PLACEMENT AND ROUTING

User's Manuals

September 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

048-0065-03
000-0104-01

COPYRIGHT

TRADEMARKS

000-0104-01

PREFACE

These manuals describe PC-PLACE, P-CAD's program for automatically placing components on a printed circuit board, and PC-ROUTE, P-CAD's program for automatically routing the board.

PC-PLACE and PC-ROUTE are used with digital printed circuit board designs that are created from a PC-CAPS schematic and packaged with PC-PACK. The introduction to these manuals, "Placement and Routing Introduction," describes how PC-PLACE and PC-ROUTE are an integral part of the PCB-3 end-to-end design system.

NOTATION

These manuals gives step-by-step procedures and examples. To make it easy for you to follow these procedures, we use the following notation.

<xxxxx> Angle brackets around lowercase letters indicate a variable name that may be entered by the system or by you. For example:

<filename>.SCH

[ ] Square brackets indicate the name of a key. For example:

[Return]

[Return] [Return] indicates the key that is used to execute a command or accept an option. This key may be labeled differently, depending on your system. For example:

[RETURN], [↵], [Enter], [Enter ↵], [ENTER].

[ ]-[ ] Square brackets connected with a hyphen indicate keys that must be pressed simultaneously. For example:

Press [Ctrl]-[Alt]-[Del].

UPPER	Uppercase letters indicate a command or an element that must be typed as shown. For example: Type PCPLOTS and press [Return].
/	A forward slash separates main menu and submenu command combinations. For example: DRAW/ARC A forward slash also begins a keyboard command. For example: /EXE
*	An asterisk in a filename or in a filename extension indicates that any character(s) can occupy that position and all the remaining positions in the filename or extension. For example, the DOS command DIR *.SYM displays a list of all the filenames with the .SYM extension in the current directory.
Select	Select means move the cursor with the mouse or the arrow keys to a command in the menu or to a point in the drawing area and press Button 1 or the space bar.

CONTENTS

CONTENTS (Continued)

PC-ROUTE USER'S MANUAL

FIGURE

PLACEMENT AND ROUTING

Introduction

September 1986

p-cad®

PERSONAL CAD SYSTEMS INC.

000-0104-01

COPYRIGHT

TRADEMARKS

000-0104-01

PLACEMENT AND ROUTING INTRODUCTION

This introduction provides an overview to Personal CAD (P-CAD) Systems' placement and routing programs, PC-PLACE and PC-ROUTE. It also describes system requirements and P-CAD filename conventions.

OVERVIEW

PC-PLACE and PC-ROUTE are an integral part of P-CAD's PCB-3 system for end-to-end printed circuit board (PCB) design.

With PC-PLACE, you can take a PCB database that was created from a PC-CAPS schematic and packaged with PC-PACK and automatically place components and interactively improve the component placement. You can also use PC-CARDS to move and swap components, but PC-PLACE provides histogram and force vector analysis tools to help you decide on moves and swaps for optimum placement.

Then, you can use PC-CARDS to manually route the board or pre-wire some traces before routing, or you can use PC-ROUTE to automatically route the board, and then finish any unrouted subnets in PC-CARDS.

Figure 1 shows the end-to-end PCB design flow using PC-PLACE and PC-ROUTE.

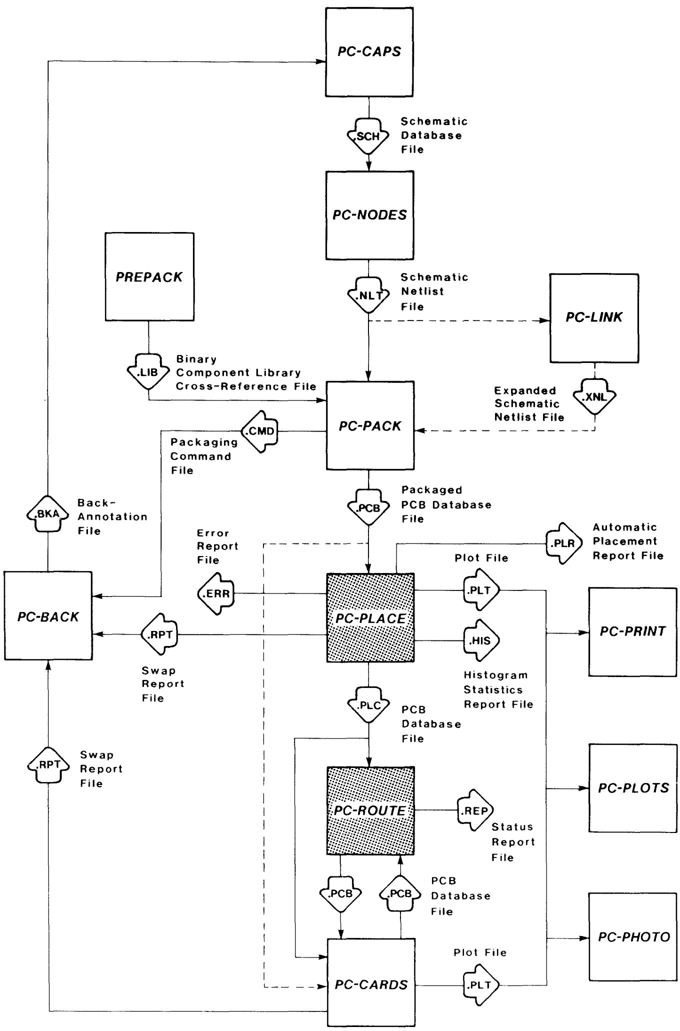


Figure 1. End-to-End PCB Design Flow Using PC-PLACE and PC-ROUTE

The recommended design flow from a packaged schematic to a routed board is described below.

First, use PC-PLACE to: 1) put in the board outline; 2) automatically and interactively place components; and 3) put in routing barriers. Refer to the section titled "Preparing Your Database" in the *PC-PLACE User's Manual* for instructions and special requirements.

> NOTE: If you use PC-CARDS to put in the board outline and/or routing barriers, refer to the section titled "Preparing Your Database for PC-ROUTE" in the *PC-ROUTE User's Manual* for instructions and special requirements.

Next, use PC-CARDS to pre-wire traces; for example, bus and ground nets.

Then, use PC-ROUTE to automatically route the board.

Finally, use PC-CARDS to complete any unrouted subnets.

SYSTEM REQUIREMENTS

PC-PLACE AND PC-ROUTE use loadable driver files (.DRV extension). These files must be present in the \PCAD\DRV directory. P-CAD recommends you use its directory structure. See the *System Autoloader* instruction sheet for more information.

The amount of memory your computer holds determines how large a routing task PC-ROUTE can handle.

If you are using an IBM PC-XT, P-CAD highly recommends using a TURBO card for better performance with PC-ROUTE.

P-CAD FILENAME CONVENTIONS

P-CAD PC-PLACE and PC-ROUTE programs use the following filename conventions.

<filename>.PCB - Packaged PCB database file undergoing PC-PLACE component placement (output from PC-PACK).

<filename>.PLC - Placed PCB database file (placed, unrouted board output from PC-PLACE).

<filename>.PLR - PC-PLACE automatic placement report (output from PC-PLACE).

<filename>.HIS - Histogram statistics report (output from PC-PLACE).

<filename>.RPT - Swap report file (output from PC-PLACE or PC-CARDS).

<filename>.CTL - PC-ROUTE routing strategy file (output from edit routing strategy phase).

<filename>.RTE* - File that contains the subnets that need to be connected on the PCB by the router (output from the extraction phase of PC-ROUTE).

<filename>.RTS* - File that contains the connected subnets (output from the router phase of PC-ROUTE).

<filename>.PCB - Routed PCB database file (output from the create routed database phase of PC-ROUTE).

<filename>.REP* - Status report file (output from PC-ROUTE).

<filename>.PLT - Plot file used to generate printer or plotter output (output from PC-PLACE or PC-CARDS SYS/PLOT command).

* If you route two boards concurrently with PC-ROUTE and you use the same input database filename for both boards but with different extensions, PC-ROUTE assigns a unique name to the output files of the second board you route. For example, if you run two boards concurrently named XYZ.PLC and XYZ.PL1, then the output files of the first board will have the root name XYZ and the output files of the second board will have the root name XY1.

PLACEMENT & ROUTING USER'S MANUALS 6

000-0104-01

PC-PLACE™
User's Manual

September 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

000-0126-01

COPYRIGHT

TRADEMARKS

000-0126-01

ABOUT THIS MANUAL

This manual describes PC-PLACE, the P-CAD PCB automatic placement program.

PC-PLACE operates in an environment very similar to that of PC-CARDS. Before reading this manual, you should be familiar with PC-CARDS operation; that is, you should read the *PC-CARDS User's Manual* and use the program. The *PC-CARDS User's Manual* contains information that is not duplicated in this manual but is necessary for understanding and running PC-PLACE.

This manual is designed to serve as a learning aid and as a reference manual.

Chapter 1, INTRODUCTION, provides an overview of PC-PLACE. Read this chapter first to become familiar with PC-PLACE.

Chapter 2, USING PC-PLACE, explains how to use PC-PLACE to automatically place components and interactively improve the placement of components on your board. It is divided into two parts. Read Part A for an overview of the main commands and concepts. Then, for step-by-step hands-on practice, use the example in Part B.

Chapter 3, COMMANDS, describes the purpose and operation of the commands unique to PC-PLACE. It supplements Chapter 3, "Commands" of the *PC-CARDS User's Manual*, which contains descriptions of commands used in both programs. When you start your own design placement, use this chapter to refer to commands you are not familiar with.

Appendix A, ERROR MESSAGES, lists error messages that may appear on the screen during PC-PLACE program operation and in the error report. It includes the cause of each error and the appropriate user action. It also lists warnings that do not interrupt program operation, but that you may want to investigate.

Appendix B, SAMPLE PLACEMENT REPORT, describes the contents of the automatic placement report and shows a sample report.

Appendix C, SAMPLE HISTOGRAM REPORT, describes the contents of the histogram report and shows a sample report.

An index is also provided at the back of the manual.

NOTATION

This manual gives step-by-step procedures and examples. To make it easy for you to follow these procedures, we use the following notation.

<xxxxx> Angle brackets around lowercase letters indicate a variable name that may be entered by the system or by you. For example:

<filename>.SCH

[ ] Square brackets indicate the name of a key. For example:

[Return]

[Return] [Return] indicates the key that is used to execute a command or accept an option. This key may be labeled differently, depending on your system. For example:

[RETURN], [↵],
[Enter], [Enter ↵],
[ENTER].

[ ]-[ ] Square brackets connected with a hyphen indicate keys that must be pressed simultaneously. For example:

Press [Ctrl]-[Alt]-[Del].

UPPER	Uppercase letters indicate a command or an element that must be typed as shown. For example: Type PCPLOTS and press [Return].
/	A forward slash separates main menu and submenu command combinations. For example: DRAW/ARC A forward slash also begins a keyboard command. For example: /EXE
*	An asterisk in a filename or in a filename extension indicates that any character(s) can occupy that position and all the remaining positions in the filename or extension. For example, the DOS command DIR *.SYM displays a list of all the filenames with the .SYM extension in the current directory.
Select	Select means move the cursor with the mouse or the arrow keys to a command in the menu or to a point in the drawing area and press Button 1 or the space bar.

CONTENTS

Contents (Continued)

Contents (Continued)

FIGURES

Figures (Continued)

CHAPTER 1. INTRODUCTION

This chapter introduces you to Personal CAD (P-CAD) Systems' PC-PLACE, which enables you to automatically place components and interactively improve the placement of components on your board.

This chapter also defines terms used in this manual. For explanations of other terms and operations, refer to the *PC-CARDS User's Manual.*

DEFINITIONS

The following terms are defined as they are used in this manual.

cutlines - The horizontal and vertical lines used by the automatic placement program to analyze net crossings and place components close to each other with regard to physical trace length.

footprint attribute - An attribute assigned to each component that is used to group the component into a category by its "footprint" or physical outline.

force vector - A line or "vector" drawn from the center of each component to a theoretical "perfect" placement location based on the current board layout.

histogram - A graphical representation of the relative board density based on the average number of nets (ratsnest lines) that cross each defined resource line.

lattice points - The locations at which components can be placed.

merit factor - A number, displayed with the histogram, that indicates the relative improvement made to the overall placement with each component swap, move, or rotate, and after running the automatic placement program.

outline - A highlighted line around the component that indicates the shape and size of the component.

ratsnest - The graphic representation of pin-to-pin circuit connectivity.

resource lines - A group of horizontal and vertical grid lines used to calculate the number of nets that cross those lines and used to update the histogram when the placement is changed.

routing channel - The space required for the width of each trace and its surrounding clearance.

OVERVIEW

PC-PLACE is an automatic program for the placement of PCB components that runs on personal computers. PC-PLACE is an integral part of P-CAD's PCB design system.

With PC-PLACE, you can take a database that was created in PC-CAPS and packaged in PC-PACK and make an initial placement of all the PCB components. Before running the automatic placement program, you can specify parameters, including a lattice on which to place the components, the "association" and position of discrete components to major components, component clearances, and horizontal and vertical cutlines used for analyzing net crossings. You can also "fix" the position of components prior to running the automatic placement program.

PC-PLACE also provides interactive analysis tools for improving the initial automatic placement. The histogram display and the merit factor indicator provide a visual reference to indicate any improvement in the initial placement. Force vectors can be displayed to indicate the theoretical "perfect" location of each component. Furthermore, ratsnest lines can help in analyzing the placement. You can display ratsnest lines individually pin to pin, one component at a time, or if desired, a total display is always available. These analysis tools can be used either before or after the automatic placement.

At any time during the placement process you can use the query commands to see the status of a component, its pin data, or the name of the net connected to the pin.

PC-PLACE runs in an environment similar to that of PC-CARDS. It contains most of the PC-CARDS commands that allow you to move parts and swap gates, and additional commands for using the PC-PLACE automatic placement program and interactive placement analysis tools.

After using PC-PLACE, you can route the PCB database either automatically using PC-ROUTE or manually using PC-CARDS.

Input and output for PC-PLACE are shown in Figure 1-1.

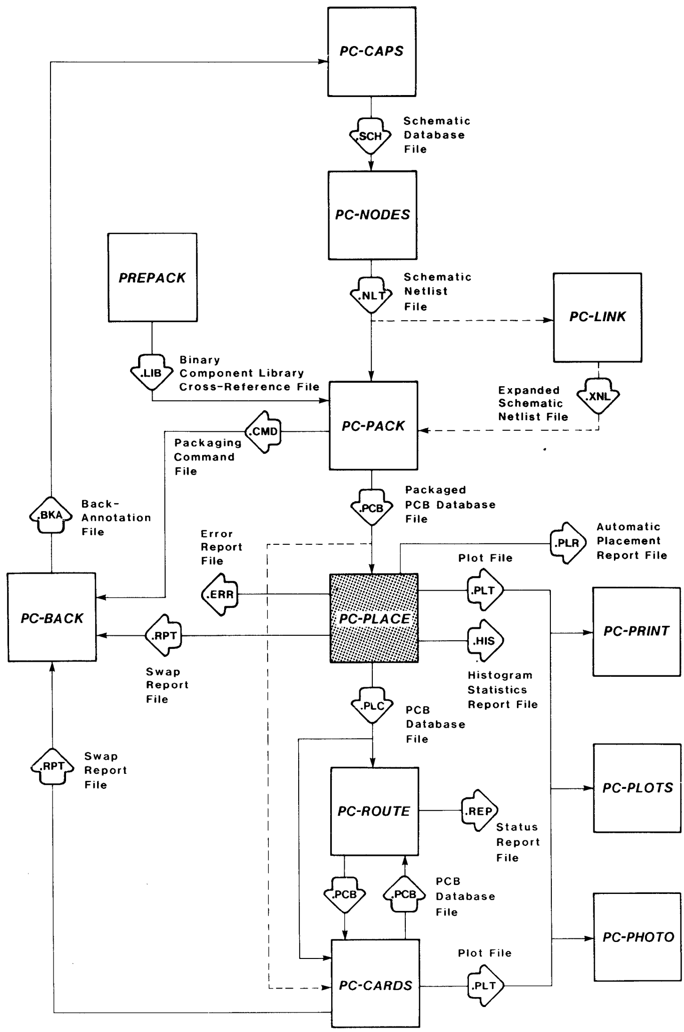


Figure 1-1. PC-PLACE Input and Output

Input

Input to PC-PLACE is a PC-PACK generated PCB database file and user-specified parameters.

PCB Database File

The database file (<filename>.PCB) produced by PC-PACK contains the graphic representation of the PCB and all the packaging and logical netlist information. The analysis features of PC-PLACE require a ratsnested database; that is, one containing full netlist connectivity (such as is created by PC-PACK). If you use a database without full connectivity, PC-PLACE will produce the graphical analysis results only for the netlist connectivity specified.

User-Specified Parameters

You can specify several parameters, such as lattice and cutline parameters for automatic placement and histogram display and resource line parameters for interactive placement.

Output

PC-PLACE output files are the placed PCB database, the placement report, the histogram report, the error report, the plot file, the swap report, and the command log.

Placed PCB Database File

The placed PCB database file (<filename>.PLC) consists of the input PCB database file with the modifications made using PC-PLACE.

Placement Report

The placement report (<filename>.PLR) is an ASCII file that is created when you run the automatic placement program using the PLCE command. This report contains statistics of the component placement, including the cutline locations, total trace length, and the placed and unplaced components. It lists the total number of components, equivalent integrated circuits, and the board density. The report also lists any warning messages.

Histogram Report

The histogram report (<filename>.HIS) is an ASCII file that is created when you use the HIST/STAT command. It contains statistics for horizontal and vertical resource lines and a summary of the change in the merit factor. It also contains statistics that indicate routing channel availability.

Error Report

If any errors occur during program operation, error messages and warnings are displayed on the screen and listed in the error report file (<filename>.ERR).

Plot File

If you want to print or plot a PC-PLACE database file, you can use the SYS/PLOT command to create a plot file (<filename>.PLT), then use PC-PRINT or PC-PLOTS to plot the PCB.

Swap Report

When you swap components, gates, or pins, PC-PLACE makes a record of each swap in the swap report file (<filename>.RPT), which you can use to back annotate the PC-CAPS schematic.

Command Log File

PC-PLACE automatically produces the command log file (PCPLACE.CMD), a log of the operations performed during a program session. This file is in the same format as the PC-CARDS PCCARDS.CMD file. If the PCPLACE.CMD file already exists when PC-PLACE is started, the file is renamed PCPLACE.CM$.

CHAPTER 2. USING PC-PLACE

This chapter explains how to use PC-PLACE to automatically place components and interactively improve the placement of components on your board.

Part A provides an overview to using PC-PLACE. As you read this part, feel free to select menu commands and enter data into the drawing area. Then, if you want more practice, use the example in Part B.

Part B provides an example using PC-PLACE with step-by-step instructions.

PART A. OVERVIEW TO USING PC-PLACE

This part explains how to start the program, prepare your database for PC-PLACE, automatically place components, interactively improve the placement, and prepare your database for PC-ROUTE.

The information in this part is presented in the sequence in which you will probably use the program, although many of the PC-PLACE features can be used at any time during program operation.

This section provides a general description of how to use PC-PLACE. For step-by-step instructions, refer to the example in Part B.

STARTING THE PROGRAM

Before starting PC-PLACE, be sure that:

- For larger databases (in excess of 100K bytes), you have at least 590K RAM. This is not required, but highly recommended.
- The program files PCPLACE.EXE and PCPLACE.OVL are in the EXE directory.
- The packaged PCB database file (<filename>.PCB) is in the appropriate project directory.

First, change the current directory to the project directory you plan to use. Then, to start PC-PLACE, type:

PCPLACE [Return]

The system displays the PC-PLACE Title Screen. Press any key to display the opening menu shown in Figure 2-1.

```
                 PC-PLACE


      Options:

          Configure PC-PLACE

          Edit Part Footprints

      >>  Edit database          <<

          Exit PC-PLACE





Press: [SPACE] for next option; [RETURN] to accept
```

Figure 2-1. PC-PLACE Opening Menu

This menu provides the following options.

Configure PC-PLACE - Allows you to call the configuration screen for configuring PC-PLACE for your system.

Edit Part Footprints - Allows you to call the Footprint Attribute Menu to assign, modify, or display a list of part footprint attributes. This option is described further in the section "Editing the Part Footprint Attributes."

Edit Database - Allows you to start program operation.

Exit PC-PLACE - Allows you to return to DOS.

To move from one option to the next, press the space bar.

To accept and initiate the selected option, press [Return].

CONFIGURING PC-PLACE

To configure the program for your system, select **Configure PC-PLACE** at the opening menu. The system displays the screen shown in Figure 2-2.

PC-PLACE CONFIGURATION

Database filename: None

Directory path:
Current

Security Device port .... port 1
Mouse port .... port 1

Video save to disk .... Off

Enter the filename; Press: [RETURN] to accept

Figure 2-2. PC-PLACE Configuration Screen

The configuration screen allows you to set several parameters.

Database filename - The name of a database file that the program automatically loads each time you start program operation. You can leave the default of "None" or enter the name of the database file you are currently processing.

Directory path - The directories in which the program will look for files you specify using the FILE/LOAD command. If the path includes more than one directory, use a semicolon to separate the directory names. For example:

\PCAD\PROJ1;\PCAD\PROJ2

The default is "Current."

Security device port - The port to which your security device is connected. The options are "port 1" and "port 2." The default is "port 1." You can use the space bar to toggle between the two options.

Mouse port - The port to which your mouse is connected. The options are "port 1" and "port 2." The default is "port 1." You can use the space bar to toggle between the two options.

Video save to disk - Allows you to store a pixel image to disk when using certain commands such as VLYR, LPAN, MASK, and STO/MAP. The options are "On" or "Off." The default is "Off." When you are working with a large database file, this feature saves time. For example, if the **video save to disk** option is "On," and you use the STO/MAP command to store a map view of your PCB, and then use the RCL command to recall the view, the MAP view comes up automatically; you don't need to wait for the system to redraw the screen. If the **video save to disk** option is "Off," and you use RCL to recall the MAP view, you must wait for the system to redraw the screen before continuing.

Make the appropriate entry for each option. To accept a default option, press [Return]. To change an option, enter the new data or press the space bar and press

[Return]. When you accept the last option, if you have made any changes, the system prompts:

Save this configuration? No

To use the configuration for this program session only, press [Return] to accept "No." To use this configuration for all program sessions (or until you edit this screen again), type Y for Yes, then press [Return]. The system returns you to the opening menu.

EDITING THE PART FOOTPRINT ATTRIBUTES

Before you run PC-PLACE, make sure the parts used in your database have been assigned a footprint attribute. Footprint attributes are used to group components that have the same outline or "footprint" into categories for automatic placement.

Each component part in our system has a footprint. You can view the physical footprint area by selecting certain commands, such as MOVE or MOVE/COMP.

For example, when you select MOVE and then select the component to be moved, a highlighted line appears around the component as shown in Figure 2-3.

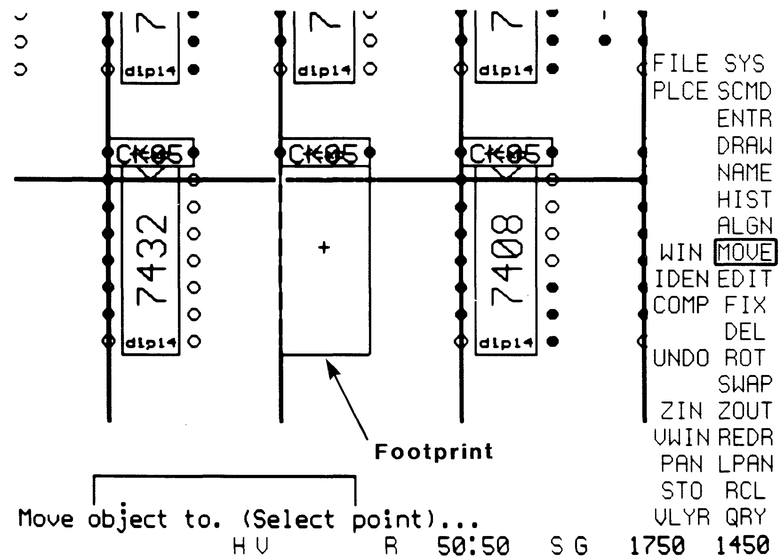


Figure 2-3. The Part Footprint

PC-PLACE looks at the footprint of a part to determine the physical size of the part for automatic placement.

You can display the footprint attribute of a component by selecting the QRY/COMP (Query/Component) command and then selecting the component.

Footprint attributes can be assigned in either of two ways: (1) using the **Edit Part Footprint** option on the PC-PLACE Opening Menu; or (2) using the ATTR/ACOM (Attribute/Add Component Attribute) command in PC-CARDS.

NOTE: If you use the **Edit Part Footprint** option in PC-PLACE, you must assign the footprint attributes to your parts before you package your database using PC-PACK. The attributes assigned to parts using this method cannot be viewed on the screen.

To assign or modify the part footprint attributes within PC-CARDS, select the ATTR/ACOM command and assign the footprint attribute using the following format:

FP=<attribute>

You can add a new layer to the layer screen, for example FP, and assign the footprint attribute to that layer. The attributes assigned to the parts using this method can be viewed on the screen by turning on the footprint layer.

To assign or modify the part footprint attributes within PC-PLACE, use the space bar to move to the **Edit Part Footprint** option on the Opening Menu and then press [Return]. The system displays the Footprint Attribute Menu.

Figure 2-4 shows a sample of the Footprint Attribute Menu.

```
                PC-PLACE Part Footprint Editor

Directory path : \pcad\proj0


Footprint Commands:

   1 - Add/Modify the footprint.
   2 - View/Modify all footprints.
   3 - List the footprints of all library parts.
   4 - List the footprint of a library part.
   5 - Quit.

Selection ( 1 - 5 ) ? 1



      Enter the selection number.  Press [RETURN] when ready
```

Figure 2-4. Footprint Attribute Menu

The first prompt is for a directory path to where the parts are located. The default is the current directory. Type the path and press [Return].

The next prompt is for a selection of one of the following five options:

1 - Allows you to assign or modify a footprint attribute.

2 - Displays the footprint attribute of each library part and allows you to modify each footprint attribute one at a time.

3 - Displays a list of all the footprint attributes of all the library parts.

4 - Displays the footprint attribute of a single library part.

5 - Returns you to the PC-PLACE Opening Menu.

To select an option, type the appropriate number and press [Return].

Adding or Modifying Footprint Attributes

If you select 1 on the Footprint Attribute Menu, the system displays the Add/Modify Footprint Attribute Screen.

Figure 2-5 shows a sample of the Add/Modify Footprint Attribute Screen with entries.

Add/Modify Footprint

Library part name(s): 7400 7404

Footprint: DIP14

Press [ESC] to exit
Or Press [RETURN] to re-enter the part names
Or Enter the footprint. Press [RETURN] when ready

Figure 2-5. Add/Modify Footprint Attribute Screen

Type the part filenames and press [Return]. You do not have to type the filename extension. Separate each part filename entry with a space or a comma. You can enter up to two lines of part filenames (approximately 16 entries).

After you enter the library part filenames, the system prompts you for the footprint attribute. Type the footprint attribute you want assigned to the parts specified and press [Return].

Viewing and Modifying All Footprint Attributes

If you select 2 on the Footprint Attribute Menu, the system displays the View/Modify All Footprints Screen.

