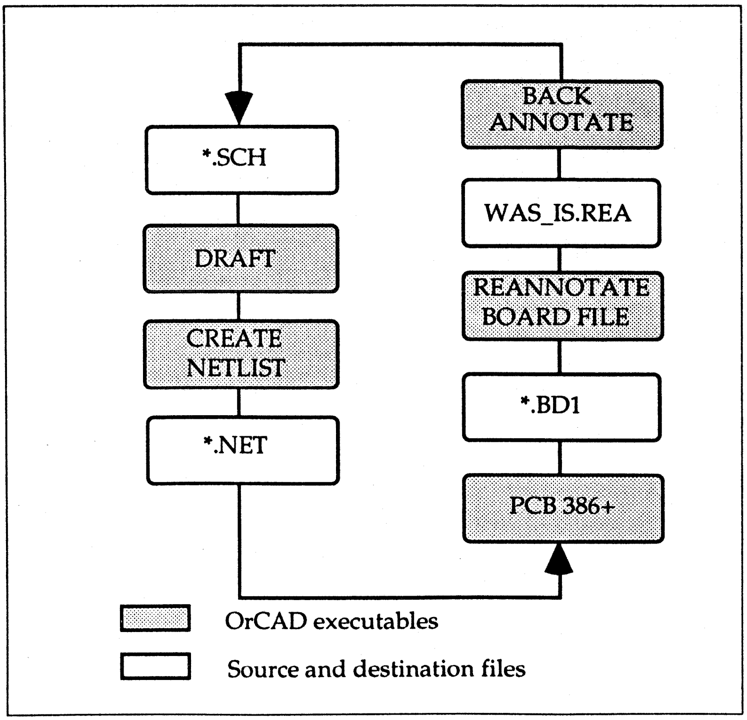
- Make sure module names do not contain spaces.
- Configure IFORM to produce the flat EDIF netlist format by selecting **FEDIF.EXE**.
- Configure IFORM to output pin numbers, rather than pin names.

See *Chapter 3: Transferring from schematic to layout* in the *PC Board Layout Tools 386+ User's Guide* for a complete description of the process. See the *Schematic Design Tools Reference Guide* for information about creating netlists and configuring IFORM.

Netlist generation and annotation processes

In concurrent design work, where a printed circuit board is being developed in parallel with the schematic logic, it is often necessary to update the annotation on both the board and schematic files.

Figure 4-1 illustrates how OrCAD executable, source, and destination files interact.



OrCAD EDA flowchart.

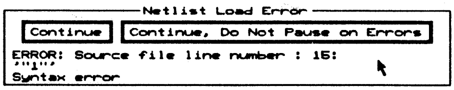
The design information is stored in the schematic files (*.SCH), then a netlist is produced (**Create Netlist**) and the resulting netlist (*.NET) contains most of the information necessary to design the board. If parts are added, deleted, or edited, or if the electrical connectivity is changed in the schematic files (**Draft**), a new netlist must be created and loaded into the board file (*.BD1).

Forward annotation of a board is performed automatically when **GO TO FUNCTION Netlist Loader** is selected. If the module reference designators are changed on the board by selecting **Reannotate Board File**, they should be back annotated onto the schematic files by selecting **Back Annotate.**

For more detailed information, see the *Schematic Design Tools Reference Guide.*

Netlist Load Error dialog box

This dialog box displays when **Edit Layout** encounters an error while parsing the netlist. The message at the bottom of the dialog box shows the error that occurred and any additional diagnostic information that is available.



Continue Select to have **Edit Layout** continue parsing the netlist until it encounters another error.

Continue, Do Not Pause on Errors Select to have **Edit Layout** finish parsing the netlist, record any errors it encounters, and display the number of errors found after the netlist is parsed.

The following netlist load errors are defined.

Duplicate time stamp

`A module in the netlist has the same time stamp as another module in the netlist or a module in the board file. The last module with that time stamp found in the netlist replaces any other modules with that time stamp.`

`Replaced module reference designator -` *name*

Cause

Older versions of **Draft** (**SDT Release IV** and versions 1.00 and 1.01 of **SDT 386+**) created duplicate time stamps during **BLOCK Save** and **BLOCK Get** operations.

SDT Release IV solution

Use **BLOCK Export** to store the schematic in a temporary file, then use **BLOCK Import** to load the temporary file onto a new schematic.

SDT 386+ v1.10 solution

For a single-sheet design or a design that consists of just a few sheets, use **BLOCK Export** to store the schematic in a temporary file, then use **BLOCK Import** to load the temporary file onto a new schematic.

For designs consisting of numerous sheets, run **Complex to Simple** on the design (see **Design Management Tools**). Then, run **Create Netlist** on the new "simplified" design.

Module ID already assigned

`A module ID (either a reference designator or a time stamp) in the netlist is already assigned to a non-module object in the board file. The module is described below.`

`Module reference designator -` *name*

`Module part value -` *name*

`Module name -` *name*

`Module time stamp -` *name*

Cause

This is probably caused by customizing the names of objects in the board file. Copper tools, pad stacks, and via stacks are the most likely objects to cause this conflict.

Solution

To locate the module ID string that has already been assigned to a non-module object in the board file, select **FIND** and enter **?** in the **Find** entry box. The **Find** dialog box displays. Select **All Strings**. Review the **All Strings** list and find a string that matches the module ID. Change the name of the non-module object.

You can also use **Module Report** to obtain an ASCII report of the board file. Search the file for the duplicate names.

Reference designator already assigned

`The reference designator of the module described below is already assigned to a non-module object in the board file.`

`Module reference designator -` ***name***

`Module part value -` ***name***

`Module name -` ***name***

`Module time stamp -` ***name***

Cause

This is probably caused by customizing the names of objects in the board file. Copper tools, pad stacks, and via stacks are the most likely objects to cause this conflict.

Solution

To locate the module ID string that has already been assigned to a non-module object in the board file, select **FIND** and enter **?** in the **Find** entry box. The **Find** dialog box displays. Select **All Strings**. Review the **All Strings** list and find a string that matches the module ID. Change the name of the non-module object.

You can also use **Module Report** to obtain an ASCII report of the board file. Search the file for the duplicate names.

Net refers to non-module object

`A net references a module reference designator, but that reference designator is already assigned to a non-module object.`

`Netname -` *name*

`Module reference designator -` *name*

`Module pad name -` *name*

Cause

The netlist has a pin and a module electrically connected. The netlist loader can find the defined reference designator string, but the string is assigned to something other than a module.

This is probably caused by customizing the names of objects in the board file.

Solution

To locate the module ID string that has already been assigned to a non-module object in the board file, select **FIND** and enter **?** in the **Find** entry box. The **Find** dialog box displays. Select **All Strings**. Review the **All Strings** list and find a string that matches the module ID. Change the name of the non-module object.

You can also use **Module Report** to obtain an ASCII report of the board file. Search the file for the duplicate names.

Netname already assigned

`The netname described below is already assigned to a non-net object in the board file.`

`Netname -` *name*

Cause

This is probably caused by customizing the names of objects in the board file. Copper tools, pad stacks, and via stacks are the mostly likely objects to cause this conflict.

Solution

To locate the module ID string that has already been assigned to a non-module object in the board file, select **FIND** and enter **?** in the **Find** entry box. The **Find** dialog box displays. Select **All Strings**. Review the **All Strings** list and find a string that matches the module ID. Change the name of the non-module object.

You can also use **Module Report** to obtain an ASCII report of the board file. Search the file for the duplicate names.

Module not found

`The module described below cannot be found in any configured library.`

`Module reference designator -` *name*

`Module part value -` *name*

`Module name -` *name*

`Module time stamp -` *name*

Cause

The library in which the module resides has not been configured.

Solution

Configure the correct library. Use **View Reference** to display the file MODULE.LST, which contains a list of all OrCAD-provided module libraries and their contents, and determine which library to configure.

Reference designator not found

`A net references a module reference designator that cannot be found.`

`Netname -` *name*

`Module reference designator -` *name*

`Module pad name -` *name*

Cause

The netlist has a pin and a module electrically connected, but the netlist loader cannot find a module with the specified reference designator. This is caused by a previous netlist load error that prevented the netlist loader from loading the specified module.

Solution

Correct previous netlist load error and then reload the netlist.

Pad name not found

`A net references a module pad name that cannot be found.`

`Netname -` *name*

`Module reference designator -` *name*

`Module pad name -` *name*

Causes

Pin-to-pad references are "out of sync" for one of the following reasons:

- The wrong device library file was used to load parts into the schematic. The SDT device library file DEVICE.LIB was used instead of the SDT-for-PCB device library file PCBDEV.LIB.
- **Output pin names (instead of pin numbers)** is enabled in the **Format Specific Options** area of the **Configure Netlist Format** screen.
- The netlist file does not contain the external library definition.

Solution

The pin number or pin name in the schematic has to match the pad name of the PCB 386+ module. The preferred solution is to change the pad name of the specified module. Change the pad name in the module library, not on the board, by using the library editor's Edit Pad dialog box. The pin number or pin name can also be changed in the schematic.

Note that if Netname displays the message "Pad Name not found in Cell definition" do not enable **Do not create the 'external' library declaration** in the **Format Specific Options** area of the **Configure Netlist Format** screen.

Unable to add module (insufficient memory)

`There is insufficient virtual memory. Unable to add new module.`

`Additional bytes required -` *number*

Cause

Insufficient virtual memory.

Solution

Increase your system's virtual memory by the number of bytes displayed.

Use the DOS CHKDSK command to determine how much free disk space is available. Free more disk space on the drive containing the virtual memory swap file.

The **Virtual Memory Options** area of the **Configure PC Board Layout** screen shows where the virtual memory swap file is stored.

See *Technical Note #46: Memory considerations for 386+ programs* for a description of the Phar Lap™ memory extender.

Unable to create net (insufficient memory)

`There is insufficient virtual memory. Unable to create the net described below.`

`Netname -` *name*

Cause

Insufficient virtual memory.

Solution

Increase your system's virtual memory.

Use the DOS CHKDSK command to determine how much free disk space is available. Free more disk space on the drive containing the virtual memory swap file.

The **Virtual Memory Options** area of the **Configure PC Board Layout** screen shows where the virtual memory swap file is stored.

See *Technical Note #46: Memory considerations for 386+ programs* for a description of the Phar Lap™ memory extender.

EDIF error

`While processing the EDIF netlist, an error has occurred.`

`Line number -` *lineNumber*

`'`*lineText*`'`

description

Causes

Either a syntax error has occurred, or there is insufficient virtual memory to continue netlist processing.

Solutions

If the *description* field displays a reference to a syntax error, an unexpected error has occurred during netlist generation. If you are using an OrCAD-generated EDIF 2 0 0 netlist, call OrCAD technical support.

If the *description* field displays a reference to insufficient virtual memory, increase your system's virtual memory.

Use the DOS CHKDSK command to determine how much free disk space is available. Free more disk space on the drive containing the virtual memory swap file.

The **Virtual Memory Options** area of the **Configure PC Board Layout** screen shows where the virtual memory swap file is stored.

See *Technical Note #46: Memory considerations for 386+ programs* for a description of the Phar Lap™ memory extender.

Mismatched parentheses

`Parentheses in the EDIF netlist are mismatched.`

`Excess opening parentheses found -` *number*

or

`Parentheses in the EDIF netlist are mismatched.`

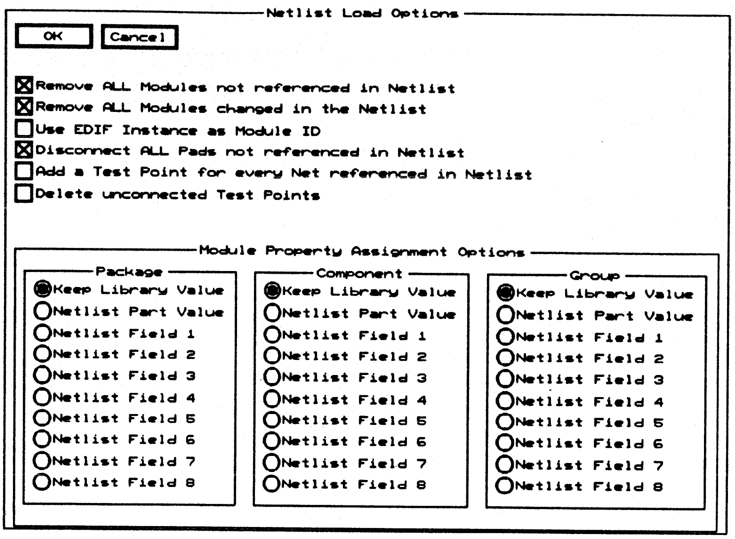
`Excess closing parentheses found -` *number*

Cause An unexpected error has occurred during netlist generation.

Solution If you are using an OrCAD-generated EDIF 2 0 0 netlist, call OrCAD technical support.

Netlist Load Options dialog box

Use this dialog box to specify conditions for netlist loading.



See also *Loading netlists.*

Check boxes

Remove ALL Modules not referenced in Netlist Enable if any modules on the board do not appear in the netlist. *This option is recommended for forward annotation.*

Remove ALL Modules changed in the Netlist Enable if you change the packaging on the schematic. This removes the old instances and loads the new packaging.

Use EDIF Instance as Module ID Enable to use the part's reference designator, instead of the time stamp, as the module's ID.

△ *NOTE: Once you enable this option on a design,* ***Edit Layout*** *no longer uses the time stamps and, for that design,* ***Use EDIF Instance as Module ID*** *must always be enabled.*

Disconnect ALL Pads not referenced in Netlist Enable to prevent all pads not referenced in the netlist from receiving a net assignment. If this option is disabled, such pads retain their old values.

△ *NOTE: This option is highly recommended for forward annotation.*

Add a Test Point for every Net referenced in Netlist Enable to add a test point for every net that appears in the netlist.

Delete unconnected Test Points Enable to remove all test points not connected to a net.

Module Property Assignment Options area

Use this area either to keep the library values or to select a field value created by **Configure Schematic Design Tools** and have **Netlist Loader** place this value in the **Package, Component,** and **Group** fields.

See also *Edit Other Module Properties dialog box.*

Netlist Loader command

Appears on the board editor **GO TO FUNCTION** menu.

Loads a netlist file into **Edit Layout's** board editor.

New button

Appears on the **Place Module** dialog box.

Displays the **New Module** dialog box.

New command

Appears on a number of menus.

Completes the object being drawn and leaves **Edit Layout** in the drawing mode. Select **Begin** to start drawing another object of the same type.

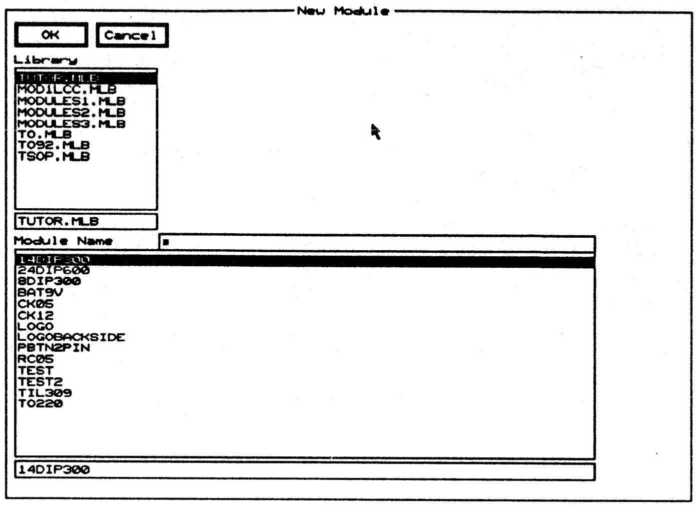
New Module command

Appears on the board editor **PLACE** menu.

Displays the **New Module** dialog box.

New Module dialog box

Use this dialog box to get a module from any current module library and place it in the **Module Name** list box on the **Place Module** dialog box.



Library Contains a list of module libraries. Note that if you have not configured any libraries in the **Configure PC Board Layout** screen's **Library Options** area, this list box contains all files in the library path. If libraries were configured, this list box contains only the names of those libraries.

Module Name Contains a list of the modules in the selected library.

Use any combination of wildcards (* or ?) and other characters in the entry box directly above the **Module Name** list box to restrict the list of modules shown.

No Autoroute Zone command

Appears on the **PLACE** menu.

Loads the current zone segment settings.

See also *Autoroute zones* and *Placing zones*.

No Fill Zone command

Appears on the **PLACE** menu.

Loads the current zone segment settings.

See also *Fill Zone command* and *Placing zones*.

No Through Zone command

Appears on the **PLACE** menu.

Loads the current zone segment settings.

A *no-through* zone defines an area in which vias cannot be created. Like autoroute and no-autoroute zones, no-through zones control the autorouter.

See also *Placing zones*.

Notice dialog boxes

A number of **Notice** dialog boxes may appear during the execution of **Edit Layout**. They provide specific information about any encountered errors. Select **OK** to dismiss these dialog boxes.

OK button

Appears on a number of dialog boxes.

Closes the dialog box and incorporates any changes.

OK to All button

Appears on a number of the **?** dialog boxes.

Prevents the dialog box from displaying during any subsequent attempts to autoroute the board.

ORIGIN command

Appears on a number of menus.

Sets the coordinates of the pointer's current position to (0.0000", 0.0000") and places the **Origin** bookmark at that location. To see the bookmark, enable **Show Bookmarks** in the **Global Options** dialog box.

You cannot delete the **Origin** bookmark. To reset it to the upper-left corner of the work space, move the pointer to that location and select **ORIGIN**.

△ *NOTE: Most dialog boxes report the distance from the top-left corner of the work space, regardless of the current origin.*

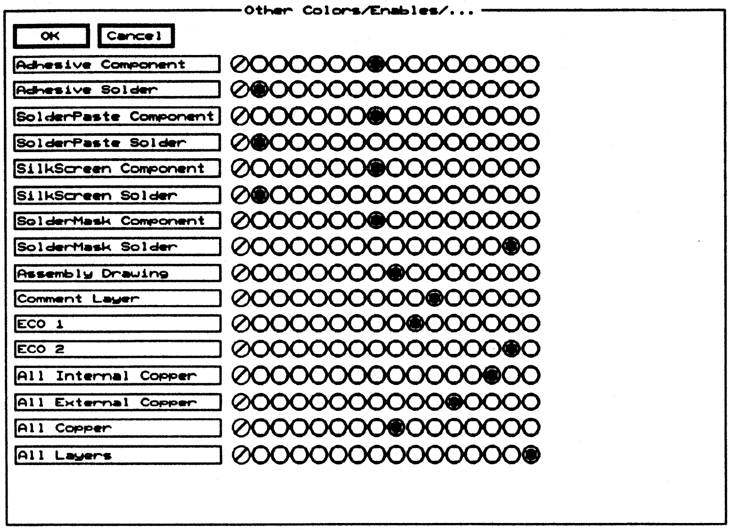
Other Colors/Enables/... button

Appears on the **Layer** dialog box.

Displays the **Other Colors/Enables/...** dialog box.

Other Colors/Enables/... dialog box

Use this dialog box to edit a layer name, and select layer colors.



Entry boxes

Use the entry boxes to change the name of a layer.

Color radio buttons

Use the color radio buttons to change the layer colors. Layer colors help you distinguish one layer from another when viewing the board. There are 16 colors available.

Other Module Properties button

Appears on the **Edit Module Properties** dialog box.

Displays the **Edit Other Module Properties** dialog box.

Out command

Appears on the **ZOOM** menu.

Out decreases the size of displayed objects and moves the current pointer position to the center of the screen. Use the **Zoom Change Factor** entry box in the **Global Options** dialog box to specify the factor by which the size of the displayed objects decreases when you select **ZOOM Out**.

Outline command

Appears on the **PLACE** menu.

Loads the current outline settings.

See also *Placing outlines*.

Outline Pads check box

In the **Global Options** dialog box, the **Outline Pads** check box controls the display of pads, vias, and test points.

See also *Outlines*.

Outline Text check box

In the **Global Options** dialog box, the **Outline Text** check box controls the display of:

- Dimension objects
- Layer markers
- Alignment targets
- Circles (except those drawn as four arcs)
- Text

See also *Outlines*.

△ *NOTE: If **Stick Text** in the **Global Options** dialog box is enabled, text characters are drawn with fine lines ("sticks"), regardless of the setting of **Outline Text**.*

Outline Tracks check box

In the **Global Options** dialog box, the **Outline Tracks** check box controls the display of:

- Segments (outlines, zones, and nets)
- Arcs (outlines, zones, and nets)
- Circles drawn as four arcs

See also *Outlines*.

Outlines

You can configure **Edit Layout** to display objects as solids or as outlines. Looking at outlines, you can more easily recognize short segments and distinguish overlapping objects.

Select **SET** from the main menu or click on the **Global** button in most dialog boxes to open the **Global Options** dialog box, where three check boxes control the display of different objects:

- **Outline Tracks**
- **Outline Pads**
- **Outline Text**

See the entry for each check box for more information on its use and the objects it controls.

△ *NOTE: Holes always show as outlines.*

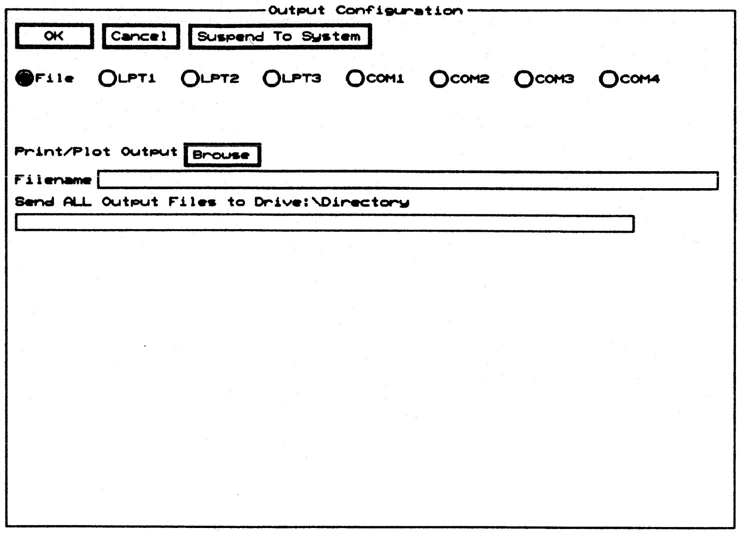
Output button

Appears on the **Printing and Plotting** dialog box.

Displays the **Output Configuration** dialog box.

Output Configuration dialog box

Use this dialog box to configure the output for printing and plotting.



Suspend To System Will only become active if your system includes an 80387 or 80487 math coprocessor, or has coprocessing functions. This button clears the dialog box, and displays the system prompt.

File Use to send the output to a file. When **File** is selected, the following items display in the **Destination** area.

Print/Plot Setup

Browse Displays the **Load Print/Plot Setup from File** dialog box.

Filename Use to enter the desired load setup filename, which may include the drive and path. If you specify the drive and path of the load setup file, and then select **Browse**, the **Output Filename** dialog box will display that drive, in that directory, and a list of filenames.

Send ALL Output Files to Drive:\Directory Typically, use to enter the drive and directory where you want all of your files sent. For example, if you want to view your output in a WISE viewer, enter the path to the viewer.