Figure 2-6 shows a sample of what would be displayed.

```
                    View/Modify All Footprints

Part File Name:  \pcad\proj0\7400.PRT

Footprint:  DIP14

New Footprint:




         Press [ESC] to exit
      Or Press [END] to view next file
      Or Enter the new footprint.  Press [RETURN] when ready
```

Figure 2-6. View/Modify All Footprints Screen

This screen provides the following options:

Part File Name - Allows you to toggle through the part names in the part directory specified on the Footprint Attribute Menu.

Footprint - Allows you to display the footprint attribute for the part filename specified above.

New **Footprint** - Allows you to specify a new footprint attribute.

To view the footprint attribute of a part, use the [End] key to toggle through the part filenames. Then press [Return] to view the footprint attribute of the part filename selected.

To change the footprint attribute, press [Return] after the footprint displayed, type the new footprint, and press [Return].

Viewing All Footprint Attributes

If you select 3 on the Footprint Attribute Menu, the system displays a list of all the library parts and their footprint attributes.

> NOTE: You can print your library listing by pressing [Ctrl]-[P] and selecting option 3 on the Footprint Attribute Menu. When the library listing is printed, press [Ctrl]-[P] to turn off the print function.

Figure 2-7 shows a sample of what would be displayed.

Part File Name	Footprint
\pcad\proj0\7400.PRT	DIP14
\pcad\proj0\7404.PRT	DIP14

Figure 2-7. List of Footprint Attributes of All Library Parts

Viewing One Footprint Attribute

If you select 4 on the Footprint Attribute Menu, the system prompts you for input of the part filename. Type the part filename and press [Return]. You do not have to type the filename extension.

After you enter the part filename, the system lists the footprint attribute for that part at the bottom of the Footprint Attribute Menu as shown in Figure 2-8.

PC-PLACE Part Footprint Editor

Directory path : \pcad\proj0

Footprint Commands:

1 - Add/Modify the footprint.
2 - View/Modify all footprints.
3 - List the footprints of all library parts.
4 - List the footprint of a library part.
5 - Quit.

Selection (1 - 5) ? 4

Part filename ? 7400

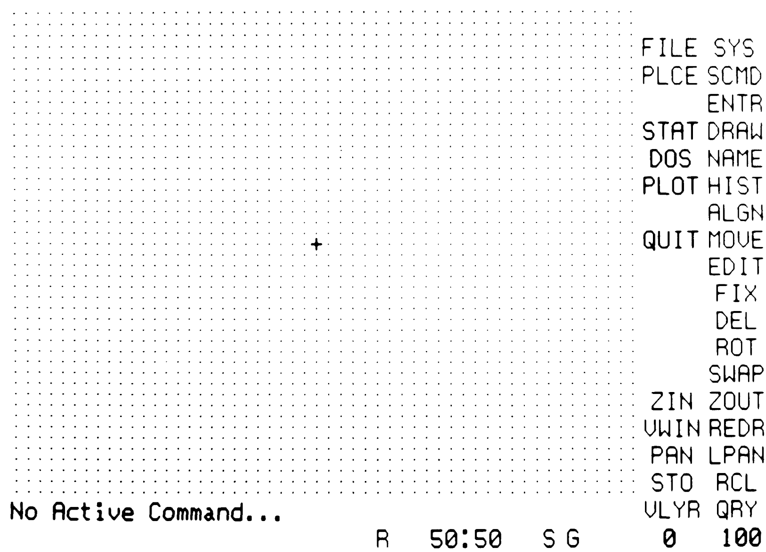
Footprint = DIP14

Press any key when ready .....

Figure 2-8. Footprint Attribute of a Part

RUNNING PC-PLACE

To run PC-PLACE, select **Edit Database** at the opening menu. The PC-PLACE Program Screen is displayed as shown in Figure 2-9. The graphic display on the screen is identical to the PC-CARDS display, and the commands, status line parameters, and cursor movement operate in the same manner as in PC-CARDS. The main menu is on the right side of the screen.



86383

Figure 2-9. PC-PLACE Program Screen

PC-PLACE COMMANDS

The PC-PLACE commands include many of the PC-CARDS commands and subcommands. PC-PLACE also includes many different commands and subcommands, which are described in the following sections and further in Chapter 3, "Commands."

NOTE: All PC-PLACE operations are done in the equivalent of PC-CARDS detail mode. Because there is only one mode, the DETL and SYMB menu selections are not present.

CONFIGURING YOUR DATABASE FOR THE HISTOGRAM DISPLAY

PC-PLACE provides a number of options for the histogram display. You can select configuration options for each database file you process. The configuration is stored with the file and is used each time you process the file unless you specify any changes.

To specify configuration options for the file that is currently loaded, use the HIST/CNFG command. The system displays the screen shown in Figure 2-10.

HISTOGRAM SPECIFICATIONS: Textual Input

Discrete Part Force Vectors	NO
Percent of Screen for Histo	8
Histo Axis Orientation	TOP and LEFT
Histo Block Width	NARROW
Routing Grid	50
Number of Signal Layers	2

Press: [SPACE] for the next option; [RETURN] to accept

Figure 2-10. Histogram Specifications Screen

To accept an option, press [Return]. If you want to change an option, enter the value you want in the manner explained below, then press [Return] to accept the new value. You can press [Esc] at any time to accept all the displayed options.

Discrete Part Force Vectors - Specifies whether or not the program will display force vectors from discrete parts. (Force vectors are always displayed for DIP-type parts and connectors.) The options are "YES" and "NO."

The default is "NO." Press the space bar to toggle the options.

> NOTE: A discrete part is defined as a part with a component attribute, assigned using the PC-CARDS SCMD/SCAT command, of 11000 through 11999 and a connector is defined as a part with a component attribute of 12000 through 12999.

Percent of Screen for Histo - Specifies the percent of the screen that the histogram display will occupy. The default is "8." To change it, enter the percent you want. Your entry must be a number from 5 to 20.

Histo Axis Orientation - Specifies which screen edges will be used for the histogram display. The options are "TOP and LEFT," "TOP and RIGHT," "BOTTOM and LEFT," and "BOTTOM and RIGHT." The default is "TOP and LEFT." Press the space bar to cycle through the options.

Histo Block Width - Specifies the width of the histogram display graphics. "NARROW" is a line display and "WIDE" is a bar display. The width of each "WIDE" bar is determined by the spacing of the resource line grid. The default is "NARROW." Press the space bar to toggle the options.

Routing Grid - Specifies the size of the grid on which you plan to route the board. The default is "50." This means that when the program determines resource line channel usage, it will allow one trace per 50 DBU length of resource line for each signal layer. To change this parameter, enter the grid size you want. Your entry must be a number from 1 to 100.

Number of Signal Layers - Specifies the number of layers to be used for signal routing (not including power or ground planes). The program uses this number with the routing grid parameter to determine the total possible routing channels for each resource line. The default is "2." To change it, enter the appropriate number of layers. Your entry must be a number from 1 through 20.

When you accept a value for the last option, the system prompts:

Save this configuration? (Y or N): No

Type Y and press [Return] to save the changes or press [Return] to accept "No" and exit without saving.

If you do not save the configuration, the changes are not recorded. The graphics screen appears, and normal program operation resumes.

When you save the configuration, the system returns to the graphics screen, redraws the placement database, and overlays it with a default resource line grid pattern.

The default resource line grid pattern is 100 DBU by 100 DBU. This pattern produces a good histogram display for most boards that have a fairly regular row and column configuration. We recommend that you use the default resource line grid. However, if you want to change the resource line grid, an explanation of how to do it is described below. Figure 2-11 shows the default grid.

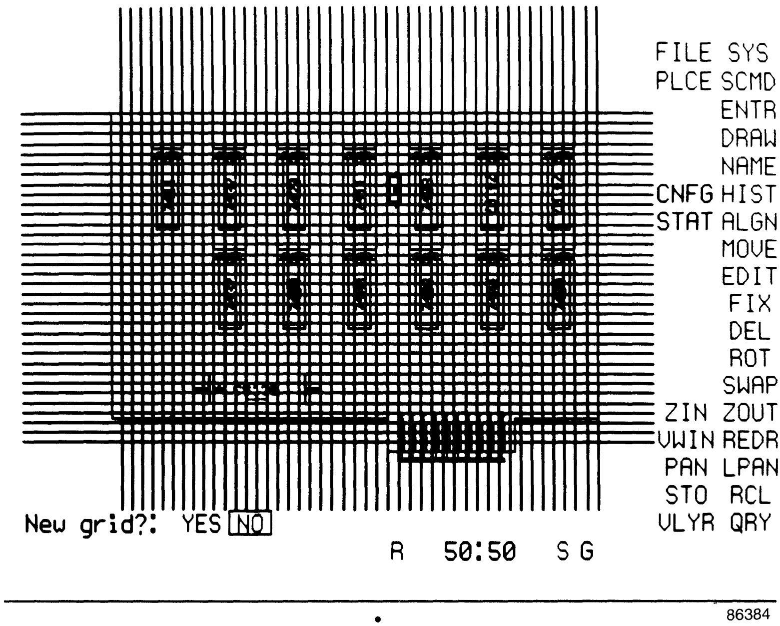


Figure 2-11. Default Resource Line Grid

The system prompts:

NEW GRID?: YES NO

To use the default resource line grid pattern, press Button 1 or the space bar to accept the default answer "No." The system stores the configuration you selected and the default grid in the database file.

If you do not want to use the default grid pattern, use the mouse or the cursor key to select "Yes," and press Button 1 or the space bar. The system prompts you to enter additional information to create a unique resource line grid pattern. The next prompt asks:

Uniform Grid? YES NO

For a uniform grid, you specify a resource line pattern, which is repeated throughout the placement window. A nonuniform grid can be any arrangement of resource lines where you must specify each individual resource line. Examples of uniform and nonuniform grids are shown in Figures 2-12a and 2-12b.

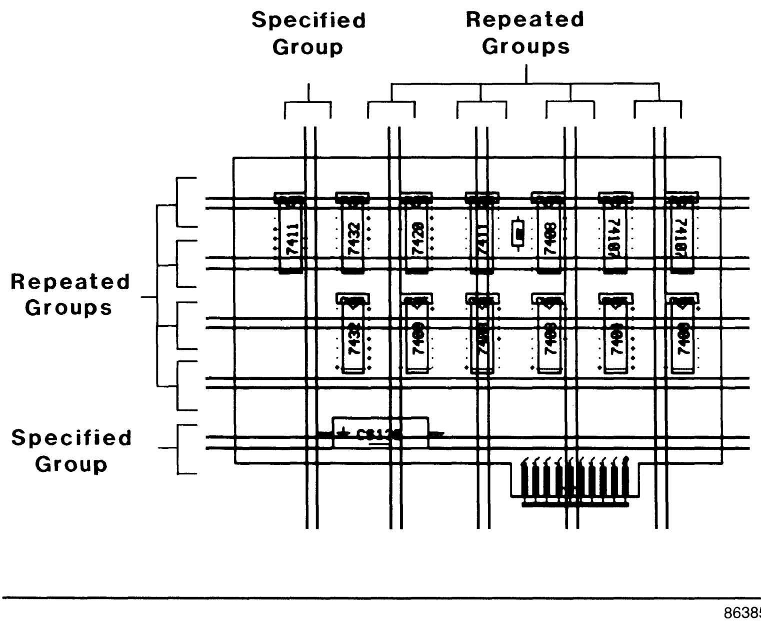
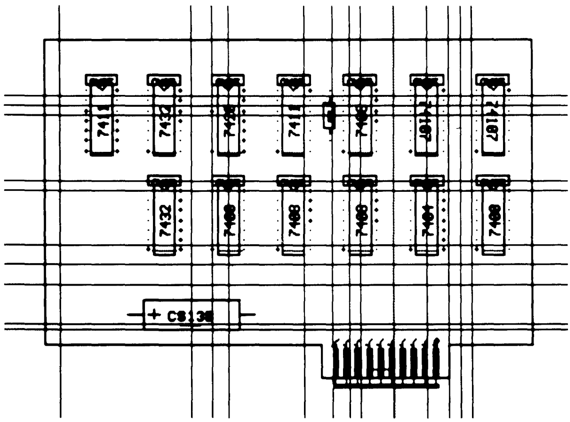


Figure 2-12a. Uniform Resource Line Grid



86386

Figure 2-12b. Nonuniform Resource Line Grid

The default is "No." If you want to change it to "Yes," use the mouse or the cursor key. When the selection you want is highlighted, press Button 1 or the space bar to accept it.

If you selected a uniform grid, you must specify a "placement window" to tell the system in what area the pattern you specify is to be repeated.

> NOTE: When you select the corners of the placement window, select corners that extend beyond the board outline.

The next prompt asks for window coordinates.

Corner 1....
Enter grid window:

Select the first corner either by moving the cursor to the point you want and pressing Button 1 or by moving the cursor to the X Y cursor-coordinate box on the status line in the lower right corner of the screen, pressing Button 1, entering the numeric coordinates, and pressing [Return]. Select the second corner in the same manner.

To create the new grid, you must next specify locations for the resource lines. The system prompts for a point on the x axis. This is the horizontal axis, and the resulting resource lines will be vertical.

X Point
Enter point 1 :

Enter a point, either by moving the cursor to it and pressing Button 1 or by moving the cursor to the cursor coordinate box on the status line, pressing Button 1, entering the coordinates, and pressing [Return]. As soon as you select the point, the system displays the resource lines you created and asks for another point on this axis.

For a uniform grid, the left edge of the window helps determine the pattern. Enter one or more points to create a pattern. The system will repeat the pattern throughout the placement window in the x direction.

For a nonuniform grid, enter a point for each resource line you want.

When you have entered all the necessary points on the x axis, press Button 2 or [Esc]. The system prompts:

Y Point
Enter point 1 :

Use the same process as for the x axis to enter these points. For a uniform grid, the bottom edge of the placement window helps determine the pattern, so enter at least one point to define a pattern in the down-to-up direction.

> NOTE: You can enter grid lines in any order; you do not have to enter them in sequence from left to right or top to bottom. When you are entering the grid lines for an axis, you can delete a line you have already entered on that axis by moving the cursor to any point on the line and pressing Button 1 or the space bar. You must enter at least one point on each axis; if you do not, the program will be unable to load the histogram display.

When you are finished, the system paints the entire grid pattern (both x and y) on the screen and then removes it. (If the grid pattern is too fine in any area, the system may not paint the grid, but it is recorded in the database.)

The grid pattern and the configuration you chose are now stored in the database file and will be used each time you load the file. You can change the file's configuration parameters or grid pattern at any time by selecting HIST/CNFG and repeating this process.

USING THE HISTOGRAM

To use the histogram analysis tool, select the HIST command from the main menu. Two letters, H and V, appear on the status line in red. Place the cursor over the H and press Button 1 or the space bar to toggle H to green. The system responds with:

Loading Histogram

Figure 2-13 shows the histogram display.

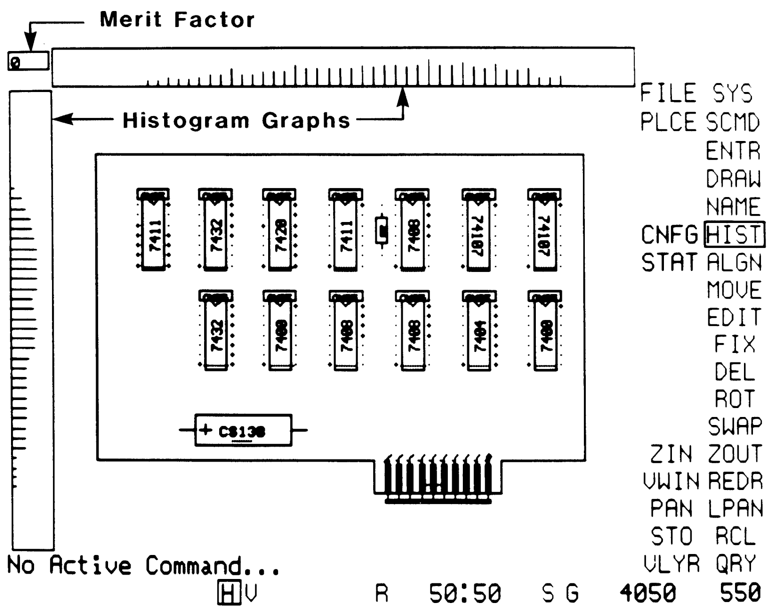


Figure 2-13. Histogram Display

The histogram display graphs appear along the axes specified during configuration. The default (top and left) locates the graphs along the top and left edges of the screen. The merit factor number, set to an initial value of 0 the first time you load a file, appears in the corner of the axes. The histogram graphs and the merit factor are described in the next sections.

To remove the histogram display from the screen, move the cursor to the letter H on the status line and press Button 1 or the space bar to toggle the letter H to red. The system keeps the histogram data in the database. Whenever you toggle the letter H back to green, the histogram data that appears reflects the changes you made while the histogram was not displayed.

We recommend that you leave the histogram display on while doing second and subsequent automatic placements of your PCB database. This allows you to compare different placements by viewing the change in the histogram graph and merit factor.

Histogram Display Graphs

The histogram program analyzes the number of nets (or ratsnest lines) and unused pins that cross each resource line, calculates that number against the total number of available routing channels, and gives a graphic display of the results.

Each histogram graph shows a line or bar (depending on whether you chose NARROW or WIDE during configuration) for each of the resource lines. The length and color of each line represent the percentage of channel usage across that resource line.

The color of the line represents the degree of congestion as follows:

green	=	0 - 35% channel usage
yellow	=	35 - 50% channel usage
red	=	over 50% channel usage

The length of the line or bar increases as the channel congestion along the resource line increases. You can estimate the percent of channel usage within each color range by noticing which lines are longer.

Merit Factor

The merit factor is displayed in the corner between the two histogram graphs (in the default configuration, this is the upper left corner). This number displays the relative improvement each placement change makes to the histogram.

When a change makes an improvement to the overall placement, the merit factor increases; for example, it might change from 0 to 2. When a change makes the placement less favorable, the merit factor decreases; for example, it might change from 0 to -1, or from 4 to 2. If the merit factor decreases when you make a swap, you should swap the parts back to their previous locations. You can use the SWAP/UNDO command to reverse a swap easily.

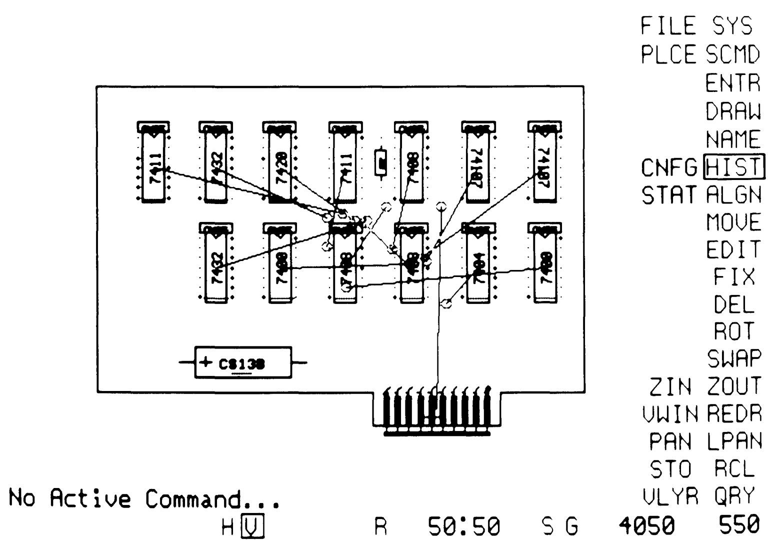
The merit factor is set to 0 the first time a histogram is loaded for a file. When you turn off the histogram display or save the file, the merit factor value is saved. If you redefine the histogram specifications for the file, the program resets the merit factor to 0.

USING THE FORCE VECTORS

To use the force vector indicators, place the cursor over the V on the status line and press Button 1 or the space bar to toggle the V to green. The system responds with the message:

Loading Force Vectors

Figure 2-14 shows the force vector display.



86389

Figure 2-14. Force Vectors

The force vector lines originate in the center of each component. (When you configure your database, you can choose whether or not discrete parts have force vectors.) The lines have varying angles and lengths, and each line has a circle at its end. The circle represents a theoretical "perfect" location for the component in the current placement arrangement. Of course, you cannot move each part to the indicated spot, but the force vector provides an indication of how far the part is from where it theoretically should be (by the length of the line) and the direction in which you should move it, if possible.

Force vectors are most helpful when the placement is random, as is the case when the components are placed by PC-PACK. After you have automatically placed the components, the force vectors are less useful and you should rely more on the histogram and merit factor as indicators.

The force vector graphic information is located on a layer named $FORC, which is automatically created by the program. You can easily change the color of the force vector display using the VLYR command.

To remove the force vector display from the screen, move the cursor to the letter V on the status line and press Button 1 or the space bar to toggle the V to red.

PREPARING YOUR DATABASE FOR AUTOMATIC PLACEMENT

Before you run the automatic placement program, prepare your database by creating the board outline, preplacing and fixing the position of certain components, such as connectors, and specifying parameters for placement. The following sections describe these steps.

> NOTE: You can also prepare your database before packaging in PC-PLACE and PC-CARDS and use this file as input for the PCB layer database file in PC-PACK. See the *PC-PACK User's Manual* for more information.

Creating the Board Outline

First, load your database. If you entered the database filename on the PC-PLACE Configuration Screen, the database is automatically loaded when you select **Edit Database**. If you did not enter the database filename on the configuration screen, use the FILE/LOAD command and enter the filename. Include the directory designator if necessary.

When the database is loaded, use the DRAW/LINE or DRAW/RECT command to create the board outline. The board outline must be on the BRDOUT layer so that PC-PLACE and PC-ROUTE, the automatic routing program, will recognize it. PC-PLACE will not place any parts closer than 100 mils from the board outline.

You can use the EDIT commands to edit the board outline.

Fixing Components

Use the MOVE or MOVE/COMP command and the FIX command to preplace and fix the position of components, such as connectors.

The MOVE command allows you to move a component by selecting the component to be moved and then selecting the location to place the component.

The MOVE/COMP command allows you to move a component by entering the component's reference designator to identify the component and then selecting the location to place the component.

After you have moved the components to the desired location, use the FIX command to fix the location of the components.

Specifying Lattice Points

Use the ENTR/LATP (Enter/Lattice Points) command to enter lattice points.

When you enter lattice points, the system draws a series of intersecting horizontal and vertical lines on the screen to form a lattice. Each intersecting point is a lattice point.

A component is placed on a lattice point at its origin. PC-PLACE is designed assuming your parts are created with the origin at pin 1. If your parts are created with an origin other than pin 1, you will need to allow for this when you specify the lattice points. For example, if the origin of your parts is at the lower left pin, you should enter the lattice points that define the first lattice point and the area of the lattice lower than you would if the origin was at the upper left pin.

Before you enter lattice points, add a lattice layer to the layer screen using the VLYR command. The lattice layer name must have the prefix LAT. Then enable and set it active.

When you select ENTR/LATP, the lattice layer is automatically set active.

> NOTE: If you do not add a lattice layer to the layer screen, the system will automatically add a layer named LAT1 and set it ABL and active.

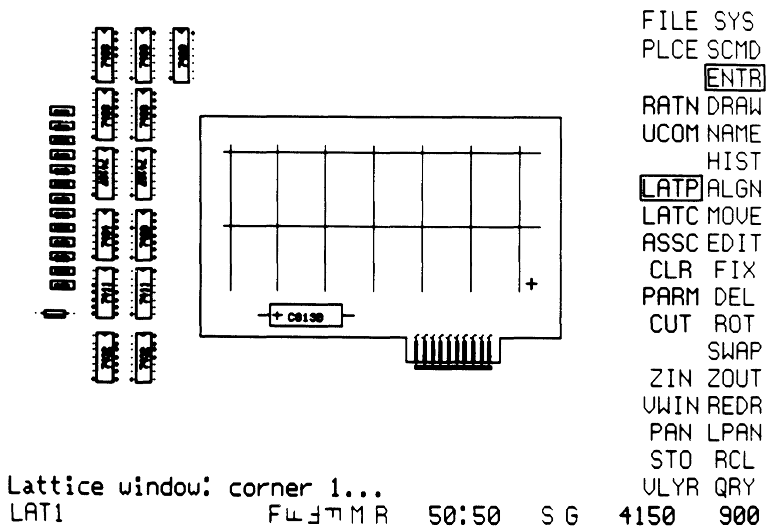
To enter lattice points, specify the distance vertically and horizontally between points in mils, and specify a window to define the area of the lattice.

To determine the lattice point specifications, consider the size and the number of components on your board. For example, if the components are 300-mils wide and you want the horizontal spacing between components to be 300 mils, then enter 600 for the x spacing.

If the components are 600-mils high and you want the horizontal spacing between components to be 400 mils, then enter 1000 for the y spacing. For the window, if you have 12 components, you need to specify a window large enough to create a lattice with at least 12 lattice points.

Using the specifications described in the example above, PC-PLACE will enter lattice points in increments of 600 mils in the horizontal direction and in increments of 1000 mils in the vertical direction within the window specified.

Figure 2-15 shows an example of a lattice with the specifications described above.



86390

Figure 2-15. Lattice

You can define more than one lattice for the placement of components. For example, you might want the automatic placement program to place all the major components on one lattice and all the discrete components on another lattice.

NOTE: Major components are integrated circuits.

To define more than one lattice, add a lattice layer name to the layer screen, then enable and set it active. The lattice layer name must have the prefix LAT. Assign the layer a different color to distinguish it from other lattice layers. Then enter lattice points using the ENTR/LATP command.

> NOTE: Before you run the automatic placement program, you must turn on all the lattice layers you are using.

In addition to specifying multiple lattices, you can also specify the orientation that parts should be placed on the lattices in increments of 90 degrees. To set the orientation, toggle the appropriate "F" on the status line to green before you run the automatic placement.

Creating Placement Barriers

After you have defined the lattice, you can place barriers around lattice points to prevent the automatic placement program from placing components on the lattice points within those barriers. If you have more than one lattice defined, the barriers will affect all the lattices.

Use the DRAW/RECT command to draw rectangles on the BARPLC layer to represent the barriers.

Figure 2-16 shows an example of a barrier around a lattice point.

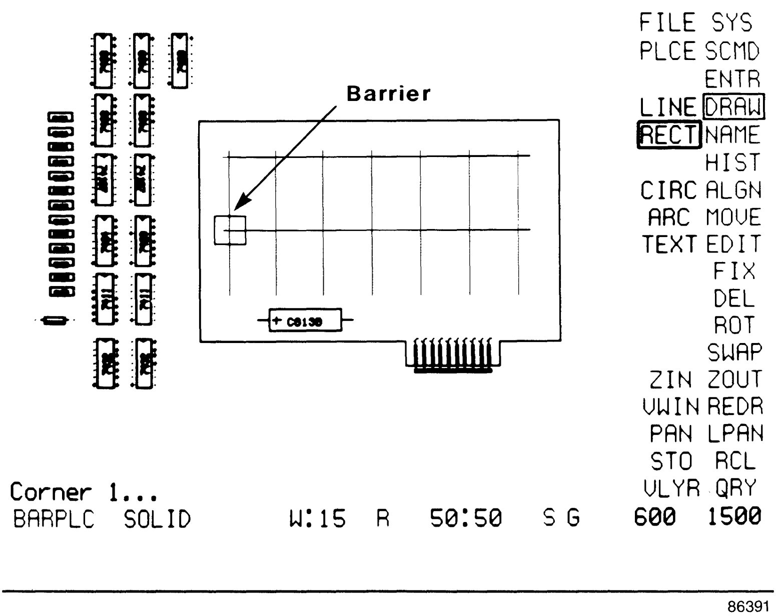


Figure 2-16. Barrier

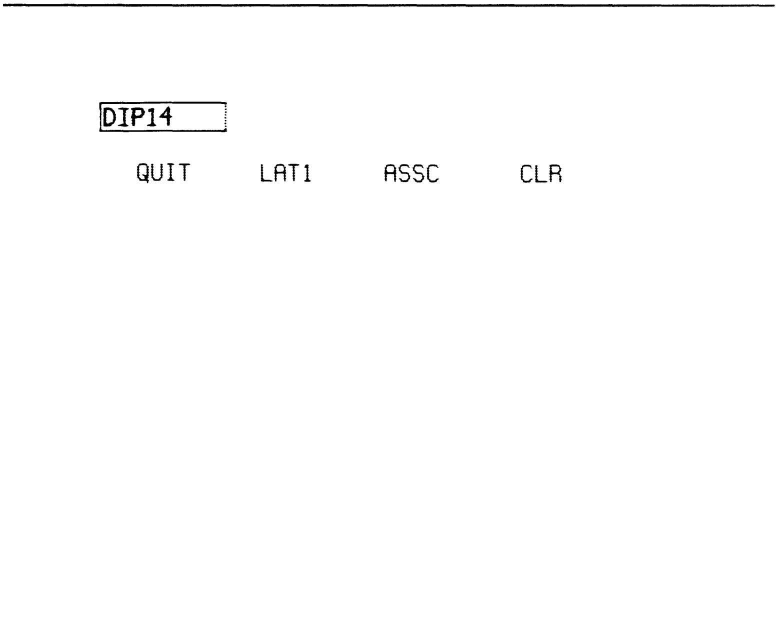
Specifying Components for Placement

Use the ENTR/LATC (Enter/Lattice Component) command to specify the major and discrete components to be placed on each lattice.

Components can be identified by part filename (for example, 7400), footprint attribute (for example, DIP14), or "ALL" for all major components.

You can specify different components for different lattices. For example, you may want all the major components on the LAT1 layer and all the discrete components on the LAT2 layer.

Figure 2-17 shows a sample of the Lattice Component Screen.



86392

Figure 2-17. Lattice Component Screen

Notice the additional entries LAT1, ASSC, and CLR. These parameters allow you to toggle between lattice layers, access the Component Association Screen, and access the Component Clearance Screen, respectively, without returning to the PC-PLACE Program Screen.

Specifying Component Clearance

Use the ENTR/CLR (Enter/Clearances) command to specify the minimum horizontal and vertical clearance required from the outline of a major component to another major component.

The clearance specified with this command specifies the minimum distance a major component can be placed from other major components, regardless of lattice points that may be closer to the component than the clearance specified.

Major components can be identified by part filename or footprint attribute.

Figure 2-18 shows a sample of the Component Clearance Screen.

```
DIP14     100  100

  QUIT   ASSC LATC
```

86393

Figure 2-18. Component Clearance Screen

Notice the additional entries ASSC and LATC. These parameters allow you to access the Component Association Screen and the Lattice Component Screen, respectively, without returning to the PC-PLACE Program Screen.

Specifying Component Association

Use the ENTR/ASSC (Enter/Association) command to specify the association of discrete components to major components.

Discrete components can be identified by part filename or footprint attribute.

You can specify the position (ABOVE, BELOW, LEFT, and/or RIGHT) of discrete components to major components. You can also specify the distance of discrete components from major components. The distance specified is from component outline to component outline. This distance will not be violated, regardless of lattice point locations that are closer to the components than the distance specified.

Figure 2-19 shows a sample of a Component Association Screen.

```
FCAP     ABOVE    BELOW    LEFT     RIGHT    0
RES      ABOVE    BELOW    LEFT     RIGHT    [75  ]

  QUIT      CLR      LATC     PGFOR     PGBAK
```

86394

Figure 2-19. Component Association Screen

Notice the additional entries CLR, LATC, PGFOR, and PGBAK. CLR and LATC allow you to access the Component Clearance Screen and the Lattice Component Screen, respectively. PGFOR and PGBAK allow you to access the following or previous page on your screen when the entries on the page exceed 15 lines.

Specifying Cutline Placement

Use the ENTR/CUT (Enter/Cutlines) command to specify cutlines across the board in horizontal and vertical directions.

Cutlines divide the board into partitions. The automatic placement program tries to place components that are heavily interconnected in the same partition to minimize the number of nets that cross the cutlines.

When you select ENTR/CUT, a layer named CUTPLC is added to the layer screen, which is enabled and set active. A status line parameter is also displayed that allows you to toggle between HORZ (horizontal) and VERT (vertical) directions.

To determine the number and the location of the cutlines, consider the shape and size of your board, and the location of the lattice points.

You can enter multiple cutlines in both the horizontal and vertical directions.

You can move cutlines using the MOVE or EDIT/MOVS commands, or you can delete cutlines using the DEL or EDIT/DELS commands.

If you do not specify cutlines, the system specifies them for you. System-generated cutlines are not displayed on the screen. You can check the location of the cutlines by viewing or printing the placement report. We recommend that you let the system specify cutlines for you.

Figure 2-20 shows cutlines on a board.

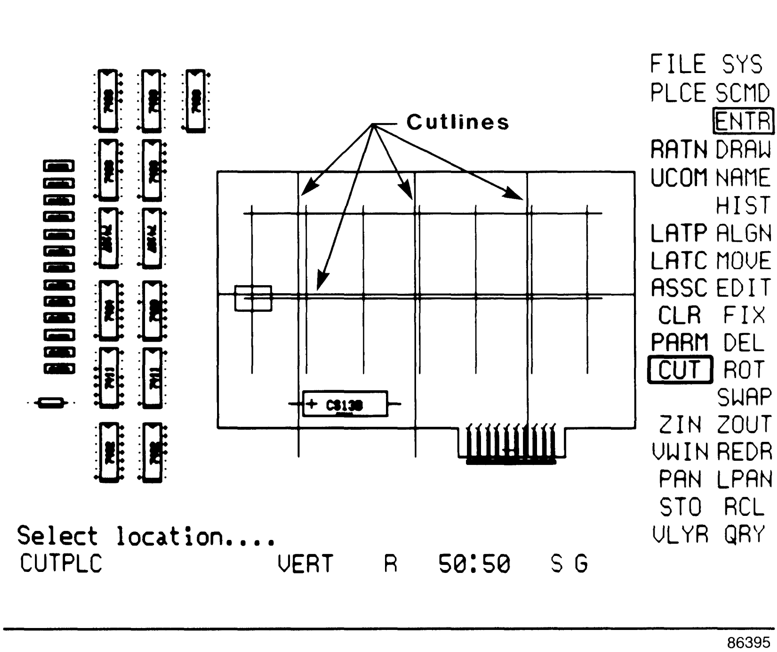


Figure 2-20. Cutlines

Specifying Sorting Parameters

Use the ENTR/PARM (Enter/Parameters) command to specify parameters that affect the order in which components are placed.

PC-PLACE usually sorts and places from the largest to the smallest component. However, there are other sorting parameters you can specify that affect this order.

You can specify the order in which components are placed by height or by width on the lattice. For example, if you have defined a lattice that is higher than it is wide, the automatic placement program will be restricted by the width of the lattice more than by the height. Therefore, you should specify "width" as the sorting parameter.

You can also specify the order in which components are placed by the constraint index.

As mentioned above, PC-PLACE usually sorts and places components from the largest to the smallest. However, if you have only a few small components of a certain component type and only a few lattice points available on that lattice layer, you may want to specify a constraint index that will place these components first. Otherwise, these lattice points may be blocked by larger components on another lattice layer.

The constraint index must be equal to or less than the constraint ratio of the components you want placed first. The constraint ratio of a component type describes how heavily constrained the placement of those components are on the lattice. PC-PLACE determines the constraint ratio of components by dividing the number of components of that type by the total number of lattice points available. If the constraint ratio is greater than or equal to the constraint index, then those component types will be placed before less constrained components.

For example, if you have two overlapping lattices, one small lattice that has only 10 lattice points available for 9 DIP14s and therefore the DIP14s have a constraint ratio of 9/10 or 90 percent, and another larger lattice specified for all other major components, you should specify a constraint index less than 90 percent, such as 80 percent. Then these components will be placed first before other less constrained components.

STARTING THE AUTOMATIC PLACEMENT PROGRAM

Before you start the automatic placement program, be sure that:

- You have specified all the required parameters.
- Each part has a footprint attribute.
- The correct orientation is set on the status line.
- All lattice layers you will be using are turned on.

To start the automatic placement of components, select the PLCE (Place) command. When you select PLCE, the system prompts:

Ready to Place? YES NO

Select YES to start the program. The following messages are displayed sequentially:

Beginning Placement

Extracting database

Processing cutline <n>....

Placing major devices

Placing discrete devices

Updating database

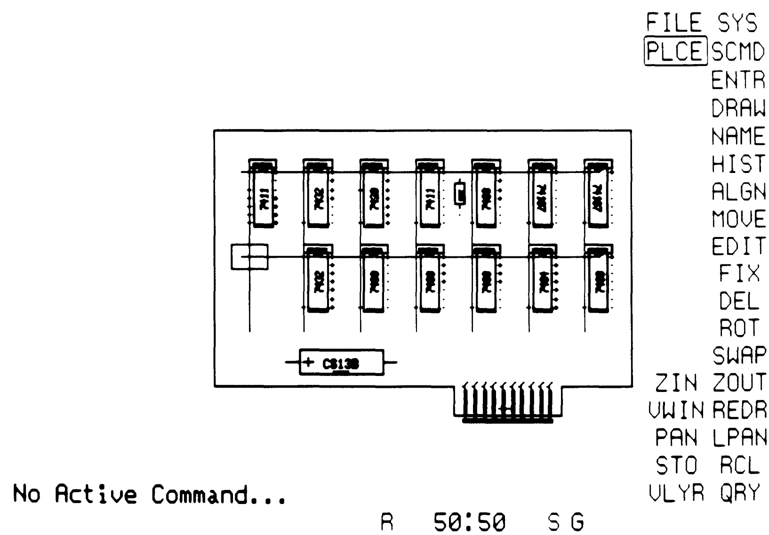
Creating report

Placement complete

When the automatic placement is complete, the system redraws the screen and the components are now initially placed inside the board outline.

Any components not placed, either because there were not enough lattice points specified or because there was no association specified, are moved below the board outline.

Figure 2-21 shows a board that was placed with the automatic placement program.



86396

Figure 2-21. Placed PCB Database

EVALUATING THE PLACEMENT OF COMPONENTS

After you have automatically placed the components inside the board outline, view or print the placement report as described in the following section "Viewing and Printing Report Files" to examine the statistics of the placement. Refer to Appendix B for a description and a sample of the placement report. If you are not satisfied with the component placement, you can specify different parameters, such as a new lattice or new cutlines and rerun the automatic placement program, or you can improve the placement interactively using the histogram, force vectors, and merit factor display.

If you rerun the automatic placement program, be sure to first save the placed database and report files under a unique name so you have a record of the placement.

Before you rerun the automatic placement program, turn on the histogram display by toggling the H on the status line to green. The merit factor should be set to 0. Each time you run the automatic placement, you can see the relative improvement of the placement by the histogram display and the merit factor.

> NOTE: If you have a large database, you may want to leave the histogram display off during automatic placement to shorten the program runtime and to save memory.

You can use the HIST/STAT command to record the current histogram and merit factor before running the automatic placement. Then turn the histogram display off by toggling the H on the status line to red. When the automatic placement is complete, turn the histogram display back on to display the change in the placement.

After you run the automatic placement program, view or print the placement report for that run, compare the data with the previous run, and choose the optimal placement.

Viewing and Printing Report Files

You can view or print the output report files. These reports include the placement report (<filename>.PLR), the histogram report (<filename>.HIS), the swap report (<filename>.RPT), and the error report (<filename>.ERR).

To display or print a report file from the DOS prompt C:\>, use the DOS TYPE and [Ctrl]-[P] commands. For example, to display a file, type:

TYPE TUTORPLC.PLR [Return]

Or, to print a file, type:

TYPE TUTORPLC.PLR [Ctrl]-[P] [Return]

When the file is printed, press [Ctrl]-[P] to turn off the print function.

You can also use the DOS PRINT command to print a file.

To display or print an output file from within the PC-PLACE environment, select the SYS/DOS (System/DOS) command on the main menu. Then enter the DOS command TYPE and the filename to display the file, or enter TYPE, the filename, and [Ctrl]-[P] [Return] to print the file.

Analyzing Part-to-Part Interconnections

At any time during the placement process, you can display a single ratsnest line from pin to pin, the ratsnest lines of a particular component, or the full ratsnest of all the components on the board. Use the QRY/NET (Query/Net) command to help you analyze the part-to-part interconnections. The QRY/NET command displays the net name and ratsnest line connected to a pin, indicating what component pin the pin queried is connected to.

Use the SWAP/COMP (Swap/Component) command to display all the connections of a component.

Toggle the R (Ratsnest) status line parameter to green to turn on the full ratsnest display of all the components on the board.

Improving Component Placement

The histogram and force vectors are tools to make it easier for you to improve the placement. You must make the actual changes by moving and aligning components and swapping components and gates. The histogram display indicates the areas of the board that need the most improvement and the force vectors suggest directions and distances for placement changes. The program immediately shows you the relative improvement your changes make.

You can turn either the histogram or the force vectors on and off at any time. To make your changes, use the menu commands as you would in PC-CARDS. Each time you perform a move using the MOVE or MOVE/COMP command, alignment using the ALIGN commands, or swap using the SWAP commands, the system displays one or both of the following messages, depending on which of the options you are using:

Updating Histogram

then

Updating Force Vectors

The histogram display is automatically updated to the new placement arrangement and the merit factor reflects the change. Then the force vectors are updated for the new arrangement.

PREPARING YOUR DATABASE FOR PC-ROUTE

If you plan to use PC-ROUTE to route a PCB, and there are areas on the PCB that you want to block from routing, you must first use PC-PLACE or PC-CARDS to add routing barriers.

PC-ROUTE recognizes nine layers for routing barriers and one layer for via barriers. Add to your database as many of these layers as you require. Use the VLYR command to display the layer screen. To add a new layer, move the cursor to the space below the last layer and press Button 1. Enter the name and color for the new layer and set it to ABL.

Use the DRAW/RECT (Draw/Rectangle) command to draw rectangles to represent the barriers. Be sure to enter each barrier on the appropriate layer. The barrier layers and their functions are listed below.

> **BARALL** - Prevents PC-ROUTE from routing traces within the barrier rectangle on all routing layers.
>
> **BARCMP** - Prevents PC-ROUTE from routing traces within the barrier rectangle on the component layer only.
>
> **BARSLD** - Prevents PC-ROUTE from routing traces within the barrier rectangle on the solder layer only.

BARIN1 through BARIN6 - Prevents PC-ROUTE from routing traces within the barrier rectangle on the internal layers 1 through 6, respectively.

BARVIA - Prevents PC-ROUTE from placing vias within the barrier rectangle, but allows routing of traces through it.

NOTE: PC-PLACE does not include the ENTR/WIRE command. If you want to pre-wire traces before running PC-ROUTE, you must use PC-CARDS.

EXITING PC-PLACE

When you have finished editing a database, use the FILE/SAVE command to save the updated database. Change the filename extension of the updated database to .PLC. This extension is not required for program operation, but it is recommended to help you keep your files clearly organized.

To exit PC-PLACE, select the SYS/QUIT command. The opening menu appears. Select **Exit PC-PLACE.** The system returns you to DOS.

PART B. EXAMPLE USING PC-PLACE

This part provides step-by-step instructions for using PC-PLACE. In this example, you will automatically place components on a board. The steps in this example are:

Step 1. Start the Program
Step 2. Configure Your System
Step 3. Configure the Histogram Display
Step 4. Create the Board Outline
Step 5. Fix the Components
Step 6. Specify Lattice Points
Step 7. Create the Placement Barriers
Step 8. Specify Components for Placement
Step 9. Specify Component Association
Step 10. Specify Component Clearance
Step 11. Start the Automatic Placement
Step 12. Evaluate the Placement

If you have not already done so, load the example data files into your project directory.

At the DOS prompt C:\> type:

DIR <directory name>

to confirm the copies. Your directory listing should include the following files:

TUTORPLC.PCB
7400.PRT
7404.PRT
7408.PRT
7411.PRT
7420.PRT
7421.PRT
7432.PRT
74107.PRT
CON20.PRT
RC07.PRT
CK05.PRT
CS13B.PRT

Step 1. Start the Program

First, change the current directory to the project directory you plan to use. Then, to start PC-PLACE, type:

PCPLACE [Return]

The system displays the PC-PLACE Title Screen. Press any key to display the opening menu.

Step 2. Configure Your System

1. Press the space bar to move to the **Configure PC-PLACE** option and press [Return]. The PC-PLACE Configuration Screen is displayed.

2. Select the appropriate parameters. When you select the last option, the system returns you to the PC-PLACE Opening Menu.

Step 3. Configure the Histogram Display

In this step, you will configure the database for the histogram display.

1. Select **Edit Database** on the PC-PLACE Opening Menu. The PC-PLACE Program Screen is displayed.

2. Select HIST/CNFG to display the file configuration parameter options.

3. Press [Return] after each option to accept the defaults displayed.

   After you accept the last option, the system prompts:

   Save this configuration?
   (Y or No

4. Press [Return] to accept these parameters for this session.

Step 4. Create the Board Outline

1. Load TUTORPLC.PCB using the FILE/LOAD command.

   NOTE: When you load the example data file, you may need to zoom out using the ZOUT command or zoom in using the ZIN command, depending on the resolution of your monitor, to match the screen display shown Figure 2-22.

   You may also need to change the current viewing window using the PAN command.

   Figure 2-22 shows what your screen should look like.

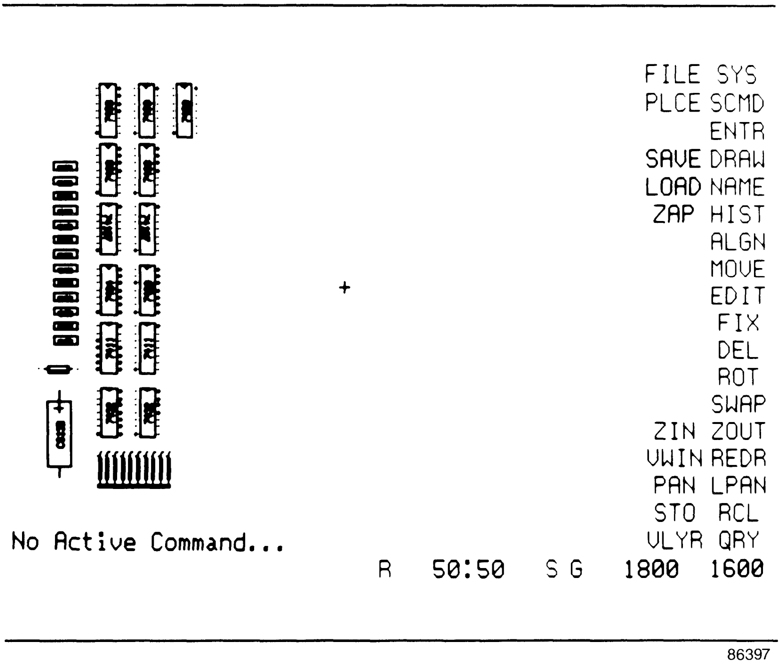


Figure 2-22. Screen Display After TUTORPLC.PCB is Loaded

2. Select DRAW/LINE. The system prompts:

   Select start point....

3. Set the BRDOUT layer active.

4. Select the X Y cursor coordinate field on the status line to set the cursor location to 0 150 for the first point. The system prompts:

   Enter x position:

5. Type 0 150 and press [Return]. The system prompts:

   Select next point...

6. Select the rest of the corners of the board outline in the same manner as described above. Enter the cursor locations indicated by the arrows in Figure 2-23.

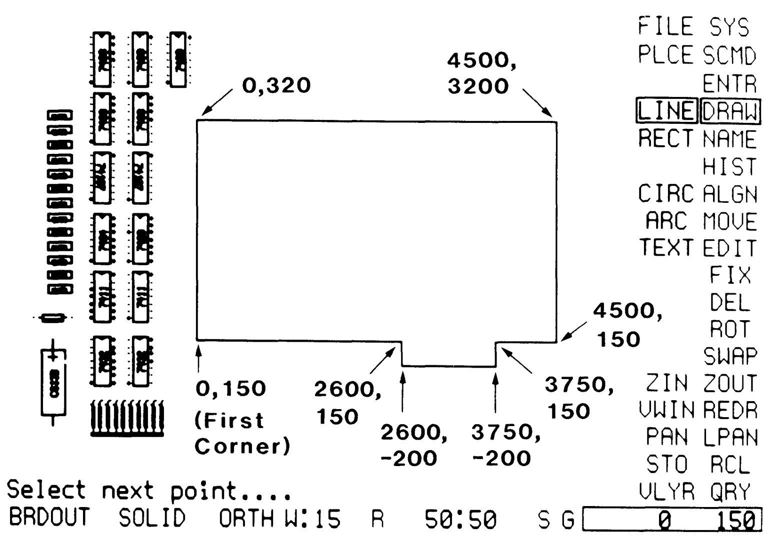


Figure 2-23. Board Outline

Step 5. Fix the Components

1. Select MOVE/COMP to move the connector and capacitor (C1) to inside the board outline. The system prompts:

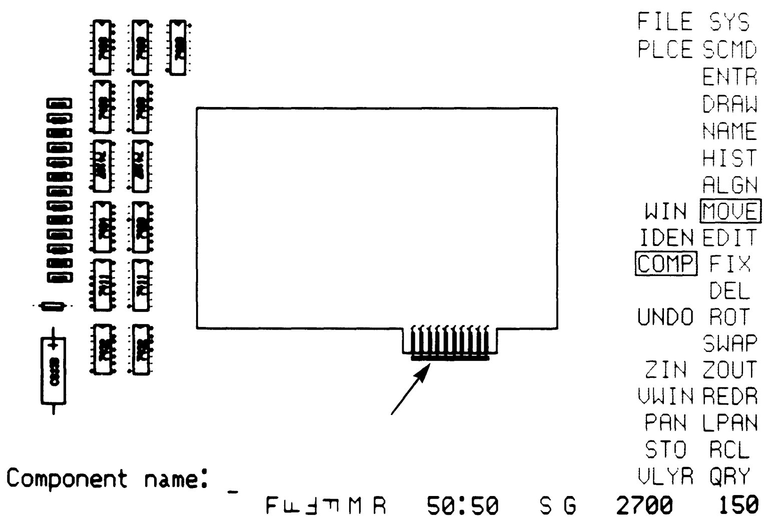
    Component name:

2. Type P1 and press [Return] for the reference designator of the connector. The system prompts:

    Select loc to place comp.
    (Orientation OK?).

3. Select the location shown in Figure 2-24. The system continues prompting:

    Component name:



86399

Figure 2-24. Location for Connector

4. Type C1 and press [Return] for the reference designator of the capacitor. The system prompts:

   Select loc to place comp.
   (Orientation OK?).

5. Select the location shown in Figure 2-25.

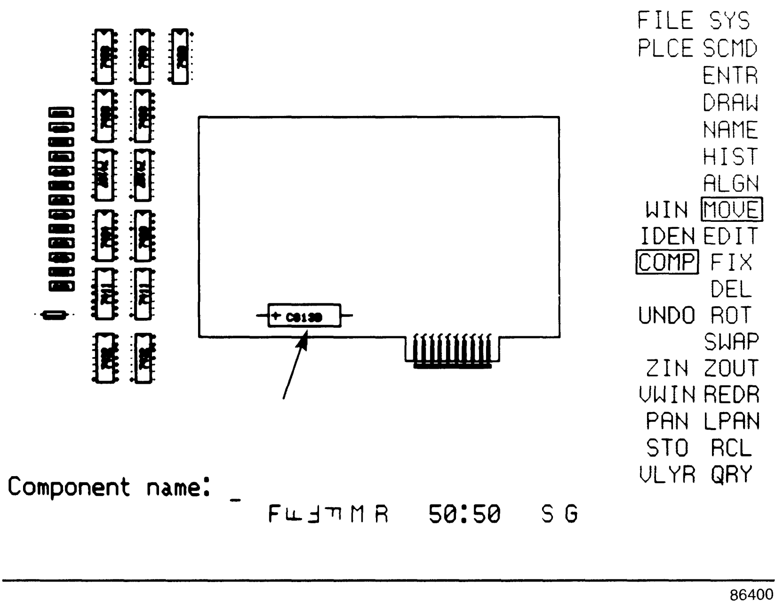


Figure 2-25. Location for Capacitor

6. Select FIX to fix the position of the components prior to automatic placement. The system prompts:

   Select a component....

7. Select the connector and the capacitor. The fixed components are highlighted.

Step 6. Specify Lattice Points

In this step, you will define a lattice to describe the locations at which the components can be placed.

1. Select ENTR/LATP to enter lattice points. A layer named LAT1 is added to the layer screen, which is set ABL and active. The system prompts:

   Enter x spacing:

2. Type 600 and press [Return] for the horizontal spacing between lattice points. The system prompts:

   Enter y spacing:

3. Type 1000 and press [Return] for the vertical spacing between lattice points. The system prompts:

   Lattice window: corner 1...

   NOTE: When you specify corner 1 of the lattice window, you are also specifying the location of the first lattice point.

4. Position the cursor for corner 1 as shown in Figure 2-26 and press Button 1. The system prompts:

   Lattice window: corner 2...

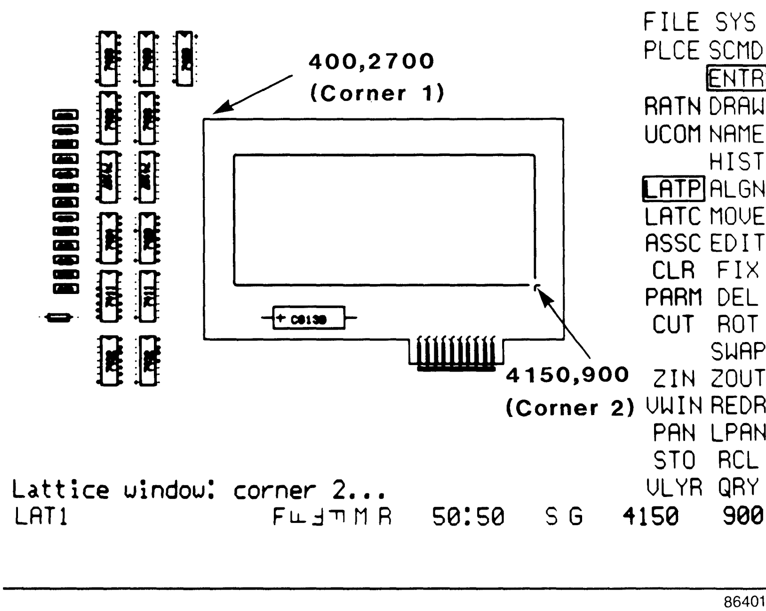


Figure 2-26. Cursor Location for Corner 1 and 2

5. Position the cursor for corner 2 as shown in Figure 2-26 above and press Button 1.

   Figure 2-27 shows what your screen should look like.