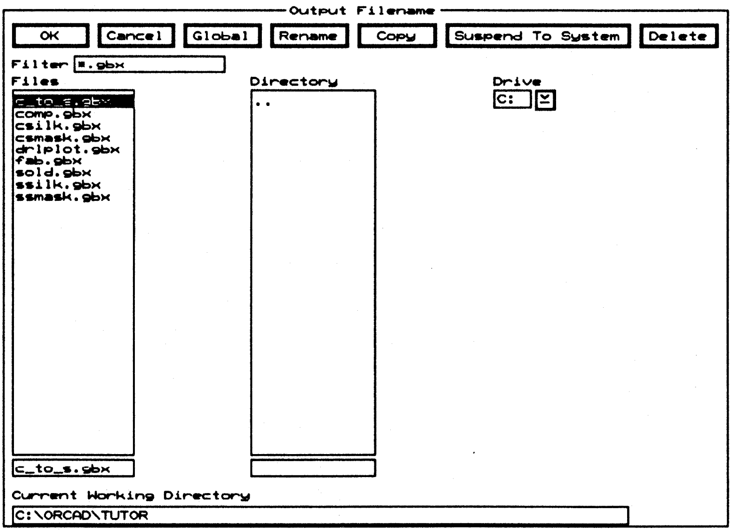
NOTE: If you have any pages defined with outputs in the ***Pages*** *list box in the* ***Printing and Plotting*** *dialog box, each and every one of them will be updated and redirected to the specified directory as soon as you select* ***OK****.*

LPT*n* Use to select a specific line printer.

COM*n* Use to select a specific serial communications port. When a communications port is selected, the **Destination** area displays the following communications parameters: speed, parity, data bits, and stop bits. Set these parameters as desired.

Output Filename dialog box

Use this dialog box to view a list of printer and plotter output files.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes the file highlighted in the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to enter a new filename.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Pad command

Appears on the library editor **PLACE** menu.

Loads the current pad stack.

See also *Placing pads*.

Pad Array Alphabet button

Appears on the **Edit Pad Array Settings** dialog box.

Displays the **Edit Pad Array Alphabet** dialog box.

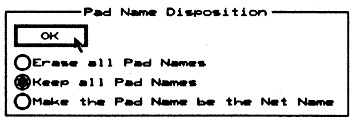
Pad Array Settings button

Appears on the **Edit Pad** dialog box.

Displays the **Edit Pad Array Settings** dialog box.

Pad Name Disposition dialog box

Use this dialog box to name the pads in a board file which is being imported into the library editor.



Erase all Pad Names Deletes all the pad names in the imported board file.

Keep all Pad Names Maintains all the pad names in the imported board file.

Make the Pad Name be the Netname Changes all the pad names to the netnames they are associated with.

Pad Stack Editor button

Appears on a number of dialog boxes.

Displays the **Edit Pad Stack** dialog box.

Pad Stack Editor command

Appears on the **GO TO FUNCTION** menu.

Displays the **Edit Pad Stack** dialog box.

Permanently Delete command

Appears on the **SELECTIVE** menu.

Removes an object from the undelete buffer. The object cannot be restored with **SELECTIVE** or **UNDELETE.**

PLACE command

Appears on a number of menus.

On the board editor main menu

Displays the menu shown at right.

Selecting a command from this menu loads the current settings of the object.

```
Module
New Module
Text
Hole
Circle
Dimension
Layer Marker
Alignment Target
Test Point
Outline
Polygon
Fill Zone
No Fill Zone
No Through Zone
Autoroute Zone
No Autoroute Zone
```

On the library editor main menu

Displays the menu shown at right.

Selecting a command from this menu loads the current settings of the object.

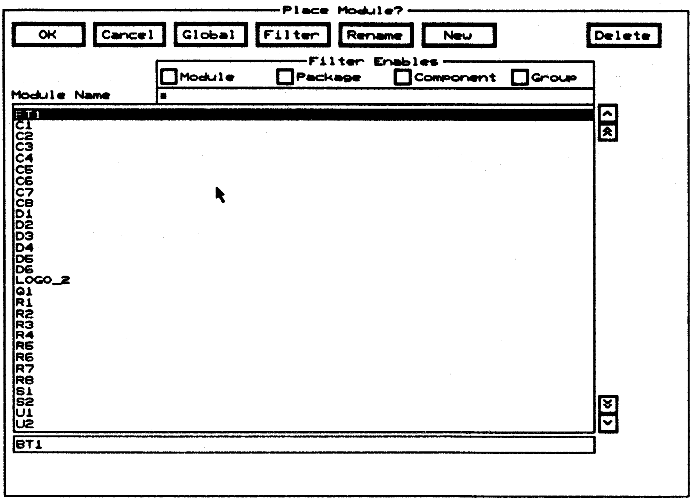
```
Pad
Text
Hole
Circle
Dimension
Layer Marker
Alignment Target
Outline
Polygon
Fill Zone
No Fill Zone
No Through Zone
No Autoroute Zone
```

On other menus

Places the selected objects.

Place Module dialog box

Use this dialog box to place a new module or move an existing module on the board editor.



Filter Displays the **Edit Filter** dialog box.

Rename Displays the **Rename Module** dialog box.

New Displays the **New Module** dialog box.

Delete Removes the selected module from the **Module Name** list box. If the module has been placed on a board, selecting **Delete** also removes it from the board.

Filter Enables Use this area to view the modules shown in the **Module Name** list box by type.

- **Module** Enable to list all the modules which exactly match the value shown in the **Module** droplist box on the **Edit Filter** dialog box.
- **Package** Enable to list all the modules which exactly match the value shown in the **Package** droplist box on the **Edit Filter** dialog box.
- **Component** Enable to list all the modules which exactly match the value shown in the **Component** droplist box on the **Edit Filter** dialog box.
- **Group** Enable to list all the modules which exactly match the value shown in the **Group** droplist box on the **Edit Filter** dialog box.

Module Name Contains a list of modules.

Use any combination of wildcards (* or ?) and other characters in the entry box directly above the **Module Name** list box to restrict the list of modules shown.

Placing alignment targets

1. Select **PLACE Alignment Target**.
2. If desired, select **Set** to display the **Edit Alignment Target** dialog box and change the copper tool, alignment target style, or radius, and then select **OK**.
3. Move the pointer to the desired location and select **Place**.

Placing circles

1. Select **PLACE Circle**. An × displays to indicate the center of the circle.
2. Move the pointer to the location where you want the center of the circle and select **Begin**.
3. Move the pointer away from the × to enlarge the circle to the desired size and select **End**.

Placing dimension objects

1. Select **PLACE Dimension.** A dimension object displays.
2. If desired, select **Set** to display the **Edit Dimension Text** dialog box and change the copper tool, angle, height, or other characteristic, and then select **OK.**
3. Move the pointer to the location where you want the first end bar and select **Begin.**
4. Move the pointer to the location where you want the second end bar and select **End.**

Placing holes

1. Select **PLACE Hole.** A hole displays.
2. If desired, select **Set** to display the **Edit Hole** dialog box and select a size from the **Drill Diameter** droplist box, and then select **OK.**
3. Move the pointer to the desired location and select **Place.**

Placing layer markers

1. Select **PLACE Layer Marker.**
2. Move the pointer to the desired location and select **Place.**

Placing a layer marker is a convenient way to display the number of layers on the board.

Placing outlines

1. Select **PLACE Outline.**
2. If desired, select **Set** to display the **Edit Outline Segment** dialog box and change the copper tool or drawing method or select any of the various options, and then select **OK.**
3. Select **Begin** and start drawing the outline. As you move the pointer, notice that you can draw two perpendicular segments at a time.
4. Select **Begin** again to start each new segment in a continuous outline, and **New** to start a new outline.
5. Continue until you reach the final position, and then select **End.**

Placing pad arrays

1. In the library editor, select **PLACE Pad Set**. The **Edit Pad** dialog box displays.
2. Select **Pad Array Settings**. The **Edit Pad Array Settings** dialog box displays.
3. Select a pad array style in the **Style** area. A graphic representation of the selected pad array style displays in the **Style Sample** area, and the appropriate options become available in the **X Direction, Y Direction,** and **Options** areas.
4. Set the options in the **X Direction, Y Direction,** and **Options** areas as desired.
5. Select **OK** to close the **Edit Pad Array Settings** dialog box, and then select **OK** to close the **Edit Pad** dialog box.
6. Move the pointer to the desired location and select **Place**.

Placing pads

1. In the library editor, select **PLACE Pad**. A pad stack displays.
2. If desired, select **Set** to display the **Edit Pad** dialog box and change the pad stack or angle, and then select **OK**.
3. Move the pointer to the desired location and select **Place**.

See also *Placing pad arrays*.

Placing polygons

1. Select the desired drawing method on the **Global Options** dialog box.
2. Select **PLACE,** and select **Polygon** from the menu that displays.
3. Select **Begin** and start drawing the polygon's boundary.
4. Select **End.**

Placing test points

1. In the board editor, select **PLACE Test Point.** A test point displays.
2. If desired, select **Set** to display the **Edit Test Point** dialog box and change the pad stack or netname, and then select **OK.**
3. Move the pointer to the desired location and select **Place.**

Placing text

1. Select **PLACE Text.** The **Text** entry box displays.
2. Enter the text. The text string displays at the pointer.
3. If desired, select **Set** to display the **Edit Text** dialog box and change the copper tool, angle, or character height, and then select **OK.**
4. Move the pointer to the desired location and select **Place.**

Placing vias

1. In the board editor, select **ROUTE.**
2. Place the pointer on a pad that has a net associated with it, and then select **Begin.**
3. Move the pointer to the desired location and select **Via** or **/ other.** The current layer automatically changes to this copper layer's pair, as specified in the **Layer** dialog box.

Placing zones

1. Select the desired drawing method on the **Global Options** dialog box.
2. Select **PLACE**, and select the desired zone type from the menu that displays.
3. Select **Begin** and start drawing the zone's boundary.
4. Select **End**.

See also *Assigning nets to fill zones* and *Viewing fill zones*.

△ *NOTE: You cannot place a fill zone inside another fill zone, and two fill zones cannot share a boundary.*

Plane layers

Select **LAYER** and then select **Copper Colors/Enables/...** to display the **Copper Colors/Enable/...** dialog box, in which you designate a layer as a plane. A *plane layer* is a layer of copper that is dedicated to a single net. On a plane layer, pads with the same netname as the plane layer are connected to the plane layer.

The autorouter cannot create routes on plane layers, but it can route through them, where necessary, using vias. If you need to place routes on a given layer, use a fill zone for the copper area, and don't designate the layer as a plane.

Plane names

Because plane names are restricted to 21 characters, netnames longer than 21 characters cannot be used for plane layers.

Pads

You should use simple pad shapes—circles, squares, rectangles, and ovals—on a plane layer. Where routing requires more complex shapes on a layer, don't designate the layer as a plane. Use fill zones, instead.

Plotting

See *Printing and plotting* and *Printing and Plotting dialog box*.

Polygon command

Appears on the **PLACE** menu.

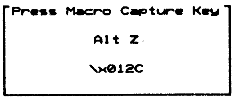
Loads the current zone settings.

A *polygon* is a continuous outline shape. Unlike other zones, it has no exterior or interior properties.

See also *Placing zones* and *Placing polygons.*

Press Macro Capture Key dialog box

Use this dialog box to assign a key or key combinations to run macros.



The pressed key or key combination displays, along with the macro internal code, in the dialog box. If you press an invalid key or combination, it does not display.

See also *Creating macros* and *% MACRO command.*

Previous command

Appears on the **ZOOM** menu.

Toggles between the current and previous zoom scales, and moves the current pointer position to the center of the screen.

Printing and plotting

In **PCB 386+ 2.00**, each output device is categorized as a plotter or a printer, depending on the type of input it requires.

Plotters accept vector commands. A *vector* is one or more points and an associated function. For example, a line has two points and a "shortest-distance" function; a circle has a center point and a "radius" function. Such devices do not need information about every point along the vector.

Printers accept raster commands. A *raster* is an array of dots, which may define any shape. Such a device needs instructions for each and every dot.

1. In the board editor, select **GO TO FUNCTION Printing and Plotting**. The **Printing and Plotting** dialog box displays.
2. If desired, select **Driver** to display the **Driver Configuration** dialog box, in which you can specify and configure the destination. See *Selecting output devices*.
3. Select **Begin All** to print or plot all pages defined in the **Pages** list box, or select **Begin** to print or plot the page contents defined in the **Page Contents** list box. See also *Configuring pages*, *Printing and Plotting dialog box*, and *Load Print/Plot Setup from File dialog box*.

Printing and Plotting command

Appears on the board editor **GO TO FUNCTION** menu.

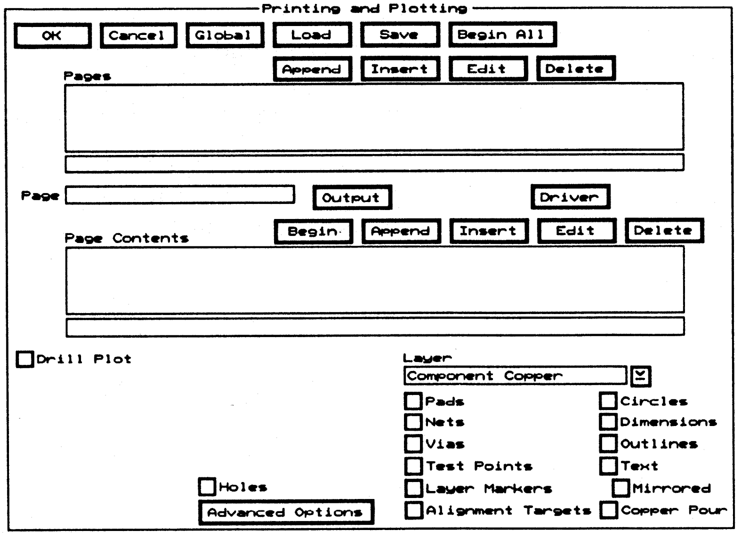
Displays the **Printing and Plotting** dialog box.

See also *Printing and plotting* and *Printing and Plotting dialog box*.

Printing and Plotting dialog box

Use this dialog box to select pages, and to print or plot the board.

See also *SolderMask plots* and *Configuring pages.*



Load Displays the **Load Setup from File** dialog box.

Save Displays the **Save Setup to File** dialog box.

Begin All Select to print or plot all the pages in the **Pages** list box.

Pages

Pages Contains a list of page names. Each line in the list box represents a sheet of paper, either a print, or a plot. Each line consists of three items: a page description, an output, and the driver format.

Use the **Page** entry box directly beneath the **Pages** list box to create a page name.

Append Adds the page name below the highlighted page in the **Pages** list box.

Insert Adds a page name above the highlighted page name in the **Pages** list box.

Edit Loads the page contents items defined by the highlighted page name into the **Page Contents** list box.

Delete Deletes the highlighted page from the **Pages** list box.

Output Displays the **Output Configuration** dialog box. If a filename has been entered in the **Filename** entry box in the **Output Configuration** dialog box, that filename will display to the right of the **Output** button.

Driver Displays the **Driver Configuration** dialog box. If a driver has been selected in the **Driver Configuration** dialog box, that driver name will display to the right of the **Driver** button.

Page Contents

Page Contents Contains a numbered list of page content items. Typically, each line either represents a layer, or a drill plot (which is a pair of layers).

Begin Prints the items displayed in the **Page Contents** list box. This button only becomes available when you define an output destination, a driver, and at least one **Page Contents** line.

Append Adds the page contents item defined by the options below the **Page Contents** list box below the highlighted line.

Insert Adds the page contents item defined by the options below the **Page Contents** list box above the highlighted page contents item.

Edit Sets the options below the **Page Contents** list box to the values associated with the selected page contents item.

Delete Removes the selected page contents item from the **Page Contents** list box.

Drill Plot Enable to display drill plot options.

Between and **Layer** Use both **Layer** list boxes to designate the layers for the drill plot.

Drill Legend Designates the location of the drill legend.

Drill Target Designates the type of drill target.

Fixed Label Size If non-zero, all drill targets are printed or plotted at this size. If zero, all drill targets are printed or plotted at the same size as the hole they represent.

Objects included

Holes Enable to include holes in the output.

Pads Enable to include pads in the output.

Nets Enable to include nets in the output.

Vias Enable to include vias in the output.

△ *NOTE: For solder mask plots, do not include vias if you wish to tent your vias with solder mask.*

Test Points Enable to include test points in the output.

△ *NOTE: Pads, vias, test points, and nets should not be plotted on layer planes. Layer planes are plotted as negative images, and PCB 386+ 2,00 will automatically plot the isolation for each of these objects.*

Layer Markers Enable to include layer markers in the output.

Alignment Targets Enable to include alignment targets in the output.

Circles Enable to include circles in the output.

△ *NOTE: The **Circles** check box does not affect circles drawn as four arcs, which are controlled by the **Outlines** check box.*

Dimensions Enable to include dimension objects in the output.

Outlines Enable to include outlines in the output. This check box also pertains to circles drawn as four arcs.

△ *NOTE: For the **SilkScreen** layers, enabling **Outlines** will only produce module outlines.*

Text Enable to include text in the output.

Mirrored Enable to print or plot mirrored text. Typically, the only time this is enabled is for the solder side silkscreen layer.

Copper Pour Enable to print or plot the copper pour area of a fill zone. Disable **Copper Pour** to print or plot copper zones as outlines.

Advanced Options Displays the **Advanced Printing and Plotting Options** dialog box.

QUIT command

Appears on the main menu.

On the board editor main menu

Displays the menu shown at right.

Loads, updates, and writes board files, erases all routes, flushes the undelete buffer, suspends to the operating system, and exits **Edit Layout**.

```
Update Board File
Write Board File
Initialize Board File
Erase All Routes
Flush Undelete Buffer
Cleanup Stubs
Suspend to System
Abandon Program
```

On the library editor main menu

Displays the menu shown at right.

Loads, updates, and writes library files, flushes the undelete buffer, suspends to the operating system, and displays the board editor.

```
Update Library File
Write Library File
Initialize to Library
Flush Undelete Buffer
Suspend to System
Leave Library Editor
```

Quit Selective Undelete command

Appears on the **SELECTIVE** menu.

Exits selective-undelete mode and returns you to the board or library editor.

RatsNest Block command

Appears on the **Block End** menu.

Displays the complete ratsnest for all the nets of all the pads within the block and intersecting block layer. However, nets that have been designated **Do Not Route Net** will not display a ratsnest.

See also *Ratsnests*.

Ratsnests

A ratsnest is a graphic representation of the electrical connections between pairs of pads in a layout. The ratsnest display for pads that connect to a power or ground layer are shown as circles to indicate that the pad is going to connect through a via.

To display the ratsnest for a group of modules, use **BLOCK** and **Block End** to enclose them, and then select **RatsNest Block**. All modules in the block display their connections.

To clear the ratsnest display, move the pointer to a location free of board objects, and select **X SHOW RATSNEST**. The ratsnest disappears, and the message "Show RatsNest Cleared" displays in the lower-right corner of the screen.

To display the ratsnest for a single pad, position the pointer on the desired pad, and select **X SHOW RATSNEST**.

To clear the ratsnest, position the pointer anywhere on the net, and select **X SHOW RATSNEST** again.

△ *NOTES: Positioning the pointer on a non-net object, and selecting X SHOW RATSNEST will neither display nor clear ratsnests.*

Ratsnests are cumulative. You may enter any combination of block or single pad ratsnests and they will all display until cleared.

Refresh command

Appears on the **ZOOM** menu.

Redraws the screen without changing the zoom scale or viewing area.

Rename button

Appears on a number of dialog boxes.

On file dialog boxes

On the **Write Board File, Initialize to Board File, Write Library File, Initialize to Library File,** and various **Load . . . from File** and **Save . . . to File** dialog boxes, displays the **Rename File** dialog box.

On the Edit Net Properties dialog box

Displays the **Rename Net Objects** dialog box.

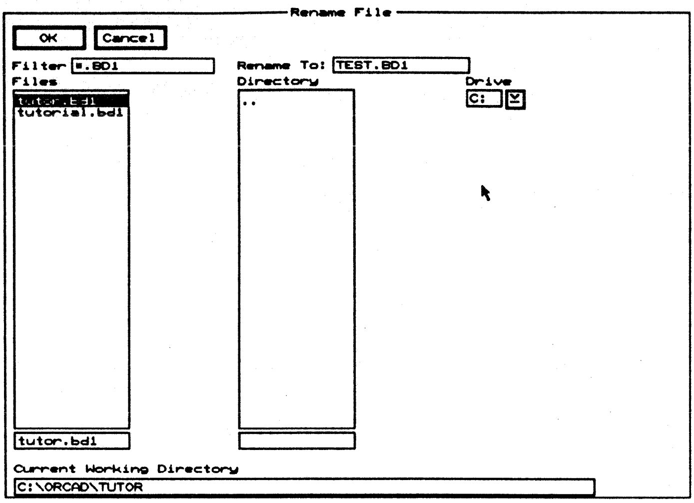
On module dialog boxes

On the **Place Module** and **Get Module** dialog boxes, displays the **Rename Module** dialog box.

Rename File dialog box

Use this dialog box to rename a file.

See also *Changing file names.*



Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown.

Rename To Use to enter the new filename.

Files Contains a list of the files in the current working directory.

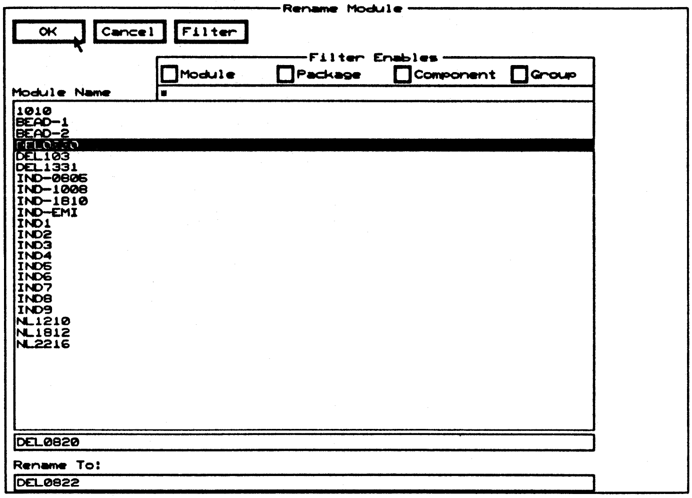
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Rename Module dialog box

Use this dialog box to rename a module.



Filter Displays the **Edit Filter** dialog box.

Filter Enables Use this area to view the modules shown in the **Module Name** list box by type.

- **Module** Enable to list all modules which exactly match the value shown in the **Module** droplist box on the **Edit Filter** dialog box.
- **Package** Enable to list all modules which exactly match the value shown in the **Package** droplist box on the **Edit Filter** dialog box.
- **Component** Enable to list all modules which exactly match the value shown in the **Component** droplist box on the **Edit Filter** dialog box.
- **Group** Enable to list all modules which exactly match the value shown in the **Group** droplist box on the **Edit Filter** dialog box.

Module Name Contains a list of modules.