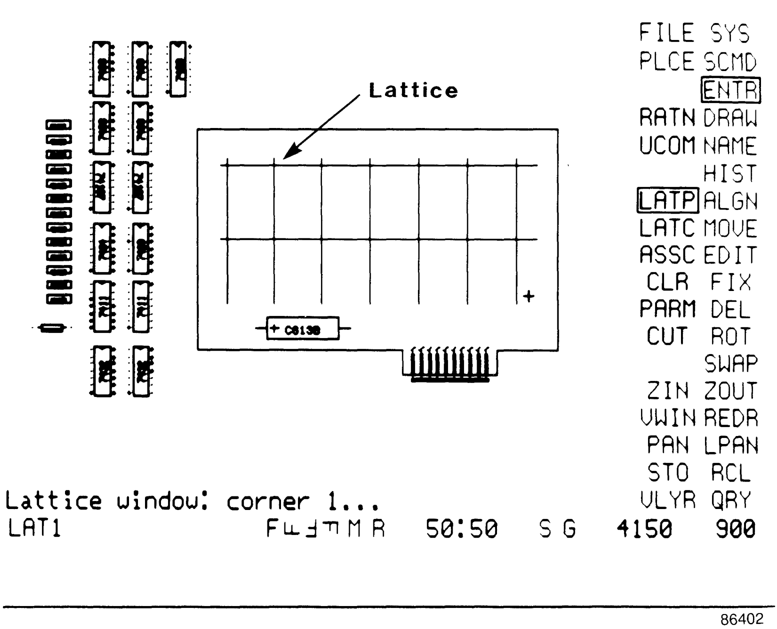


Figure 2-27. Lattice Defined

6. Press Button 2 to confirm the lattice selection.

Notice that there are 13 integrated circuits to be placed and 14 lattice points available.

Step 7. Create the Placement Barriers

In this step, you will place a barrier around a lattice point to prevent PC-PLACE from placing a component there.

1. Using the VLYR command, add a layer named BARPLC to the layer screen. Assign it a color, and set it ABL and active.

2. Select DRAW/RECT. Select two points to draw the rectangle shown in Figure 2-28.

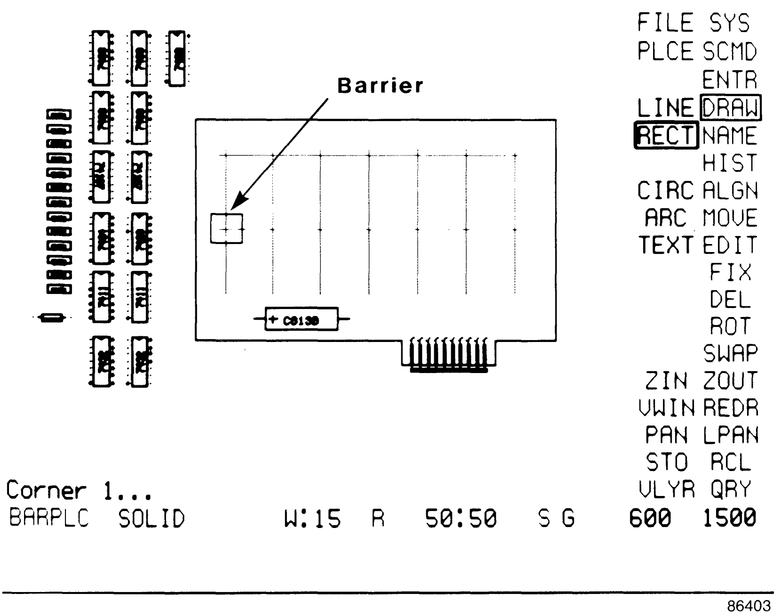


Figure 2-28. Barrier Placed

Step 8. Specify Components for Placement

In this step, you will specify the components that should be placed on the lattice.

1. Select ENTR/LATC. The system displays the Lattice Component Screen as shown in Figure 2-29.

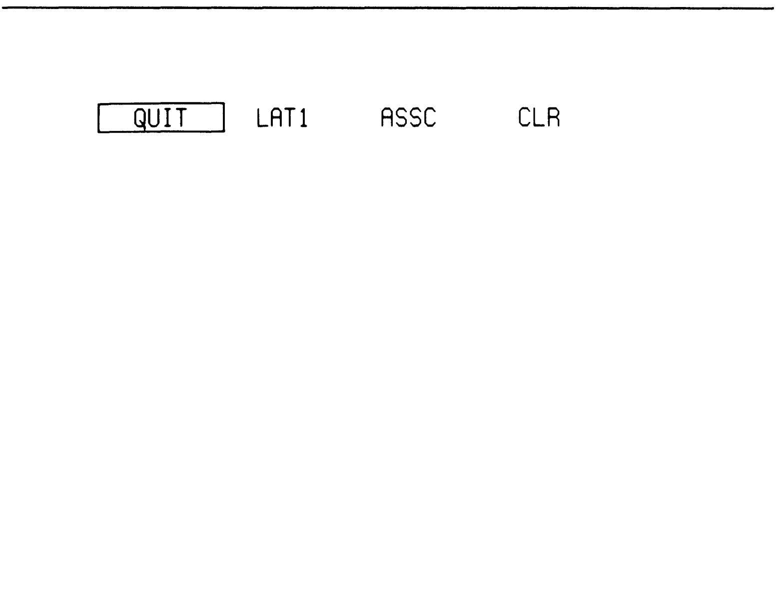


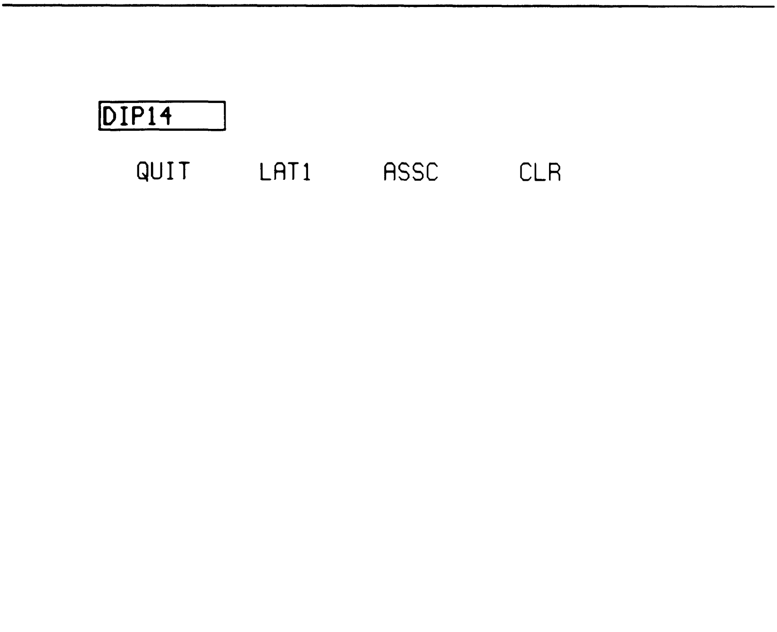
Figure 2-29. Lattice Component Screen

2. Move the cursor to the empty space above QUIT and press Button 1. The system prompts:

   Comp name:

3. Type DIP14 for the footprint attribute of the components to be placed on the LAT1 lattice and press [Return].

Figure 2-30 shows what your screen should look like.



86405

Figure 2-30. Lattice Component Screen After DIP14 is Entered

Step 9. Specify Component Association

In this step, you will specify the position and the distance of the filter capacitors and the resistors to the DIP14s.

1. Select ASSC on the Lattice Component Screen. The system displays the Component Association Screen as shown in Figure 2-31.

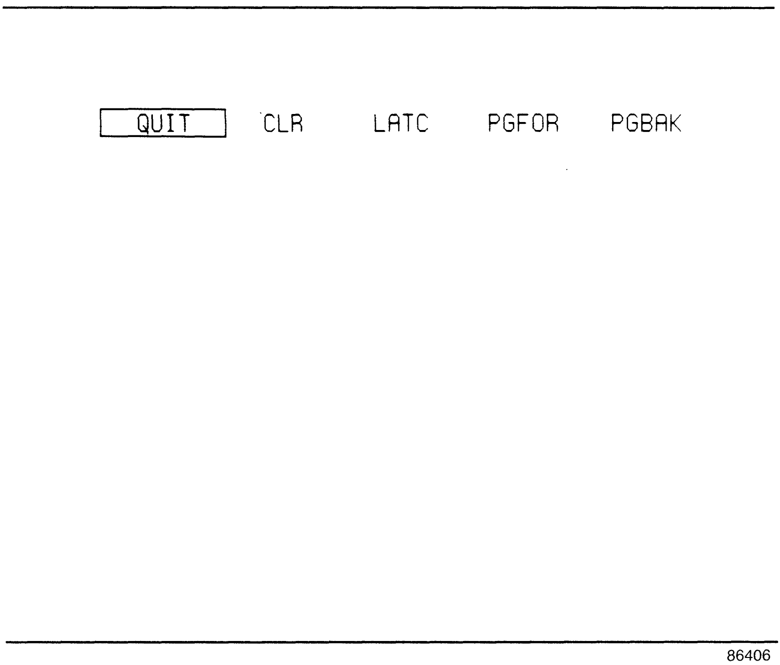


Figure 2-31. Component Association Screen

2. Move the cursor to the empty space directly above QUIT and press Button 1. The system prompts:

   Comp name:

3. Type FCAP and press [Return].

   Figure 2-32 shows what your screen should look like.

```
[FCAP    ]  ABOVE    BELOW    LEFT     RIGHT    100
     QUIT      CLR      LATC     PGFOR    PGBAK
```

86407

Figure 2-32. Component Association Screen After FCAP Entered

4. Select ABOVE to toggle the position on. ABOVE should be green.

5. Select the number on the right side of the screen. The system prompts:

   Enter distance:

6. Type 0 and press [Return] to set the distance.

NOTE: A distance of 0 means the filter capacitor will be placed outline to outline with a major component.

7. Add RES to the list on the Component Association Screen using the same procedure described above. Then set the position to LEFT and RIGHT, and set the distance to 75.

   Figure 2-33 shows what your screen should look like.

```
FCAP    ABOVE    BELOW    LEFT    RIGHT   0
RES     ABOVE    BELOW    LEFT    RIGHT   75

  QUIT     CLR     LATC     PGFOR    PGBAK
```

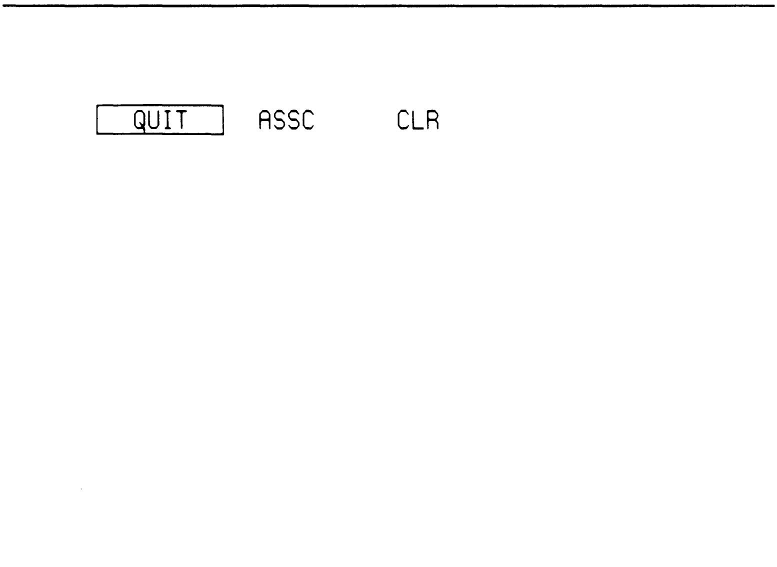
86408

Figure 2-33. Component Association Menu After FCAP and RES Entered

Step 10. Specify Component Clearance

In this step, you will specify the horizontal and vertical clearance required from the outline of the DIP14s to other components.

1. Select CLR on the Component Association Menu. The system displays the Component Clearance Screen as shown in Figure 2-34.



86409

Figure 2-34. Component Clearance Screen

2. Select the empty space directly above QUIT. The system prompts:

   Comp name:

3. Type DIP14 and press [Return]. The horizontal and vertical distance of these major components is set to 100.

   For this example, the defaults are used.

   Figure 2-35 shows what your screen should look like.

```
DIP14     100   100

   QUIT    ASSC  LATC
```

86410

Figure 2-35. Component Clearance Screen After DIP14 Entered

4. Select QUIT and the PC-PLACE Program Screen is displayed.

For this example, the system default cutlines and sorting parameters are used.

Now that you have specified the placement parameters, you are ready to start the automatic placement program.

Step 11. Start the Automatic Placement

1. Select PLCE to start the automatic placement of components. The system prompts:

   Ready to place? YES NO

2. Select YES. The program starts executing and the following messages are displayed:

   Beginning Placement

   Extracting database

   Processing cutline 1.....

   Processing cutline 2.....

   Processing cutline 3.....

   Processing cutline 4.....

   Placing major devices

   Placing discrete devices

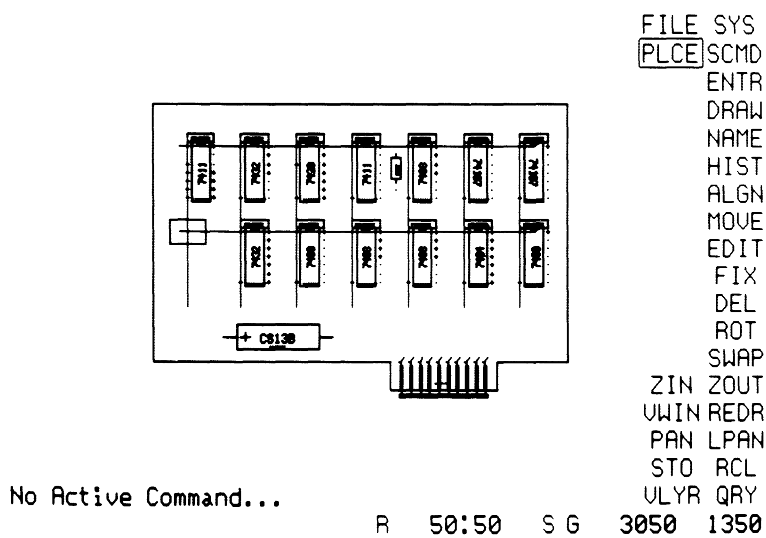
   Updating database

   Creating report

   Placement complete

The system then redraws the screen and the components are placed inside the board outline.

Figure 2-36 shows what your screen should look like.



86411

Figure 2-36. Placed PCB Database

3. Select FILE/SAVE to save the database. The system prompts:

   Enter file name:

4. Type TUTORPLC.PLC for the database name and press [Return].

5. Select SYS/QUIT to exit the PC-PLACE Program Screen. The PC-PLACE Opening Menu is displayed.

6. Select Exit PC-PLACE to exit the program.

Step 12. Evaluate the Placement

In this step, you will print the placement report.

1. From the DOS prompt, type:

    TYPE TUTORPLC.PLR [Ctrl]-[P]
    [Return]

2. Type [Ctrl]-[P] to turn off the print function.

3. Use the placement report to analyze the placement of the components.

CHAPTER 3. COMMANDS

This chapter supplements Chapter 3, "Commands" of the *PC-CARDS User's Manual.* It lists only the PC-PLACE commands that are not included in PC-CARDS. For information about other PC-PLACE commands, refer to the *PC-CARDS User's Manual.*

The commands are listed in alphabetical order. Each command description is divided into three parts:

FUNCTION: A short description of the purpose of the command.

REMARKS: Some things you need to know about the command.

OPERATION: Detailed procedures for using the command, including system prompts and user responses.

In PC-PLACE, all commands are used in DETL mode; PC-PLACE has no SYMB mode. You can use either of two procedures to select a command from the menu:

- Use the mouse or the cursor keys to move the cursor to the command, then press Button 1 or the space bar.

- Enter the command from the keyboard by typing a slash (/). The system prompts:

  Menu command:

  Type the command and press [Return].

  To select a submenu command, again type /, the subcommand, and [Return].

To cancel a command, you can select another command, press Button 2, or press [Esc].

ALGN

(Align)

FUNCTION:

Allows you to align components.

REMARKS:

When more than one component must be moved to achieve the alignment, use the IDEN or WIN subcommand. Used alone, the ALGN command allows you to align only one component.

OPERATION:

1. Select ALGN. The system prompts:

   Select the component(s)....

2. Select the component to align. The system prompts:

   Select alignment point...

3. Select a point to define the alignment.

   The ALGN command moves components along one axis only. The point you select determines whether the move is horizontal or vertical. If the point is farther horizontally, it defines the new x coordinate for the component's origin; if the point is farther vertically, it defines the y coordinate for the component's origin.

ALGN

The system moves the component to the specified location and prompts you again to select a component.

4. Select and align another component or terminate the command.

ALGN/IDEN

(Align/Identify)

FUNCTION:

Aligns a number of selected components.

REMARKS:

With this command, you select components and the system arranges them into rows or columns. The system selects horizontal or vertical alignment depending on the predominant orientation of the components selected.

OPERATION:

1. Select ALGN/IDEN. The system prompts:

   Select component. (BUT2 to term. selection)

2. Place the cursor on the component and press Button 1 to select the component. The system highlights the component and displays the same prompt again.

3. Repeat step 2 until you are finished with your selection.

4. Press Button 2 to terminate the selection. The system prompts:

   Select alignment point...

ALGN/IDEN

5. Select a point to define the alignment location. If alignment is vertical, the point defines the x coordinate of the component's origin. If alignment is horizontal, the point defines the y coordinate of the component's origin.

   The system aligns the selected components.

   This command moves the components along one axis only, so components may overlap once aligned. If components overlap, use the MOVE or MOVE/COMP command to place them correctly.

   The system prompts again to select components.

6. Continue selecting and aligning objects or terminate the command.

ALGN/UNDO

(Align/Undo)

FUNCTION:

Reverses the last change that was made using the ALGN command.

REMARKS:

To reverse the change, you must use UNDO before performing any other operation, including using Button 2 to terminate the current command. (You can use the viewing commands, such as ZOUT or PAN, and still use UNDO.)

OPERATION:

Select UNDO to reverse the last change. The system moves the components back to their original positions and updates the histogram and force vectors if they are displayed.

ALGN/WIN

(Align/Window)

FUNCTION:

Aligns a number of components enclosed in a window.

REMARKS:

All the components fully or partially enclosed in the window are arranged into rows or columns. The system selects horizontal or vertical alignment depending on the predominant orientation of the components.

OPERATION:

1. Select ALGN/WIN. The system prompts:

   Corner 1....

2. Select the first corner of the window by using the cursor or by entering grid coordinates. The system prompts:

   Corner 2....

3. Select the opposite corner of the window. The system highlights the enclosed components and prompts:

   Select alignment location...

ALGN/WIN

4. Select a point to define the alignment location. If alignment is vertical, the point defines the x coordinate of the origin of the components. If alignment is horizontal, the point defines the y coordinate of the origin of the components.

   The system aligns the components.

   This command moves the components along one axis only, so components may overlap once aligned. If components overlap, use the MOVE command to place them correctly.

   The system prompts again to select another window.

5. Continue windowing and aligning objects or terminate the command.

ENTR

(Enter)

FUNCTION:

Enables the submenu for entering ratsnests, uncommitted pins, lattice points, lattice component data, component associations, horizontal and vertical component clearances, sorting parameters, and cutlines.

OPERATION:

Select ENTR to enable the following submenu options:

RATN	(Ratsnest)
UCOM	(Uncommit a Pin)
LATP	(Lattice Points)
LATC	(Lattice Component Types)
ASSC	(Association)
CLR	(Clearances)
PAR	(Parameters)
CUT	(Cutlines)

These commands are described in this chapter, except for the ENTR/RATN and ENTR/UCOM commands. Refer to Chapter 3, "Commands" in the *PC-CARDS User's Manual* for more information on those commands.

ENTR/ASSC

(Enter/Association)

FUNCTION:

Specifies the association of discrete components to major components.

REMARKS:

The ENTR/ASSC command allows you to specify the position (ABOVE, BELOW, LEFT, or RIGHT) of discrete components to major components.

You can also specify the distance of discrete components from major components. The distance is measured in mils from discrete component outline to major component outline.

Discrete components can be identified by part filename (for example, RC07) or footprint attribute (for example, RES).

The ENTR/ASSC command is useful for placing filter capacitors or "pull-up" resistors which are connected to a specific integrated circuit.

OPERATION:

1. Select ENTR/ASSC. The system displays the Component Association Screen as shown below:

QUIT CLR LATC PGFOR PGBAK

ENTR/ASSC

where:

QUIT returns you to the PC-PLACE program screen.

CLR displays the Component Clearance Screen. Refer to the ENTR/CLR command description for more information.

LATC displays the Lattice Component Screen. Refer to the ENTR/LATC command description for more information.

PGFOR allows you to page forward through entries when the entries on the screen exceed 16.

PGBAK allows you to page backward through entries when the entries on the screen exceed 16.

2. You can make the following changes on the Component Association Screen:

   - To add a discrete component, select the empty space directly above QUIT. The system prompts:

     Comp name:

     Type the part filename or the footprint attribute and press [Return]. The following options are displayed:

     ABOVE BELOW LEFT RIGHT 100

ENTR/ASSC

where:

ABOVE means the discrete component specified can be placed above a major component.

BELOW means the discrete component specified can be placed below a major component.

LEFT means the discrete component specified can be placed to the left of a major component.

RIGHT means the discrete component specified can be placed to the right of a major component

- To change the position of a discrete component to a major component, select ABOVE, BELOW, LEFT, or RIGHT to toggle them on or off. If the position is highlighted, the position is on. You can specify more than one position to be on.

- To change the distance between a discrete component type and a major component, select the number on the right side of the screen. The system prompts:

  Enter distance:

  Type the distance desired.

  The default is "100."

  NOTE: A distance of 100 means the discrete component will be placed 100 mils from its outline to the outline of a major component.

ENTR/ASSC

A distance of 0 means the discrete component will be placed outline to outline with a major component.

- To delete a discrete component name from the list, select the component part filename or footprint attribute. The system prompts:

  OK to delete? YES NO

  Select YES to delete the name.

- To display the Component Clearance Screen, select CLR. Refer to the ENTR/CLR command description for more information.

- To display the Lattice Component Screen, select LATC. Refer to the ENTR/LATC command description for more information.

ENTR/CLR

(Enter/Clearances)

FUNCTION:

Specifies the horizontal and vertical clearance required from the outline of a major component to another component.

REMARKS:

The ENTR/CLR command allows you to specify the minimum distance a component can be placed from other components, regardless of lattice points that may be closer to the component than the clearance specified.

Major components can be identified by part filename or footprint attribute.

OPERATION:

1. Select ENTR/CLR. The system displays the Component Clearance Screen as shown below.

   QUIT ASSC LATC

   where:

   QUIT returns you to the PC-PLACE Program Screen.

ENTR/CLR

ASSC displays the Component Association Screen. Refer to the ENTR/ASSC command description for more information.

LATC displays the Lattice Component Association Screen. Refer to the ENTR/LATC command description for more information.

2. You can make the following changes on the Component Clearance Screen:

   - To add a major component, select the empty space directly above QUIT. The system prompts:

     Comp name:

     Type the component filename or footprint attribute and press [Return]. The following options are displayed next to the component filename or footprint attribute:

     100 100

     where:

     The leftmost **100** means the major component will be placed a minimum of 100 mils from other major components in the horizontal direction.

ENTR/CLR

The rightmost **100** means the major component specified will be placed a minimum of 100 mils from other major components in the vertical direction. If the number of entries exceeds 16, the entries will appear in columns on the screen.

- To change the clearance of major components in the horizontal direction, select the leftmost number. The system prompts:

  Enter new x clearance:

- To change the clearance of components in the vertical direction, select the rightmost number. The system prompts:

  Enter new y clearance:

- To delete a clearance definition, select the component part filename or footprint attribute. The system prompts:

  OK to delete? YES NO

- To return to the PC-PLACE program screen, select QUIT.

- To display the Component Association Screen, select ASSC. Refer to the ENTR/ASSC command description for more information.

- To display the Lattice Component Screen, select LATC. Refer to the ENTR/LATC command description for more information.

ENTR/CUT

(Enter/Cutlines)

FUNCTION:

Specifies the horizontal and vertical cutlines across the board outline.

REMARKS:

Cutlines divide the board into partitions. PC-PLACE tries to place components that are heavily interconnected in the same partition to minimize the number of nets that cross the cutlines.

When you select ENTR/CUT, a layer named CUTPLC is added to the layer screen, which is enabled and active. A status line parameter is also displayed that allows you to toggle between HORZ (horizontal) and VERT (vertical) directions.

To determine the number and the location of the cutlines, consider the shape and size of your board and the location of connectors and other fixed components.

You can enter multiple cutlines in both the horizontal and vertical directions.

You can move cutlines using the EDIT/MOVS command, or you can delete the cutlines using the EDIT/DELS command. If you do not specify cutlines, the system will specify them for you.

ENTR/CUT

System-generated cutlines are not displayed on the screen. You can check the location of the cutlines by viewing or printing the placement report. We recommend that you let the system specify cutlines for you.

OPERATION:

1. Select ENTR/CUT. The CUTPLC layer is enabled and set active.

   Select location....

2. You can make the following changes:

   - To enter a cutline in the horizontal direction, select the status line parameter to toggle it to HORZ for horizontal. Then select a point for the location inside the border outline.

   - To enter a cutline in the vertical direction, select the status line parameter to toggle it to VERT for vertical. Then select a point for the location inside the border outline.

3. Continue entering cutlines or terminate the command.

ENTR/LATC

(Enter/Lattice Component)

FUNCTION:

Specifies the major and discrete component parts that can be placed on a lattice.

REMARKS:

You can define more than one lattice for different components. For example, you can specify that all DIP14s be placed on the LAT1 layer and all DIP20s be placed on the LAT2 layer.

Components can be identified by part filename, footprint attribute, or "ALL" for all major components.

OPERATION:

1. Select ENTR/LATC. The system displays the Lattice Component Screen as shown below.

   QUIT <lattice layer> ASSC CLR

   where:

   QUIT returns you to the PC-PLACE Program Screen.

   <lattice layer> is the lattice layer name that is currently active. If no lattice layer is active, then

ENTR/LATC

the first lattice layer the system finds will be displayed.

ASSC displays the Component Association Screen. Refer to the ENTR/ASSC command description for more information.

CLR displays the Component Clearance Screen. Refer to the ENTR/CLR command description for more information.

2. You can make the following changes on the Lattice Component Screen:

   - To add a component, first select the lattice layer name to toggle through the options. Then select the empty space directly above QUIT. The system prompts:

     Comp name:

     Type the component part filename, footprint attribute, or "ALL" and press [Return]. If the entries on the screen exceed 16, the entries are displayed in columns.

   - To delete a component name, select the name on the list. The system prompts:

     OK to delete? YES NO

     Select YES to delete the name.

ENTR/LATC

- To return to the PC-PLACE program screen, select QUIT.
- To display the Component Association Screen, select ASSC.
- To display the Component Clearance Screen, select CLR.

ENTR/LATP

(Enter/Lattice Points)

FUNCTION:

Enters lattice points.

REMARKS:

The ENTR/LATP command allows you to specify the location and spacing between lattice points on a lattice layer.

When you enter lattice points, the system draws a series of intersecting horizontal and vertical lines on the screen to form a lattice. Each intersecting point is a lattice point.

A component is placed on a lattice point at its origin. PC-PLACE is designed assuming your parts are created with the origin at pin 1. If your parts are created with an origin other than pin 1, you will need to allow for this when you specify lattice points.

Before you enter lattice points, add a lattice layer to the layer screen using the VLYR command. The lattice layer name must have the prefix LAT. Then enable and set it active.

When you select ENTR/LATP, the lattice layer is automatically set active.

ENTR/LATP

> NOTE: If you did not add a lattice layer to the layer screen, the system will automatically add a layer named LAT1 and set it ABL and active.

To enter lattice points, specify the distance vertically and horizontally between points in mils and specify a window to define the area of the lattice.

To determine the lattice point specifications, consider the size and the number of components on your board. For example, if the components are 300-mils wide and you want the horizontal spacing between components to be 300 mils, then enter 600 for the x spacing. If the components are 600-mils high and you want the horizontal spacing between components to be 400 mils, then enter 1000 for the y spacing. If you have 12 components to be placed, you need to specify a window large enough to create a lattice with at least 12 lattice points.

Using the specifications described in the example above, PC-PLACE will enter lattice points in increments of 600 mils in the horizontal direction and in increments of 1000 mils in the vertical direction within the window specified.

You can specify more than one lattice for the placement of components.

To define more than one lattice, add a lattice layer name to the layer screen, then enable and set it active. The lattice layer name must have the prefix LAT. Assign the layer a different color to distinguish it from other lattice layers. Then enter lattice points using the ENTR/LATP command.

ENTR/LATP

You delete the lattice using the EDIT/DELS command or the DEL command.

You can also place barriers on the BARPLC layer, if desired, around lattice points to prevent PC-PLACE from placing components on those lattice points.

OPERATION:

1. Select ENTR/LATP. The LAT1 lattice layer is enabled and set active.

   Enter x spacing:

2. Type the horizontal spacing between lattice points in mils and press [Return] or Button 1. The system prompts:

   Enter y spacing:

3. Type the vertical spacing between lattice points in mils and press [Return] or Button 1. The system prompts:

   Lattice window: corner 1...

4. Position the cursor for corner 1 and press Button 1. The system prompts:

   Lattice window: corner 2...

ENTR/LATP

5. Position the cursor for corner 2 and press Button 1. The lattice is displayed and the system continues prompting:

   Lattice window: corner 1...

   NOTE: You can change the lattice window by selecting another location for corner 1 and corner 2.

6. Define another lattice or terminate the command.

ENTR/PARM

(Enter/Parameters)

FUNCTION:

Specifies sorting parameters that affect the order in which components are placed.

REMARKS:

PC-PLACE usually sorts and places from the largest to the smallest component. However, there are sorting parameters you can specify that will affect this order.

You can specify the order in which components are placed by height or by width.

For example, if you have defined a lattice that is higher than it is wide, the automatic placement program will be restricted by the width of the lattice more than the height. Therefore, you should specify "width" as the sorting parameter. The default is "height."

If all of your components are the same size this parameter has no affect on the placement.

You can also specify the order in which components are placed by the constraint index.

As mentioned above, PC-PLACE usually sorts and places components from the largest to the smallest components. However, if you have only a few smaller components of a certain component type and only a few lattice points available on that lattice layer, you may

ENTR/PARM

want to specify a constraint index that will place these components first. Otherwise, these lattice point locations may be blocked by larger components on another lattice layer.

The constraint index must be equal to or less than the constraint ratio of the components you want placed first. The constraint ratio of a component type describes how heavily constrained the placement of those components are on the lattice. PC-PLACE determines the constraint ratio of components by dividing the number of components of that type by the total number of lattice points available. If the constraint ratio is greater than or equal to the constraint index, then those component types will be placed before less constrained components.

For example, if you have two overlapping lattices, one small lattice that has only 10 lattice points defined for 9 DIP14s and therefore the DIP14s have a constraint ratio of 9/10 or 90 percent, and another larger lattice defined for all other major components, you should specify a constraint index less than 90 percent, such as 80 percent, which is the default. Then these components will be placed first before other less constrained components.

OPERATION:

1. Select ENTR/PARM. The system prompts:

   Sort by height or width? Height

ENTR/PARM

2. To change this parameter, select "Height" to toggle to "Width," or select "Width" to toggle to "Height." The system prompts:

   Constraint index (0 to 100) 80

3. To change this parameter, enter a new number between 0 and 100.

FIX

FUNCTION:

Fixes the position of components prior to automatic placement.

REMARKS:

The FIX command can be used with the MOVE and MOVE/COMP command to preplace components which will remain in that location during automatic placement.

This command allows you to toggle between fixing and unfixing a component by selecting the component.

OPERATION:

1. Select FIX. All currently fixed components are highlighted. The system prompts:

   Select a component....

2. You can make the following changes:

   - To fix a component, move the cursor to the component desired and press Button 1. The fixed component is highlighted.

   - To unfix a component, move the cursor to the fixed highlighted component and press Button 1. The unfixed component is no longer highlighted.

HIST

(Histogram)

FUNCTION:

Enables the submenu for specifying histogram configuration parameters and listing histogram statistics.

OPERATION:

Select HIST to enable the CNFG and STAT submenu functions.

HIST/CNFG

(Histogram/Configure)

FUNCTION:

Calls the file configuration parameters and resource line grid options to the screen for your input.

REMARKS:

The configuration parameters control the appearance of the histogram display.

The resource line grid arrangement determines the locations that provide the histogram data; the system counts the nets that cross each resource line in the grid. You can change the grid if you expect to get more helpful information from locations other than the current ones.

PC-PLACE stores configuration parameters and a set of resource lines with each file. Default values are used unless this command is used to change them.

OPERATION:

Procedures for using this command are explained in detail in "Configuring Your Database for the Histogram Display" in Chapter 2.

HIST/STAT

(Histogram/Statistics)

FUNCTION:

Provides a listing of histogram statistics.

REMARKS:

The listing is saved to the histogram report file (<filename>.HIS), which you can view or print at any time. Appendix C provides a sample histogram statistics file.

Be sure to print out the histogram statistics report file before making another listing for the same database. The histogram statistics report file overwrites any existing file with the same name.

OPERATION:

Select HIST/STAT. The program writes the statistics to the histogram statistics report file, then returns to normal program operation. To view or print the histogram statistics report file from within PC-PLACE, use the SYS/DOS command, then use the DOS TYPE or PRINT command. Or from DOS, use the TYPE or PRINT command.

MOVE

FUNCTION:

Moves data or graphics from one location to another.

REMARKS:

The MOVE command can be used as a single command to move one object at a time, or can be used with the subcommands, WIN and IDEN to move several components at a time. The MOVE command can also be used with the subcommand COMP to move a component.

OPERATION:

1. Select MOVE. The system prompts:

   Select object(s).

2. Select an object. The system highlights the object and prompts:

   Move object to. (Select point)...

3. Select the point where you want to move the object.

MOVE/COMP

(Move/Component)

FUNCTION:

Moves a component from one location to another.

REMARKS:

The MOVE/COMP command is used to move and fix components before running the automatic placement program.

This command allows you to identify and move a component by entering its reference designator, rather than zooming in to identify and select a component using the MOVE command.

When you identify the component by entering its reference designator, the component does not have to be within the screen area.

OPERATION:

1. Select MOVE/COMP. The system prompts:

   Comp name:

2. Type the component's reference designator and press [Return]. The system prompts:

   Select loc to place comp.
   (Orientation OK?)

MOVE/COMP

NOTE: Make sure the component is set to the desired orientation as specified by the green F on the status line.

3. Select the point where you want to move the component.

4. Type another component reference designator to move another component, or terminate this command.

MOVE/UNDO

(Move/Undo)

FUNCTION:

Reverses the last change that was made using the MOV command.

REMARKS:

To reverse the change, you must use UNDO before performing any other operation, including using Button 2 to terminate the current command. (You can use viewing commands, such as ZOUT or PAN, and still use UNDO.)

To reverse a MOVE/COMP operation, you must press Button 2 to end the command when the systems prompts for the next component name before you select UNDO.

OPERATION:

Select UNDO to reverse the last change. The system moves the components back to their original positions and updates the histogram and force vectors if they are displayed.

PLCE

(Place)

FUNCTION:

Starts the automatic placement of components.

OPERATION:

1. Select PLCE. The system prompts:

   Ready to place? YES NO.

2. Select YES.

   The program starts executing. The following messages are displayed:

   Beginning Placement

   Extracting database

   Processing cutline <n>...

   Placing major devices

   Placing discrete devices

   Updating database

   Creating report

   Placement complete

PLCE

The system then redraws the screen and the components are now placed inside the board outline.

Any unplaced components are moved below the board outline.

If the histogram or the force vectors are turned on, they are updated to reflect changes to the placement.

QRY

(Query)

FUNCTION:

Enables the submenu for checking the status of a component, net, or pin.

REMARKS:

The QRY command displays information that can be used when setting up a lattice.

OPERATION:

Select QRY to enable the following submenu options:

COMP (Component)
PIN
NET

QRY/COMP

(Query/Component)

FUNCTION:

Displays the reference designator, the part filename, and the footprint of a component.

REMARKS:

The QRY/COMP command displays information about a component that can be used when setting up a lattice.

OPERATION:

1. Select QRY/COMP. The system prompts:

   Select a component....

2. Move the cursor to the desired component and press Button 1. The selected component is highlighted.

   The system displays the reference designator, the part filename, and the footprint attribute. Following is an sample of what would be displayed on the status line:

   NAME=U3 Type=7400.PRT FP=DIP14

3. Select another component to query or terminate the command.

QRY/NET

(Query/Net)

FUNCTION:

Displays the net name and ratsnest line connected to the pin.

REMARKS:

The QRY/NET command allows you to selectively display ratsnest lines.

This command can be used when analyzing the placement of components.

OPERATION:

1. Select QRY/NET. The system prompts:

   Select a pin....

2. Move the cursor to the desired pin and press Button 1.

   The system displays the corresponding ratsnest showing all the points of connection of the selected pin.

   The system also displays the name of the net connected to the selected pin.

QRY/NET

Following is a sample of what would be displayed on the status line:

Net name = UN000007

3. Select another pin or terminate the command.

QRY/PIN

(Query/Pin)

FUNCTION:

Displays the name, type, and logic equivalency of a pin.

REMARKS:

When you select the pin, if no net name is assigned, the system displays the pin number.

OPERATION:

1. Select QRY/PIN. The system prompts:

   Select a pin....

2. Move the cursor to the desired pin and press Button 1.

   The pin is highlighted and the system displays the pin name, pin type, and pin logic equivalency.

   Following is a sample of what would be displayed:

   Name = 1 type = 21 equiv = 1

3. Select another pin or terminate the command.

ROT/UNDO

(Rotate/Undo)

FUNCTION:

Reverses the last change that was made using the ROT command.

REMARKS:

To reverse the change, you must use UNDO before performing any other operation, including using Button 2 to terminate the current command. (You can use viewing commands, such as ZOUT or PAN, and still use UNDO.)

OPERATION:

Select UNDO to reverse the last change. The system moves the components back to their original positions and updates the histogram and force vectors if they are displayed.

SWAP/UNDO

(Swap/Undo)

FUNCTION:

Reverses the last change that was made using the SWAP command.

REMARKS:

To reverse the change, you must use UNDO before performing any other operation, including using Button 2 to terminate the current command. (You can use viewing commands, such as ZOUT or PAN, and still use UNDO.)

OPERATION:

Select UNDO to reverse the last change. The system moves the components back to their original positions and updates the histogram and force vectors if they are displayed.

APPENDIX A. ERROR MESSAGES

This appendix lists and explains error messages that may appear on the screen and that are stored in the error report file (<filename>.ERR). These messages describe warnings and system failures.

If an error message is displayed that is not listed here, refer to Appendix A of the *PC-CARDS User's Manual* for a list of additional messages or call P-CAD customer support.

Following is a list of warning messages that do not interrupt program operation and that do not necessarily require any action. This list includes the cause of each error and the appropriate user action.

Message:	**No lattice points defined for component type <filename>**
Cause:	The component type specified is not assigned to a lattice; therefore, it can't be placed.
Action:	Assign component to a lattice using the ENTR/LATC command.

Message: **Connector (component <reference designator>) must be fixed**

Cause: The connector specified was not fixed.

Action: Fix the connector.

Message: **Component <reference designator> is fixed outside board outline**

Cause: The fixed component specified is not completely within the board outline.

Action: Correct board outline or move or unfix the component if necessary.

Message: **Component <reference designator> is fixed on a placement barrier**

Cause: The fixed component specified is on a placement barrier.

Action: Move or unfix the component if necessary.

Message: **Component <reference designator> overlaps component <reference designator>**

Cause: The fixed components specified overlap each other.

Action: Move or unfix one of the components.

Message: **Component <reference designator> could not be placed**

Cause: Not enough lattice points, or lattice points too close together.

Action: Specify appropriate parameters and select PLCE to run the automatic placement program.

Message: **Component <reference designator> has a scat number <n> out of range**

Cause: The component specified has a system component attribute that is out of the required range. This number is used to determine the component type, such as a major component, discrete component, or a connector. PC-PLACE considers all system component attribute numbers that are out of range to be discrete components.

Action: Change the system component attribute of the component to a number within the following range:

10000 - 10999 for major components

11000 - 11999 for discrete components

12000 - 12999 for connectors

Message: **No association definitions found**

Cause: No discrete association definitions exist in the database.

Action: Define associations using the ENTR/ASSC command.

Message: **No clearances defined**

Cause: No major component clearances exist in the database. The default values of 100 were used.

Action: Define clearances using the ENTR/CLR command if necessary.

Message: **Invalid cutline from <d> <d> to <d> <d>**

Cause: The cutline connecting the two given points is not horizontal or vertical or is outside the board outline. The cutline will be ignored during placement. If all cutlines are invalid, the system will generate its own cutlines.

Action: Delete or edit the cutline.

Message: **Illegal lattice line from <n> <n> to <n> <n>**

Cause: The lattice line connecting the specified vertices is not horizontal or vertical. Any lattice points on this line will be ignored.

Action: Delete or edit the line.

Following is a list of messages that interrupt program operation. This list includes the cause of each error and the appropriate user action.

Message: **No footprint defined for component <reference designator>**

Cause: The component specified has no footprint defined.

Action: Define footprint for this component using ATTR/ACOM command in PC-CARDS, or define the footprint for the part file using the Edit Part Footprints command, and then rerun PC-PACK.

Message: **No components in database**

Cause: Database contains no components.

Action: Check database for components and rerun PC-PLACE.

Message: **Component <filename> has no external name**

Cause: The component specified has no part filename.

Action: Assign a part filename.

Message: No lattices defined

Cause: No placement lattices have been defined on which to place components.

Action: Define lattice using the ENTR/LATP command.

Message: No nets in database

Cause: No nets committed to pins in database.

Action: Enter ratsnests in PC-CARDS using the ENTR/RATN command.

Message: Lattice is too large

Cause: Too many lattice points have been defined.

Action: Edit lattice using the ENTR/LATP command or DELETE command.

Following is a list of messages that interrupt program operation. These error messages are caused by a database that is too large. Reduce the size of your database or get more memory.

Unable to allocate a cell entry
Unable to allocate a net cell entry
Unable to allocate addresses for gain value lists
Unable to allocate barrier list
Unable to allocate cell table
Unable to allocate component list
Unable to allocate component table
Unable to allocate cutline list entry

Unable to allocate cutline structure
Unable to allocate gain list sentinel
Unable to allocate footprint associations table
Unable to allocate footprint clearance table
Unable to allocate footprint table
Unable to allocate lattice
Unable to allocate lattice layer table
Unable to allocate lattice line list entry
Unable to allocate list entry for free list
Unable to allocate linked list entry
Unable to allocate map structure
Unable to allocate net table
Unable to allocate package footprint table
Unable to allocate partitioner net table
Unable to allocate package part table
Unable to allocate part list entry
Unable to allocate partname associations table
Unable to allocate partname clearance table
Unable to allocate pin structure
Unable to allocate section table
Unable to allocate single linked list record
Unable to allocate split component structure
Unable to allocate tree node structure
Unable to allocate header list structure
Unable to allocate list block structure

Following is a list of messages that interrupt program operation and indicate a program error. These error messages are unlikely to appear on your screen or in the error report, but if they do, call P-CAD customer support.

Could not find lattice row in plgtrnum
Database file format incompatible
Database too small
DM function failed in ptlenrpt
Failed to free association list
Failed to free barrier list

Failed to free component list
Failed to free component table
Failed to free cutline list
Failed to free footprint association table
Failed to free footprint clearance table
Failed to free footprint package table
Failed to free gain list structure
Failed to free lattice
Failed to free lattice column list
Failed to free lattice layer table
Failed to free lattice row list
Failed to free linked list in plfreels
Failed to free net table
Failed to free partname association table
Failed to free partname clearance table
Failed to free partname list
Failed to free partname package table
Failed to free pinlist in ptlenrpt
Failed to free the net cell_list
Failed to free the net list
Failed to free the net mapping table
Failed to free the partitioner net table
Failed to free the partition tree
Failed to free the pin list
Failed to free the section table
Failed to free the split component list
Failed to free the cell net_list
Failed to free the cell table
Failed to free header list structure
Failed to free list block structure
Fatal return code from dmlscpic
Fatal return code from dmlstpin in plckcnct
Fatal return code from dmlstpin in plgtpins
Fatal return code from dmlstpin in plgtntyp
Fatal return code from dmlstpin in plntcntn
Fatal return code from dmgacnam
Fatal return code from dmgtcbat
Fatal return code from dminit

Fatal return code from dmlodall
Fatal return code from dmnxtcmp in plupdndx
Fatal return code from dmnxtnet in plupdndx
Fatal return code from dmnxtpar in plgtparm
Fatal return code from dmnxtpin in plckcnct
Fatal return code from dmnxtpin in plgtpins
Fatal return code from dmnxtpin in plgtntyp
Fatal return code from dmnxtpin in plntcntn
Fatal return code from dmowncmp in plntcntn
Fatal return code from dmownnet in plckcnct
Fatal return code from dmownnet in plgtpins
Fatal return code from dmowncmp in plgtpins
Fatal return code from dmpexnam
Fatal return code from ptimpprt
Fatal return code from ptestbal
Fatal return code from ptimprov
Fatal return code from ptinitga
Fatal return code from ptinitft
Fatal return code from ptpartit
Fatal return code from ptupdsct
Fatal return code from rxbrdout
Footprint out of range in plmkfptb
Invalid picture block for placement barrier
Invalid row <n> or column <n> in pllatpt
Invalid row <n> or column <n> in pllatx
Invalid row <n> or column <n> in pilaty
Invalid row <n> or column <n> in plltxmin
Invalid row <n> or column <n> in plltymin
Invalid row <n> or column <n> in plltxmax
Invalid row <n> or column <n> in plltymax
Invalid row <n> or column <n> in plltused
Invalid value for list block offset
Invalid value for net structure pointer
System heap is full
Unable to extract database file name

Following is a list of error messages that interrupt program operation. These errors are for the histogram and force vector features and are stored in the PCPLACE.DBG file.

The format of these messages is:

Error: <error no.> in <routine name>:
<error message>

Error 00 requires you to make sure you have 640K of memory available. The remaining errors indicate a problem in your database.

If you cannot easily correct the error, you may have to use your most recent backup copy of the file.

00 Memory allocation error
01 Error in deleting Histo specs
02 Error in initial Histo process
03 Error in getting Histo specs
04 Error in initial histogram process
05 Error in getting histogram specs
06 Write histo specs. to database error
07 Init. Histo specs. error
08 Error in drawing all component vectors
09 Error in calculating netsizes
10 Error in adding force vector
11 Error in deleting force vector
12 Error in collecting swap data
14 Error in collecting more comp data
15 Error in collecting unused pin data

APPENDIX B. SAMPLE PLACEMENT REPORT

This appendix contains a sample placement report. The report contains five sections:

HEADER - This section lists the database filename and the date and time the automatic placement program began processing.

CUTLINES - This section lists the coordinate locations of the cutlines.

PLACEMENT STATISTICS - This section gives statistics of the placement. It lists the time the automatic placement program completed the placement and the total runtime. It lists an estimate of the combined length of the traces on the board. It also lists the total number of components on the board, the number of integrated circuits, the number of discrete components, the number of connectors, the number of equivalent integrated circuits, and the board density.

> WARNING: This section lists any warning messages that occurred during program operation.

UNPLACED COMPONENTS - This section gives the reference designator and the part filename of any unplaced components.

PLACED COMPONENTS - This section gives the reference designator, the part filename, and the x,y coordinate location of each placed component.

```
Database file name:  TUTORPLC.PLC
Date:  2-26-1986
Beginning placement at 14:52:47

**************************CUT LINES**************************
HORIZONTAL        2200
VERTICAL          2200
VERTICAL          1300
VERTICAL          3100

********************PLACEMENT STATISTICS********************
Completed placement at 14:53:31
Elapsed Time:  00:00:44
Estimated trace length (mils) = 172052
Total number of components = 29
Number of I.C.'s =      13
Number of Discretes =   15
Number of Connectors =   1
Number of equivalent IC's = 14.50
Board density = 0.97 sq inch per equivalent I.C.

**********************PLACED COMPONENTS*********************
       Name        Part      X coordinate        Y coordinate

     U1          74107          4000              2700
     U2          74107          3400              2700
     P1          CONN20         2700               150
     U4          7408           2800              2700
     U3          7408           2800              1700
     U5          7408           2200              1700
     U6          7400           1600              1700
     U7          7400           4000              1700
     U8          7432           1000              1700
     U9          7432           1000              2700
     U10         7411            400              2700
     U11         7411           2200              2700
     U12         7404           3400              1700
     U13         7420           1600              2700
     R1          RC07           2625              2250
     C1          CS13B           800               300
     C2          CK05            400              2800
     C3          CK05           1000              1800
     C4          CK05           1000              2800
     C5          CK05           1600              1800
     C6          CK05           1600              2800
     C7          CK05           2200              1800
     C8          CK05           2200              2800
     C9          CK05           2800              1800
     C10         CK05           2800              2800
     C11         CK05           3400              1800
     C12         CK05           3400              2800
     C13         CK05           4000              1800
     C14         CK05           4000              2800
************************************************************
```

APPENDIX C. SAMPLE HISTOGRAM STATISTICS REPORT

This appendix contains a sample histogram statistics report. The report contains four sections:

Header - This section contains information identifying the report, including the date and time created and the filename.

Vertical Grid Demand (X-AXIS) - This section gives statistics for the vertical histogram axis. It first shows the supply of routing channels. It then shows the high, low, mean, and average of channels used along the cutlines, both in actual channels used and in percentage of available channels. The coordinates column shows the x axis coordinate of the cutlines with the high, low, and mean proportions of channels used.

Horizontal Grid Demand (Y-AXIS) - This section gives the same statistics for the horizontal grid as the previous section gives for the vertical grid.

PLACEMENT MERIT FACTOR - This section shows the current value of the merit factor for the database.

```
-------------------------------------------------------------
      Copyright (C) 1985 Personal CAD Systems Inc.


      Program    :    PC-PLACE
      Date       :    2-21-1986
      Time       :    14:41:09
      File Name  :    TUTORPLC.his

      Predicted Routing Congestion Histogram :
                 Statistics Summary

-------------------------------------------------------------

Vertical Grid Demand (X-AXIS)
             (Supply = 97 Routing Grids)       Coordinates (dbus)

HIGH                  31       ( 31 %)              3150
LOW                    0       (  0 %)             -2050
MEAN                   8       (  8 %)               850
AVERAGE               11       ( 11 %)


Horizontal Grid Demand (Y-AXIS)
             (Supply = 116 Routing Grids)      Coordinates (dbus)

HIGH                  53       ( 45 %)              2100
LOW                    0       (  0 %)              -100
MEAN                  20       ( 17 %)               400
AVERAGE               18       ( 15 %)

PLACEMENT MERIT FACTOR =  -5
```

INDEX

C

D

PC-ROUTE™
User's Manual

September 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

000-0079-02

000-0079-02

ABOUT THIS MANUAL

This manual is designed to serve as a learning aid and as a reference manual.

Chapter 1, INTRODUCTION, provides an overview to PC-ROUTE. Read this chapter first to become familiar with features of the program.

Chapter 2, USING PC-ROUTE, describes how to prepare the database, start the program, edit the routing strategy, run the router, and read the status report.

Appendix A, ERROR MESSAGES, provides a list of possible error messages, the cause of each error, and the appropriate user action.

Appendix B, SAMPLE STATUS REPORT, provides a sample of the PC-ROUTE status report. It lists any errors that ocurred during processing, the subnets that could not be connected by the router, and other statistics of the route.

An index is also provided at the back of this manual.

CONTENTS

CONTENTS (Continued)

FIGURES

Figures (Continued)

TABLE

CHAPTER 1. INTRODUCTION

This chapter introduces you to Personal CAD (P-CAD) Systems' PC-ROUTE, an autorouting program that runs on personal computers. This chapter describes terms as they are used in this manual, general program operation, and routing strategy features.

DEFINITIONS

The following terms are defined as they are used in this manual.

air gap - The distance from the edge of a trace, pad, or via to the edge of another trace, pad, or via.

air line - A line that represents the pin-to-pin connection.

bus - A strip of metal that distributes a common signal, such as ground or power.

checkpoint - A point at which data is saved for restarting the router.

grid - A point formed by the intersection of horizontal and perpendicular lines.

grid line - A vertical or horizontal line along which a metal trace can potentially be placed. Commonly, grid lines are evenly spaced at 50- or 25-mil increments.

lattice - A regular array of points used to describe a set of legal grid locations.

layer pair - Two route layers that are paired together for routing.

net - A group of electrically equivalent features, such as pins, traces, and vias.

pin pair - A pair of pins that are to be connected.

route layer - A layer on which there are metal traces that conduct signals. Also called a signal layer.

routing barrier - A rectangle that specifies what area should be blocked from routing.

routing box - A rectangle within which the router is constrained in its attempt to connect a subnet.

routing pass - A single cycle of routing. In each routing pass, PC-ROUTE attempts to connect every unconnected subnet based on user-specified routing parameters and rules.

run time - The amount of time it takes the router to complete processing.

stub - A common path shared by traces for a short distance.

subnet - A part of a net; for example, a pin pair.

trace - A line that conducts a signal on a PCB.

via - A plated hole that connects metal traces on different layers.

wavefront - A set of points that represent the far ends of all routes being attempted. Also referred to as the routing search path.

OVERVIEW

PC-ROUTE reads a PCB database file containing the physical placement, packaging, and logical information and automatically routes traces on the PCB. You can specify routing parameters and rules for routing the PCB, or you can use the PCAD1 default parameters and rules.

On some designs, PC-ROUTE may not be able to route all the nets. For these designs, the PCB database can be manually edited in PC-CARDS to finish the routing. PC-ROUTE marks the unrouted subnets on the $CONT layer so you can easily locate the unrouted subnets in PC-CARDS.

Input and output of PC-ROUTE are shown in Figure 1-1.