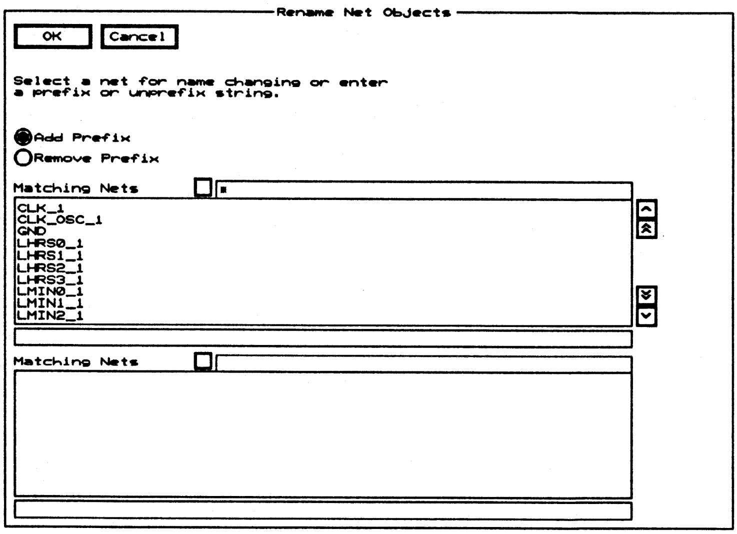
Use any combination of wildcards (* or ?) and other characters in the entry box directly above the **Module Name** list box to restrict the list of modules shown.

Rename To Use this entry box to change the name of the module selected in the **Module Name** list box.

Rename Net Objects dialog box

Use this dialog box to rename net objects.

See also *Renaming Net Objects.*



Radio buttons

The **Add Prefix** and **Remove Prefix** radio buttons become active when a prefix is entered into the entry box below the first **Matching Nets/Not Matching Nets** list box.

Add Prefix Adds the prefix string in the entry box to all netnames displayed in the **Matching Nets/Not Matching Nets** list boxes.

Remove Prefix Removes the prefix string in the entry box from all netnames displayed in the **Matching Nets/Not Matching Nets** list boxes.

△ *NOTE: You cannot add a prefix which matches an existing netname.*

List boxes

Matching Nets/Not Matching Nets Contains a list of netnames. Use any combination of wildcards (* or ?) and other characters in the entry box above the **Matching Nets/Not Matching Nets** list box to restrict the list of netnames shown.

The check box toggles the filter to only display those netnames which match the filter value (**Matching Nets**) or only those netnames which do not (**Not Matching Nets**).

There are two **Matching Nets/Not Matching Nets** list boxes on this dialog box. When a netname is selected in the first list box, the second list box becomes active.

Entry boxes

The entry box below the first **Matching Nets/Not Matching Nets** list box is used to add or remove netname prefixes to all filtered netnames.

The entry box below the second **Matching Nets/Not Matching Nets** list box is used to create a new netname.

Renaming Net Objects

Use the **Rename Net Objects** dialog box to rename a netname to a new netname, rename a netname to an existing netname, and add or remove netname prefixes to all filtered netnames.

Renaming a netname to a new netname

1. Display the **Rename Net Objects** dialog box.
2. Select the desired netname in the first **Matching Nets/Not Matching Nets** list box.
3. In the entry box below the second **Matching Nets/Not Matching Nets** list box enter the new netname.
4. Select **OK**.

Renaming a netname to an existing netname

1. Display the **Rename Net Objects** dialog box.
2. Select the desired netname in the first **Matching Nets/Not Matching Nets** list box.
3. Select the replacement netname in the second **Matching Nets/Not Matching Nets** list box.
4. Select **OK**.

Adding a netname prefix

1. Display the **Rename Net Objects** dialog box.
2. In the entry box below the first **Matching Nets/Not Matching Nets** list box, enter a netname prefix. The **Add Prefix** radio button becomes active.
3. Select **OK**.

△ *NOTE: You cannot add a prefix which matches an existing netname.*

Removing a netname prefix

1. Display the **Rename Net Objects** dialog box.
2. In the entry box below the first **Matching Nets/Not Matching Nets** list box enter netname prefix. The **Add Prefix** radio button becomes active.
3. Select the **Remove Prefix** radio button.
4. Select **OK**.

Reversing the mouse buttons

In the **Processing Options** area of the **Configure Edit Layout** screen, select **Left hand mouse operation.** The functions (<Enter> and <Esc>) of the mouse buttons reverse. Select **Left hand mouse operation** again to return the mouse buttons to their default functions.

Rotating arcs

Edit Layout supports 90° arcs that are contained in a single quadrant (0°–90°, 90°–180°, 180°–270°, 270°–360°). If you rotate an arc so that either condition no longer applies, **Edit Layout** breaks the arc into four segments. Note that you cannot recreate the arc from these four segments.

See also *Rotating modules.*

Rotating modules

You can rotate modules to a specific angle or in preset increments.

See also *Rotating arcs* and *Rotating pads*.

To a specific angle

1. In the **Global Options** dialog box, make sure **Allow Move/Edit/Delete of Module Elements** is disabled and **Stay On Grid** is enabled.
2. Place the pointer on a module label or pad and select **MOVE Set**. The **Set Block Parameters** dialog box displays.
3. In the **Angle** entry box, enter the number of degrees by which the module is to be rotated.

   You can rotate a module in increments of one-hundredth of a degree (0.01°), and the allowed range is –359.99 to 359.99. To rotate clockwise, enter a minus sign before the rotation value.
4. Select **OK** to dismiss the dialog box. The module displays at its new orientation.

△ *NOTE: Rotation uses the current pointer position as the pivot point.*

In preset increments

Before you use > **Rotate Clockwise** or < **Rotate Counter Clockwise** to rotate a module, you must specify the amount by which the module is to be rotated. The rotation step angle is initially set to 90.00 degrees.

1. In the **Global Options** dialog box, make sure **Allow Move/Edit/Delete of Module Elements** is disabled and **Stay On Grid** is enabled.
2. Place the pointer on a module label or pad and select **MOVE Set**. The **Set Block Parameters** dialog box displays.
3. In the **Rotation Step Angle** entry box, enter the number of degrees by which the module is to be rotated each time you use > **Rotate Clockwise** or < **Rotate Counter Clockwise**.

   You can set the angle to a resolution of one-hundredth of a degree (0.01°), and the allowed range is 0.00 to 359.99.
4. Select **OK** to dismiss the **Set Block Parameters** dialog box.
5. Select > **Rotate Clockwise** or < **Rotate Counter Clockwise**. The module rotates in the corresponding direction by the specified number of degrees.
6. Continue selecting > **Rotate Clockwise** or < **Rotate Counter Clockwise** until the module is oriented correctly
7. Select **Place.**

△ *NOTE: Rotation uses the current pointer position as the pivot point.*

Rotating pads

Except for circular pads, rotating a pad rotates its thermal relief along with it. Thermal relief for circular pads is always drawn parallel to the X and Y axes.

See also *Rotating modules.*

ROUTE command

Appears on a number of menus.

On the main menu

Begins the process of creating net segments.

On the Delete Block menu

Deletes only the net segments and arcs enclosed or intersected by the block boundary.

Run button

Appears on the **Macro Maintenance** dialog box.

Executes the selected macro.

Running macros

In **Edit Layout**, you typically run a macro by pressing the assigned key or combination. You can also run a macro from within the **Macro Maintenance** dialog box by following these steps:

1. Select a macro in the **Defined Macros** list box.
2. Select **Run**. The **Macro Maintenance** dialog box closes and the macro executes, starting at the current pointer location.

See also *Macros* and *Macro Maintenance dialog box.*

Save button

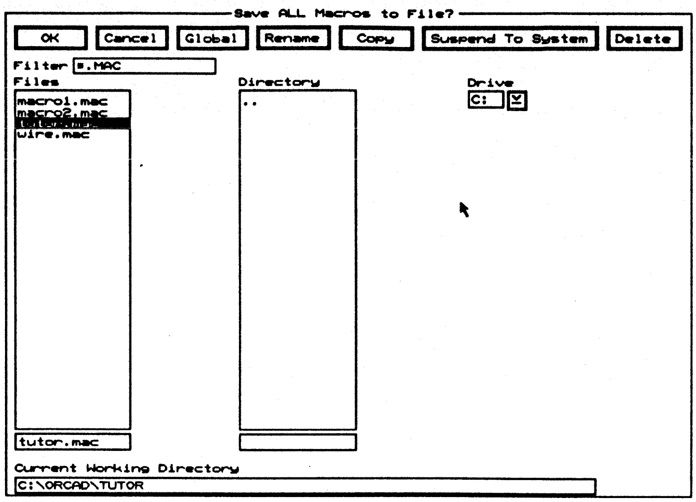
Appears on a number of dialog boxes.

Displays the appropriate **Save... to File** dialog box.

Save ALL Macros to File dialog box

Use this dialog box to save all the defined macros to a file.

See also *Saving macros*.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes a file from the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown in the **Files** list box.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to enter a new filename.

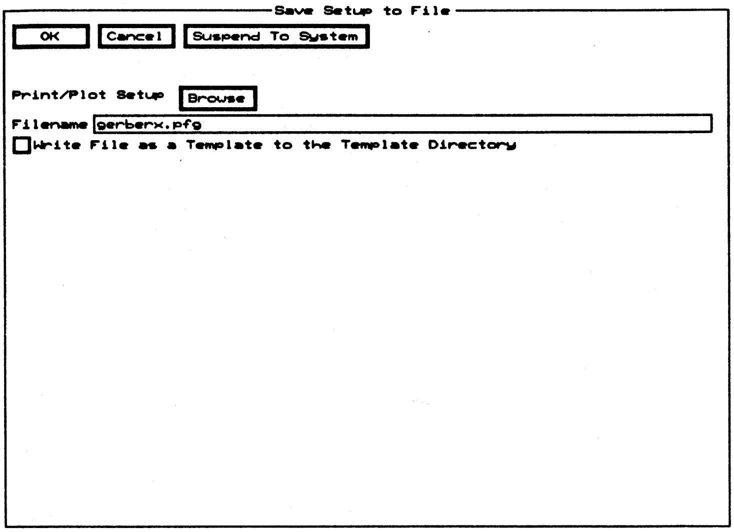
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Save Setup to File dialog box

Use this dialog box to specify the setup filename conditions.



Suspend To System Will only become active if your system includes an 80387 or 80487 math coprocessor, or has coprocessing functions. This button clears the dialog box, and displays the system prompt.

Print/Plot Setup

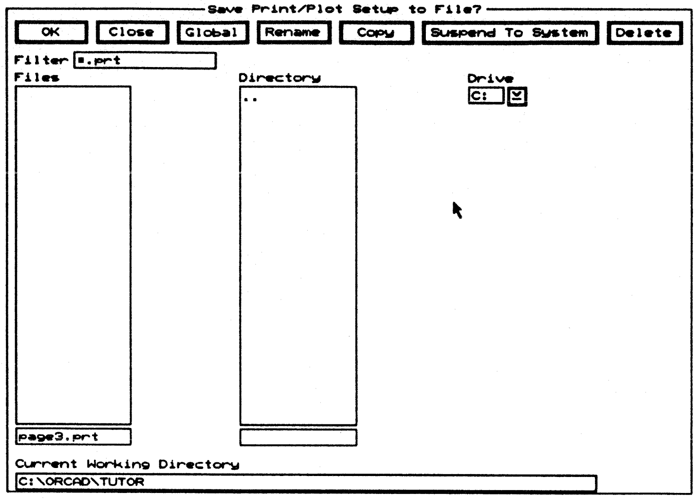
Browse Displays the **Save Print/Plot Setup to File** dialog box.

Filename Use to enter the desired save setup filename, which may include the drive and path. If you specify the drive and path of the save setup file, and then select **Browse**, the **Save Print/Plot Setup to File** dialog box will display that drive, in that directory, and that filename.

Write File as a Template to the Template Directory Enable to

Save Print/Plot Setup to File dialog box

Use this dialog box to save the printer or plotter setup to a file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Deletes a file from the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown.

Files Contains a list of the files in the current working directory. Use the entry box directly beneath the **Files** list box to enter a new filename.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Saving macros

Saved macros can be reused the next time you run **Edit Layout.** In **Edit Layout,** you can save all defined macros in one file or export single macros to separate files.

See also *Save* 'macroName' *Macro to File dialog box* and *Save ALL Macros to File dialog box.*

All macros

Follow these steps to save any defined macros in a single file:

1. Select **GO TO FUNCTION Macro Maintenance.** The **Macro Maintenance** dialog box displays. All defined macros display in the **Defined Macros** list box. The macros listed are those now stored in memory.
2. Select **Save.** The **Save ALL Macros to File** dialog box displays.
3. Select a filename from the **Files** list box or enter a name in the entry box below it. You can also select a different directory or drive from the corresponding list box.
4. Select **OK** to save all defined macros to the specified filename in the current working directory. The **Save ALL Macros to File** dialog box closes and the **Macro Maintenance** dialog box displays.

One macro

Follow these steps to export a single macro to a file:

1. Select **GO TO FUNCTION Macro Maintenance.** The **Macro Maintenance** dialog box displays. All defined macros display in the **Defined Macros** list box. The macros listed are those now stored in memory.
2. Select the macro to save from the **Defined Macros** list box, and then select **Export**. The **Export *'macroName'* Macro to File** dialog box displays. *macroName* is the macro you selected in step 2.
3. Select a filename from the **Files** list box or enter a name in the entry box below it. You can also select a different directory or drive from the corresponding list box.
4. Select **OK** to save the selected macro to the specified filename in the current working directory. The **Export *'macroName'* Macro to File** dialog box closes and the **Macro Maintenance** dialog box displays.

Saving work settings

Many of the configurations are saved with the board file when you select **QUIT Update Board File** or **QUIT Write Board File**. When you load the file into **Edit Layout**, all previously saved configurations for the board are used.

Selected Net button

Appears on the **Edit Net Properties** dialog box.

For each **Apply To All Filtered Nets** or **Apply** check box that is enabled, **Selected Net** applies the corresponding property as marked (selected, entered, or enabled/disabled) to all nets.

This button also applies all properties as marked (selected, entered, or enabled/disabled), regardless of the state of the corresponding **Apply To All Filtered Nets** check box, to the net whose name appears in the entry box below the **Matching Nets/Not Matching Nets** list box.

Selecting modules

You use the **BLOCK** and **Block End** commands to select a module. You must enclose the connection point of each of the module's pads in the block boundary. (A pad's connection point is the location shown for the pad in the **Edit Pad** dialog box.) The active layer must also intersect all of the module's pads.

See *Set Block Parameters dialog box* for more information about controlling what types of objects are selected when you define a block. See also *Moving modules*.

Selecting output devices

1. In the **Printing and Plotting** dialog box, select **Driver** to display the **Driver Configuration** dialog box.
2. Select a destination device:
   - ❖ **Raster Device.** The printer driver specified on the **Configure PC Board Layout** screen displays in the **Raster Device** entry box. Select the paper orientation and specify the page overlap.

     See *Chapter 1: Configure Layout Tools* for details on configuring **PCB 386+ 2.00.**
   - ❖ **Vector Device.** Select a vector device, such as Gerber (274-X), HPGL2, and PostScript, in the **Vector Device** droplist box. Some vector device selections display additional options that affect the output.

     See also *Gerber format.*
3. Select **OK** to dismiss the **Driver Configuration** dialog box. The **Printing and Plotting** dialog box displays.

See also *Driver Configuration dialog box.*

SELECTIVE command

Appears on a number of menus.

Sets **Edit Layout** in a special mode that displays the objects on the screen in dark gray and the objects in the undelete buffer in the color assigned to their layer.

In this selective-undelete mode, you can restore items in the undelete buffer to the screen or permanently delete them from the buffer. Use the following commands to modify the undelete buffer.

When you select **SELECTIVE**, the following commands become available:

Permanently Delete removes an object from the undelete buffer. The object cannot be restored with **SELECTIVE** or **UNDELETE**.

Quit Selective Undelete exits selective-undelete mode and returns you to the board editor.

Undelete restores an object to the board and changes its color to dark gray.

△ *NOTE: You cannot selectively restore a module's objects if you deleted the entire module in one operation. Instead, restore the module, and then delete any of the module's objects you do not want.*

SET command

Appears on a number of menus.

On the main menu

Displays the **Global Options** dialog box.

On object menus

Displays the appropriate dialog box for editing the object. For example, selecting **PLACE Alignment Target Set** displays the **Edit Alignment Target** dialog box.

On the autorouter menu

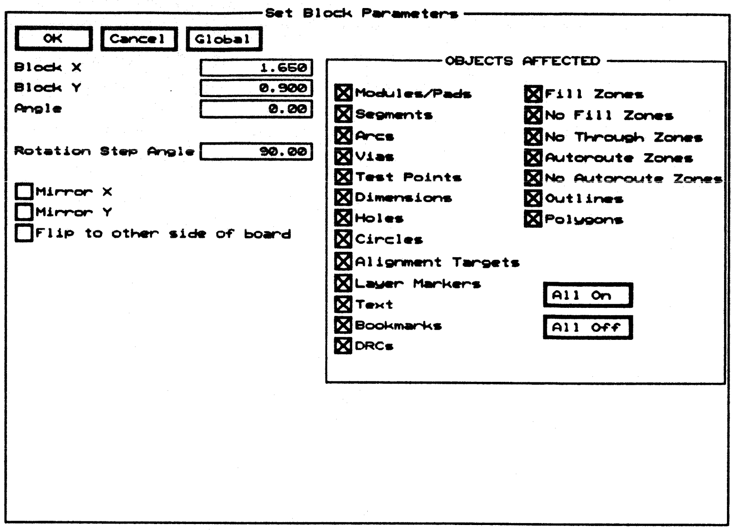
Displays the **Autoroute Options** dialog box.

On the BLOCK and Move menus

Displays the **Set Block Parameters** dialog box.

Set Block Parameters dialog box

Use this dialog box to move, rotate, and mirror objects enclosed or intersected by the block boundary.



Block X Moves the objects enclosed or intersected by the block boundary to the left or right of their current location.

Block Y Moves the objects enclosed or intersected by the block boundary up or down from their current location.

△ *NOTE: In the Block entry boxes, the allowed range is 0.0000 in. (0.0000 mm) to 33.0000 in. (838.2000 mm). The block's lower-left corner is the reference point. Note that the value shown is the distance from the top-left corner of the work space, regardless of the current origin.*

Angle Rotates the objects enclosed or intersected by the block boundary. The allowed range is –359.99 to 359.99 degrees.

Rotation Step Angle Sets how many degrees **> Rotate Clockwise** and **< Rotate Counter Clockwise** rotate the objects enclosed or intersected by the block boundary. The allowed range is 0.00 to 359.99 degrees and the default is 90.00 degrees.

△ *NOTE: Test points and vias cannot be rotated.*

Mirror X Mirrors all enabled objects within the block along the X axis. **Mirror X** is not active if the block encloses a module when **Allow Move/Edit/Delete of Module Elements** is disabled in **Global Options**.

Mirror Y Mirrors all enabled objects within the block along the Y axis. **Mirror Y** is not active if the block encloses a module when **Allow Move/Edit/Delete of Module Elements** is disabled in **Global Options**.

Selecting both **Mirror X** and **Mirror Y** mirrors all enabled objects within the block along both axes.

Flip to other side of board Does a mirror Y and places the objects on a different layer. Note that you can flip a module to the other side of the board.

OBJECTS AFFECTED Use these check boxes to determine which objects enclosed or intersected by the block boundary are affected by a **BLOCK** command sequence. For example, to **BLOCK Delete** just segments, select only the **Segments** check box. Note that these check boxes are not active once the box is drawn.

All On Use to enable all the **OBJECTS AFFECTED** check boxes.

All Off Use to disable all the **OBJECTS AFFECTED** check boxes.

Set Scale command

Appears on the **ZOOM** menu.

Displays the **Set Zoom Scale** dialog box.

Set Sweep Window command

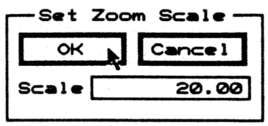
Appears on the **Whole Board** and **Sweep Window End** menus.

Begins the process of defining a sweep window.

Note that, for whole board sweeps, the sweep windows overlap by 25%.

Set Zoom Scale dialog box

Use this dialog box to set a specific zoom scale.



Scale Enter a value between 0.01 (maximum magnification) and 100 (minimum magnification).

Selecting **OK** dismisses the dialog box and if possible, moves the current pointer position to the center of the screen.

SolderMask plots

There is a SolderMask for the component layer and a SolderMask for the solder layer. SolderMask plots are negative image plots. Typically, you will only want to plot the pads.

Because this is a negative image plot, enabling **Vias** and **Test Points** prevents SolderMask from being deposited on your vias and test points.

If you enable **Vias**, your vias will not be tented.

If you do not want SolderMask deposited on your test points (so you can probe them) then enable **Test Points**.

See also *Printing and Plotting dialog box*.

Spacing/DRC Check Block command

Appears on the **Block End** menu that displays when you define the lower right corner of the autoroute block boundary.

Scans the block for spacing violations. If a violation is found, a DRC is placed on the board at the point of violation. When the DRC check is completed the **Finished** dialog box displays.

Select **DRCs** in the **Jump To** dialog box to list the DRCs on the layout and jump to a specific DRC marker. Place the pointer on a DRC marker and select **INQUIRE** to display the associated error message.

See *Spacing/DRC Check Whole Board command* for a list of defined DRC errors.

△ *NOTE: When doing spacing checks for pad to non-net objects, the pad must have a netname.*

Spacing/DRC Check Whole Board command

Appears on the **Whole Board** and **Sweep Window End** menus.

Scans the board for spacing violations. If a violation is found, a DRC is placed on the board at the point of violation. When the DRC check is completed the **Finished** dialog box displays.

Select **DRCs** in the **Jump To** dialog box to list the DRCs on the layout and jump to a specific DRC marker. Place the pointer on a DRC marker and select **INQUIRE** to display the associated error message. The following DRC errors are defined.

Bad Via Location

The via is inside a no-through zone or outside the autoroute zone.

Bad Via Type

The via is not of the type to which the net is restricted.

Off Grid Via

The via is not on the specified via grid.

Pad Spacing Error

The pad is too close to an object that is not in the same net.

△ *NOTE: For a pad stack defined to be on both external copper layers, a drill diameter of zero (0) presents no obstacle to the autorouter, but a nonzero drill diameter causes the autorouter to maintain a distance from the hole equivalent to the specified copper-to-copper spacing.*

Pad To Pad Spacing Error

The pad is too close to another pad.

Segment Spacing Error

The segment is too close to an object that is not in the same net.

SMD Pad Not Connected To Plane The surface-mount pad is not connected to the plane layer by a via.

Via Spacing Error The via is too close to an object that is not in the same net.

△ *NOTES: Use **Show DRCs** in the **Global Options** dialog box to show and hide DRCs.*

When doing spacing checks for pad to non-net objects, the pad must have a netname.

Standard JEDEC Alphabet button

Appears on the **Edit Pad Array Alphabet** dialog box.

Enables the appropriate check boxes to create a standard JEDEC alphabet.

Stripe Width Copper Tool

To achieve your goals with respect to copper thieving and venting, you may require a particular zone stripe width and spacing through the body of the zone which is too wide to squeeze between your pads.

For example, if you need a twenty mil zone stripe on a forty mil pitch or spacing, and you have pads which are forty mils edge-to-edge, then a fifteen mil copper to copper isolation from each pad edge does not leave enough room to pass a twenty mil zone stripe between the pads.

PCB 386+ 2.00 supports a separate fill copper tool which should be evenly divisible into your chosen stripe width copper tool. If you choose a fill copper tool of 10 mils with a stripe width copper tool of 20 mils, then the body of the zone will contain 20 mil wide stripes while allowing the stripe to neck down to 10 mils.

However, if in the example used above, the pads are not placed exactly on the correct grid with respect to the zone stripes, the 10 mil fill copper tool width will not line up with the pads. Using a 5 mil fill copper tool width will provide better results.