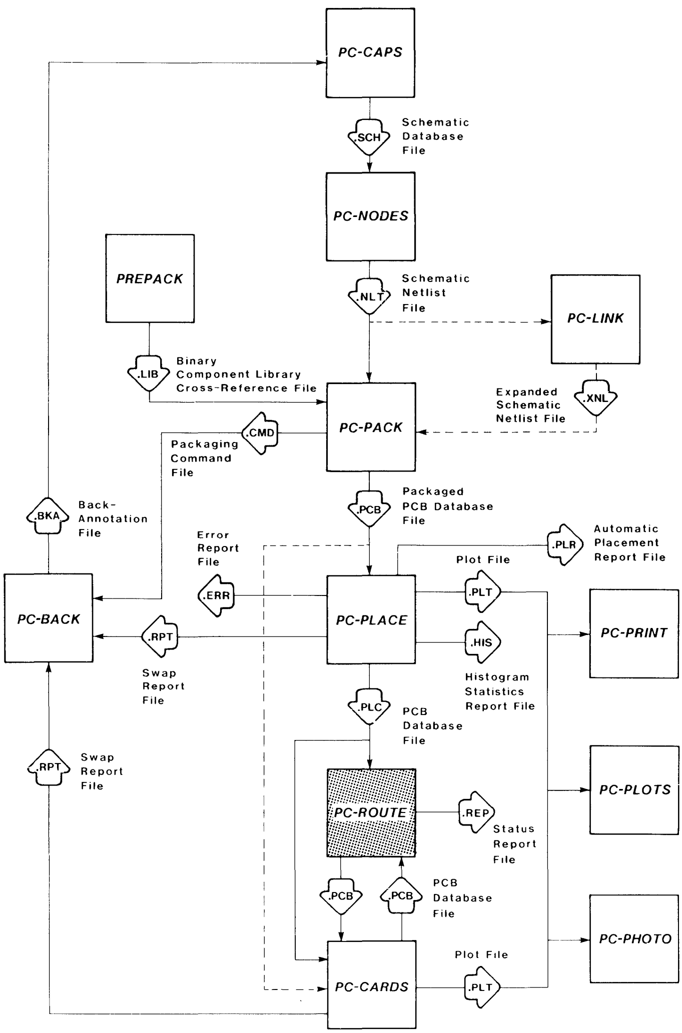


Figure 1-1. PC-ROUTE Input and Output

Input

Input to PC-ROUTE is a PCB database file, created with PC-CARDS, and user-specified routing parameters and rules.

PCB Database File

The PCB database file (<filename>.PLC) contains the graphic representation of the physical layout of a PCB and the logical netlist information.

For best routing, PC-ROUTE requires the following conditions in the PCB database:

- The components should be placed in the optimum position.

  NOTE: Although you can use PC-CARDS to manually place components, we recommend that you use PC-PLACE. PC-PLACE, with its automatic placement and histogram tools, can help you achieve good component placement, which optimizes the routing results.

- Logical netlist connectivity must exist in the database. Netlist connectivity can be established using PC-PACK to generate the PCB database from the schematic netlist or by entering ratsnest data in PC-CARDS.

- All nets must be named.

- The PCB board outline must be drawn on the database layer named BRDOUT. The system does not recognize shapes on any other layer as board outlines.

- Any routing barriers must be drawn as rectangles on layers BARALL, BARCMP, BARSLD, BARVIA, and BARIN1 through BARIN8. Refer to Chapter 2, "Using PC-ROUTE," for more information.

- Pre-wire any desired traces in PC-CARDS using the ENTR/WIRE command.

  NOTE: We recommend prewiring bus structures, such as power and ground buses, in PC-CARDS.

User-Specified Routing Parameters

Before starting the router, you can specify parameters and rules for routing the PCB by selecting options and entering values on the routing strategy menus. The routing strategy is stored in a file with the extension .CTL. You can use the PCAD1 routing strategy (PCAD1.CTL), or you can create a new routing strategy (<filename>.CTL) and select it as input to PC-ROUTE.

Output

PC-ROUTE outputs a routed PCB database file and a status report file.

Routed PCB Database File

The routed PCB database file (<filename>.PCB) consists of the input database with the fully or partially routed nets.

The routed PCB database file can be edited in PC-CARDS to complete any unrouted nets.

Status Report File

PC-ROUTE outputs a status report file (<filename>.REP) that lists the start and end time of each phase of PC-ROUTE, the routing completion percentage achieved, the metal layed per layer, the number of vias used, and any errors detected during processing. It also lists the unrouted subnets. See Appendix B for a sample status report.

Other Files

PC-ROUTE also outputs two other files during processing. After the extraction phase, PC-ROUTE outputs a file (<filename>.RTE) that contains the subnets that need to be connected by the router. After the router phase, PC-ROUTE outputs a file (<filename>.RTS) that contains the connected subnets. These files and the status report file can be deleted from your working directory after the board has been routed and the files are no longer needed. Use the File Maintenance Menu to delete these files.

OPERATION

Running PC-ROUTE consists of:

1. Editing the routing strategy through the routing strategy menus (optional).

2. Selecting options on the Routing Options Menu and running the router.

3. Erasing PC-ROUTE generated files and routing strategy files on the File Maintenance Menu after the routing process has been completed (optional).

These steps are described in more detail in Chapter 2, "Using PC-ROUTE."

Menus

The PC-ROUTE menus store data from the current session and from previous sessions and highlight the next logical step in the routing process or display options based on the current status of the database. For example, when you select the Routing Options Menu, the current status of the database is displayed at the top of the menu and the system displays options based on the current status of the database. This feature saves you keystrokes and lets you know where you are in the routing cycle.

PC-ROUTE provides on-line help screens. If you do not understand a menu or a screen, press the [F1] key to call the corresponding help screen that explains the menu option or the parameter currently selected.

Special Keys

Some keys on the keyboard provide quick access to options or parameters on the menus and provide quick access to other menus.

The [Home] key moves the cursor to the parameter or option on the top of the menu.

The [End] key moves the cursor to the parameter or option on the bottom of the menu. If your keyboard does not have an [End] key, use the [F2] key; it performs the same function as the [End] key.

The [F1] key calls the help screen for the parameter selected.

The [Esc] key exits a menu or screen.

Display Control

Two types of screens can be displayed during routing: status and graphics.

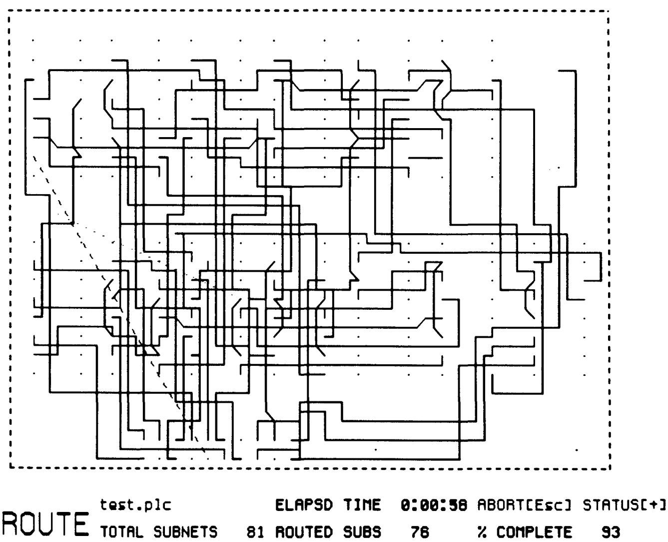
The status display shows statistics about the routing process as shown in Figure 1-2.

```
PC-ROUTE

test.plc

TOTAL LAY PAIRS  1
PASSES/LAY PAIR  2
TOTAL SUBNETS   81

ELAPSD TIME  0:01:09
ROUTED SUBS    77
 % COMPLETE    95

CURRENT LAY PAIR  1
CURRENT PASS 2
CURRENT SUBNET   81

ROUTE COMPLETED[CR]
```

85030

Figure 1-2. Status Display

The graphics display shows an up-to-date picture of the PCB database being routed, the routed traces, and the net "air line" during routing as shown in Figure 1-3.

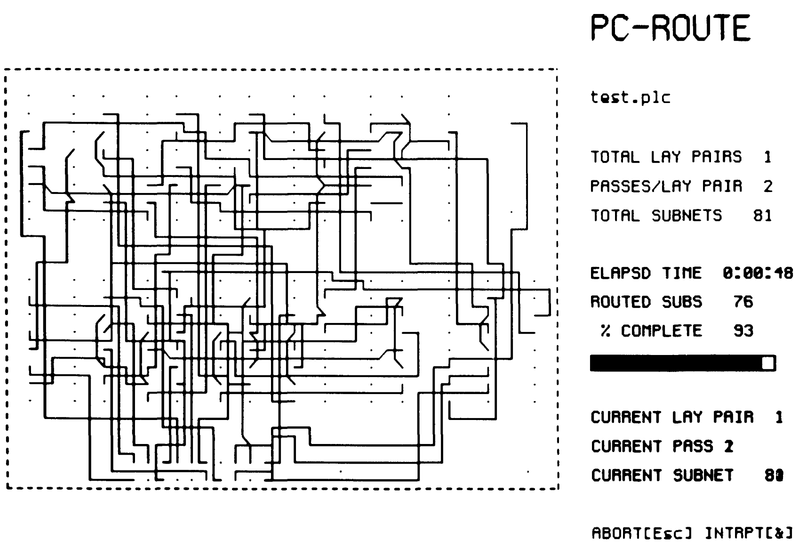


85031

Figure 1-3. Graphics Display

You can choose to see the "wavefront" (routing search path) also. The graphics screen includes an abbreviated status display that appears below the PCB database being routed.

You can choose to display each screen type individually or both screen types in a split screen format as shown in Figure 1-4.



85032

Figure 1-4. Split Screen Format

When an individual screen type is displayed, you can toggle to the other screen type by pressing the [+] key. The advantage of the individual screen display is a higher resolution picture. The advantage of the split screen display is more information.

Table 1-1 summarizes the display options.

Table 1-1. Display Options

Options	Screen Type
Status	Status
Route	Route
Route + Wavefront	Route
Status + Route	Split
Status + Route + Wavefront	Split

Refer to Chapter 2, "Using PC-ROUTE," for more information.

ROUTING STRATEGY FEATURES

PC-ROUTE allows you to specify a custom routing strategy through the routing strategy menus in order to meet individual requirements of circuit rules, manufacturing rules, and time constraints, or you can use the PCAD1 default routing strategy. These features are described below and in the descriptions of the menus to which they apply in Chapter 2, "Using PC-ROUTE."

Grid Structure

PC-ROUTE routes traces on grid lines only. You can specify the grid structure by specifying the distances between grid points.

You can specify a simple grid structure by specifying major grids, the distance over which the grid structure repeats. The simplest grid structure is a 50-mil by 50-mil grid.

You can also specify an irregular grid structure by specifying not only the major grid, but also minor grids, which are the grid locations offset from the major grid. Since the distance between IC pins, 100 mils, does not divide evenly to allow the proper spacing for two trace routing between pins, you can specify minor grid offsets from the major grid. Figure 1-5 shows an example of irregular grid parameters.

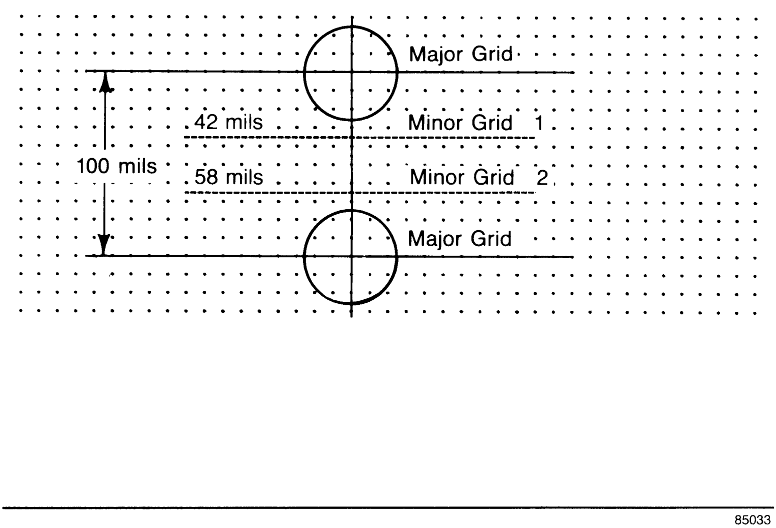


Figure 1-5. Irregular Grid Parameters

Multiple Layers

PC-ROUTE allows you to specify the number of layers the router should use. PC-ROUTE routes traces on layer pairs. The first layer pair is COMP and SOLDER, the second is INT1 and INT2, and the third is INT3 and INT4, and so on.

Routing Speed

PC-ROUTE allows you to specify whether processing time or quality of the route is more critical. The quality of the route refers to how many connections are completed, the number of vias used, and how much wire is used for the traces. For less dense board layouts, a "Fast" routing speed normally provides acceptable quality and completion rates in a much quicker run time. For more dense board layouts, a "Slow" routing speed provides a better quality route and provides a better completion rate, but at the expense of a longer run time.

Interconnection Methods

PC-ROUTE allows you to choose among three interconnection methods:

- Chained pin pairs (Daisy-Chain)
- Pin pairs with "T" connections (Steiner)
- Pin pairs without "T" connections (Min-span)

The method you choose is dependent on circuit rules and/or manufacturing rules. The pin pairs with "T" connections method is the most efficient interconnection method and should be selected if the design rules allow it.

Figure 1-6 shows the three interconnection methods.

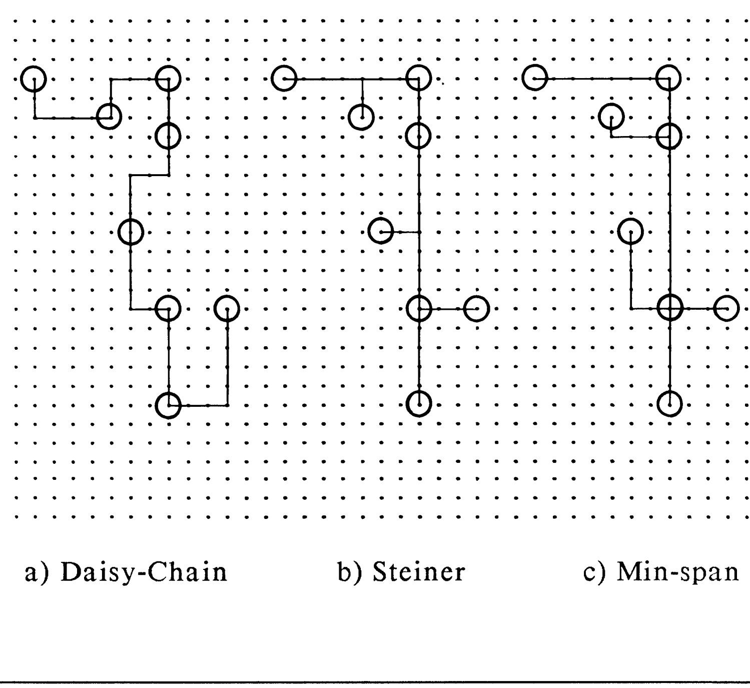


Figure 1-6. Interconnection Methods

With the Daisy-Chain and the Min-span methods of routing, traces connecting to a pin can share a common path for a short distance. This common path is called a stub. PC-ROUTE allows you to specify the length of stubs. If you do not want stubs, use the value 0. Figure 1-7 shows a stub.

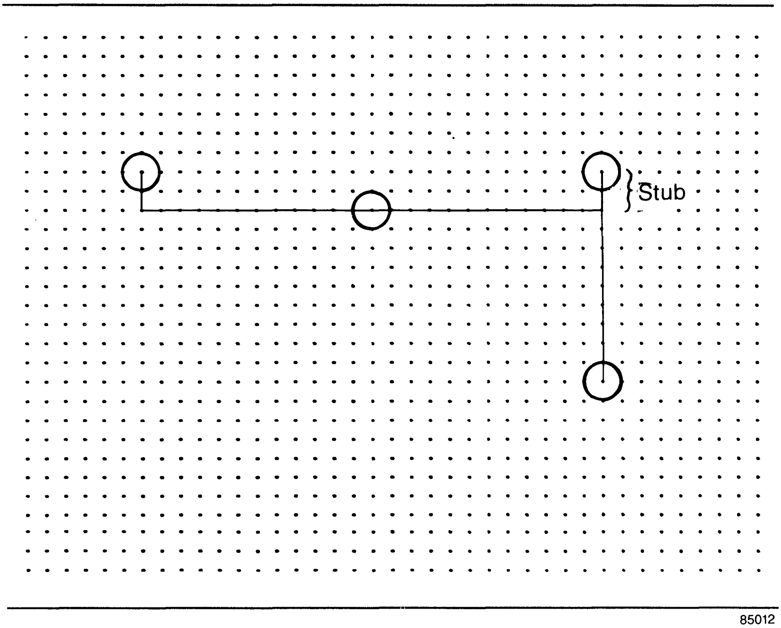


Figure 1-7. Stub

Routing Order

PC-ROUTE allows you to specify which routing order to use:

- Shortest connections first (Short-Long)
- Longest connections first (Long-Short)

If the shortest connections are routed first, the route tends to result in fewer unconnected subnets, but the unrouted subnets may be longer and therefore more difficult to connect manually.

If the longest connections are routed first, the resulting unrouted subnets are usually shorter subnets and therefore easier to connect manually, but the route usually results in more unrouted subnets.

45 Degree Routing

PC-ROUTE allows you to specify that colinear pin connections be routed at 45 degree angles. Colinear pin connections are pin pairs that are directly across from each other horizontally or vertically.

This option can be selected for horizontal and/or vertical pin connections, or neither.

45 degree routing gives PC-ROUTE greater use of the board area than orthogonal routing, increasing the density of traces that can be laid. This feature is especially useful in routing large memory sections of boards where a high density of pin pair connections exists.

Figure 1-8 shows 45 degree routing of pin pair connections.

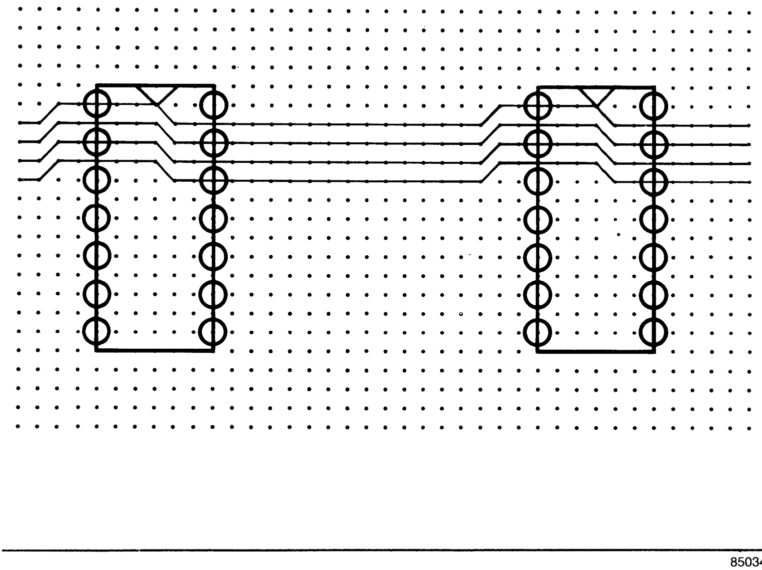


Figure 1-8. 45 Degree Routing

Routing Search Area

PC-ROUTE allows you to specify the number of grids to enlarge the routing box search area. The routing box refers to the rectangle that surrounds the pair of points to be routed, with the two points making up two corners of the box. This box can be enlarged to give the router more area to search during subsequent passes, but the run time will be longer.

The value entered is doubled by the router for the second pass and quadrupled for the third pass. Figure 1-9 shows an example of a routing search area size with a value of 3.

Notice that the box extends three grids from the routing box in the first pass and three more grids in the second pass.

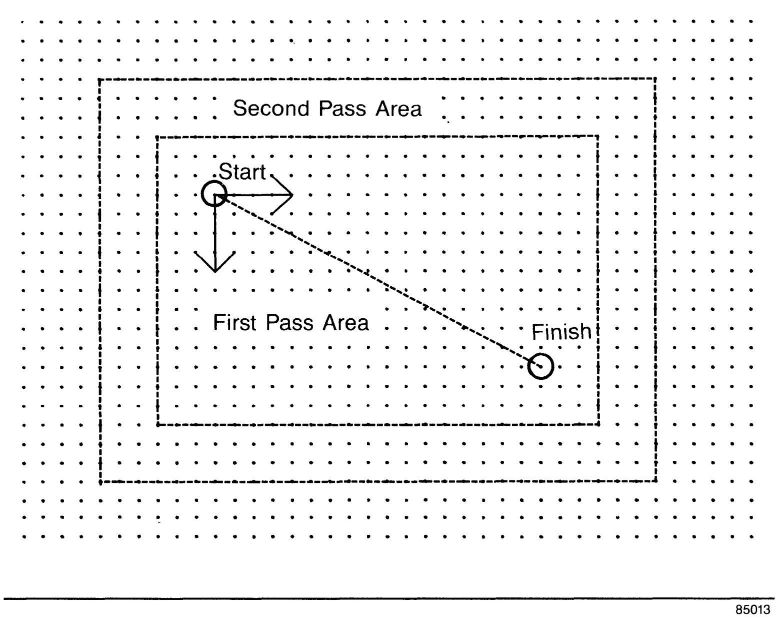


Figure 1-9. Routing Search Area

Normally, the router will extend the route search area to the entire board in the last pass. In order to speed up routing at the expense of several fewer connections, you can limit the search area on the last pass.

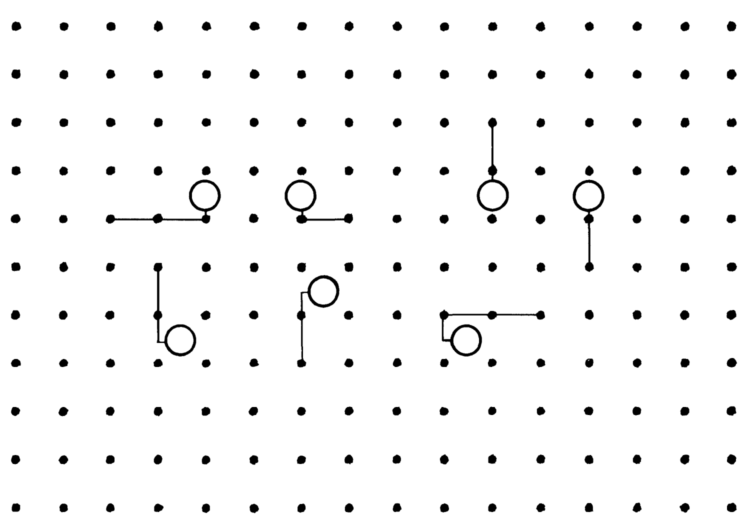
Pad Blocking Rules

PC-ROUTE allows you to specify separate pad blocking rules for each pin type used in the PCB design to prevent shorts. For standard circle or square pad shapes, you define the size and the shape of the pad. For irregular pad shapes (such as connectors), you create a model for each pad type that shows the grid locations, surrounding the pad, that are illegal for placing vias and routing traces.

Also, each pin type can be given different blocking rules for different layers.

Modeling of Pins Off the Grid

PC-ROUTE will route traces to pins that are not on a grid point. For each axis on which the pin is off the grid line, the trace is routed to the closest lower left grid point and then the trace snaps to the pin orthogonally. Figure 1-10 shows examples of how the router would route traces to pins that are off the grid.



85014

Figure 1-10. Routing of Pins Off the Grid

Wiring Rules

PC-ROUTE allows you to specify separate wiring rules that define the width of the trace and the spacing or the clearance required between traces and pads.

Net Class Definition

PC-ROUTE allows you to assign wiring rules to specific nets (such as ground and power) through net class definition. You can also set priorities for the routing of different groups of nets and specify which layers they can be routed on. Refer to Chapter 2, "Using PC-ROUTE," for more information.

Via Lattices

PC-ROUTE places vias randomly; that is, any unused grid point is a potential via site. When grid size or other parameters require it, you can define a via lattice to describe all the legal via sites.

Routing Cost Factors

PC-ROUTE allows you to specify cost factors to right way routes, wrong way routes, and vias. The router predominantly routes horizontally on one layer and vertically on the adjacent layer in a layer pair. Right way routes are traces routed horizontally on a horizontal layer. Wrong way routes are traces routed vertically on a horizontal layer.

By assigning a higher cost value to vias than wrong way routes, you can minimize the number of vias placed by the router. For example, if you assign a value of 1 to right way routes, 1 to wrong way routes, and 2 to vias, the router will try to avoid placing vias, and will route the trace in the other direction on the same layer as shown on the top of Figure 1-11. Note, however, that usually the higher the cost value assigned to vias, the lower the route completion rate.

By assigning a higher cost value to wrong way routes than vias, you can minimize the number of traces routed in the other direction. For example, if you assign a value of 1 to right way routes, 4 to wrong way routes, and 2 to vias, the router will put in a via and will route the trace on the adjacent layer as shown on the bottom of Figure 1-11.

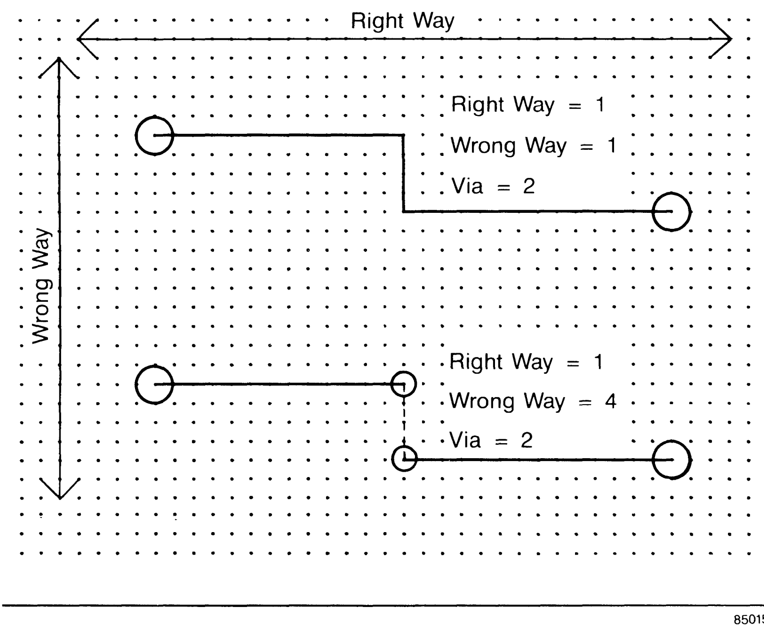


Figure 1-11. Cost Factors

Unconnected Subnets

After routing, PC-ROUTE marks unconnected subnets on the $CONT layer so you can easily locate the unrouted subnets in PC-CARDS.

The ratsnest displayed in PC-CARDS for a routed database is also a display of unrouted nets. However, the $CONT layer created by PC-ROUTE indicates the optimum pin pair connections of a net to be routed rather than the random pin pairing of a PC-CARDS ratsnests.

CHAPTER 2. USING PC-ROUTE

This chapter describes how to use PC-ROUTE to automatically route a PCB. It describes how to prepare the database, start the program, specify rules and parameters for routing on the routing strategy menus, and run the router. It also describes how to read the status report.

Following each menu is a description of the possible options you can select and parameters you can enter and the procedure for selecting an option or entering a parameter.

PREPARING YOUR DATABASE

Before you use PC-ROUTE, you must prepare the database by adding a board outline and, if desired, routing barriers. The database can be prepared in PC-CARDS as well as PC-PLACE. This section provides instructions and special requirements for preparing the database. This information is also included in the *PC-PLACE User's Manual.*

Adding the Board Outline

Before you use PC-ROUTE, you must add a board outline and rearrange the components in the database into the board outline.

First, load your database using the FILE/LOAD command and enter the filename.

When the database is loaded, use the DRAW/LINE or DRAW/RECT commands to create the board outline. The board outline must be drawn in DETL (Detail) mode and must be on the BRDOUT layer so that PC-ROUTE, the automatic routing program, will recognize it.

If some components are not inside the board outline, use the MOVE command to place the components within the outline.

Adding Routing Barriers

If you plan to use PC-ROUTE to route a PCB, and there are areas on the PCB that you want to block from routing, you must first use PC-PLACE or PC-CARDS to add routing barriers.

PC-ROUTE recognizes eleven layers for routing barriers and one layer for via barriers. Add to your database as many of these layers as you require. Use the VLYR command to display the layer screen. To add a new layer, move the cursor to the space below the last layer and press Button 1. Enter the name and color for the new layer and set it to ABL.