Suspend To System button

Appears on a number of dialog boxes. The button is active if your system includes an 80387 or 80487 math coprocessor, or has coprocessing functions.

Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Suspend to System command

Appears on the **QUIT** menu.

Clears the screen and displays the system prompt.

Enter `exit` at the system prompt to return to **Edit Layout**.

Suspending to the operating system

If your system includes an 80387 or 80487 math coprocessor, or has coprocessing functions, follow these steps to temporarily suspend **Edit Layout** and display the operating system prompt:

1. Select **QUIT Suspend to System**. The **Edit Layout** screen disappears and the DOS prompt displays. A right angle bracket (>) is appended to the DOS prompt, indicating that **Edit Layout** is suspended in the background.
2. Enter `exit` to return to **Edit Layout**.

△ *NOTE: Suspend to System is inactive unless your system has a math coprocessor chip or coprocessing functions.*

You can also suspend to the operating system by selecting the **Suspend To System** button on a number of dialog boxes. When you enter `exit` at the DOS prompt you return to the dialog box.

Sweep Window Begin command

Appears on the **Set Sweep Window** menu.

Uses the current pointer position as the starting point of a sweep window. As you move the pointer, **Edit Layout** displays the sweep window boundary.

It is advisable to create the sweep window in the densest area of the board, and to select a sweep routing direction that leads into the next densest area.

See *Autoroute Options dialog box.*

Sweep Window End command

Appears on the **Sweep Window Begin** menu.

Completes the sweep window boundary and displays the menu shown at right.

```
Autoroute Whole Board
Spacing/DRC Check Whole Board
Set Sweep Window
```

Note that, unlike a block boundary, the sweep window boundary does not display once you finish drawing it by selecting **Sweep Window End.**

Template files

If you run **Edit Layout** without configuring a board file, or if you specify a missing or invalid file, **Edit Layout** loads the template named in the **Miscellaneous Options** area of the **Configure Layout Tools** screen. Likewise, if you don't specify a file, or if you specify a missing or invalid file, in the **QUIT Initialize to Board File** or **QUIT Initialize to Library File** dialog box, **Edit Layout** loads the template.

ORCADPCB._T_ is the template file provided with **PCB 386+ 2.00.** ORCADPCB._T_ serves as the template board file and the template library file. You can create as many template files as you like to meet your needs.

See also *Configuring template files* and *Chapter 11: Make Board Template.*

Test Point command

Appears on the board editor **PLACE** menu.

Loads the current pad stack.

See also *Placing test points.*

Text command

Appears on a number of menus.

On the Delete Block menu

Deletes only the comment text enclosed or intersected by the block boundary.

On the PLACE menu

Displays the **Text** entry box.

See also *Placing text.*

Text entry box

Use the **Text** entry box to enter comment text.

Move the pointer to the desired location and select **Place** to place the comment text. The **Text** entry box displays again. Press <Esc> to dismiss the entry box.

TRACK DELETE command

Appears on a number of menus.

A track is all the net segments between two terminal points. A terminal point is defined as a pad or a point where three net segments meet on the board.

Selecting **TRACK DELETE** deletes a track and stores it in the undelete buffer.

UNDELETE command

Appears on a number of menus.

Restores individual objects and blocks of objects from the undelete buffer. **UNDELETE** restores your latest deletion first, followed by your other deletions in reverse order. If the buffer is empty, **UNDELETE** displays the message "Nothing to Undelete."

On the SELECTIVE menu

Restores an object to the board and changes its color to dark gray. See *SELECTIVE command* for more information.

Understanding an EDIF 2 0 0 netlist

In response to a **Netlist Load Error** message, you may wish to review the OrCAD-generated EDIF 2 0 0 netlist.

Viewing the netlist file

1. From the **PC Board Layout Tools** screen, select **Edit File.**
2. Select **Execute** from the menu that displays. The **Edit File** dialog box displays.
3. Select the desired netlist file from the **Files** list box.
4. Select **OK**. The netlist file displays.

Reading the netlist file

The figure below shows selected portions of the TUTOR4.NET EDIF 2 0 0 netlist file. Some of the lines are shown in bold text and are labeled with numbered callouts.

```
(edif &TUTORORC
 (edifVersion 2 0 0)
 (edifLevel 0)
 (keywordMap (keywordLevel 0))
 (status
  (written
   (timeStamp 0 0 0 0 0 0)
   (program "IFORM.EXE")
   (comment "Original data from OrCAD/SDT"))
  (comment "Digital clock schematic")
  (comment "    November 2, 1993")
        :
[several lines omitted]
        :
[Part List portion of netlist file]
 (external OrCAD_LIB
  (edifLevel 0)
  (technology
   (numberDefinition
    (scale 1 1 (unit distance))))
  (cell &CAP ①
   (cellType generic)
   (comment "From OrCAD library TUTOR.LIB")
   (view NetlistView
    (viewType netlist)
    (interface
     (port &1 (direction INOUT)) ②
     (port &2 (direction INOUT)))))
```

Example of EDIF 2 0 0 netlist format from TUTOR4.NET (continues).

```
        :
[several lines omitted]
        :
   (cell (rename &4SW_SPST "4SW SPST") ③
     (cellType generic)
     (comment "From OrCAD library TUTOR.LIB")
        :
[several lines omitted]
        :
     (instance &C4 ④
      (viewRef NetlistView
       (cellRef &CAP ⑤
        (libraryRef OrCAD_LIB)))
       (property PartValue (string ".01UF")) ⑥
       (property ModuleValue (string "CK12"))
       (property TimeStampValue (string "6EFD5407"))
       (property Field1Value (string ""))

        (property Field2Value (string ""))
       (property Field3Value (string "CK12"))
       (property Field4Value (string ""))
       (property Field5Value (string ""))
       (property Field6Value (string ""))
       (property Field7Value (string ""))
       (property Field8Value (string ""))
        :
[several lines omitted]
        :
[Netlist portion of the netlist file]
     (net N00001 ⑦
      (joined
       (portRef &2 (instanceRef &R2))
       (portRef &1 (instanceRef &U1))
       (portRef &3 (instanceRef &U1))
       (portRef &5 (instanceRef &U1))))
      (net &CLK_1
      (joined
       (portRef &10 (instanceRef &U1))
        :
[several lines omitted]
        :
```

Example of EDIF 2 0 0 netlist format from TUTOR4.NET.

TUTOR4.NET line descriptions

The following describes each line shown in bold in the previous figure.

① `CAP` is the name of a part in the SDT library. Every part on the schematic has a cell definition in the netlist file that defines the part.

② Defines the type of pin for pin number `1` of the `CAP` cell in the netlist file. This information comes from the SDT library file. A connectivity definition is indicated by `port`.

③ `4SW SPST` is the name of the part in the SDT library. However, the EDIF 2 0 0 netlist format has a limited character set which excludes space characters in any string. In this example, the space character was changed to an underscore character. A warning message originating from IFORM often accompanies this kind of renaming.

④ `C4` is an instance of the `CAP` cell in the netlist file. There may be several instances that reference the `CAP` cell in the netlist, but each will have a unique reference designator.

⑤ `cellRef` continues the definition of the `C4` object by referencing the `CAP` cell previously defined in the netlist.

⑥ `property` is used to associate a name like `PartValue`, or `ModuleName`, with a number or string value.

⑦ The netlist portion of the netlist file defines the connectivity. These lines indicate that net `N00001` connects pin number `2` of the instance with the reference designator `R2` with pin number `1` of the instance with the reference designator `U1`.

EDIF 2 0 0 names, identifiers, and characters

The following defines the terms that most frequently appear in an OrCAD-generated EDIF 2 0 0 netlist.

cell — The cell construct is used to define a part in the netlist file. The definition consists of information from the SDT library. This construct may be used again in another cell to build a design hierarchy.

cellRef — Used to reference a previously defined cell.

view — Used to specify a data representation of a cell.

contents — Consists of a detailed use of a view of a cell.

instance — Allows a view to be referenced within another view to create the instance design hierarchy.

port — Used as the basic means of communicating signal information between cells, and is the subject of all connectivity statements.

net — The net construct is provided within the contents of a view in order to describe its connectivity.

joined — Used to specify that certain ports are connected.

ref — Indicates a reference to a previously defined name.

property — Used to associate names with EDIF objects.

& — The ampersand is an identifier. Identifiers are used in an EDIF description to uniquely identify objects.

" " Strings are delimited by quotation marks.

( ) Parentheses enclose object descriptions.

Up button

Appears on the **Driver Configuration** dialog box when **HPGL, HPGL2,** or **HI29** are the selected vector devices.

Use to scroll the list of pen properties up.

Update Board File command

Appears on the board editor **QUIT** menu.

Writes the latest edits to the board file currently loaded in **Edit Layout**. If you did not enter a filename when you initialized the board file, the **Write Board File** dialog box displays when you select **Update Board File.**

To update the loaded board file, select **QUIT Update Board File. Edit Layout** saves the file to the same file name in the current working directory.

Update Board File also creates a backup file, which is the last saved version of the board. The backup file has a .BCK extension, and is also in the current working directory.

Update Library File command

Appears on the library editor **QUIT** menu.

Writes the latest edits to the library file currently loaded in **Edit Layout**. If you did not enter a filename when you initialized the library file, the **Write Library File** dialog box displays when you select **Update Library File.**

VERBOSE INQUIRE command

Appears on a number of menus.

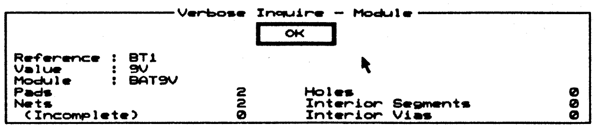
Displays information about the object at the pointer location:

- ❖ Module. Displays the **Verbose Inquire – Module** dialog box.
- ❖ Net. Displays the **Verbose Inquire – Net** dialog box.
- ❖ Other object. Displays appropriate information in the lower-right corner of the screen.

Verbose Inquire – Module dialog box

Shows the reference designator; value; module name; and the number of pads, complete nets, incomplete nets, holes, interior segments, and interior vias on the module.

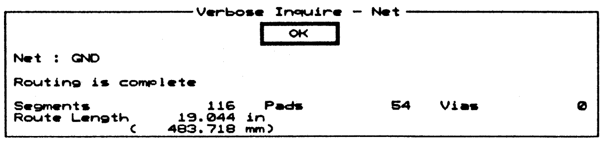
Select **OK** to dismiss the dialog box.



Verbose Inquire – Net dialog box

Shows the netname; the routing status; the number of pads, segments, and vias; and the net route length in inches and millimeters.

Select **OK** to dismiss the dialog box.



Via command

Appears on the **Begin** menu.

Places a via at the location of the pointer. The current layer automatically changes to the current layer's pair, as specified in the **Layer** dialog box.

Note that **/ other** has the same function as **Via** when routing.

Via Stack Editor button

Appears on a number of dialog boxes.

Displays the **Edit Via Stack** dialog box.

Via Stack Editor command

Appears on the **GO TO FUNCTION** menu.

Displays the **Edit Via Stack** dialog box.

Viewing fill zones

In the **Global Options** dialog box, enable **Show Copper Pour** to display fill zones. When **Show Copper Pour** is disabled, fill zones display as outlines.

Viewing module information

Modules are constructed of many graphic objects, such as outline segments, pads, text, holes, and zones. Select **INQUIRE, VERBOSE INQUIRE,** or **EDIT** to display information about these objects.

Whole Board command

Appears on the **Autorouter** menu.

Displays the menu shown at right.

```
Autoroute Whole Board
Spacing/DRC Check Whole Board
Set Sweep Window
```

Window command

Appears on the ZOOM menu.

Window works the same as the **WINDOW ZOOM** command.

WINDOW ZOOM command

Appears on a number of menus.

Zooms in or out on a precise area of the board, as specified by the window zoom boundary you draw.

See also *Defining zoom windows.*

Window Zoom End command

Appears on the **WINDOW ZOOM** menu.

Sets the location of the lower-right corner of the window zoom boundary.

Write Board File command

Appears on the board editor **QUIT** menu.

Displays the **Write Board File** dialog box.

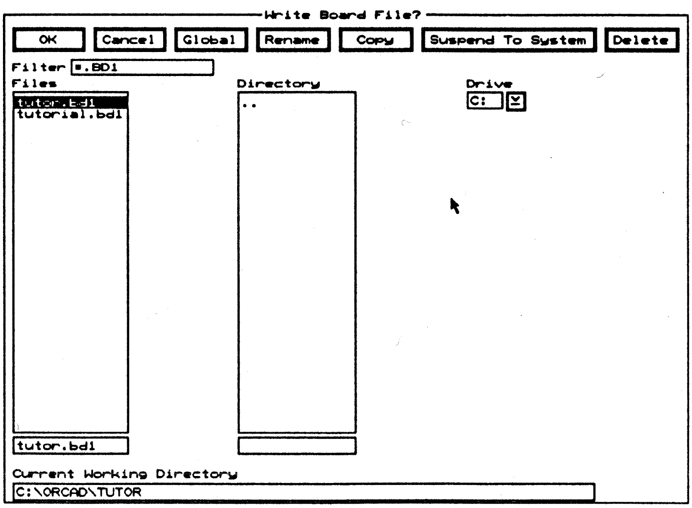
You can use **QUIT Write Board File** to create incremental backups of your board file. To save a board file under a different file name, follow these steps:

1. Select **QUIT Write Board File.** The **Write Board File** dialog box displays.
2. Enter a new filename in the entry box below the **Files** list box. You can also select a different directory or drive from the corresponding list box.
3. Select **OK** to save the new file to the destination shown in the **Current Working Directory** box. The dialog box closes and the **Edit Layout** screen displays.

See also *Write Board File dialog box.*

Write Board File dialog box

Use this dialog box to write the currently loaded board file to a specific board file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Removes the selected file from the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown.

Files Select a filename from the **Files** list box or create a new board file by entering a filename in the entry box below it.

If the file already exists, **Edit Layout** displays the **File Exists - OK to Overwrite** dialog box.

Enable **Do Not Prompt On OverWrite** in the **Global Options** dialog box to prevent the **File Exists - OK to Overwrite** dialog box from being displayed.

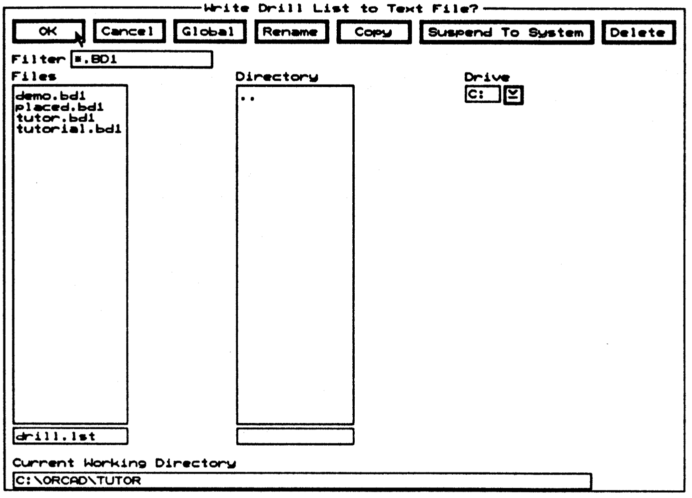
Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Write Drill List to Text File dialog box

Use this dialog box to write the currently loaded drill list to a specific text file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Removes the selected file from the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown.

Files Select a filename from the **Files** list box or create a new file by entering a filename in the entry box below it.

If the file already exists, **Edit Layout** displays the **File Exists - OK to Overwrite** dialog box.

Enable **Do Not Prompt On OverWrite** in the **Global Options** dialog box to prevent the **File Exists - OK to Overwrite** dialog box from being displayed.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

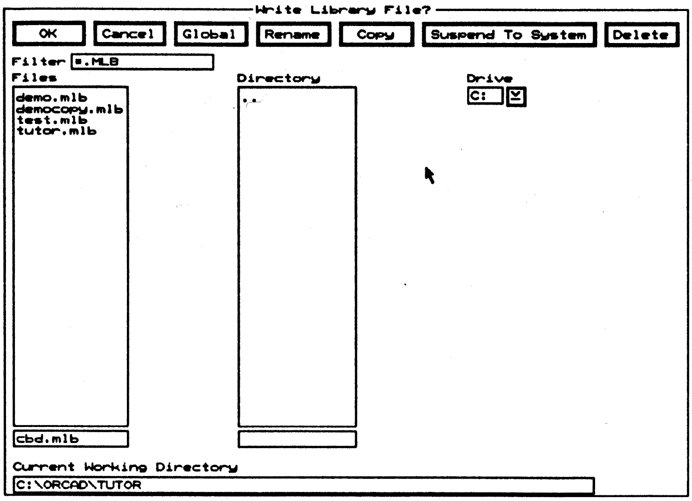
Write Library File command

Appears on the library editor **QUIT** menu.

Displays the **Write Library File** dialog box.

Write Library File dialog box

Use this dialog box to write the currently loaded library file to a specific library file.



Rename Displays the **Rename File** dialog box.

Copy Displays the **Copy File** dialog box.

Suspend To System Clears the dialog box and displays the system prompt.

Enter `exit` at the system prompt to return to the dialog box.

Delete Removes the selected file from the **Files** list box.

Filter Use any combination of wildcards (* or ?) and other characters in the entry box to restrict the list of files shown.

Files Select a filename from the **Files** list box or create a new library file by entering a filename in the entry box below it.

Directory Contains a list of all the subdirectories under the directory shown in the **Current Working Directory** entry box.

Drive Use to select another drive.

Current Working Directory Shows the path to your current working directory.

Write List button

Appears on the **Edit Drill List** dialog box.

Displays the **Write Drill List to Text File** dialog box.

X SHOW RATSNEST command

Appears on a number of menus.

See also *Ratsnests* and *RatsNest Block command.*

On the board editor main menu

If your pointer is on a pad that has a netname assigned to it, **X SHOW RATSNEST** displays a full ratsnest for that net.

If your pointer is on a reference designator, part name, or module name, **X SHOW RATSNEST** displays a full ratsnest for all the nets associated with that module.

To remove ratsnests from a pad or a module, place the pointer in an unoccupied area of the board and select **X SHOW RATSNEST** again.

On the library editor main menu

This command does not apply to the library editor.

Zone Properties button

Appears on a number of dialog boxes.

Displays the **Edit Zone Properties** dialog box.

Zone types

Zones in **PCB 386+ 2.00** are bounded by a continuous series of segments and arcs. The following types of zones are available:

- Fill
- No-fill
- Autoroute
- No-autoroute
- No-through
- Polygon

Fill and no-fill zones control copper pours. Autoroute, no-autoroute, and no-through zones control the autorouter.

See also *Placing zones.*

ZOOM command

Appears on a number of menus.

Displays the menu shown at right.

```
Center
In
Out
Previous
Refresh
Set Scale
  1
  2
  3
  4
  5
  6
  7
  8
  9
 10
 20     T
 50     F
100     H
Window
```

Zooms in on or out, changing the size of the displayed objects and thus the amount of detail you see on the screen.

The numeric values in the menu represent the number of mils per displayed pixel. A zoom scale of 1, then, means 1 pixel = 1 mil (0.001 in.); a scale of 5 means 1 pixel = 5 mils (0.005 in.); a scale of .01 means 1 pixel = 0.01 mil (0.00001 in.), or 100 pixels = 1 mil. You can set the zoom scale to any number from 0.01 (maximum magnification) to 100 (minimum magnification).

The current zoom scale is displayed near the bottom-left corner of the screen, just to the right of the pointer coordinates.

See also *Defining zoom windows.*

1 through 100 H commands

Appear on the ZOOM menu.

1	Changes the display to a scale of 1 pixel = 1 mil (0.001 in.).
2	Changes the display to a scale of 1 pixel = 2 mils (0.002 in.).
3	Changes the display to a scale of 1 pixel = 3 mils (0.003 in.).
4	Changes the display to a scale of 1 pixel = 4 mils (0.004 in.).
5	Changes the display to a scale of 1 pixel = 5 mils (0.005 in.).
6	Changes the display to a scale of 1 pixel = 6 mils (0.006 in.).
7	Changes the display to a scale of 1 pixel = 7 mils (0.007 in.).
8	Changes the display to a scale of 1 pixel = 8 mils (0.008 in.).
9	Changes the display to a scale of 1 pixel = 9 mils (0.009 in.).
10	Changes the display to a scale of 1 pixel = 10 mils (0.01 in.).
20 T	Changes the display to a scale of 1 pixel = 20 mils (0.02 in.).
50 F	Changes the display to a scale of 1 pixel = 50 mils (0.05 in.).
100 H	Changes the display to a scale of 1 pixel = 100 mils (0.1 in.).

= BOOKMARK command

Appears on a number of menus.

Displays the **Bookmark** dialog box.

+ LAYER command

Appears on a number of menus.

Use **+ LAYER** to move the current layer "down" through the enabled copper layers. For example, if the current layer is Internal Copper 5, and Internal Copper 6 is enabled, select **+ LAYER** to change the current layer to Internal Copper 6. Note that when you have reached the last enabled copper layer **+ LAYER** will move you back up to the "top" enabled copper layer.

– LAYER command

Appears on a number of menus.

Use **– LAYER** to move the current layer "up" through the enabled copper layers. For example, if the current layer is Internal Copper 5, and Internal Copper 4 is enabled, select **– LAYER** to change the current layer to Internal Copper 4. Note that when you have reached the last enabled copper layer **– LAYER** will move you back down to the "bottom" enabled copper layer.

* LAYER command

Appears on a number of menus.

Use *** LAYER** to set the current layer to All Layers.

/ OTHER command

Appears on a number of menus.

On the main menu

Use **/ OTHER** to toggle the current layer back and forth between the layers selected in the **Copper Pairs** area on the **Layer** dialog box.

On the Begin menu

Places a via at the pointer's location and automatically changes the current layer to the current layer's pair, as specified in the **Layer** dialog box.

? dialog boxes

During autorouting, any of the following messages may display.

More than one autoroute zone found on layer *n*. Results on that layer unpredictable.

There are multiple autoroute zones on the named layer. An autoroute zone will be created around all copper objects, which may not be optimal.

Select **OK** to continue the autoroute, but the results are unpredictable.

Select **Cancel** to cancel the autoroute, and display the board editor.

Select **OK to All** to prevent this message from displaying during any subsequent attempts to autoroute the board.

Net *n* has no enabled layer pads/test points.

There are no pads or test points on enabled layers for the named net, which may be acceptable. Delete the net to prevent the dialog box from displaying again.

Select **OK** to continue the autoroute.

Select **Cancel** to cancel the autoroute, and display the board editor.

Select **OK to All** to prevent this message from displaying during any subsequent attempts to autoroute the board.

Net *n* has only one enabled layer pad/test point.

There is only one pad or test point on enabled layers for the named net, which may be acceptable. Place additional modules or test points to prevent the dialog box from displaying again.

Select **OK** to continue the autoroute.

Select **Cancel** to cancel the autoroute, and display the board editor.

Select **OK to All** to prevent this message from displaying during any subsequent attempts to autoroute the board.

No autoroute zone found. A temporary one will be created.

There are no autoroute zones on any copper layer. An autoroute zone will be created around all copper objects, which may not be optimal.

Select **OK** to have the autorouter create an autoroute zone on the named layer.

Select **Cancel** to cancel the autoroute, and display the board editor.

Select **OK to All** to prevent this message from displaying during any subsequent attempts to autoroute the board.

No autoroute zone found on layer *n*. A temporary one will be created.