Use the DRAW/REC command to enter rectangles to represent the barriers. Be sure to enter each barrier on the appropriate layer. The barrier layers and their functions are listed below.

BARALL - Prevents PC-ROUTE from routing traces within the barrier rectangle on all routing layers.

BARCMP - Prevents PC-ROUTE from routing traces within the barrier rectangle on the component layer only.

BARSLD - Prevents PC-ROUTE from routing traces within the barrier rectangle on the solder layer only.

BARIN1 through BARIN8 - Prevents PC-ROUTE from routing traces within the barrier rectangle on the internal layers 1 through 8, respectively.

BARVIA - Prevents PC-ROUTE from placing vias within the barrier rectangle, but allows routing of traces through it.

STARTING THE PROGRAM

Before starting PC-ROUTE, be sure that:

- The following files are in the EXE directory on your hard disk.

  PCROUTE.EXE
  PCROUTE1.EXE
  PCROUTE2.EXE
  PCROUTE3.EXE
  PCROUTE4.EXE
  PCROUTE.HLP
  PCAD1.CTL

- The PCAD1.CTL file is also in the working directory.

  NOTE: Be sure to copy the new 1.35 version PCAD1.CTL file into your working directory and the EXE directory. Version 1.35 of PC-ROUTE utilizes a new control file structure. Control files (<filename>.CTL) created under previous versions of PC-ROUTE are not compatible with version 1.35. However, you can convert an old control file to the 1.35 level by editing the old routing strategy and then saving it. Refer to Appendix C, "Converting a Control File," for details.

Also, be sure to remove your old ROUTCTFL.CTF files from the EXE and working directories before using the 1.35 version of PC-ROUTE.

- The PCB database file (<filename>.PLC) is in the working directory on the hard disk.
- The driver files (<filename>.DRV) are in the \PCAD\DRV directory.
- The PCADDRV.SYS file is in the root directory.

First, change the current directory to the working directory.

Then, to start PC-ROUTE, type:

PCROUTE [Return]

The system displays the PC-ROUTE title screen. Press any key to display the PC-ROUTE Main Menu, shown in Figure 2-1.

PC-ROUTE

Enter database name: __ (Database reference name)

Select routing strategy:

Options:

Edit routing strategy

Route

Exit PC-ROUTE

Select Router File Maintenance

[SPACE] options; [RETURN] accept; [F1] help; [HOME] database name; [END] maint.

Figure 2-1. PC-ROUTE Main Menu

This menu provides the following options.

Edit routing strategy - Calls the Routing Strategy Menu to create a new or change an existing routing strategy.

Route - Calls the Routing Options Menu to start the routing process.

Exit PC-ROUTE - Returns you to DOS.

Select Router File Maintenance - Calls the File Maintenance Menu to delete routing strategy files and/or router created files for a specified database.

First, type the database name of the PCB database file to be routed and press [Return]. If you do not enter a filename extension, a .PLC extension is added when you press [Return]. The database name defaults to the file used during the last run of PC-ROUTE, if any.

NOTE: The "Database reference name" is the unique name that PC-ROUTE assigns to the file (<filename>.RTE) output from the extraction phase of PC-ROUTE, the file (<filename>.RTS) output from the routing phase of PC-ROUTE, and the status report file (<filename>.REP) output from the routing phase of PC-ROUTE. This name is displayed after you enter the database name and select an option on the PC-ROUTE Main Menu. For example, if you entered TEST.PLC for the database name, the database reference name TEST would be displayed.

Next, select a routing strategy by using the space bar to cycle through the routing strategies, and then press [Return]. The routing strategy is a set of parameters and rules for routing a PCB. P-CAD provides a default routing strategy named PCAD1. The routing strategy defaults to the strategy used during the last run of PC-ROUTE for the selected database, if any. If a new database is specified, the routing strategy defaults to PCAD1.

NOTE: If you have not created a new routing strategy, only the PCAD1 default routing strategy is displayed.

When you have selected a database name and routing strategy, the next option that you should run is highlighted. To select this option, press [Return], or use the space bar to cycle to another option, and then press [Return].

If you already have a routing strategy set up with the parameters and rules you want, or you have chosen the PCAD1 routing strategy, select **Route** and press [Return] to access the Routing Options Menu.

If you want to add to, change, or review parameters and rules for a particular routing strategy set, press the space bar to cycle to the **Edit routing strategy** option and press [Return]. The edit routing strategy options are described in the next section, "Editing the Routing Strategy."

To delete files created by the router or to delete a routing strategy, press [End] and then [Return] to access the PC-ROUTE File Maintenance Menu. This menu is described in the section titled "Maintaining Your Files."

> NOTE: If your keyboard does not have an [End] key, use [F2]; it performs the same function.

EDITING THE ROUTING STRATEGY

To edit the routing strategy, select the **Edit routing strategy** option on the PC-ROUTE Main Menu. The Routing Strategy Menu is displayed, as shown in Figure 2-2, with the current routing strategy name at the top of the menu. This name corresponds to the routing strategy you selected on the PC-ROUTE Main Menu.

PC-ROUTE - Editing strategy TEST

Options:

<< Edit routing parameters <<

Edit pad blocking rules

Edit wiring rules

Edit net class definitions

[SPACE] options; [RETURN] accept; [F1] help; [ESC] quit

Figure 2-2. Routing Strategy Menu

This menu provides the following options.

Edit routing parameters - Calls the Routing Parameters Menu to change parameters that control the routing.

Edit pad blocking rules - Calls the Pad Blocking Rules Menu to change pad blocking rules and/or define new blocking rules.

Edit wiring rules - Calls the Wiring Rules Menu to change wiring rules and/or define new wiring rules.

Edit net class definitions - Calls the Net Class Definitions Menu to assign wiring rules, priority, and layer pair to different groups of nets.

To edit the existing routing strategy (the filename displayed at the top of the menu) or to create a new strategy, use the space bar to cycle through the options, and then press [Return] to select the option. The corresponding menu is displayed. Enter new parameters and rules where desired.

After you finish entering new parameters or rules on one of the routing strategy menus, press [Esc] to return to the Routing Strategy Menu.

Continue selecting options and entering new parameters or rules until you have selected all the desired options on the Routing Strategy Menu and have specified all the parameters and rules you want for the routing strategy set on the routing strategy menus.

When you finish editing the current routing strategy, press [Esc] to return to the Routing Strategy Menu. Press [Esc] again and the following options for saving the strategy you just edited are displayed on the bottom of the Routing Strategy Menu as shown in Figure 2-3.

```
PC-ROUTE - Editing strategy TEST

Options:

    Edit routing parameters

    Edit pad clearance rules

    Edit wiring rules

    Edit net class definitions

Select option for saving the strategy just edited:

>>  Save under current name  <<

    Save under new name

    Do not use or save

[SPACE] options; [RETURN] accept; [F1] help.
```

Figure 2-3. Options for Saving the Routing Strategy

The options for saving the routing strategy just edited are described below.

Save under current name - Saves the new values you entered on the routing strategy menus under the routing strategy that is currently selected.

Save under new name - Saves the new values you entered on the routing strategy menus under the name you specify.

Do not use or save - Ignores any new values you entered on the routing strategy menus.

Use the space bar to cycle through the options, and then press [Return] to select an option.

If you select the **Save under current name** option, and the current routing strategy name is PCAD1, the system displays the following message:

Cannot write over PCAD1
Select another option

This feature protects the PCAD1 routing strategy so that you always have the PCAD recommended routing strategy available to use as a guideline for creating your own customized routing strategy.

If you select the **Save under new name** option, the system prompts you for the new name as follows:

Enter new name:

Type the new routing strategy name and press [Return]. The PC-ROUTE Main Menu is displayed and the new strategy name is added to the routing strategy options. If you enter the name of a routing strategy file that already exists, the system prompts:

This strategy already exists
Enter new name

The following sections describe the routing strategy menus that are displayed when you select the editing routing strategy options.

Editing the Routing Parameters

To edit routing parameters, select the **Edit routing parameters** option on the Routing Strategy Menu. The Routing Parameters Menu is displayed with the values of the current routing strategy. Figure 2-4 is an example of the Routing Parameters Menu with the PCAD1 routing strategy values displayed.

PC-ROUTE

Edit routing parameters for strategy PCAD1.

Routing grid x: 50 y: 50

Number of routing layers: 2

Route speed: Fast

Route type: Steiner

Route order: Short-Long

Display type: Status + Route + Wavefront

Perform 45 degree route: Both

Perform via minimization: Yes

Define detailed parameters

Enter parameter; [RETURN] to accept; [F1] help; [END] detail; [ESC] quit.

Figure 2-4. Routing Parameters Menu

This menu allows you to specify the following parameters to be used during routing:

Routing grid - Specifies the spacing between the x and y routing grids in database units (DBUs). We recommend that you use a routing grid of 50 DBUs in both axes whenever possible. The default is "50."

For the routing grid parameters, press [Return] to accept the values displayed, or type in new values and press [Return].

Often it is better to specify a larger grid even if the design rules allow for a smaller grid. When you specify a larger grid, PC-ROUTE runs faster and requires less memory.

A common technique you might want to use is to route on a 50-mil grid and manually complete unconnected subnets on a 25-mil grid.

> NOTE: The x and y values do not have to be the same. For example, you can specify a 50-mil grid for the x axis and a 25-mil grid for the y axis for a special routing situation.

Number of routing layers - Specifies the number of layers the router should use to route traces. PC-ROUTE routes traces in layer pairs; therefore, this number must be an even number (for example, 2, 4, 6, and so on). The maximum number of layers PC-ROUTE can handle is 10. The default is "2."

> NOTE: The router routes traces on the COMP and SOLDER layer pair first, the INT1 and INT2 layers second, the INT3 and INT4 layers third, and so on.

For the number of routing layers parameter, press [Return] to accept the value displayed, or type in a new value and press [Return].

Route speed - Specifies the speed routine to be used by the router. The options are "Slow" or "Fast." The default is "Fast."

For the route speed parameter, use the space bar to toggle between the options, and press [Return] to select an option.

If you select "Slow," the router uses algorithms that search more carefully and thoroughly for the best path, but take longer than the fast routing speed. This method usually produces a better quality route and a higher completion rate.

If you select "Fast," the router uses algorithms that find the quickest solution; but this routing speed could produce a lower quality route and route fewer traces than the slow routing speed.

Route type - Specifies the interconnection method to be used by the router. The options are "Daisy-Chain," "Steiner," or "Min-span." The default is "Steiner."

For the route type parameter, use the space bar to cycle through the options, and then press [Return] to select an option.

If you select "Daisy-Chain," PC-ROUTE will route one or two connections per connection point.

If you select "Steiner," PC-ROUTE will route T junctions and any other orthogonal connection that minimizes the run length between points. The Steiner method is a more general-purpose approach.

If you select "Min-span," PC-ROUTE will route up to four connections per connection point but does not permit T junction connections. Min-span is often required with Mil-Spec designs.

Route order - Specifies the order in which the subnets will be routed. This parameter applies only to nets with the same routing priority that you specify on the Edit

Net Class Definitions Menu. The options are "Short-Long" or "Long-Short." The default is "Short-Long."

For the route order parameter, use the space bar to toggle between the options, and then press [Return] to select an option.

If you select "Short-Long," PC-ROUTE will route the shortest subnets first, and then route the longest subnets. The advantage of this order is that the route tends to result in fewer unconnected subnets. The disadvantage of this order is that the unconnected subnets may be longer and therefore more difficult to connect manually.

If you select "Long-Short," PC-ROUTE routes the longest subnets first and routes the shortest subnets last. The advantage of this order is that the resulting unconnected subnets are usually shorter subnets and therefore easier to connect manually. The disadvantage of this order is that the route usually results in more unconnected subnets.

Display type - Specifies the type of display you want on the screen during the routing phase. The options are "Status," "Route," "Status + Route," "Route + Wavefront," or "Status + Route + Wavefront." The default is "Status + Route + Wavefront."

For the display type parameter, use the space bar to cycle through the options, and then press [Return] to accept an option.

If you select "Status," PC-ROUTE will display only the statistics of the route during routing.

If you select "Route," PC-ROUTE will display the routed traces and the "air line" (ratsnest connection) for the net being currently routed.

> NOTE: If you select "Status" or "Route" during routing, you can toggle between these display types by pressing the [+] key. When the status display is turned off, the screen will still display a status line indicating the time elapsed and route completion percentage.

If you select "Status + Route," PC-ROUTE will display the statistics of the route, the routed traces, and the air line for the net being currently routed.

If you select "Route + Wavefront," PC-ROUTE will display the routed traces, the air line for the net being currently routed, and the routing search path for the current net.

If you select "Status + Route + Wavefront," PC-ROUTE will display the statistics of the route, the routed traces, the air line, and the routing search path for the current net.

> NOTE: If you have a large board, you can select a display type that does not include the status display ("Route" or "Route + Wavefront"), and then a larger picture of the board will be visible during routing.

Perform 45 degree route - Specifies that colinear pin connections can be routed at 45 degree angles. Colinear pin connections are pin pairs that are directly across from each other horizontally at a 0 or 180 degree angle, or vertically at a 90 or 270 degree angle. These colinear pin connections must meet the air gap and blocking requirements specified in the Wiring Rules Menu and Pad Blocking Rules Menu, such that a grid next to each of the pins in the colinear connection is open for a trace. The 45 degree router will only route on the grid immediately above or below the grid line it begins on; in order to get around a pin, the grid immediately above or below the pin must be open. Note that this is not the case during standard 25-mil routes, since standard size pads (for example, 40, 50, 60 mils wide) normally block the adjacent grid points.

The options are "Horizontal," "Vertical," "Both," or "None." The default is "Both."

For the perform 45 degree route parameter, use the space bar to cycle through the options, and then press [Return] to select an option.

If you select "Horizontal," PC-ROUTE will first attempt to make horizontal pin pair connections using the 45 degree routing pass.

If you select "Vertical," PC-ROUTE will first attempt to make vertical pin pair connections using the 45 degree routing pass.

If you select "Both," PC-ROUTE will attempt to make both horizontal and vertical pin pair connections using the 45 degree routing pass.

If you select "None," PC-ROUTE will attempt to make all pin pair connections using only the orthogonal routing pass.

If you select one of the 45 degree routing options, PC-ROUTE will attempt to make all horizontal and/or vertical pin pair connections first using the 45 degree routing pass. PC-ROUTE will then attempt the remaining unrouted connections using the orthogonal routing passes. Note that the router will only perform the 45 degree routing algorithm once per layer pair, at the beginning of the routing sequence for that layer pair. Also, PC-ROUTE will only route nets assigned to the DEFAULT rule on the Wiring Rules Menu with the 45 degree routing option. If you assign a group of nets with colinear connections to a wiring rule other than DEFAULT when defining net classes, this net class will be routed with orthogonal traces.

Perform via minimization - Specifies a post routing process that minimizes vias. The options are "Yes" or "No." The default is "Yes."

For the perform via minimization parameter, use the space bar to toggle between the options, and then press [Return] to select and option.

Define detailed parameters - Calls the Detailed Routing Parameters Menu.

After you finish, press [Esc] to return to the Routing Strategy Menu or press the [End] key to access the **Define detailed parameters** option, and then press [Return] to call the Detailed Routing Parameters Menu.

Editing the Detailed Routing Parameters

To edit the detailed routing parameters, select the **Define detailed parameters** option on the Routing Parameters Menu. The following menu is displayed.

```
                        PC-ROUTE

    Edit detailed routing parameters for strategy TEST.

ROUTING GRID                Major grid x:   50        y:   50
 DEFINITION               Minor grid 1 x:   0         y:   0
                          Minor grid 2 x:   0         y:   0
                          Minor grid 3 x:   0         y:   0

VIA SITES                   All grid points
VIA LATTICE                  First via x:   0         y:   0
 DEFINITION                Via spacing x:   0         y:   0
                         Number of vias x:   0         y:   0

ROUTE SEARCH AREA SIZE      Number of grids to enlarge:    5
PASSES PER LAYER PAIR                 Number of passes:    2
OPEN ROUTE SEARCH AREA TO FULL BOARD ON LAST PASS     :    Yes

ROUTE AREA CLEARANCE FROM BOARD EDGE          distance:    100

                Continued detailed parameter menu

Enter parameter; [RETURN] to accept; [F1] help;
[END] detail; [ESC] quit.
```

Figure 2-5. Detailed Routing Parameters Menu

```
                         PC-ROUTE

   Edit detailed routing parameters for strategy TEST.

SELECT HORIZONTAL/VERTICAL LAYERS             :  COMP = horz

COSTING                         Right way cost:  1
                                Wrong way cost:  4
                                      Via cost:  2

STUB LENGTH       Horizontal layer length in x: 100    y:  100
 DEFINITION         Vertical layer length in x: 100    y:  100




  [SPACE] options; [RETURN] accept; [F1] help; [ESC] quit.
```

Figure 2-5 Continued

The Detailed Routing Parameters Menu is divided into ten sections:

Routing Grid Definition
Via Sites
Via Lattice Definition
Route Search Area Size
Passes Per Layer Pair
Open Route Search Area to Full Board on Last Pass
Route Area Clearance from Board Edge
Select Horizontal/Vertical Layers
Costing
Stub Length Definition

Routing Grid Definition

These parameters allow you to specify an irregular grid size in which the grid you want to use is not evenly spaced.

Major grid - Specifies the x and y values of the major routing grid. The values entered on the Routing Parameters Menu are displayed. You can overwrite this value by typing in a new value. The default is "50" for both x and y.

Minor grid - Specifies the x and y distance each minor grid is from the major grid. Three minor grid specifications are allowed. The default is "0" for all three minor grid specifications.

For example, if you want two grid lines between each 100-mil increment in the y direction and a standard 50-mil grid in the x direction, enter 50,100 for the x,y value of the major grids. Then, for the minor grids, enter 0,42 for the x,y value for the first minor grid offset and enter 0,58 for the x,y value for the second minor grid offset.

For the routing grid definition parameters, press [Return] to accept the values displayed, or type in new values, and then press [Return].

Via Sites

This parameter allows you to specify whether via sites are allowed on all grid points or on a defined via lattice only. This parameter toggles between "All grid points" and "Via lattice points only."

For the via sites parameter, use the space bar to toggle between "All grid points" and "Via lattice points only," and then press [Return] to select an option.

If you select "All grid points," the cursor bypasses the parameters under the Via Lattice Definition section and moves directly to the Route Search Area Size parameter. If you select "Via lattice points only," the cursor moves to the first parameter in the Lattice Definition section to define the via lattice. Type in new values and then press [Return].

Via Lattice Definition

These parameters allow you to define the via lattice by specifying what grid locations are legal for placing vias when using a nonstandard or reduced grid size.

First via x and y - Specifies the x and y coordinates from the PCB, which is the beginning grid location that is legal for placing vias. These coordinates must be within the routable area of the board.

Via spacing x and y - Specifies the spacing between legal via locations in the x and y directions in mils.

Number of vias - Specifies the number of legal via locations in the x and y directions. This value determines how far across the board the via grid extends. This value must not be too large so as to enter into the board edge clearance area or exceed the board outline limits.

For the via lattice definition parameters, press [Return] to accept the values displayed, or type in new values, and then press [Return].

To determine the number of legal via locations, divide the distance of the legal via location span across the board by the distance between the legal via locations. For example, on a 10-inch board, you might want the span of the legal via locations across the board to be 9800 mils (9.8 inches). If you specified 50 mils (0.50 inches) for the x value for the distance between legal via locations, then the number of repeats is 196 (9.800 divided by 0.050 = 196 vias). Add 1 to the total number of repeats. Therefore, in this example, you would enter 197 for the number of vias in the x direction.

Another way to determine the number of legal via locations in both the x and y directions is to count them. For example, on the board shown in Figure 2-6, there are 19 legal via locations across the board on the x axis (shown by the intersecting points of the dashed lines) and 13 vias on the y axis.

You should determine the dimensions of the via lattice before entering PC-ROUTE. If the via lattice exceeds the legal limits of the PCB, as defined by the board outline, or does not fit within the defined grid size, the system will output an error message in the status report file (<filename>.REP) and processing will stop.

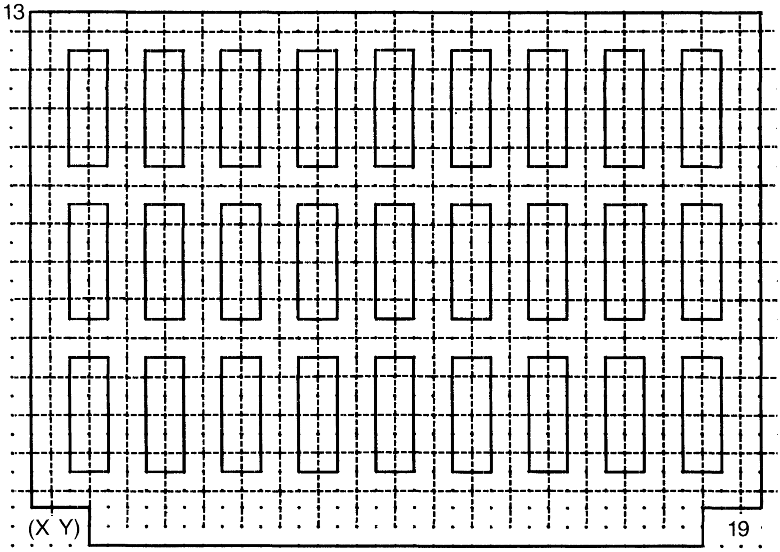


Figure 2-6. Via Lattice

Route Search Area Size

This parameter allows you to specify the number of grids to enlarge the routing box search area. The routing box refers to the rectangle that surrounds the pair of points to be routed, with the two points making up two corners of the box. This box can be enlarged to give the router more area to search during subsequent passes. However, the enlarged routing box should be kept small enough so the router doesn't waste time searching an area that is unlikely to produce a connection, but large enough so that the probability of a connection is high.

For the route search area size parameter, press [Return] to accept the value displayed, or type in new values, and then press [Return].

The value entered here is doubled by the router for the second pass and quadrupled for the third pass. Figure 2-7 shows how the route search area size is enlarged if you specify search area = 3 for each pass at a connection.

Note that the search area is also affected by the "Open Route Search Area to Full Board on Last Pass" option. This parameter is explained below.

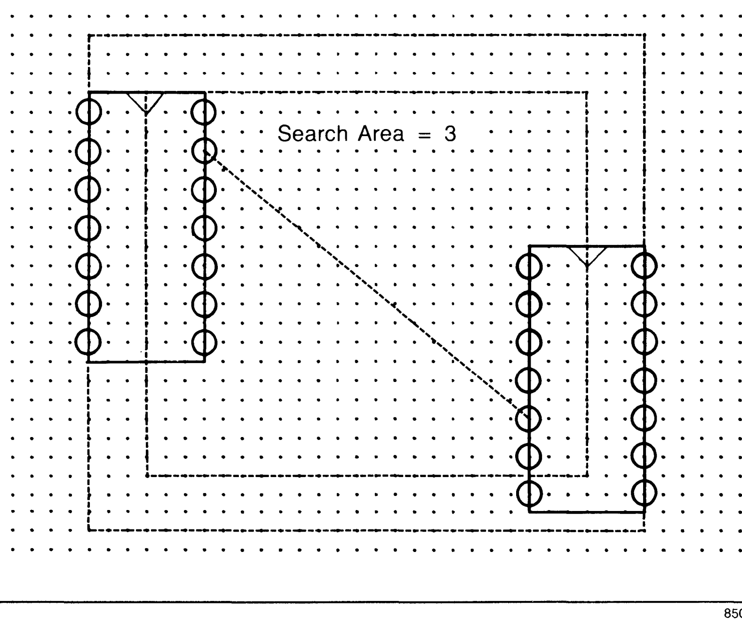


Figure 2-7. Route Search Area Size

Passes Per Layer Pair

This parameter allows you to specify the number of times PC-ROUTE cycles through the remaining unrouted subnets, attempting to finish the routing for each layer pair. PC-ROUTE allows you to specify a maximum of 3 passes per layer pair. The default is "2."

For the passes per layer pair parameter, press [Return] to accept the value displayed, or type in a new value and then press [Return].

Open Route Search Area to Full Board on Last Pass

This parameter expands the board area available for routing to its full extent on the last pass the router takes in completing layer pair connections. The options are "YES" or "NO." The default is "YES."

For the open route search area to full board on last pass parameter, use the space bar to toggle between the options, and then press [Return] to select an option.

If you select "YES," the number of connections will be maximized. If you select "NO," the run time will be decreased.

Route Area Clearance from Board Edge

This parameter specifies the board outline clearance in which PC-ROUTE cannot route. The default is 100 mils, which restricts the router from routing any closer to the board outline than 100 mils.

For the route area clearance from board edge parameter, press [Return] to accept the value displayed, or type in a new value, and then press [Return].

Select Horizontal/Vertical Layers

This parameter specifies the predominant direction in which PC-ROUTE routes each layer. The options are "COMP = horz" or "COMP = vert." The default is "COMP = horz."

For the horizontal/vertical layers parameter, use the space bar to toggle between the options, and press [Return] to select an option.

If you select "COMP = horz," PC-ROUTE will route odd numbered layers with horizontal traces and all even numbered layers with vertical traces.

If you select "COMP = vert," PC-ROUTE will route all odd numbered layers with vertical traces and all even numbered layers with horizontal traces.

Odd numbered layers are COMP, INT1, INT3, and so on. Even numbered layers are SOLD, INT2, INT4, and so on.

Costing

These parameters allow you to assign cost values to right-way routes, wrong-way routes, and vias. The range of values that can be specified for these parameters is 0 to 10, with 10 being the highest cost.

Right way cost - Specifies the cost of routing in the correct direction; for example, a horizontal route on a horizontal layer. The default is "1." The higher the cost specified, the less likely PC-ROUTE will route in the "right way."

Wrong way cost - Specifies the cost of routing in the wrong direction; for example, a vertical route on a horizontal layer. The default is "4." The higher the cost specified, the less likely PC-ROUTE will route in the "wrong way."

The default cost ratio of 1 to 4 indicates it is 4 times as costly to route in the wrong way as in the right way.

Via cost - Specifies the cost of putting in a via. The default is "2." The higher the cost, the less likely PC-ROUTE will place vias.

The router predominately routes horizontally on one layer and vertically on the adjacent layer on a layer pair.

The router determines which path to choose based on the total cost of routing a path. The least costly path is the one chosen by the router.

For example, if you want to minimize the number of vias the router places on your board, enter a high value for **Via cost**. Although a high value tends to produce fewer vias, it also tends to route the layers independently, thus resulting in more unconnected subnets.

> NOTE: The higher the highest cost you specify for a right-way cost, a wrong-way cost, and a via cost, the longer the run time.

We recommended that if you are an inexperienced user, you use the default cost parameters.

For the costing parametes, press [Return] to accept the values displayed, or type in new values, and then press [Return].

Stub Length Definition

These parameters allow you to specify the maximum length of a stub on both the horizontal and vertical layers, when using daisy chain or min-span route wiring types.

Horizontal layer length - Specifies the stub length allowed in the x and y directions in mils on the horizontal layers.

Vertical layer length - Specifies the stub length allowed in the x and y directions in mils on the vertical layers.

For the stub length definition parameters, press [Return] to accept the values displayed, or type in new values, and then press [Return].

To get on-screen help with any of the detailed routing parameters, use the arrow keys to move the cursor to the section you want help with and press [F1]. The help screen for that section will be displayed.

After you finish editing the detailed routing parameters, press [Esc] twice to return to the Routing Strategy Menu.