One or more autoroute zones have been found, but not on the named copper layer. An autoroute zone will be created around all copper objects, which may not be optimal.

Select **OK** to have **Edit Layout** create an autoroute zone on all layers or all copper layers.

Select **Cancel** to cancel the autoroute, and display the board editor.

Select **OK to All** to prevent this message from displaying during any subsequent attempts to autoroute the board.

Pad Stack *n* is not defined on any copper layers.

The named pad stack is not defined on any copper layer, which may be acceptable. Redefine or delete the pad stack to prevent the message from displaying again.

Select **OK** to continue the autoroute.

Select **Cancel** to cancel the autoroute, and display the board editor.

Via Stack *n* is not defined on any copper layers.

The named via stack is not defined on any copper layer, which may be acceptable. Redefine or delete the via stack to prevent the message from displaying again.

Select **OK** to continue the autoroute.

Select **Cancel** to cancel the autoroute, and display the board editor.

? CONDITIONS command

Appears on a number of menus.

Displays the **Conditions** dialog box.

% MACRO command

Appears on a number of menus.

Displays the **Press Macro Capture Key** dialog box.

See also *Macros*.

> Rotate Clockwise command

Appears on a number of menus.

Rotates the selected objects clockwise by the current rotation step angle.

<Rotate Counter Clockwise command

Appears on a number of menus.

Rotates the selected objects counterclockwise by the current rotation step angle.

Edit File

Edit File runs a text editor. When you receive the design environment from OrCAD, it is configured to run a text editor called M2EDIT. However, you can configure the design environment to run the text editor of your choice.

For instructions on how to configure the design environment to run your text editor, see the *ESP Design Environment User's Guide*. To use the M2EDIT editor, see the *Stony Brook M2EDIT Text Editor User's Guide.*

Execution

With the **PC Board Layout Tools** screen displayed, select **Edit File**. Select **Execute** from the menu that displays. The **Edit File** dialog box displays. Select a file from the **Files** list box, or enter a name in the **File to Edit** entry box, and then select **OK**. The screen for the configured text editor displays.

View Reference

View Reference runs a text editor in a reference material directory provided by OrCAD. This directory contains supplemental "read me" files of product information. These files contain information about:

❖ Drivers supported by the design environment

❖ Drivers that can be made using GENDRIVE

❖ Libraries included and modules found in each

When you receive the design environment from OrCAD, it is configured to run a text editor called M2EDIT; however, you can configure it to run the text editor of your choice.

For instructions on how to configure the design environment to run your text editor, see the *ESP Design Environment User's Guide.* To use the M2EDIT editor, see the *Stony Brook M2EDIT Text Editor User's Guide.*

Execution

With the **PC Board Layout Tools** screen displayed, select **View Reference**. Select **Execute** from the menu that displays. The **Edit File** dialog box displays. Select a file from the **Files** list box, or enter a name in the **File to Edit** entry box, and then select **OK**. The screen for the configured text editor displays. Use the text editor to open and read the reference file of your choice.

PART III: PROCESSORS

PCB 386+ 2.00 includes processors that read and modify the design database. Some of the processors also create reports.

Part III: Processors describes processors and provides instructions for their use.

Chapter 5: *Modify Modules* describes how **Modify Modules** modifies pad shape, pad size, and drill size for modules in a board file.

Chapter 6: *Create NC Drill File* describes how **Create NC Drill File** generates a file containing drilling information, including location and drill size, for a board file.

Chapter 7: *Reannotate Board File* describes how **Reannotate Board File** reannotates your board file so the modules are numbered sequentially.

Chapter 8: *Fix Time Stamps* describes how **Fix Time Stamps** sets the time stamps in your board file to match the time stamps in a netlist file.

Note that the processors take advantage of EMS, if it is present on your system. Also, you can interrupt any of the processors by pressing <Ctrl><Break>.

Modify Modules

Modify Modules modifies the shape, size, type, orientation, drill size, and layers of pads in modules in a board file. **Modify Modules** also creates a module information report. The changes can be made to all pads or to a specific set of pads.

An advantage to using **Modify Modules** to modify pads is that **Modify Modules** changes pads globally. Using **Modify Modules**, you can easily change the characteristics of a group of pads.

It is also easy to create new modules using **Modify Modules** by changing particular characteristics of existing modules.

△ *NOTE: Modify Modules affects only pad characteristics. It does not affect routes or other board features.*

Execution

With the **PC Board Layout Tools** screen displayed, select **Modify Modules**. Select **Execute** from the menu that displays.

While **Modify Modules** runs, messages display at the bottom of the screen. When **Modify Modules** is complete, the **PC Board Layout Tools** screen displays.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Modify Modules.** Select **Local Configuration** from the menu that displays.

Select **Configure MODMOD_.** The **Modify Modules** local configuration screen displays (figure 5-1).

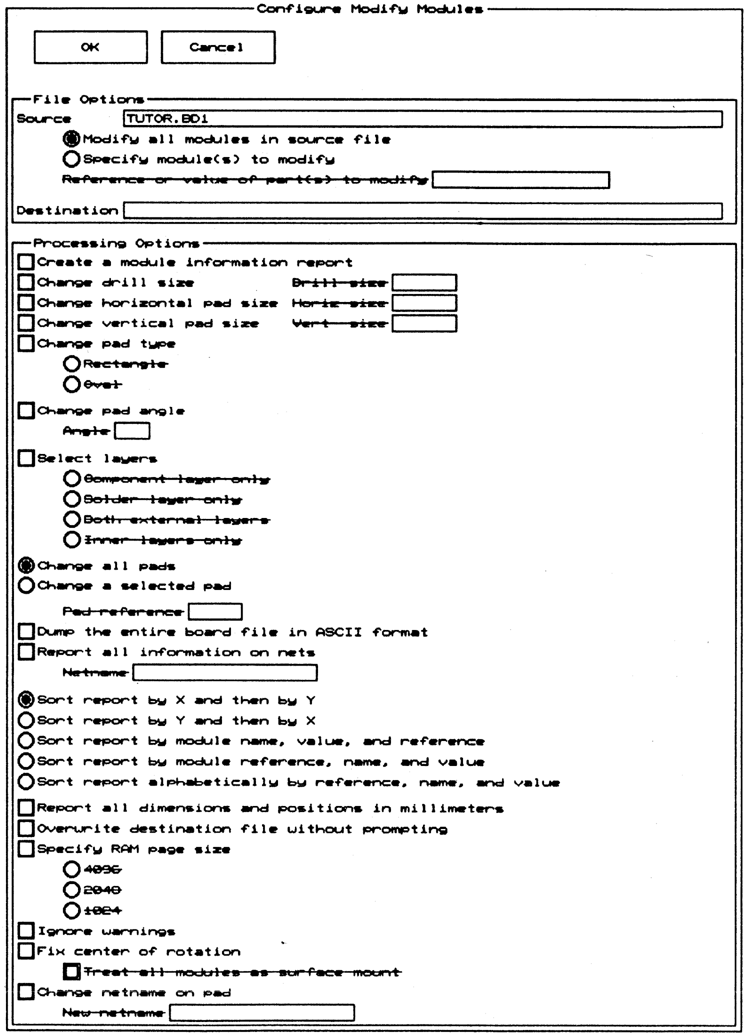


Figure 5-1. Local configuration screen for ***Modify Modules****.*

File Options The **File Options** area defines the source and destination files.

Source **Source** is a board file to modify. It may have any valid pathname. The source is originally set to *rootSheet*.BD1. Source can also be and ASCII file created by **MODMOD_**. If this is the case, output must then be the name of the board file to be created from the ASCII file.

Enter the source filename, then select one of the following options:

○ `Modify all modules in source file`

Tells **Modify Modules** to modify all the modules in the source file.

○ `Specify module(s) to modify`

`Reference or value of part(s) to modify` [ ]

Tells **Modify Modules** to modify only specific modules. When you select this option, **Reference or value of part(s) to modify** becomes available. Enter a character string representing the module reference designator, module name, or any character string associated with the modules to modify.

Use commas to separate multiple strings. You can enter up to 18 characters in the entry box.

△ *NOTE: In **PC Board Layout Tools**, module names can be up to 63 characters in length.*

Use asterisks or question marks as wildcards in this entry box. An asterisk represents multiple characters, while a question mark indicates a single character wildcard.

▲ *CAUTION: Wildcards can be dangerous because you might modify modules you do not want to modify. Before using wildcards, be sure you have a backup of your board file and modules.*

To see which modules match a wildcard, use **Module Report** with the following configuration:

```
Source          ORIGINAL.BD1

    ○ Specify module(s) to report

    Reference or value
    of part(s) to report    wildcard

Destination     MATCHES.LOC

  ❑ List module names only in report file
```

Edit the report file, and delete any module names you do not want to modify. Then run **Modify Modules** with the following configuration in addition to any other options you select:

```
Source          ORIGINAL.BD1

    ○ Specify module(s) to modify

    Reference or value
    of part(s) to modify    @MATCHES.LOC

Destination     NEW.BD1
```

Only the modules listed in the file MATCHES.LOC are affected.

Destination

The **Destination** is the board file or report file where the output is placed. It may have any valid pathname. Once a report is created, you can view it using **Edit File** or another text editor.

You must specify a destination for **Modify Modules** to work properly.

Processing Options

Select any combination of the following options:

- ❑ `Create a module information report`

  Tells **Modify Modules** to output a report with the information about the modules in the source board file. When you select this option, several options that don't apply to reporting become unavailable.

- ❑ `Change drill size` `Drill size` [ ]

  Tells **Modify Modules** to change the drill size of the designated pads. When you select this option, the **Drill size** entry box becomes available.

- ❑ `Change horizontal pad size` `Horiz size` [ ]

  Tells **Modify Modules** to change the horizontal pad size of the designated pads. When you select this option, the **Horiz size** entry box becomes available.

- ❑ `Change vertical pad size` `Vert size` [ ]

  Tells **Modify Modules** to change the vertical pad size of the designated pads. When you select this option, the **Vert size** entry box becomes available.

- ❑ `Change pad type`
  - ○ `Rectangle`
  - ○ `Oval`

  Tells **Modify Modules** to change the pad type of the selected pads. When you select this option, the pad type options become available. Select **Rectangle** or **Oval.**

- ❑ `Change pad angle`
  `Angle` [ ]

  Tells **Modify Modules** to change the pad orientation of the designated pads. When you select this option, the **Degrees** entry box becomes available.

- ❑ `Select layers`
  - ○ `Component layer only`
  - ○ `Solder layer only`
  - ○ `Both external layers`
  - ○ `Inner layers only`

  Tells **Modify Modules** which sides of the layout to modify. When you select this option, the side options become available.

Select either of the following pad selection options:

- ○ `Change all pads`

  Tells **Modify Modules** to make the selected changes to all pads in the board file.

- ○ `Change a selected pad`
  `Pad reference` [ ]

  Tells **Modify Modules** to make the selected changes to only the specified pads. When you select this option, the **Pad Reference** entry box becomes available.

  You can use commas and hyphens to specify multiple pads, as in "2-3" and "1,3–5."

- ❑ `Dump the entire board file in ASCII format`

  Tells **Modify Modules** to create an ASCII version of the entire board file.

- ❑ `Report/Edit all information on nets`
  `Netname` [ ]

  Tells **Modify Modules** to report all information about the nets on the board file. When you select this option, the **Netname** entry box becomes available. If you do not specify a netname, **Modify Modules** reports on all nets.

  This option can also limit pad changes to only those pads on a specified net.

Select one of the following five report sorting options:

- ○ `Sort report by X and then by Y`

  Tells **Modify Modules** to sort modules by X coordinate, then by Y coordinate.

- ○ `Sort report by Y and then by X`

  Tells **Modify Modules** to sort modules by Y coordinate, then by X coordinate.

- ○ `Sort report by module name, value, and reference`

  Tells **Modify Modules** to sort modules by name, then by value and reference.

- ○ `Sort report by module reference, name, and value`

  Tells **Modify Modules** to sort modules by reference, then by name and value. Reference strings are assumed to be a letter followed by a number.

- ○ `Sort report alphabetically by reference, name, and value`

  Tells **Modify Modules** to sort modules by reference, then by name and value. Reference strings are collated alphabetically.

- ❑ `Report all dimensions and positions in millimeters`

  Tells **Modify Modules** to interpret and report sizes, dimensions, and positions in millimeters.

- ❑ `Overwrite destination file without prompting`

  Tells **Modify Modules** to overwrite any existing version of the destination file without prompting for permission.

❑ `Specify RAM page size`

○ `4096`

○ `2048`

○ `1024`

Tells **Modify Modules** to use a smaller RAM page size to free memory for processing. When you select this option, three page-size radio buttons become available. Select decreasing page sizes until **Modify Modules** completes successfully.

❑ `Ignore warnings`

Tells **Modify Modules** to leave the return code set to 0 (i.e., continue processing) after issuing warning messages.

❑ `Fix center of rotation`

Tells **Modify Modules** to set the rotation center to the X and Y coordinates of pin 1 for through-hole devices and to the center of gravity for surface mount devices. When you select this option, the following option becomes available.

❑ `Treat all modules as surface mount`

Tells **Modify Modules** to set the rotation center to the center of gravity for all specified parts, as if all are surface mount devices.

❑ `Change netname on pad`

`New netname` [ ]

Tells **Modify Modules** to add the specified netname or change the netname to the specified netname for all specified pads.

Example

The following example demonstrates how to use **Modify Modules**.

Changing the shape of a pin on each module

You can use **Modify Module** to change the pad characteristics of modules that are already placed in a layout. For example, suppose you want to change the pad characteristics of modules in a board file.

To change the shape of pad 3 on every 74LS00 module on the layout, run **Modify Modules** with the following configuration:

```
Source        [SAMPLE.BD1                              ]

     O Specify module(s) to modify

     Reference or value of part(s)
     to modify [74LS00  ]

Destination   [NEW.BD1                                 ]

❑ Change pad type
     O Rectangle

❑ Change pad angle
     Angle [90 ]

O Change a selected pad
     Pad reference [3    ]
```

Modify Modules creates a new board file called NEW.BD1. All the 74LS00 modules in this file have the pad 3 shape set to rectangle. The original board file, SAMPLE.BD1, is unaffected.

To view the pad characteristics of the 74LS00 modules on NEW.BD1, run **Modify Modules** with the following configuration:

```
Source        [NEW.BD1]

    O Specify module(s) to modify

    Reference or value of part(s)
    to modify [74LS00]

Destination   [NEW.TXT]

❑ Create a module information report
```

Create NC Drill File

Create NC Drill File generates a file containing drilling information, including location and diameter, for a board file. **Create NC Drill File** also generates a tool list (.TOL) file and a tool count (.TC) file.

Execution

With the **PC Board Layout Tools** screen displayed, select **Create NC Drill File**. Select **Execute** from the menu that displays.

While **Create NC Drill File** runs, messages display at the bottom of the screen. When **Create NC Drill File** is complete, the **PC Board Layout Tools** screen displays.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Create NC Drill File.** Select **Local Configuration** from the menu that displays.

Select **Configure NCDRILL_. Create NC Drill File's** local configuration screen displays (figure 6-1).

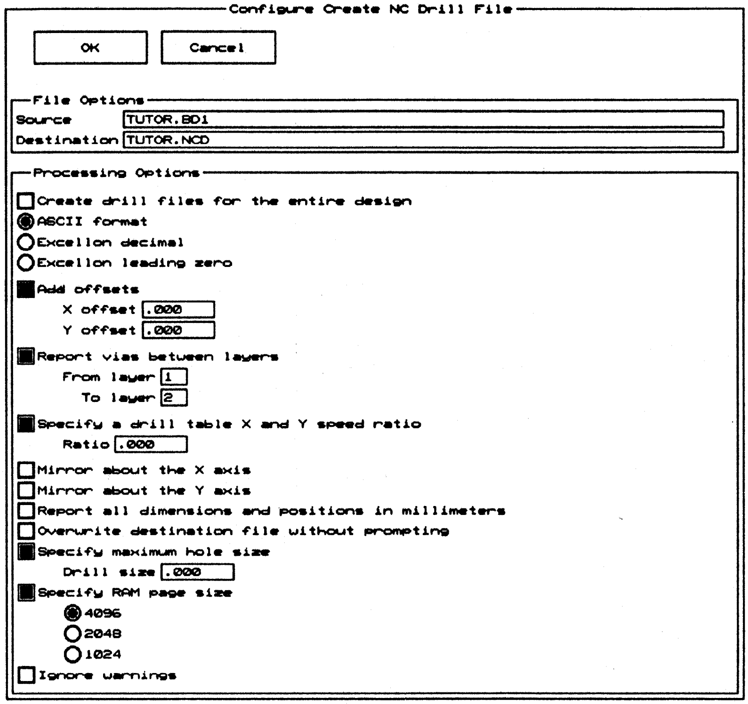


Figure 6-1. Create NC Drill File's local configuration screen.

File Options

The **File Options** area defines the source and destination files.

Source

Source is the board file on which to report. It may have any valid pathname. The source is originally set to *rootSheet*.BD1.

Destination

Destination is the file where the report is placed. It may have any valid pathname. The destination is originally set to *rootSheet*.NCD. Once the report is created, you can view it using **Edit File** or another text editor.

You must specify a destination for **Create NC Drill File** to work properly.

Processing Options

The **Processing Options** area specifies various characteristics of the output.

❑ `Create drill files for the entire design`

Tells **Create NC Drill File** to create drill files for the entire design. When you select this option, **Create NC Drill File** makes the **Report vias between layers** option and the **Destination** entry box unavailable. It creates a file named *startLayer_stopLayer*.NCD for each layer pair that contains a hole to be drilled.

Select one of the following three output format options:

- ○ `ASCII format`

  Tells **Create NC Drill File** to produce output in ASCII format.

- ○ `Excellon decimal`

  Tells **Create NC Drill File** to produce output in decimal format, with the decimal point explicitly placed.

- ○ `Excellon leading zero`

  Tells **Create NC Drill File** to produce output in leading-zero format, with trailing zeros suppressed.

- ○ `Excellon trailing zero`

  Tells **Create NC Drill File** to produce output in trailing-zero format, with leading zeros suppressed.

- ❑ `Add offsets`

  `X offset` [ ]

  `Y offset` [ ]

  Tells **Create NC Drill File** to add offsets to all locations. When you select this option, the **X offset** and **Y offset** entry boxes become available.

- ❑ `Report vias between layers`

  `From layer` [ ]

  `To layer` [ ]

  Tells **Create NC Drill File** to report all vias between the layers specified in the **From layer** and **To layer** entry boxes, which become available when you select this option.

  The allowed values are C (or 0) for the component layer, 1–14 for inner layers, and S (or 15) for the solder layer.

- `Specify a drill table X and Y speed ratio`

  `Ratio` [ ]

  Tells **Create NC Drill File** to sort the output to minimize the time required to drill the board on a drill table having the specified ratio of X speed over Y speed. When you select this option, the **Ratio** entry box becomes available.

- `Mirror about the X axis`

  Tells **Create NC Drill File** to flip drill information with respect to the X axis.

- `Mirror about the Y axis`

  Tells **Create NC Drill File** to flip drill information with respect to the Y axis.

- `Report all dimensions and positions in millimeters`

  Tells **Create NC Drill File** to interpret and report sizes, dimensions, and positions in millimeters.

- `Overwrite destination file without prompting`

  Tells **Create NC Drill File** to overwrite any existing version of the destination file without prompting for permission.

- `Specify maximum hole size`

  `Drill size` [ ]

  Tells **Create NC Drill File** to generate a route command, rather than a drill command, for holes larger than the specified drill size.

- `Ignore warnings`

  Tells **Create NC Drill File** to leave the return code set to 0 (i.e., continue processing) after issuing warning messages.

Examples

The following examples demonstrate how to use **Create NC Drill File**.

Drill size report

To create a drill size report, run **Create NC Drill File** with the following configuration:

```
Source       [SAMPLE.BD1]
Destination  [SAMPLE.NCD]

O ASCII format

❑ Report vias between layers
     From layer [S]
       To layer [C]
```

Create NC Drill File creates a text file called SAMPLE.NCD and writes the drill size and location for each pad and via on the board.

Each line begins with the word ASCII, followed by three numbers in parenthesis. The first number is the drill diameter, the second number is the X coordinate of the hole, and the third number is the Y coordinate of the hole. The report is listed in order of size first, and then in an order that minimizes drilling time.

Conveying drill hole information to a drilling machine

Suppose you want to convey drill hole information to a drilling machine that requires Excellon trailing-zero format. Run **Create NC Drill File** with the following configuration:

```
Source       [SAMPLE.BD1]
Destination  [SAMPLE.NCD]

O Excellon decimal

O Overwrite destination file without prompting
```

If SAMPLE.NCD already exists, you must tell **Create NC Drill File** to overwrite it; otherwise, the message "Destination file already exists, over write it? (y/n)" displays.

Reannotate Board File

Reannotate Board File reannotates your board file so the modules are numbered sequentially. You can reannotate specific modules, or all modules in a board file.

Normally, only modules with references that begin with alpha characters and three asterisks (such as ABC*** and R***) are reannotated. If you specify unconditional reannotation on the **Configure Reannotate Board File** screen, all modules are renumbered sequentially, beginning with the number 1. See *Local configuration* in this chapter for a description of the **Configure Reannotate Board File** screen.

Execution

With the **PC Board Layout Tools** screen displayed, select **Reannotate Board File.** Select **Execute** from the menu that displays.

While **Reannotate Board File** runs, messages display at the bottom of the screen. When **Reannotate Board File** is complete, the **PC Board Layout Tools** screen displays.

Updating the schematic file

In addition to the reannotated board file, **Reannotate Board File** also creates a file called WAS_IS.REA. This file has the same format as the file WAS_IS, created by **Compare Netlists**. See *Chapter 10: Compare Netlists* for a description of the WAS_IS file.

You must run **Back Annotate** in **Schematic Design Tools 386+**, using the WAS_IS.REA file, every time you use **Reannotate Board File**, so that your schematic file and your board file match. An easy way to run **Reannotate Board File** is to use the **To Schematic** transfer. See *Chapter 13: To Schematic* for information about this transfer, or see *Chapter 7: Back Annotate* in the *Schematic Design Tools Reference Guide*.

△ *NOTE: If you use an include file with* ***Reannotate Board File****, a WAS_IS.REA file is not created. In this case, use the include file with* ***Back Annotate*** *to update your schematic. Include files are discussed in the* File options *section of this chapter.*

Adding modules

If you plan to add modules to your layout, and you want your layout and schematic files to match, you should first add the parts to your schematic file and bring them forward to your layout file.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Reannotate Board File**. Select **Local Configuration** from the menu that displays.

Select **Configure REANNO_**. **Reannotate Board File's** local configuration screen displays (figure 7-1).

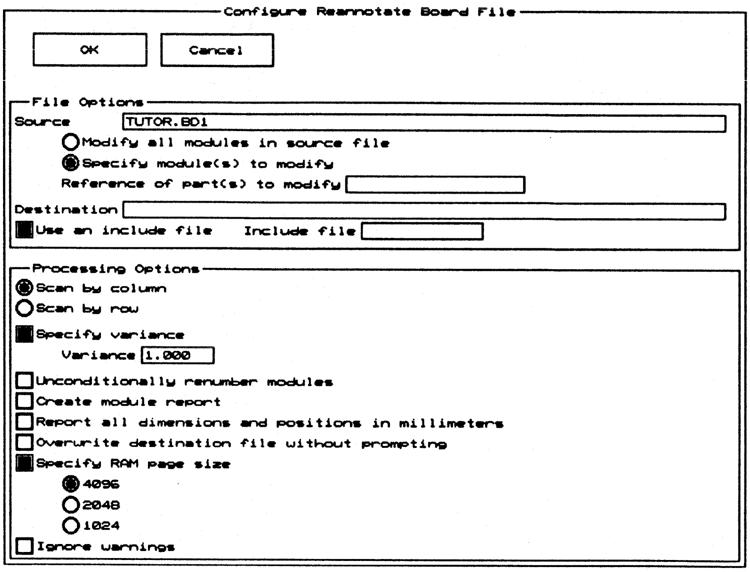


Figure 7-1. Reannotate Board File's local configuration screen.

File Options

The **File Options** area defines the source, destination, and optional include files.

Source

Source is the board file to modify. It may have any valid pathname. The source is originally set to *rootSheet*.BD1.

Enter the source filename, then select one of the following options:

○ `Modify all modules in source file`

Tells **Reannotate Board File** to modify all the modules in the source file.

○ `Specify module(s) to modify`

`Reference of part(s) to modify` [ ]

Tells **Reannotate Board File** to modify only specific modules. When you select this option, **Reference of part(s) to modify** becomes available. Enter a character string representing the module reference designators associated with the modules to modify.

Use asterisks or question marks as wildcards in this entry box. An asterisk represents multiple characters, while a question mark indicates a single character wildcard.

Use commas to separate multiple strings. You can enter up to 18 characters in the entry box.

Destination

Destination is the newly reannotated board file. It may have any valid pathname.

You must specify a destination for **Reannotate Board File** to work properly.

Include file

❑ `Use an include file` `Include file` [ ]

Tells **Reannotate Board File** to use an include file. When you select this option, the **Include file** entry box becomes available. **Reannotate Board File** modifies only the modules listed in the include file.

If the modules you want to modify have no common text string that distinguishes them from modules you wish to leave unchanged, you can specify them in an *include file* instead of selecting **Specify module(s) to modify**. The include file is an ASCII text file containing a list of reference designators and the values that should replace them.

```
U7    U1
U4    U2
U12   U3
```

An include file.

As shown in the example at right, each line of an include file contains the reference designator that **Reannotate Board File** looks for, followed by the new reference designator that will replace it. The items are separated by one or more blank spaces.

Processing Options

Select either of the following scan options:

- ○ `Scan by column`

  Tells **Reannotate Board File** to reannotate your layout by column.

- ○ `Scan by row`

  Tells **Reannotate Board File** to reannotate your layout by row.

- ☐ `Specify variance`

  `Variance` [ ]

  Tells **Reannotate Board File** to allow the specified variance in module X coordinates when calculating module sequence numbers. When you select this option, the **Variance** entry box becomes available. The default variance is one-quarter (0.25) inch.

  Reannotate Board File assigns sequence numbers by:

  1. Finding the leftmost module (the module with the smallest X coordinate) not yet reannotated.
  2. Defining a "column" starting at the X coordinate of that module's anchor-point and extending to the right by the specified distance (variance).
  3. Finding all modules not yet reannotated whose anchor point falls within that column.
  4. Assigning the next available sequence number to the uppermost module (the module with the smallest Y coordinate) of that group.
  5. Repeating steps 1 through 4 until all modules are reannotated.

- ❑ `Unconditionally renumber modules`

  Tells **Reannotate Board File** to unconditionally renumber the modules in your board file sequentially, beginning with 1, regardless of any pre-existing annotation.

  If you don't select this option, **Reannotate Board File** resequences only modules with designators that begin with a letter and three asterisks (such as R***), which are assigned any numbers not used in the sequence of existing reference designators.

- ❑ `Create module report`

  Tells **Reannotate Board File** to report module text, locations, and time stamps. The report is sorted by the first letter of the reference designator, then by row or column (as selected above).

△ ***NOTE:*** *If you select **Use an include file**, **Reannotate Board File** does not create a WAS_IS.REA file.*

- ❑ `Report all dimensions and positions in millimeters`

  Tells **Reannotate Board File** to interpret and report sizes, dimensions, and positions in millimeters.

- ❑ `Overwrite destination file without prompting`

  Tells **Reannotate Board File** to overwrite any existing version of the destination file without prompting for permission.

- ❑ `Specify RAM page size`
  - ○ `4096`
  - ○ `2048`
  - ○ `1024`

  Tells **Reannotate Board File** to use a smaller RAM page size to free memory for processing. When you select this option, three page-size radio buttons become available. Select decreasing page sizes until **Reannotate Board File** completes successfully.

- ❑ `Ignore warnings`

  Tells **Reannotate Board File** to leave the return code set to 0 after issuing warning messages.

CHAPTER 8

Fix Time Stamps

Fix Time Stamps sets the time stamp fields inside a board file to match the time stamps in a netlist file. Modules with the same reference designator in the netlist and board files are assumed to be the same module.

Sometimes the mapping between netlist modules and board file modules cannot be determined by matching reference designators. You can also use **Fix Time Stamps** to create a report, use **Edit File** to indicate the mapping by hand, and specify the modified report in place of the source board file.

Execution

With the **PC Board Layout Tools** screen displayed, select **Fix Time Stamps.** Select **Execute** from the menu that displays.

While **Fix Time Stamps** runs, messages display at the bottom of the screen. When **Fix Time Stamps** is complete, the **PC Board Layout Tools** screen displays.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Fix Time Stamps**. Select **Local Configuration** from the menu that displays.

Select **Configure FIXTIME_**. The **Fix Time Stamps** local configuration screen displays (figure 8-1).

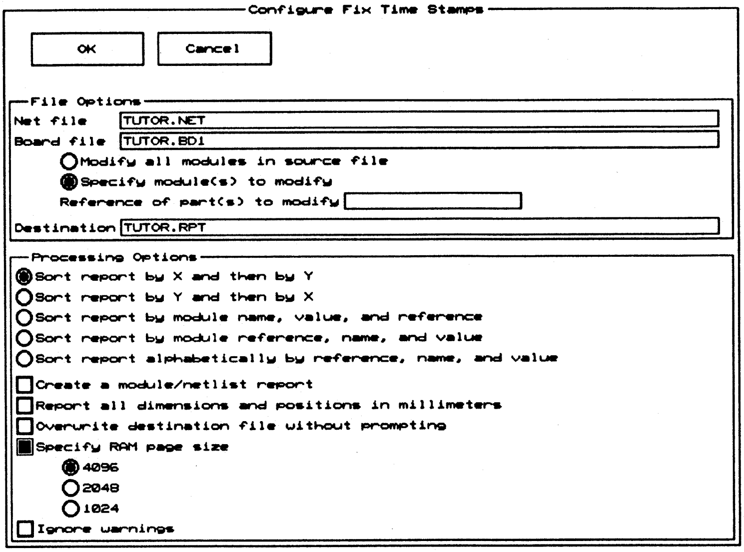


Figure 8-1. Local configuration screen for ***Fix Time Stamps.***

File Options

The **File Options** area defines the source and destination files.

Net file

Net file is the EDIF netlist file to read. It may have any valid pathname. The source is originally set to *rootSheet*.NET.

Board file

Board file is the board file to change. It may have any valid pathname. The source is originally set to *rootSheet*.BD1.

Enter the net and board filenames, then select one of the following options:

○ `Modify all modules in source file`

Tells **Fix Time Stamps** to modify all the modules in the source file.

○ `Specify module(s) to modify`

`Reference of part(s) to modify` [ ]

Tells **Fix Time Stamps** to modify only specific modules. When you select this option, **Reference of part(s) to modify** becomes available. Enter a character string representing the module reference designator.

Use asterisks or question marks as wildcards in this entry box. An asterisk represents multiple characters, while a question mark indicates a single character wildcard.

Use commas to separate multiple strings. You can enter up to 18 characters.

Destination

Destination is the file where the report or the modified board file is placed. It may have any valid pathname. The destination is originally set to *rootSheet*.RPT.

You must specify a destination for **Fix Time Stamps** to work properly.

Processing Options

The **Processing Options** area specifies various characteristics of the output.

Select one of the following five report sorting options:

- O `Sort report by X and then by Y`

  Tells **Fix Time Stamps** to sort modules by X coordinate, then by Y coordinate.

- O `Sort report by Y and then by X`

  Tells **Fix Time Stamps** to sort modules by Y coordinate, then by X coordinate.

- O `Sort report by module name, value, and reference`

  Tells **Fix Time Stamps** to sort modules by name, then by value and reference.

- O `Sort report by module reference, name, and value`

  Tells **Fix Time Stamps** to sort modules by reference, then by name and value. Reference strings are assumed to be a letter followed by a number.

- O `Sort report alphabetically by reference, name, and value`

  Tells **Fix Time Stamps** to sort modules by reference, then by name and value. Reference strings are collated alphabetically.

- ❑ `Create a module/netlist report`

  Tells **Fix Time Stamps** to report the reference text, location, and time stamp for the selected modules.

- ❑ `Report all dimensions and positions in millimeters`

  Tells **Fix Time Stamps** to interpret and report sizes, dimensions, and positions in millimeters.

- ❑ `Overwrite destination file without prompting`

  Tells **Fix Time Stamps** to overwrite any existing version of the destination file without prompting for permission.

- ❑ `Specify RAM page size`
  - ❍ `4096`
  - ❍ `2048`
  - ❍ `1024`

  Tells **Fix Time Stamps** to use a smaller RAM page size to free memory for processing. When you select this option, three page-size radio buttons become available. Select decreasing page sizes until **Fix Time Stamps** completes successfully.

- ❑ `Ignore warnings`

  Tells **Fix Time Stamps** to leave the return code set to 0 after issuing warning messages.

PART IV: REPORTERS

PCB 386+ 2.00 includes reporters that create reports without changing the design database in any way.

Part IV: Reporters describes reporters and provides instructions for their use.

Chapter 9: *Module Report* describes how **Module Report** reports module locations in your board file.

Chapter 10: *Compare Netlists* describes how **Compare Netlists** reports differences between an EDIF netlist file and a board file.

Note that the reporters take advantage of EMS, if it is present on your system. Also, you can interrupt either reporter by pressing <Ctrl><Break>.

Module Report

Module Report generates a report of module information for a board file. Depending on the format selected, the report lists information about module reference designators, name, orientation, pad locations, and netnames.

Several report formats are available. The short format lists each module's orientation and the X and Y coordinates of its pin 1. The anchor point coordinates format lists module locations by the X and Y coordinates of the module's center of rotation.

Execution

With the **PC Board Layout Tools** screen displayed, select **Module Report.** Select **Execute** from the menu that displays.

While **Module Report** runs, messages display on the screen. When **Module Report** is complete, the **PC Board Layout Tools** screen displays.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Module Report**. Select **Local Configuration** from the menu that displays.

Select **Configure MODLOC_. Module Report**'s local configuration screen displays (figure 9-1).

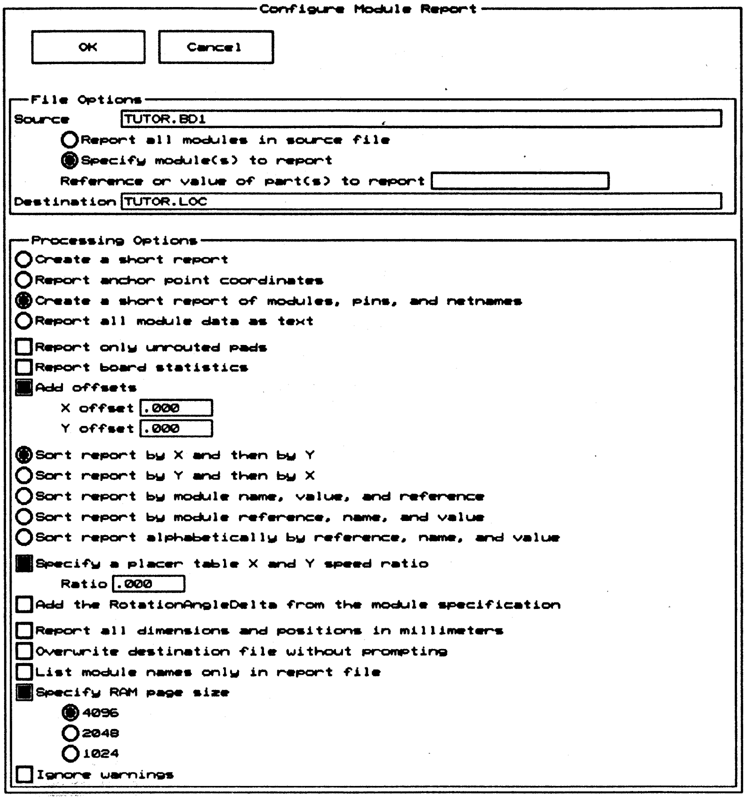


Figure 9-1. Module Report's local configuration screen.

File Options

The **File Options** area defines the source and destination files.

Source

Source is the name of the board file to report on. It may have any valid pathname. The source is originally set to *rootSheet*.BD1.

Enter the source filename, then select one of the following options:

○ `Report all modules in source file`

Tells **Module Report** to report on all the modules in the source file.

○ `Specify module(s) to report`

`Reference or value of part(s) to report` [ ]

Tells **Module Report** to report only on specific modules. When you select this option, **Reference or value of part(s) to modify** becomes available. Enter a character string representing the module reference designator, module name, or any character string associated with the modules to modify.

Use asterisks or question marks as wildcards in this entry box. An asterisk represents multiple characters, while a question mark indicates a single character wildcard.

Use commas to separate multiple strings. You can enter up to 18 characters.

Destination

Destination is the name of the file where the report is placed. It may have any valid pathname. The destination is originally set to *rootSheet*.LOC.

You must specify a destination for **Module Report** to work properly.

Processing Options

Select one of the following four report format options:

- ○ `Create a short report`

  Tells **Module Report** to list modules by location of the module's pin 1 and the module's orientation.

  If **Module Report** finds a module without a pin 1, it outputs the location of the first pin it encounters and displays a warning message. The message gives the module's name and states that it has no pin 1.

- ○ `Report anchor point coordinates`

  Tells **Module Report** to list module locations by the X and Y coordinates of the module's anchor points.

- ○ `Create a short report of modules, pins, and netnames`

  Tells **Module Report** to list each module's text, pins, location, and netname.

- ○ `Report all module data as text`

  Tells **Module Report** to list each module's name, reference, and value, and the X and Y coordinates and associated netnames of all its pins.

- ☐ `Report only unrouted pads`

  Tells **Module Report** to list only unrouted and partially routed pads. This portion of the report is sorted by netname.

- ☐ `Report board statistics`

  Tells **Module Report** to list the size, drill diameter, and layers of each via; the track length, copper area, and total number of each via type; and the overall board density.

- ❑ `Add offsets`

  `X offset` [ ]

  `Y offset` [ ]

  Tells **Module Report** to add offsets to all locations. When you select this option, the **X offset** and **Y offset** entry boxes become available.

  The allowed range is –30.00 to +30.00.

Select one of the following five report sorting options:

- ○ `Sort report by X and then by Y`

  Tells **Module Report** to sort modules by X coordinate, then by Y coordinate.

- ○ `Sort report by Y and then by X`

  Tells **Module Report** to sort modules by Y coordinate, then by X coordinate.

- ○ `Sort report by module name, value, and reference`

  Tells **Module Report** to sort modules by name, then by value and reference.

- ○ `Sort report by module reference, name, and value`

  Tells **Module Report** to sort modules by reference, then by name and value. Reference strings are assumed to be a letter followed by a number.

- ○ `Sort report alphabetically by reference, name, and value`

  Tells **Module Report** to sort modules by reference, then by name and value. Reference strings are collated alphabetically.

- ❑ `Specify a placer table X and Y speed ratio`

  `Ratio` [   ]

  Tells **Module Report** to sort the output to minimize the time required to build the board on a placer table having the specified ratio of X speed over Y speed. When you select this option, the **Ratio** entry box becomes available. The default ratio is 1.

- ❑ `Add the RotationAngleDelta from the module specification`

  Tells **Module Report** to add the RotationAngleDelta to the final placement angle given in the module report. The RotationAngleDelta is a correction factor to be applied when the placement tool picks up the module from the alignment table.

- ❑ `Report all dimensions and positions in millimeters`

  Tells **Module Report** to interpret and report sizes, dimensions, and positions in millimeters.

- ❑ `Overwrite destination file without prompting`

  Tells **Module Report** to overwrite any existing version of the destination file without prompting for permission.

- ❑ `List module names only in report file`

  Tells **Module Report** to list only the module names.

- ❑ `Specify RAM page size`
  - ❍ `4096`
  - ❍ `2048`
  - ❍ `1024`

  Tells **Module Report** to use a smaller RAM page size to free memory for processing. When you select this option, three page-size radio buttons become available. Select decreasing page sizes until **Module Report** completes successfully.

- ❑ `Ignore warnings`

  Tells **Module Report** to leave the return code set to 0 after issuing warning messages.

Examples

The examples in this section show how to get various reports on a board file called SAMPLE.BD1.

Show module text

To get a report of module text and the location and netname of all pins, run **Module Report** with the following configuration:

```
Source       SAMPLE.BD1

    O Create a short report of modules, pins,
      and netnames

Destination  SAMPLE.LOC

❑ Show the module text in the report
```

Short report

To create a short report, run **Module Report** with the following configuration:

```
Source       SAMPLE.BD1

   O Report all modules in source file

Destination  SAMPLE.LOC

O Create a short report
```

In SAMPLE.LOC, the only X and Y coordinates reported are those of pin number 1.

Report by anchor point coordinates

To create a report by anchor point coordinates, run **Module Report** with the following configuration:

```
Source       SAMPLE14.BD1

    O Report all modules in source file

Destination  SAMPLE14.LOC

O Report anchor point coordinates
```

CHAPTER 10

Compare Netlists

Compare Netlists reports differences between an EDIF netlist file and a board file. Modules which are renamed in the board file are listed in a report file. Typically, you name the report WAS_IS, and use it with **To Schematic** or with **Back Annotate** in **Schematic Design Tools 386+** to update the corresponding schematic.

The report file is a table of each module's reference, value, pin numbers, netlist netname, and board file netname.

Execution

With the **PC Board Layout Tools** screen displayed, select **Compare Netlists**. Select **Execute** from the menu that displays.

While **Compare Netlists** runs, messages display on the screen. When **Compare Netlists** is complete, the **PC Board Layout Tools** screen displays.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Compare Netlists.** Select **Local Configuration** from the menu that displays.

Select **Configure COMPNET_.** The **Compare Netlists** local configuration screen displays (figure 10-1).

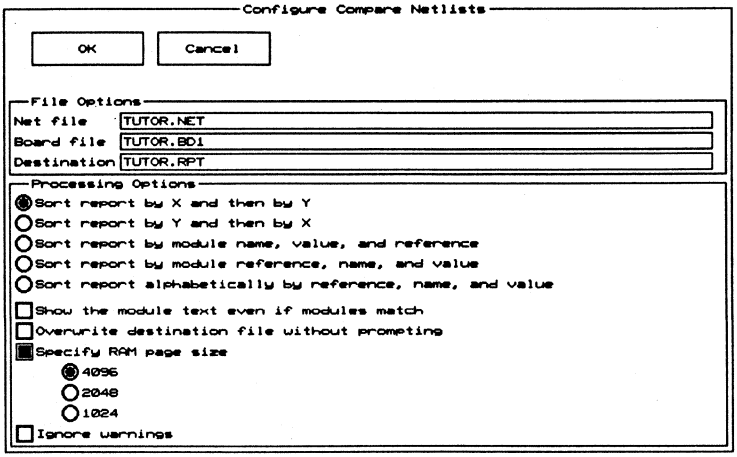


Figure 10-1. Local configuration screen for ***Compare Netlists.***

File Options

The **File Options** area defines the source and destination files.

Net file

Net file is the EDIF netlist file to read. It may have any valid pathname. The net file is originally set to *rootSheet*.NET.

Board file

Board file is the board file to compare to the EDIF netlist file. It may have any valid pathname. The board file is originally set to *rootSheet*.BD1.

Destination

Destination is the file where the report file is placed. It may have any valid pathname. The destination is originally set to *rootSheet*.RPT.

You must specify a destination for **Compare Netlists** to work properly.

Select one of the following five report sorting options:

- ❍ `Sort report by X and then by Y`

  Tells **Compare Netlists** to sort modules by X coordinate, then by Y coordinate.

- ❍ `Sort report by Y and then by X`

  Tells **Compare Netlists** to sort modules by Y coordinate, then by X coordinate.

- ❍ `Sort report by module name, value, and reference`

  Tells **Compare Netlists** to sort modules by name, then by value and reference.

- ❍ `Sort report by module reference, name, and value`

  Tells **Compare Netlists** to sort modules by reference, then by name and value. Reference strings are assumed to be a letter followed by a number.

- ❍ `Sort report alphabetically by reference, name, and value`

  Tells **Compare Netlists** to sort modules by reference, then by name and value. Reference strings are collated alphabetically.

- ❑ `Show the module text even if modules match`

  Tells **Compare Netlists** to list every module's name, reference, and value, and the X and Y coordinates and associated netnames of all its pins. By default, the report lists only modules in either file that have no match in the other file.

- ❑ `Overwrite destination file without prompting`

  Tells **Compare Netlists** to overwrite any existing version of the destination file without prompting for permission.

❑ `Specify RAM page size`

  ❍ `4096`

  ❍ `2048`

  ❍ `1024`

Tells **Compare Netlists** to use a smaller RAM page size to free memory for processing. When you select this option, three page-size radio buttons become available. Select decreasing page sizes until **Compare Netlists** completes successfully.

❑ `Ignore warnings`

Tells **Compare Netlists** to leave the return code set to 0 after issuing warning messages.

PART V: LIBRARIANS

PCB 386+ 2.00 includes librarians that create board template files and library files from **PCB 386+ 2.00** board files.

Part V: Librarians describes librarians and provides instructions for their use.

Chapter 11: *Make Board Template* describes how **Make Board Template** creates a template file from a **PCB 386+ 2.00** board file.

Chapter 12: *Make Library* describes how **Make Library** creates a library file from a **PCB 386+ 2.00** board file.

Note that the librarians take advantage of EMS, if it is present on your system. Also, you can interrupt either librarian by pressing <Ctrl><Break>.

Make Board Template

Make Board Template creates a template file from a **PCB 386+ 2.00** board file. A template file is loaded when you begin a design from scratch.

A template file can contain any of the objects a board file can contain. You might create a template for each type of board you design, for each fabrication plant you use, to establish a company standard, and so on.

You specify which template file to use on the **Configure PC Board Layout** screen. See *Chapter 1: Configure Layout Tools* for instructions on customizing your **PCB 386+ 2.00** configuration.

Execution

With the **PC Board Layout Tools** screen displayed, select **Make Board Template**. Select **Execute** from the menu that displays.

While **Make Board Template** runs, messages display on the screen. When **Make Board Template** is complete, the **PC Board Layout Tools** screen displays.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Make Board Template**. Select **Local Configuration** from the menu that displays.

Select **Configure MAKE_T. Make Board Template's** local configuration screen displays (figure 11-1).

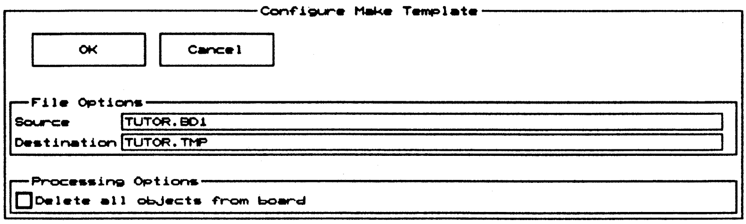


*Figure 11-1. **Make Board Template's** local configuration screen.*

File Options

The **File Options** area defines the source and destination files.

Source

Source is the name of the **PCB 386+ 2.00** board file from which the new template is to be created. It may have any valid pathname. The source is originally set to *rootSheet*.BD1.

Destination

Destination is the name of the **PCB 386+ 2.00** template file to be created from the source board file. It may have any valid pathname. The destination is originally set to *rootSheet*.TMP.

You must specify a destination for **Make Board Template** to work properly.

Processing Options

❑ `Delete all objects from board`

Tells **Make Board Template** to create an empty template by omitting all objects in the source board file.

Make Library

Make Library creates a library file from the module-related information in a **PCB 386+ 2.00** board file. In the process, **Make Library** replaces any text in the Reference and Part Value fields on each module with the contents of its Module field.

You can also use **Make Library** to create **PCB 386+ 2.00** library parts from **OrCAD/PCB II** modules. Run FROMPCB2 on an **OrCAD/PCB II** board file that contains the modules you want to translate, then run **Make Library** on the resulting **PCB 386+ 2.00** board file. See *the PC Board Layout Tools 386+ User's Guide* for information on using FROMPCB2.

Execution

With the **PC Board Layout Tools** screen displayed, select **Make Library**. Select **Execute** from the menu that displays.

While **Make Library** runs, messages display on the screen. When **Make Library** is complete, the **PC Board Layout Tools** screen displays.

Local configuration

With the **PC Board Layout Tools** screen displayed, select **Make Library**. Select **Local Configuration** from the menu that displays.

Select **Configure MAKELIB. Make Library's** local configuration screen displays (figure 12-1).

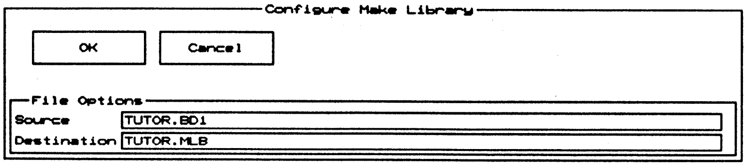


Figure 12-1. Make Library's local configuration screen.

File Options

The **File Options** area defines the source and destination files.

Source

Source is the name of the **PCB 386+ 2.00** board file from which the new library is to be created. It may have any valid pathname. The source is originally set to *rootSheet*.BD1.

Destination

Destination is the name of the **PCB 386+ 2.00** library file to be created from the source board file. It may have any valid pathname. The destination is originally set to *rootSheet*.MLB.

You must specify a destination for **Make Library** to work properly.

Part VI: Transfers

PCB 386+ 2.00 includes transfer tools that manage the steps needed to move design information from one tool set to another. Transfer tools update the database as needed, and then change from **PCB 386+ 2.00** to other OrCAD tool set screens.

Part VI: Transfers describes transfer tools and provides instructions for their use.

To Schematic

The **To Schematic** transfer changes from the **PC Board Layout Tools** screen to the **Schematic Design Tools** screen.

You can also configure **To Schematic** to run **Back Annotate,** which updates part reference designators. See the *Schematic Design Tools 386+ Reference Guide* for a complete description of **Back Annotate.**

Execution

With the **PC Board Layout Tools** screen displayed, select **To Schematic.** Select **Execute** from the menu that displays.

If you configure **To Schematic** to run **Back Annotate,** a monitor box displays at the bottom of the screen where messages report the progress of transfer. When the transfer is complete, the **Schematic Design Tools** screen (figure 13-1) displays.

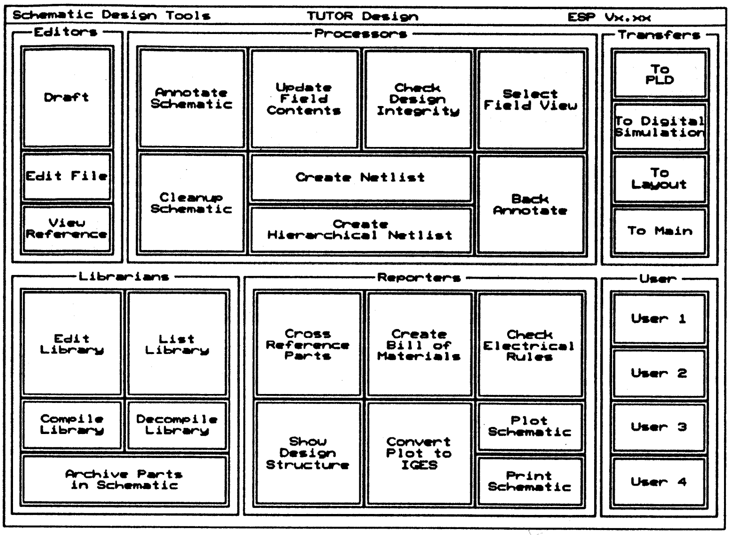


Figure 13-1. Schematic Design Tools screen.

Local configuration of To Schematic

Normally, **To Schematic** doesn't run **Back Annotate.** This section describes how to configure **To Schematic** so that it does.

With the **PC Board Layout Tools** screen displayed, select **To Schematic.** Select **Local Configuration** from the menu that displays.

```
Configure BACKANNO
BACKANNO  off
```

The menu shown at right displays. To turn **Back Annotate** on, select **BACKANNO off** from the menu. The message "Select the new status of the executable item" displays. Select **on,** and the **PC Board Layout Tools** screen displays again.

Local configuration of BACKANNO

With the **PC Board Layout Tools** screen displayed, select **To Schematic**. Select **Local Configuration** from the menu that displays.

Select **Configure BACKANNO. Back Annotate**'s local configuration screen (figure 13-2) displays.

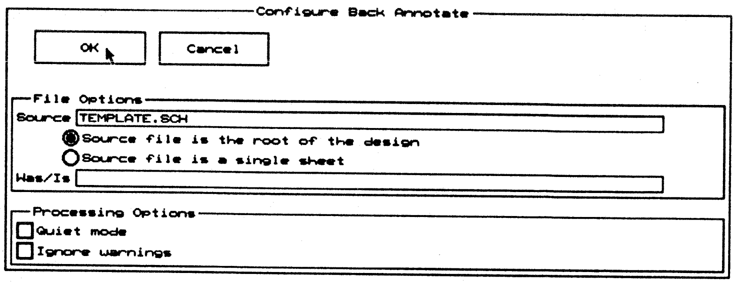


Figure 13-2. Back Annotate's local configuration screen.

File Options

The **File Options** area defines the source file and its type, and the Was/Is file.

Source

Source is the root of the design or the filename of a single sheet. It may have any valid pathname.

After entering the source filename, select one of the following options:

○ `Source file is the root of the design`

Specifies that the source file is the root sheet name of a hierarchical or flat design. If the root sheet contains sheet symbols, then the design is hierarchical. If it contains a |Link statement, it is a flat design.

○ `Source file is a single sheet`

Specifies that the source file is a single worksheet and that you want to process the single sheet only.

Was/Is

Was/Is specifies the name of the text file containing the old and new reference designator pairs. It may have any valid path and name. The format of this file is described in *Chapter 7: Back Annotate* in the *Schematic Design Tools 386+ Reference Guide.*

Processing Options

You may specify any combination of the following options:

- `Quiet mode`

  Turns quiet mode on.

- `Ignore warnings`

  Causes **Back Annotate** to continue running, rather than halt, if it encounters warnings.

To PLD

The **To PLD** transfer changes from the **PC Board Layout Tools** screen to the **Programmable Logic Tools** screen.

Execution

With the **PC Board Layout Tools** screen displayed, select **To PLD**. Select **Execute** from the menu that displays. The view changes to the **Programmable Logic Tools** screen (figure 14-1).

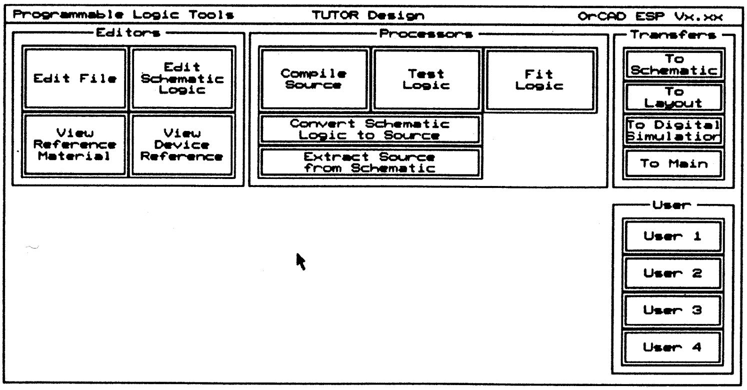


*Figure 14-1. **Programmable Logic Tools screen.***

To Digital Simulation

The **To Digital Simulation** transfer changes from the **PC Board Layout Tools** screen to the **Digital Simulation Tools** screen.

Execution

With the **PC Board Layout Tools** screen displayed, select **To Digital Simulation.** Select **Execute** from the menu that displays.

When the transfer process is complete, the **Digital Simulation Tools** screen (figure 15-1) displays.

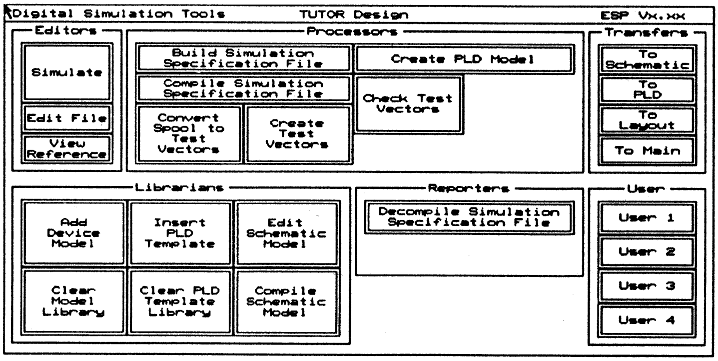


Figure 15-1. Digital Simulation Tools screen.

CHAPTER 16

To Main

The **To Main** transfer changes from the **Digital Simulation Tools** screen to the ESP design environment main screen.

Execution

With the **PC Board Layout Tools** screen displayed, select **To Main**. Select **Execute** from the menu that displays. The view changes to the design environment main screen (figure 16-1).

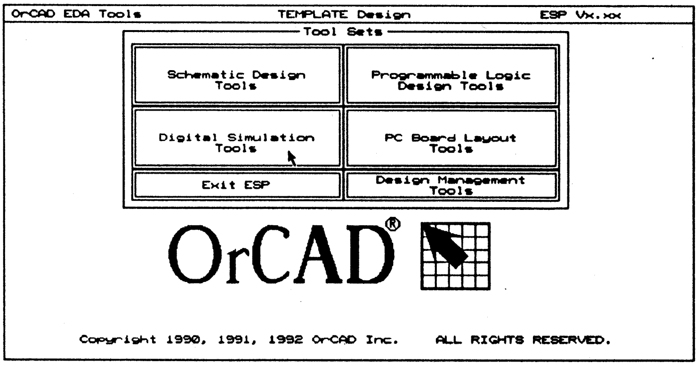


Figure 16-1. ESP design environment main screen.

APPENDIXES

This appendix provides reference information on running **PCB 386+ 2.00** utilities from the command line. A glossary and index follow this appendix.

Appendix A: *Command line controls* cross-references commands and their command line switches with the corresponding tools and local configuration buttons.

Command line controls

This appendix cross-references command line commands and their switches with the corresponding tools and local configuration buttons. This appendix is organized in alphabetical order by command.

Syntax

The syntax in this appendix follows this format:

- Parameters that you must enter exactly as shown are shown in `monospace` font.
- Variables that you must supply, such as filenames, are shown in *italic* text.
- Items in brackets are optional, and you include them only in specific circumstances. Do not type the brackets.

About switches

You can type out the full names of the switches, as they are shown in this appendix, or you can abbreviate them. You need to type only enough letters of a switch's name to distinguish it from the other switches recognized by the utility you're running.

For example, "By-Ref" doesn't distinguish "By-Reference" from "By-RefString" to the COMPNET_ utility, but "By-Refe" and "By-RefS" are unambiguous. Most switch names can be abbreviated to a single letter.

You can use any combination of uppercase and lowercase letters in switch names and options, and specify them in any order you like. The utilities will report an error if you use conflicting switches or use a switch incorrectly.

Command files

Most of the utilities accept a parameter of the form *@commandFile*, where *commandFile* is the name of an ASCII text file. Typically you include commonly used switches in *commandFile*, but it can contain any portion of the command line. For example, given the files described below, the following command lines have the same effect:

```
fixtime_ @file1.txt /w /i /by-refs
```

```
fixtime_ @file1.txt @file2.txt
```

```
fixtime_ @file3.txt
```

FILE1.TXT contains:

```
input.net input.bd1 output.bd1
```

FILE2.TXT contains:

```
/w
/i
/by-refs
```

FILE3.TXT contains:

```
@file1.txt      ; files
@file2.txt      ; switches
```

△ *NOTE: As shown above, the* @commandFile *switch is recursive.* @commandFile *switches can be nested to any depth.*

Command files can be formatted in nearly any manner, with the following provisions:

- ❖ Blank spaces are compressed; that is, multiple tabs and spaces are treated as a single space.
- ❖ Blank lines are ignored.
- ❖ Anything to the right of a semicolon (;) is considered a comment and is ignored.

You can create a simple command file easily for a given utility by running the utility without any parameters and redirecting the output to a text file, then editing the text file to "enable" the parameters you want to use.

For example, you can create the equivalent of FILE2.TXT from the example on the preceding page by following these steps:

1. Capture the output of the FIXTIME_ utility by entering the following command shown in bold:

```
C:> fixtime_ > cmdfile.txt
```

   CMDFILE.TXT contains a line for each FIXTIME_ switch. The lines have the following format:

   ; /*switch* ; *description*

2. Edit CMDFILE.TXT and delete the first semicolon on the line that contains the /WriteWithoutPrompt switch. The line should look like this:

   /WRITEWITHOUTPROMPT ; *description*

△ ***NOTE:*** *Make sure you only delete the first semicolon. The second semicolon, before the description,* must *be present. It tells FIXTIME_ to ignore the remaining text on that line.*

3. Repeat step 2 for the lines that contain the /Info and /By-RefString switches.
4. Save the file and use it as described on the preceding page:

```
C:> fixtime_ in.net in.bd1 out.bd1 @cmdfile
```

COMPNET_ *netlistFile boardFile reportFile [switches]*

Corresponding tool: **Compare Netlists**

Switch	*Description*	*Local configuration option*
/By-NameValue	Sort modules by name, then by value and reference.	❍ `Sort report by module name, value, and reference`
/By-Reference	Sort modules by reference, then by name and value. Reference strings are assumed to be a letter followed by a number.	❍ `Sort report by module reference, name, and value`
/By-RefString	Sort modules by reference, then by name and value. Reference strings are collated alphabetically.	❍ `Sort report alphabetically by reference, name, and value`
/By-XY	Sort modules by X coordinate, then by Y coordinate.	❍ `Sort report by X and then by Y`
/By-YX	Sort modules by Y coordinate, then by X coordinate.	❍ `Sort report by Y and then by X`
/Page_Size *size*	Use a smaller RAM page to free memory for processing. Set *size* to 4096, 2048, or 1024.	❑ `Specify RAM page size` ❍ `4096 bytes` ❍ `2048 bytes` ❍ `1024 bytes`
/Text	Include matching modules in the report.	❑ `Show the module text even if modules match`
/WriteWithoutPrompt	Overwrite existing report file without prompting for permission.	❑ `Overwrite destination file without prompting`

continued on next page

COMPNET_ *netlistFile boardFile reportFile [switches]*

(continued from previous page)

Switch	*Description*	*Local configuration option*
/Zero	Leave the return code set at 0 after issuing warning messages.	❑ `Ignore warnings`
@commandFile	Read *commandFile* for additional switches and options.	None

FIXTIME_ *netlistFile boardFile destinationFile* [*switches*]

Corresponding tool: **Fix Time Stamps**

Switch	*Description*	*Local configuration option*
/By-NameValue	Sort modules by name, then by value and reference.	❍ `Sort report by module name, value, and reference`
/By-Reference	Sort modules by reference, then by name and value. Reference strings are assumed to be a letter followed by a number.	❍ `Sort report by module reference, name, and value`
/By-RefString	Sort modules by reference, then by name and value. Reference strings are collated alphabetically.	❍ `Sort report alphabetically by reference, name, and value`
/By-XY	Sort modules by X coordinate, then by Y coordinate.	❍ `Sort report by X and then by Y`
/By-YX	Sort modules by Y coordinate, then by X coordinate.	❍ `Sort report by Y and then by X`
/Info	Do not generate a destination board file. Report the time stamp, reference text, and location of each module in the source netlist and board files.	❑ `Create a module/netlist report`

continued on next page

FIXTIME_ *netlistFile boardFile destinationFile* [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/LimitModules *moduleSpecifier*	Report or modify only modules that have *moduleSpecifier* in their reference text. Multiple specifiers must be enclosed in parentheses and separated by commas: (R1,R2) (SMD*,SMT*) (@*specifierFile*) where *specifierFile* is an ASCII file containing a list of module specifiers.	○ `Specify module(s) to modify` `Reference of part(s) to modify` [ ] △ *NOTE: Do not enter the parentheses in the configuration screen entry box.* △ *NOTE: The configuration screen entry box can contain up to 18 characters.*
/Metric	Output all data in millimeters.	❑ `Report all dimensions and positions in millimeters`
/Page_Size *size*	Use a smaller RAM page to free memory for processing. Set *size* to 4096, 2048, or 1024.	❑ `Specify RAM page size` ○ `4096 bytes` ○ `2048 bytes` ○ `1024 bytes`
/WriteWithoutPrompt	Overwrite existing destination file without prompting for permission.	❑ `Overwrite destination file without prompting`
/Zero	Leave the return code set at 0 after issuing warning messages.	❑ `Ignore warnings`
@commandFile	Read *commandFile* for additional switches and options.	None

MAKE_T *boardFile templateFile* [/D]

Corresponding tool: **Make Board Template**

Switch	*Description*	*Local configuration option*
/D	Create an empty template by omitting all objects in the source board file.	○ `Delete all objects from board`

MAKELIB *boardFile libraryFile*

Corresponding tool: **Make Library**

MODLOC_ *boardFile reportFile [switches]*

Corresponding tool: **Module Report**

Switch	*Description*	*Local configuration option*
/A	Report module locations by the X and Y coordinates of the module's anchor point	❍ `Report anchor point coordinates`
/By-NameValue	Sort modules by name, then by value and reference.	❍ `Sort report by module name, value, and reference`
/By-Reference	Sort modules by reference, then by name and value. Reference strings are assumed to be a letter followed by a number.	❍ `Sort report by module reference, name, and value`
/By-RefString	Sort modules by reference, then by name and value. Reference strings are collated alphabetically.	❍ `Sort report alphabetically by reference, name, and value`
/By-XY	Sort modules by X coordinate, then by Y coordinate.	❍ `Sort report by X and then by Y`
/By-YX	Sort modules by Y coordinate, then by X coordinate.	❍ `Sort report by Y and then by X`
/Degrees	Report angles in whole degrees only.	None
/DeltaAngle	Add the RotationAngleDelta from the module specification to the final placement angle given in the module report.	❑ `Add the RotationAngleDelta from the module specification`

continued on next page

MODLOC_ *boardFile reportFile* [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/LimitModules *moduleSpecifier*	Report only modules that have *moduleSpecifier* in their name, reference, value, or other associated text string. Multiple specifiers must be enclosed in parentheses and separated by commas: (10DIP100,10DIP200) (SMD*,SMT*) (@*specifierFile*) where *specifierFile* is an ASCII file containing a list of module specifiers.	❍ `Specify module(s) to report` `Reference or value of part(s) to report` [ ] △ *NOTE: Do not enter the parentheses in the configuration screen entry box.* △ *NOTE: The configuration screen entry box can contain up to 18 characters.*
/Metric	Output all data in millimeters and interpret offsets as millimeters.	❑ `Report all dimensions and positions in millimeters`
/NameList	Report only module names.	❑ `List module names only in report file`
/Page_Size *size*	Use a smaller RAM page to free memory for processing. Set *size* to 4096, 2048, or 1024.	❑ `Specify RAM page size` ❍ `4096 bytes` ❍ `2048 bytes` ❍ `1024 bytes`
/Ratio *XSpeedOverYSpeed*	Sort output to minimize the time required to build the board on a placer table having the specified speed ratio.	❑ `Specify a placer table X and Y speed ratio` `Ratio` [ ]

continued on next page

MODLOC_ *boardFile reportFile* [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/S	Report the position of each module's pin 1. If no pin 1 is present, report the position of the first pin found and issue a warning.	○ `Create a short report`
/TenthsOfADegree	Report angles in tenths of degrees (*nnn.n*).	None
/Text	Report each module's name, reference, and value, and the X and Y coordinates and associated netnames of all pins.	○ `Report all module data as text`
/ViaStatistics	Report the type, size, drill diameter, and layers of each via; the track length, copper area, and total number of each via type; and the overall board density.	☐ `Report board statistics`
/WriteWithoutPrompt	Overwrite existing report file without prompting for permission.	☐ `Overwrite destination file without prompting`
/XOffset *offset*	Add *offset* to all X coordinates in reported positions.	☐ `Add offsets` `X offset` ☐
/YOffset *offset*	Add *offset* to all Y coordinates in reported positions.	☐ `Add offsets` `Y offset` ☐

continued on next page

MODLOC_ *boardFile reportFile* [*switches*]

(continued from previous page)

/Zero	Leave the return code set at 0 after issuing warning messages.	❑ `Ignore warnings`
@*commandFile*	Read *commandFile* for additional switches and options.	None

MODMOD_ *sourceFile destinationFile* [*switches*]

Corresponding tool: **Modify Modules**

Switch	*Description*	*Local configuration option*
/AddNetName *netName*	Add *netName* (or change the netname to *netName*) on all specified pads.	❑ `Change netname on pad` `New netname` [ ]
/All	Report the entire board file in ASCII format.	❑ `Dump the entire board file in ASCII format`
/By-NameValue	Sort modules by name, then by value and reference.	○ `Sort report by module name, value, and reference`
/By-Reference	Sort modules by reference, then by name and value. Reference strings are assumed to be a letter followed by a number.	○ `Sort report by module reference, name, and value`
/By-RefString	Sort modules by reference, then by name and value. Reference strings are sorted by ASCII value.	○ `Sort report alphabetically by reference, name, and value`
/By-XY	Sort modules by X coordinate, then by Y coordinate.	○ `Sort report by X and then by Y`
/By-YX	Sort modules by Y coordinate, then by X coordinate.	○ `Sort report by Y and then by X`

continued on next page

MODMOD_ *sourceFile destinationFile* [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/ChangePad *padName*	Select the pads identified by the specified pad name. Multiple pad names must be enclosed in parentheses and separated by commas or spaces; hyphens indicate a range of pads to be selected: (1,2 5) this represents pad 1, pad 2, and pad 5 (2–20) this represents pad 2 through pad 20 (1–3,6) this represents pad 1, pad 2, pad 3, and pad 6	○ `Change a selected pad` `Pad name` ☐ △ *NOTE: In the configuration screen entry box., use commas, not spaces.* △ *NOTE: Do not enter the parentheses in the configuration screen entry box.*
/DrillDiameter *diameter*	Set the drill size of the selected pads to the specified diameter. Drill values that were originally 0.00 (zero) will remain zero or unchanged. Only non-zero drill values will be updated.	❑ `Change drill size` `Drill size` ☐

continued on next page

MODMOD_ *sourceFile destinationFile* [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/FixCenter	Set the rotation center to the X and Y of pin 1 for through-hole devices and to the center of gravity for surface mount devices.	❑ `Fix center of rotation`
/HorizontalPadSize *size*	Set the horizontal size of the selected pads to the specified size.	❑ `Change horizontal pad size` `Horiz size` [ ]
/Info	Report all information about the modules in a board file.	❑ `Create a module information report`
/KeepAllBoardOutlines	When converting an ASCII dump to binary, keep all board outlines encountered. Use in combination with the /All switch when panelizing a board dumped with offsets.	None

continued on next page

MODMOD_ *sourceFile destinationFile* [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/LimitModules *moduleSpecifier*	Report or modify only modules that have *moduleSpecifier* in their name, reference, value, or other associated text string. Multiple specifiers must be enclosed in parentheses and separated by commas: (10DIP100, 10DIP200) (SMD*, SMT*) (@*specifierFile*) where *specifierFile* is an ASCII file containing a list of module specifiers.	○ `Specify module(s) to modify` `Reference or value of module(s) to modify` [ ] △ *NOTE: Do not enter the parentheses in the configuration screen entry box.* △ *NOTE: The configuration screen entry box can contain up to 18 characters.*
/Metric	Output all data in millimeters and interpret horizontal and vertical pad sizes as millimeters.	❑ `Report all dimensions and positions in millimeters`
/Net	Report all information on the specified nets. Multiple specifiers accepted, as described for /LimitModules above.	❑ `Report all information on nets` `Netname` [ ]
/Orientation *angle*	Change the orientation of the selected pads to the specified angle, in degrees measured clockwise from the top.	❑ `Change pad angle` `Angle` [ ]

continued on next page

MODMOD_ *sourceFile destinationFile* [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/Page_Size *size*	Use a smaller RAM page to free memory for processing. Set *size* to 4096, 2048, or 1024.	❑ Specify RAM page size ○ 4096 bytes ○ 2048 bytes ○ 1024 bytes
/PrefixNetname *prefix*	Prefix all netnames in the ASCII dump with the given *prefix* string. Use when combining several boards in a single board file.	None
/Side *layer*	Make changes to the selected pads on the specified layers: Component Solder Both Inner	❑ Select layers ○ Component layer only ○ Solder layer only ○ Both external layers ○ Inner layers only
/TreatAllAsSMTs	Sets the rotation center to the center of gravity for all specified parts, as if all are surface mount devices. /FixCenter is implied.	❑ Fix center of rotation ❑ Treat all modules as surface mount
/Type *type*	Change the selected pads to the specified type: Rectangle Oval	❑ Change pad type ○ Rectangle ○ Oval
/VerticalPadSize *size*	Set the vertical size of the selected pads to the specified size.	❑ Change vertical pad size Vert size ☐

continued on next page

MODMOD_ ***sourceFile destinationFile*** **[*switches*]**

(continued from previous page)

Switch	*Description*	*Local configuration option*
/WriteWithoutPrompt	Overwrite existing destination file without prompting for permission.	❑ `Overwrite destination file without prompting`
/XOffset *offset*	Add *offset* to all X coordinates in reported positions.	❑ `Add offsets` `X offset` ☐
/YOffset *offset*	Add *offset* to all Y coordinates in reported positions.	❑ `Add offsets` `Y offset` ☐
/Zero	Leave the return code set at 0 after issuing warning messages.	❑ `Ignore warnings`
@*commandFile*	Read *commandFile* for additional switches and options.	None

NCDRILL_ *boardFile* [*drillFile*] [*switches*]

Corresponding tool: **Create NC Drill File**

Switch	*Description*	*Local configuration option*
/ASCII	Produce drill file in ASCII format.	○ `ASCII format`
/BetweenLayers *startLayer stopLayer*	Report all vias between layers specified by *startLayer* and *stopLayer*: C[omponent] 8 1 9 2 10 3 11 4 12 5 13 6 14 7 S[older]	❑ `Report vias between layers` `From layer` ☐ `To layer` ☐
/Decimal	Output data in Excellon decimal format.	○ `Excellon decimal`
/EntireDesign	Create a file named *startLayer_stopLayer*.NCD for each layer pair. If *drillFile* is specified, it contains a list of all files created and a description of their contents.	❑ `Create drill files for the entire design`
/Flip *axis*	Mirror drill information about the X or Y axis, or both. Set *axis* to X, Y, or XY.	❑ `Mirror about the X axis` ❑ `Mirror about the Y axis`
/HoleMaximum *maxHoleSize*	Don't drill holes larger than *maxHoleSize*.	❑ `Specify maximum hole size` `Drill size` ☐
/LeadingZero	Add leading zeros. Suppress trailing zeros.	○ `Excellon leading zero`

continued on next page

NCDRILL_ *boardFile* [*drillFile*] [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/Metric	Output all data in millimeters and interpret offsets as millimeters.	❑ `Report all dimensions and positions in millimeters`
/NonPlatedHoles	Drill only nonplated holes.	None
/PlatedHoles	Drill only plated holes.	None
/Ratio *XSpeedOverYSpeed*	Sort output to minimize the time required to drill the board on a drill table having the specified speed ratio. The default ratio is 1.	❑ `Specify a drill table X and Y speed ratio` `Ratio` [ ]
/TestPoints	Drill only testpoint holes.	None
/TrailingZero	Add trailing zeros. Suppress leading zeros.	None
/WriteWithoutPrompt	Overwrite existing drill file without prompting for permission.	❑ `Overwrite destination file without prompting`
/XOffset *offset*	Use the specified X offset to translate the board to the desired machine location.	❑ `Add offsets` `X offset` [ ]
/YOffset *offset*	Use the specified Y offset to translate the board to the desired machine location.	❑ `Add offsets` `Y offset` [ ]
/Zero	Leave the return code set at 0 after issuing warning messages.	❑ `Ignore warnings`
@*commandFile*	Read *commandFile* for additional switches and options.	None

PCB386 *boardFile* [*switches*]

Corresponding tool: **Edit Layout**

Switch	*Description*	*Local configuration option*
/C	Display the PCB 386+ configuration screen.	None
/L	Reverse function (<Esc> and <Enter>) of left and right mouse buttons.	❑ `Left hand mouse operation`

REANNO_ *sourceFile destinationFile* [*switches*]

Corresponding tool: **Reannotate Board File**

Switch	*Description*	*Local configuration option*
/DeltaWidth *variance*	Ignore differences in X coordinates of *variance* or less in calculating module sequence numbers.	❑ `Specify variance` `Variance` [ ]
/Include *file*	Replace listed reference numbers with those specified in *file* when reannotating.	❑ `Use an include file` `Include file` [ ]
/LimitModules *moduleSpecifier*	Report or modify only modules that have *moduleSpecifier* in their reference. Multiple specifiers must be enclosed in parentheses and separated by commas: (10DIP100,10DIP200) (SMD*,SMT*) (@*specifierFile*) where *specifierFile* is an ASCII file containing a list of module specifiers.	❍ `Specify module(s) to modify` `Reference of part(s) to modify` [ ] △ *NOTE: Do not enter the parentheses in the configuration screen entry box.* △ *NOTE: The configuration screen entry box can contain up to 18 characters.*
/Metric	Output all data in millimeters.	❑ `Report all dimensions and positions in millimeters`
/Page_Size *size*	Use a smaller RAM page to free memory for processing. Set *size* to 4096, 2048, or 1024.	❑ `Specify RAM page size` ❍ `4096 bytes` ❍ `2048 bytes` ❍ `1024 bytes`
/Row	Reannotate by row instead of by column.	❍ `Scan by row`

continued on next page

REANNO_ *sourceFile destinationFile* [*switches*]

(continued from previous page)

Switch	*Description*	*Local configuration option*
/Text	Report module text, locations, and time stamps.	❑ `Create module report`
/Unconditional	Unconditionally renumber the modules in the board file incrementally beginning with 1, regardless of any pre-existing annotation.	❑ `Unconditionally renumber modules`
/WriteWithoutPrompt	Overwrite existing destination file without prompting for permission.	❑ `Overwrite destination file without prompting`
/Zero	Leave the return code set at 0 after issuing warning messages.	❑ `Ignore warnings`
@*commandFile*	Read *commandFile* for additional switches and options.	None

A

Active layer ■ In **PC Board Layout Tools**, the layer the pointer is currently on.

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

ANSI ■ An acronym for *American National Standards Institute.*

Aperture ■ A hole, similar to the aperture of a camera, that is used in photoplotting. Apertures are available in various sizes and shapes.

Aperture list ■ A text file containing the dimensions for each of the apertures used to photoplot your PC board file.

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, etc.

ASIC ■ An acronym for *application-specific integrated circuit.*

Assembly outline ■ The segments and arcs that together define the printing or plotting area. You must define an assembly outline to print or plot a board file in **Edit Layout**.

Autorouting ■ Automatic routing performed by a computer program based on a set of rules called strategies.

Autopan ■ A feature that automatically shifts the viewing window when the cursor reaches a screen boundary.

B

BBS ■ An acronym for *bulletin board system.* See *Bulletin board system.*

BCD ■ An acronym for *binary coded decimal.*

Blind via ■ A via that reaches only one surface layer on one side of a multilayered PC board. See also *Via.*

Block ■ A specific portion of the layout that is marked and manipulated as a single entity.

Bookmark ■ A specially marked location on a layout. You can use the JUMP command to move the pointer to a bookmark.

Breadboard ■ A prototype or temporary board used for hardware testing.

Bulletin board system ■ A computer system for sending and receiving bulletins, messages, and files over telephone lines. Abbreviated *BBS.*

Buried via ■ A via not reaching a surface layer on either side of a multilayered PC board. See also *Via.*

Bus ■ A thick line that represents connecting parallel data or data in series mode grouped together as one track, rather than as individual tracks.

C

CAE ▪ An acronym for *computer aided engineering*.

CMOS ▪ An acronym for *complementary metal-oxide semiconductor*, an integrated circuit.

Complex hierarchy ▪ In **Schematic Design Tools**, a design in which more than one sheet symbol references a single worksheet. Compare *Simple hierarchy*.

Component ▪ An element; a part. PC boards are made up of components affixed to a common surface and connected by copper traces.

Component side ▪ The uppermost layer of a board on which most components are placed. See also *Solder side*.

Component silk screen ▪ The silkscreened markings of the printed circuit board which appear on the Component side. The silk screen is applied over the solder mask.

Component solder mask ▪ The colored, usually transparent, coating applied to the board over the etched copper to protect some of the copper from the soldering process.

Configuration ▪ The information a program uses to operate. The configuration can be tailored to your needs.

Contiguous segments ▪ Portions of a track drawn with the commands **ROUTE Begin Begin . . . End** or **ROUTE Begin Begin . . . New.** Compare *Discrete segments*.

Copper pour ▪ A polygon fill method by which the copper zone is filled with a specified pattern, avoiding objects that cross the zone or lie within the zone.

Copper tool ▪ A definition of the width of a segment or arc that is placed on the board.

Copper zone ▪ An area on the board designed to be covered by a layer of copper when manufactured. Also known as a "metal zone."

Current layer ▪ The layer on which current design modifications are made.

Cursor ▪ In the ESP design environment, a square or underscore; in **PCB 386+ 2.00**, a vertical bar. The cursor displays inside a text field showing where characters typed at the keyboard will display. See also *Pointer*.

D

Default ▪ A preselected parameter.

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

Design rules check (DRC) ■ A feature that checks a layout for violations of pad and track isolations. **PCB 386+ 2.00** performs DRCs in a batch operation and marks the location and type of any error with a special marker.

Discrete segments ■ Portions of a track drawn with the commands **ROUTE Begin End, ROUTE Begin End,** and so on, or **ROUTE Begin New, ROUTE Begin New,** and so on. Compare *Contiguous segments.*

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

DM/PL ■ *Digital Microprocessor Plotting Language.* Houston Instruments' plotter language.

Download ■ The process of obtaining a file from the BBS.

DRC ■ An acronym for *design rules check.* See *Design rules check.*

Drill template ■ A topographical plot of the locations where holes are to be drilled in the PC board.

DXF ■ A graphics format created by AutoCAD.

E

EDA ■ An acronym for *electronic design automation.*

EDIF ■ An acronym for *electronic design interchange format.* EDIF is a standard established by ANSI for transferring electronic data including netlists, schematics, and PC board layouts. **PCB 386+ 2.00** accepts EDIF200 data.

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

Elements ■ A numbered list of pad stack or via stack elements. Each element contains a pad or via definition, including layer, style, drill diameter, size, offset, and solder mask guard width.

Entry box ■ A box indicating a place for text or numbers entered from the keyboard: $\boxed{}$

F

Feed-through hole ■ A hole in a PC board which allows a connector to pass through the board without connecting to intermediate layers.

Feed-through pin ■ A pin connecting to the surface layers of a PC board but not to any intermediate layers.

Fill zone ■ A zone which defines an area to be filled in a copper pour.

Fire 9xxx ■ A file format used by photoplotters manufactured by Cymbolic Sciences, Inc., which accept a form of RS-274-D that includes an embedded aperture list.

Flat design ▪ In Schematic Design Tools, a schematic structure in which output lines of one sheet connect laterally to input lines of another sheet through graphical objects called *module ports.* Flat designs are practical for small designs of three or fewer sheets. See *Schematic, Hierarchical design.*

Force vector ▪ A single vector representing the mathematical sum of all the ratsnest vectors for a module. The length of the vector indicates the length of the routes and how close to optimum a module's position is on the board. The goal is to place the module so the vector is as short as possible.

FPGA ▪ An acronym for *field-programmable gate array.*

G

Gerber (274-D) ▪ A file format that can be read by Gerber and other photoplotter systems that require a separately or previously defined aperture list.

Gerber (274-X) ▪ A file format that can be read by Gerber photoplotters that accept an embedded aperture list.

Gerber photoplotting ▪ A method of transferring PC board artwork to film.

Ground ▪ The common signal at the same potential as the earth.

H

Hierarchical design ▪ In Schematic Design Tools, a schematic 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.

Highlight ▪ A feature that graphically emphasizes a particular segment, track, or net of a layout so that it stands out.

Hole ▪ In **PCB 386+ 2.00,** an absence of board.

HP-GL ▪ *Hewlett-Packard Graphics Language.* Hewlett-Packard's plotter protocol.

HP-GL2 ▪ An extension of HP-GL that supports polygon fills, wide lines, and other methods of plotting complex shapes.

I

IC ▪ An acronym for *integrated circuit.*

IGES ▪ An acronym for *initial graphic exchange specification.* A graphics format for transferring CAD/CAM information.

Inquire ▪ To display information about the pad, module, or track that the cursor is on.

Isolation ▪ The clearance around a pad, track, or via defining the nearest approach allowed by conductors of another signal set.

K

K ▪ An abbreviation for *kilobyte*. See *Kilobyte*.

Kilobyte ▪ One kilobyte is equal to 2^{10} (1024) bytes. The prefix "kilo" is taken from the metric system, where it stands for "one thousand." Abbreviated *K*.

Key field ▪ Used to tell DRAFT and the other tools which fields you want to combine and compare. A *key field* lists the part fields to combine and compare. In **Schematic Design Tools**, key fields are defined on the **Configure Schematic Tools** screen.

L

Layer ▪ A plane on which nets are laid out to connect components making up a PC board layout. A via is used to route from one layer to another. In **PCB 386+ 2.00**, a PC board layout can have up to 16 copper layers.

Layer marker ▪ An object on a board layer which indicates the copper layer's number, as counted from the component layer (layer 1) and counting only enabled layers.

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

Library ▪ A collection of standard, often-used modules stored together to speed up design work on the system.

M

Macro ▪ Series of commands you can run automatically at the touch of a key or key combination. Macros dramatically reduce the number of keystrokes required to perform complex or repetitive actions.

Manual routing ▪ Routing performed by the designer.

MB ▪ An abbreviation for *megabyte*. See *Megabyte*.

Megabyte ▪ One megabyte is equal to 2^{20} (1,048,576) bytes. The prefix "mega" is taken from the metric system, where it stands for "one million." Abbreviated *MB*.

Mil ▪ 1/1000 of an inch.

Module ▪ Footprint for a device consisting of zero or more pads, other objects, and a name. PC board layouts are made up of modules connected to a common surface and connected to each other by traces (or routes).

Module value ▪ Contains the value of the component. Module values of different components on a PC board layout can be the same or different.

N

Net ▪ All points in a circuit that are associated with the same signal name. Also, the paths connecting two or more pins on a PC board.

Net arc ▪ A net segment defined as an arc (one-quarter of a circle).

Netlist ▪ A text file that lists the interconnections of a schematic diagram by the names of the signals, modules, and pins connected together on a PC board. The nodes in a circuit.

No-Fill zone ▪ A zone which defines an area within a fill zone that is not to be filled in a copper pour.

Nominal copper tool ▪ The copper tool used by the autorouter to create net segments and arcs. Also, the copper tool used for manual routes that do not start on an existing segment.

Nominal via stack ▪ The via stack used by the autorouter and for manual routing functions that place a via (if **Via Restricted** is enabled in the **Edit Net Properties** dialog box).

O

Optimize ▪ A function in **PCB 386+ 2.00** used to improve the layout of an autorouted circuit board. The via reduction strategy reroutes the layout one connection at a time, attempting to reduce the track length and the number of vias.

P

Pad ▪ On a PC board, a copper etch shape on one or more layers, optionally with a hole, and an isolation surrounding the copper, used for connecting a component pin to the PC board. The pad indicates where pins of a component are placed.

Pad stack ▪ An object that represents all the pad stack elements; a pad stack definition. See *Elements*.

Part field ▪ In **Schematic Design Tools**, a slot for holding text or data to be associated with a part. Each part has two part fields reserved for part value and part reference. It has eight other part fields that can be used to store other useful information. See *Key fields*.

Path ▪ An unrouted, partially routed, or completely routed connection between two pads. In a net with *n* pads, there are exactly *n*–1 paths.

PCB ▪ An acronym for *printed circuit board*.

Pin ▪ The portion of a component where an electrical connection can be made.

Pin to pin spacing ▪ The physical spacing between pins on a device.

Placement ▪ The position of components on a PC board layout.

Plane layer ▪ A layer of copper that is dedicated to a single net.

PLD ▪ An acronym for *programmable logic device*. See *Programmable logic device*.

Pointer ▪ An arrow on the screen that moves as you move the mouse: ↖
See also *Cursor*.

Polygon ▪ A continuous outline shape.

PostScript ▪ A printer language.

Power plane ▪ A copper layer dedicated to a signal that is considered a power supply. The ground plane is a power plane that supplies the ground potential.

Programmable logic device ▪ A type of integrated circuit that contains fuses which 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. Abbreviated *PLD*.

PROM ▪ An acronym for *programmable read-only memory*. Introduced as computer memories, PROMs soon came to be used as general logic devices.

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

R

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

Raster ▪ An array of dots which may define any shape.

Ratsnest ▪ A straight-line connection between two or more pads in a layout that are electrically connected.

Reference designator ▪ Contains a string denoting the type of component and a number that is specific to that component. Every component on a PC board has its own distinct *reference designator*. Also known as *module name*.

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 ▪ In Schematic Design Tools, the worksheet at the top of a multiple-sheet design.

Routing ▪ Placing conductive interconnects between components on a PC board layout.

Routing strategy ▪ A set of rules for autorouting a PC board layout.

S

Schematic ▪ A graphical representation of a circuit using a standard set of electronics symbols. See also *Flat design*, *Hierarchical design*, and *Root sheet*.

Screen coordinates ▪ The X and Y coordinates reporting the location of the cursor on the screen.

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

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

Segment ▪ In PCB 386+ 2.00, a portion of a track. See also *Discrete segments* and *Contiguous segments*.

Silk screen ■ A plot of the text and/or outlines of modules for a board, used for silk screening component placement and identification information onto a PC board.

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

SMD ■ An acronym for *surface mounted device*.

SMT ■ An acronym for *surface mount technology*.

Solder mask ■ A negative plot of pads with a guard band around the pads. Also a lacquer applied to prevent solder from adhering to unwanted areas on the printed circuit board.

Solder side ■ The surface of the PC board opposite that where the components are usually mounted. See also *Component side*.

Strategy ■ See *Routing strategy*.

Stub ■ A net segment or a chain of segments, arcs, and vias that has only one end attached to a test point, a pad, or another segment.

Subnet ■ In **PCB 386+ 2.00**, a single object or a group of connected objects that is not yet connected to the rest of the net.

Surface mount ■ A technique for attaching components to a PC board whereby the pins are connected to only one surface layer without using component holes; compare *Feed-through pin*.

T

Thermal relief ■ A means of connecting a pad to a larger copper area while minimizing the amount of copper available to conduct heat during the soldering process. **PCB 386+ 2.00** uses a plus (+) shape for thermal relief.

Thermal relief copper tool ■ The copper tool used for thermally relieved pad connections when a pad is connected to a plane or fill zone. In **PCB 386+ 2.00**, all pads in a net use the same thermal relief copper tool.

Through-hole via ■ A via that connects the surface layers on a PC board. See also *Via*.

Time stamp ■ An eight-byte hexadecimal number, based on the system date and time of day, that identifies objects. Time stamps are created in **Draft**, the schematic editor, and conveyed through the netlist to **PCB 386+ 2.00**.

Trace ■ In **PCB 386+ 2.00**, a track; the copper path that carries electronic signals between components on a PC board layout.

Track ■ In **PCB 386+ 2.00**, a conductive pad-to-pad connection made up of segments.

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

U

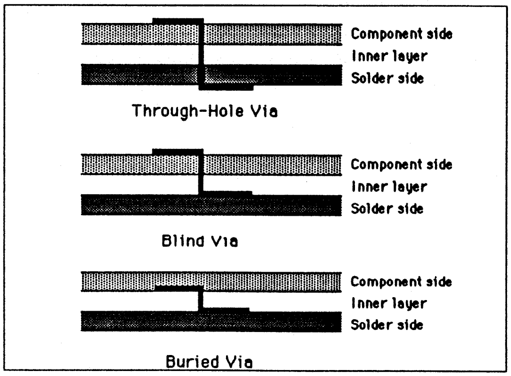
Upload ■ The process of sending a file to another computer.

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 without overwriting user button programs for other designs.

V

Vector ■ One or more points and an associated function.

Via ■ A hole connecting layers of a PC board. A through-hole via connects surface layers of a board. On multilayer boards, a via *not* reaching a surface layer on one side is called a blind via, and a via *not* reaching a surface layer on either side is called a buried via.



Via types.

Via stack ■ An object that represents all the via stack elements; a via stack definition. See *Elements*.

W

Worksheet ■ **Draft** calls the sheets of drafting paper on which the schematics are drawn *worksheets*. Worksheets display on the computer screen as a rectangular area in which you can place parts and draw wires.

Z

Zone ■ A section of a PC board which designates an area that tracks cannot pass through, in which no vias are used, in which no modules are used, or that is to be filled with copper.

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

INDEX

B

F

G

J

K

L

O

P

Q

R

T

U

V

W

X–Y

Z