Editing the Pad Blocking Rules

To edit the pad blocking rules, select the **Edit pad blocking rules** option on the Routing Strategy Menu. The menu shown in Figure 2-8 is displayed.

PC-ROUTE

Edit pad blocking rules for strategy TEST   Grid size = 50

Pin type: 0   Pad shape:

Pad diameter:

Edit detailed pads

Enter parameter; [RETURN] to accept; [F1] help; [END] detail; [ESC] quit.

Figure 2-8. Pad Blocking Rules Menu

This menu allows you to specify the shape and size assigned to the various pin types you have defined in the PCB database.

Pin type - Specifies the pin type of the pad to be defined for a part and relates to the attachment of padstacks to pin types when using the special symbol file with PC-CARDS. (See the *PC-CARDS User's Manual* for details on special symbol files.)

Pad shape - Specifies the shape assigned to the pin type. The options are "Circle," "Square," or "Irregular." "Circle" is for round pads. "Square" is for square pads.

Irregular is for all other pad shapes (for example, connector fingers).

Pad diameter or Pad side length - Specifies the size of a round or square pad, respectively, in DBUs. The maximum value is 250 DBUs.

To define or change the shape and size assigned to a pin type, use the space bar or the backspace key to cycle through pin types 0 through 24, and press [Return] to select the pin type to define. Next, use the space bar to cycle through the pad shapes and press [Return] to select the pad shape.

If you select "Circle" or "Square" for **Pad shape**, another field is displayed, **Pad diameter**, or **Pad side length**, respectively. Type the size of the round or square pad in DBUs.

If you select "Irregular" for **Pad shape**, pad blocking models are displayed that you must block to indicate the size and shape of an irregular pin type. The trace model specifies the grids surrounding the origin of the pad that need to be blocked from the router routing traces. The via model specifies the grids surrounding the origin of the pad that need to be blocked from the router placing vias. The plus sign (+) represents the origin of the pad, the 0s represent the grids surrounding the pad center that are open, and the 1s represent the grids surrounding the pad center that are blocked.

To block an irregular pad type, use the four arrow keys to move around the pad model and type a 1 at each grid that should be blocked. When the trace blocking is completely defined, press [Return]. The cursor moves to the via pad. Define this pad in the same way, and then press [Return].

The cursor returns to **Pin type** and the pad display is erased. You can select another pin type to be defined, or select the **Edit detailed pads** option, or press [Esc] to return to the Routing Strategy Menu.

You can also use the **Copy pin** option to copy an irregular pad blocking model from another irregular pin. After you select "Irregular," press the [Home] key, and then use the space bar to toggle through the pin types that have been defined, and press [Return].

> NOTE: Since blocking rules for irregular pin types are specified in grids, each routing grid size requires a new blocking rule definition.

Edit detailed pads - Calls the Detailed Pad Blocking Rules Menu.

Figures 2-9, 2-10, and 2-11 show the pad blocking defined for a circular, square, and irregular pad shape, respectively.

PC-ROUTE

Edit pad blocking rules for strategy TEST   Grid size = 50

Pin type: 0   Pad shape: Circle

Pad diameter: 50

Edit detailed pads

Enter parameter; [RETURN] to accept; [F1] help; [END] detail; [ESC] quit.

Figure 2-9. Pad Blocking Defined for a Circular Pad

PC-ROUTE

Edit pad blocking rules for strategy TEST   Grid size = 50

Pin type: 6   Pad shape: Square

Pad side length: 60

Edit detailed pads

Enter parameter; [RETURN] to accept; [F1] help; [END] detail; [ESC] quit.

Figure 2-10. Pad Blocking Defined for a Square Pad

```
                              PC-ROUTE

   Edit pad blocking rules for strategy TEST      Grid size = 50

       Pin Type: 7    Pad Shape: Irregular        Copy pin:

       0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 1 1 1 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 1 1 1 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 1 + 1 0 0 0 0      0 0 0 0 0 + 0 0 0 0 0
       0 0 0 0 1 1 1 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 1 1 1 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
       0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0

               Trace                      Via

                           Edit detailed pads

Arrow keys to move; [RETURN] accept pad; [HOME] copy; [END] detail;
[ESC] quit.
```

Figure 2-11. Pad Blocking Defined for an Irregular Pad

Editing the Detailed Pad Blocking Rules

To edit the detailed pad blocking rules, select the **Edit detailed pads** option on the Pad Blocking Rules Menu. The following menu is displayed.

PC-ROUTE

Edit detailed pad blocking rules for strategy TEST Grid size = 50

Pin type: 0 Pad shape: Layer:

[SPACE] choices; [RETURN] to accept; [F1] help; [ESC] quit.

Figure 2-12. Detailed Pad Blocking Rules Menu

This menu allows you to define different pad blocking information for each layer on the board. Only pin types that have been previously defined (with the basic pad blocking menu) can be selected from this menu.

To select a pin type to edit, use the space bar or the backspace key to cycle through the pin types and press [Return] to select the pin type. Next, use the space bar to cycle through the pad shapes and press Return. Then use the space bar to cycle through the layer options and press [Return].

If you select "Circle" or "Square" for **Pad shape**, another field is displayed, **Pad diameter** or **Pad side length**, respectively. Type the size of the round or square pad in DBUs.

If you select "Irregular" for **Pad shape**, the pad models are displayed. If desired, enter new blocking rules in the same way you did on the Pad Blocking Rules Menu.

After the Trace and Via pad blocking rules are defined, the cursor returns to Layer and the pad displayed is erased. You can select another layer to edit, or you can select another pin type to edit. To return to **Pin type**, press [Home].

> NOTE: Regular pads can be defined as a circle on some layers and a square on the others. A pad that is irregular on any layer must be defined as irregular on all layers.

Figures 2-13, 2-14, and 2-15 show the detailed pad blocking defined for a circular, square, and irregular pad shape, respectively.

PC-ROUTE

Edit detailed pad blocking rules for strategy TEST   Grid size = 50

Pin type: 0   Pad shape: Circle   Layer: 2

Pad diameter: 50

Enter parameter; [RETURN] to accept; [F1] help; [ESC] quit.

Figure 2-13. Detailed Pad Blocking Defined for a Circular Pad

PC-ROUTE

Edit detailed pad blocking rules for strategy TEST   Grid size = 50

Pin type: 6   Pad shape: Square   Layer: 1

Pad side length: 60

Enter parameter; [RETURN] to accept; [F1] help; [ESC] quit.

Figure 2-14. Detailed Pad Blocking Defined for a Square Pad

PC-ROUTE

Edit detailed pad blocking rules for strategy TEST   Grid size = 50

Pin Type: 7   Pad Shape: Irregular   Layer: 1   Copy pin:

```
0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 1 1 1 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 1 1 1 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 1 + 1 0 0 0 0      0 0 0 0 0 + 0 0 0 0 0
0 0 0 0 1 1 1 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 1 1 1 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0      0 0 0 0 0 0 0 0 0 0 0
          Trace                        Via
```

Arrow keys move; [RETURN] accept pad; [F1] help; [HOME] copy; [ESC] quit.

Figure 2-15. Detailed Pad Blocking Defined for an Irregular Pad

Editing the Wiring Rules

To edit the wiring rules, select the **Edit wiring rules** option on the Routing Strategy Menu. The following menu is displayed.

PC-ROUTE

Edit wiring rules for strategy TEST   Grid size = 50

	Rule Name	Trace Width (Database Units)	Clearance Needed (Database Units)	Rule Type
(Default Rule)	DEFAULT	12	12	Normal
	IGNORE	0	0	Ignore
	POWER	50	50	Normal

Enter parameter; [RETURN] to accept; [F1] help; [ESC] quit.

Figure 2-16. Wiring Rules Menu

This menu allows you to define wiring rules of trace width and clearance to be used during routing. The first rule, the default rule, defines all traces to be routed at a width of 12 DBUs, with a grid clearance of 12 DBUs. This rule will be used by the router for routing all nets except those specifically entered in the Net Class Definitions Menu. The second rule, the ignore rule, defines any nets assigned to that rule in the Net Class Definitions Menu to be ignored; that is, those nets will not be routed. The third rule, the power rule, defines any nets assigned to that rule in the Net Class Definitions Menu to be routed at a width of 50 DBUs with a grid clearance of 50 DBUs.

NOTE: PC-ROUTE will only use the 45 degree routing option to route nets assigned to the DEFAULT rule. If you assign a group of nets with colinear pin connections to a wiring rule other than DEFAULT when defining net classes, this net class will be routed with orthogonal traces.

Rule Name - Specifies the rule being described in terms of trace width, clearance needed, and rule type. This name must be alphanumeric with a maximum of eight characters.

Trace Width - Specifies the width of a trace to be routed in DBUs for all nets applied to a particular wiring rule.

Clearance Needed - Specifies the minimum trace-to-trace, trace-to-pad, and pad-to-pad clearance needed for the width of a trace in DBUs. PC-ROUTE uses this information to determine how much of an air gap to leave around pads and traces during its routing process.

Rule Type - Specifies how PC-ROUTE will process the wiring rule. The options are "Normal" or "Ignore."

If you select "Normal," PC-ROUTE will process the wiring rule.

If you select "Ignore," PC-ROUTE will not route the nets assigned to that wiring rule name. (See the "Editing Net Class Definitions" section for assignment of nets.)

NOTE: If you assign "Ignore" to a net with prewired traces, the traces will be ignored, and PC-ROUTE can route over those prewired traces causing shorts.

When the Wiring Rules Menu is displayed, all of the rules already defined will be displayed and the cursor will be positioned at the beginning of the next blank line.

To define a new rule, type the name of the rule under the Rule Name column and press [Return]. Next type the width of the traces in DBUs under the Trace Width column and press [Return]. Then, type the clearance needed in DBUs around traces under the Clearance Needed column and press [Return]. Finally, use the space bar to select the Rule Type option and press [Return].

> NOTE: The maximum value that can be entered for Trace Width and Clearance is 250 DBUs.

Editing Net Class Definitions

To edit net class definitions, select the **Edit net class definitions** option on the Routing Strategy Menu. The Net Class Definitions Menu is displayed as shown in Figure 2-17.

PC-ROUTE

Edit net class definitions for strategy TEST

>> List all net classes <<

Edit net classes

Delete net classes

[SPACE] options; [RETURN] accept; [F1] help; [ESC] quit.

Figure 2-17. Net Class Definitions Menu

This menu provides the following options.

List all net classes - Displays a list of all the net classes currently defined for this strategy.

Edit net classes - Calls the Edit Net Class Definitions Menu to change or create new net class definitions for the current routing strategy.

Delete net classes - Calls the Delete Net Class Definitions Menu.

Use the space bar to cycle through the options, and then press [Return] to select an option.

Listing All Net Classes

To list the names of all net classes currently defined for this strategy, select the **List all net classes** option on the Net Class Definitions Menu. Figure 2-18 shows an example of the screen display.

PC-ROUTE

Net classes defined for strategy TEST

Class1
Class2
Class3

[Esc] to quit.

Figure 2-18. List of All Net Classes Defined for Current Strategy

Editing Net Classes

To create or change net class definitions for the current routing strategy, select the **Edit net classes** option on the Net Class Definitions Menu. The following menu is displayed.

PC-ROUTE

Edit net class definitions for strategy TEST

Net class name:

Wiring rule: Priority:

Type of class: Layer pair:

Enter parameter; [RETURN] to accept; [F1] help; [ESC] quit.

Figure 2-19. Edit Net Class Definitions Menu

Defining net classes allows you to assign wiring rules, which were defined in the Wiring Rules Menu, to different groups of nets and to specify priorities for how PC-ROUTE should route different groups of nets. Net classes can be defined by net list, component list, or window area, and can be assigned to a specific layer.

To change a net class or define a new net class, type the name of the net class on this menu, and press [Return].

Then, use the space bar to cycle through the **Wiring rule** options (previously defined in the Wiring Rules Menu), and then press [Return] to select an option.

Next, use the space bar to cycle through the **Priority** options (None, High, Medium, or Low), and then press [Return] to select an option. None is no priority. High is the highest priority. Medium is routed after nets with high priority. Low is routed after nets with high or medium priority.

> NOTE: PC-ROUTE routes by this user-specified priority, unless there is a net class that is wider than this one. Then, the wider traces are routed first.

Next, use the space bar to cycle through the **Type of class** options (Netlist, Comp list, or Window), and then press [Return] to select an option. This parameter allows you to select whether you wish to define the nets belonging to this net class definition by net list, component list, or window coordinates.

Next, use the space bar to cycle through the **Layer pair** options, and then press [Return] to select an option. The layer pairs that are listed depends on the number of layers defined in the Routing Parameters Menu. If you select "Any," the router will route this net class on any layer pair. If you select a specific layer pair, the router will route this net class on those layers only.

If you select "Net list" as the type of class, you are prompted to enter the names of the nets you want assigned to this class.

If you select "Comp list" as the type of class, you are prompted to enter a list of component types (for example: 7400, 7408). Any nets assigned to these component types will be assigned to this class.

If you select "Window" as the type of class, you are prompted to enter four PCB database coordinate values, as displayed in PC-CARDS, to define the window.

These coordinate values are: llx (lower left x), lly (lower left y), urx (upper right x), and ury (upper right y). Any nets contained within this window will be assigned to this class and PC-ROUTE will complete the routing of every net that has a pin in this window.

Type the names of the nets (for example, GND or +5), the names of the component types (for example, 7400 or 7408), or the names of the x and y window coordinates to be defined in this net class, and then press [Return].

> NOTE: PC-ROUTE excepts only ten net names or component names. If you have more than ten net names or component names, you need to define several net classes using the same net class definitions but with different net names and component names.
>
> Also, PC-ROUTE will only route nets assigned to the DEFAULT rule with the 45 degree routing option. If you assign a group of nets with colinear connections to a wiring rule other than DEFAULT when defining net classes, this net class will be routed with orthogonal traces.

Figures 2-20, 2-21 and 2-22 show examples of the Net Class Menu with nets classes defined by net list, component list, and window coordinates, respectively.

PC-ROUTE

Edit net class definitions for strategy TEST

Net class name: Class1

Wiring rule: DEFAULT   Priority: None

Type of class: Net list   Layer pair: Any

Nets: +5
-12

Enter parameter; [RETURN] accept; [HOME] class name; [F1] help; [ESC] quit.

Figure 2-20. Net Classes Defined by Net List

PC-ROUTE

Edit net class definitions for strategy TEST

Net class name: Class2

Wiring rule: DEFAULT   Priority: None

Type of class: Comp list   Layer pair: Int1/Int2

Components: 7400
7408

Enter parameter; [RETURN] accept; [HOME] class name; [F1] help;
[ESC] quit.

Figure 2-21. Net Classes Defined by Component List

PC-ROUTE

Edit net class definitions for strategy TEST

Net class name: Class3

Wiring rule: POWER Priority: Medium

Type of class: Window Layer pair: Comp/Sold

Window coordinates

llx: 500 lly: 500 urx: 2500 ury: 2500

Enter parameter; [RETURN] accept; [HOME] class name; [F1] help; [ESC] quit.

Figure 2-22. Net Classes Defined by Window Coordinates

Deleting Net Classes

To delete any or all of the net classes defined for the current strategy, select the **Delete net classes** option on the Net Class Definitions Menu. Figure 2-23 shows an example of what is displayed.

PC-ROUTE

Delete net class definitions for strategy TEST

Net class name: Class1

Wiring rule: DEFAULT   Priority: None

Type of class: Net list   Layer pair: Any

Nets: +5
-12

OK to delete net class? Yes

[SPACE] choices; [RETURN] accept; [HOME] class name; [ESC] quit.

Figure 2-23. Delete Net Class Definitions Menu

Use the space bar to cycle through the net class names until the desired name is displayed. Press [Return] to select the net class. The following message is displayed:

OK to delete net class? Yes

Press [Return] to confirm the delete or use the space bar to cycle to "No" and press [Return].

When you are finished editing the current routing strategy, press [Esc] to return to the Routing Strategy Menu. Press [Esc] again and the options for saving the strategy you just edited are displayed on the bottom of the Routing Strategy Menu. Select one of the options displayed. These options are described in "Editing the Routing Strategy."

NOTE: If two net classes of the same type have been defined so that they conflict, the routing strategy will not be saved. For example, if Class1 assigns wiring rule IGNORE to net GND and Class2 assigns wiring rule POWER to net GND, there is a conflict. When you try to save the strategy, the following message is displayed:

Net classes <name1> and <name2> conflict.
Press any key to end.

When a key is pressed, the Routing Strategy Menu is displayed with an Edit Net Class Definitions option highlighted. Revise the Net Class Definitions.

EXECUTING PC-ROUTE

To run the routing phase of PC-ROUTE, move the cursor to the **Route** option on the PC-ROUTE Main Menu and press [Return]. The following menu is displayed.

PC-ROUTE

Routing Options

Route status: New route, nothing complete

>>	Start	<<
Extract data:	YES	
Route:	NEW	
Create routed database:	YES	
Database name:	TEST.PCB	

Route by window

[SPACE] options; [RETURN] accept; [F1] help; [END] window; [ESC] quit.

Figure 2-24. Routing Options Menu

This menu allows you to control the routing of the database. The status line at the top indicates the current status of the database.

This menu gives you the following options:

Start - Executes the choices selected as described below.

Extract data - Specifies the extract option. The options are "YES" or "NO."

Select "YES" if routing is being done on the database for the first time.

Select "NO" if the data has already been extracted from the database and you are restarting the route.

> NOTE: This option must be "YES" any time the routing strategy has changed.

Route - Specifies the route option. The options are "NEW," "RESTART," or "NO."

Select "NEW" if routing is being done on the database for the first time.

Select "RESTART" if you stopped the routing process for this database and you want to restart the routing process.

Select "NO" if you want to extract data from the database.

Create routed database - Specifies the create routed database option. The options are "YES" or "NO."

Select "YES" if you want the autorouting data to be added to the input database and a new database created.

Select "NO" if you do not want the autorouting data to be added to the input database and do not want a new database created.

Database name - Specifies the database name for the routed database. The name displayed is the same name as the database name entered on the PC-ROUTE Main Menu, except the filename extension is now .PCB.

Route by window - Specifies the PCB database coordinate values to define the window to be routed. These coordinate values are: llx (lower left x), lly (lower left y), urx (upper right x), and ury (upper right y). Only nets with pins within this window will be routed.

Note that this option is different from the "window" type of net class option on the Edit Net Class Menu. On the Routing Options Menu, the definition of the window area to be routed is not saved as a part of the routing strategy control file (<filename>.CTL). This option is used for one time only to route a group of nets on the board. If you do not specify coordinate values, the entire board is routed.

> NOTE: If you select this option, we recommend that you select "No" for the **Perform via minimization** option on the Routing Parameters Menu. Because you are only routing a small area of the board, the router will remove vias and change the layer the traces are on. This results in wrong way traces, which contradicts the trace direction you selected for each layer.

When the menu is displayed, the **Start** line is highlighted and the options displayed are selected by the system based on the current status of the database. If these options are the ones you want to use, press [Return]. To change an option, use the arrow keys to move the cursor down to the desired line. Use the space bar to cycle through the options for each line, and press [Return] to select an option. To change the name of the return database, type in the new name. To enter coordinate values for route by window, press [End] to move to the **Route by Window** option and press [Return]. Then, type in the coordinate values and press [Return]. If you decide not to use these values, move the cursor to "CANCEL" and press [Return]. Press [Home] to move the cursor to the **Start** line and press [Return] to start the router.

If the router is being run on the database file for the first time and you selected "YES" for the **Extract data** option, the data extraction from the database begins immediately. The screen displays the current status indicating the processing steps PC-ROUTE is going through to prepare for routing. These messages are:

Extracting the database <filename>.PLC

Reading data from the database

Writing the route data file

If you select "NEW" or "RESTART" for the **Route** option, the following messages are displayed in the sequence listed:

Allocating Model

Building Model

Allocating Maps

The type of display you see on the screen when PC-ROUTE begins routing depends on what you selected for **display type** on the Routing Parameters Screen.

NOTE: PC-ROUTE uses its own colors for the layer assignments. Therefore, the colors of the layer assignments in PC-CARDS will not match the colors of the layer assignments shown during routing.

During routing, you can use the [End] key to stop the routing of the current subnet and continue to the next subnet.

During routing, if PC-ROUTE encounters a fatal error, the program stops and the following message is displayed at the bottom of the status display:

Error Stop

Press [Return] to return to the PC-ROUTE Main Menu. The system prompts:

WARNING - Router aborted
View error report, <databaserefname>.REP

To find the error, view the status report file (<filename>.REP). Look for an error ID and a corresponding error message. Then refer to Appendix A, "Error Messages," for an explanation of the error and what you should do about it. If the extract phase or database creater phase aborts, a similar message appears when you return to the Main Menu.

Although PC-ROUTE is a batch program, you can stop processing and restart processing.

To freeze the routing at any time, type &. Then, type any key to continue routing.

> CAUTION: Be careful not to strike keys randomly because some of them may invoke debug options, such as !, @, and #.

To abort the router, press [Esc]. The system prompts you to confirm. Press [Esc] again and the following message is displayed at the bottom of the status display:

Route was halted [CR]

Press [Return]. The following message is displayed:

> **Router was stopped by user.**
> **Create new database? No.**

You can choose not to create a new database by selecting "No," or you can choose to create a new database by selecting "Yes." Use the space bar to toggle between these options. If you select "No," the PC-ROUTE Main Menu is displayed. If you select "Yes," PC-ROUTE saves the database and displays the PC-ROUTE Main Menu when processing has ended.

You can restart PC-ROUTE without having to reroute all the connections that were created before the [Esc]. Select "RESTART" for the **Route** option on the Routing Options Menu to restart the router.

> NOTE: Do not run the EXTRACT process again if you want to restart.

PC-ROUTE saves the data at checkpoints, about every 40 routes. If the router is stopped unintentionally, such as a power failure, the data is restarted from the last checkpoint. If the router is interrupted with [Esc], all routes will be recovered when restarted.

If you selected "Yes" for the **via minimization** option, this is the last step that the router performs on each layer pair. At this point, the router takes out any unnecessary vias. During this step, PC-ROUTE displays the following message:

> **Minimizing vias**

When the router has completed processing, the following message is displayed on the bottom of the status display:

Route complete [CR]

Press [Return] to continue to the next phase of processing.

NOTE: If you do not press [Return], after two minutes PC-ROUTE continues to the next phase anyway. This feature allows you to run PC-ROUTE unattended.

CREATING THE ROUTED DATABASE

If you selected "YES" for the **Create routed database** option on the Routing Options Menu, the system creates the routed PCB database with the database name you selected.

The following messages are displayed during the process:

Creating routed database <filename>.PLC

Setting up and checking

Opening database

Processing route data

Saving database

If the input database was changed in PC-CARDS after the extraction phase, the system displays the following message:

> **Mismatch between extract database/stamp and current database/stamp (routed database still being created).**
>
> **Press any key to return to main menu.**

This message lets you know the database was updated in PC-CARDS after the data was extracted, but creates the routed database anyway.

After the PCB database file is created, you can graphically view it or edit it in PC-CARDS. Use the PC-CARDS FILE/LOAD command to call up the database file on the screen. Note that PC-ROUTE added a layer named $CONT to the layer structure. The $CONT layer displays the ratsnests for the unconnected subnets PC-ROUTE could not route.

Use the ENTR/WIRE command to complete any unconnected subnets.

READING THE STATUS REPORT

The status report that is output from PC-ROUTE can be viewed with any editor, displayed on the screen with the DOS TYPE command, or printed with the DOS PRINT command. The report file has the extension .REP. See Appendix B for a sample report.

After running the router, you should check the status report for errors that occurred during routing that were nonfatal as well as fatal. Check the status report for errors that occurred during the extraction and returner phases as well as the routing phase.

MAINTAINING YOUR FILES

To erase certain files created during the routing cycle, select the **Select Router File Maintenance** option on the PC-ROUTE Main Menu. The following menu is displayed.

PC-ROUTE

File Maintenance

>> Delete router file, route report, and route solution file <<

Delete routing strategy file

[SPACE] options; [RETURN] accept; [F1] help; [ESC] quit

Figure 2-25. PC-ROUTE File Maintenance Menu

This menu lets you delete files in your working directory that are no longer needed. This menu gives you the following options.

Delete router file, route report, and route solution file - Deletes all the files that were created by the router during processing (<filename>.RTE, <filename>.RTS, and <filename>.REP files).

Delete routing strategy file - Deletes the selected routing strategy file.

Use the space bar to toggle between the options.

If you select the **Delete router file, route report, and route solution file** option, the current database selected on the PC-ROUTE Main Menu is displayed as shown below.

Database reference name: <xxxx>

Use the space bar and backspace key to cycle through the database reference names. To select the database files to be deleted, press [Return].

To delete the files for the selected database, press [Return].

If you select the **Delete routing strategy file** option, the current routing strategy name is displayed as shown below.

Strategy file name: <xxxxx>

NOTE: To protect the PCAD1 routing strategy, the PCAD1 strategy filename will not be listed as one of the options.

Use the space bar and the backspace key to cycle through the names of the existing routing strategy files. To select the strategy to delete, press [Return].

EXITING PC-ROUTE

To exit PC-ROUTE, move the cursor to the **Exit PC-ROUTE** option on the PC-ROUTE Main Menu and press [Return].

The system returns you to DOS.

APPENDIX A. ERROR MESSAGES

Following is a list of error messages that might appear in the P-CAD status report file. Error messages appear in the report alongside a corresponding error ID. This list is arranged alphabetically by error ID and includes the error message, the cause of the error, and the appropriate user action.

Several types of error IDs and error messages might appear in the P-CAD status report file. Some messages are "trace back" messages. A trace back message refers you to a program error ID that lists a message that indicates what the actual error is and the appropriate user action.

If you encounter an error that is not listed here, call P-CAD.

In this list <n> is a variable number and <xxx> is a character string.

Calxmnet

Message: FATAL calling dmlsnpic.

Cause: System failure.

Action: Call P-CAD.

Cmcalnxm

Message: FATAL calling dmnxnpic.

Cause: System failure.

Action: Call P-CAD.

Cmwir2db

Message: Error in dmadnpic <n>.

Cause: System failure.

Action: Call P-CAD.

Main (extracter)

Message: Could not open file <xxx>.

Cause: The database <xxx> could not be opened; the database did not exist.

Action: Check that the database <xxx> exists.

Message: Could not read file <xxx>.

Cause: An I/O error was encountered reading a database stamp record; database was corrupted.

Action: Check the database.

Message: Could not open file <xxx>.

Cause: The file <xxx> (created by edit rules interface) could not be opened, the file was deleted.

Action: Reedit the parameters.

Message: Could not read rule count line.

Cause: System error.

Action: Call P-CAD.

Message: Could not read grid x data line.

Cause: System error.

Action: Call P-CAD.

Message: Could not read grid y data line.

Cause: System error.

Action: Call P-CAD.

Message: DMINIT returned <n>.

Cause: System error.

Action: Call P-CAD.

Message: DMLODALL returned <n> for open file <xxx>.

Cause: System error.

Action: Call P-CAD.

Message: Could not find pin <n> in the pin list.

Cause: System error.

Action: Call P-CAD.

Message: Could not open file <xxx>.

Cause: The file <xxx> (created by edit rules interface) could not be opened, the file was deleted.

Action: Reedit the parameters.

Message: Could not find the route type.

Cause: System error.

Action: Call P-CAD.

Message: Unknown route type.

Cause: System error.

Action: Call P-CAD.

Message: Pin outside of the routable area.

Cause: A pin was within 100 mils of the board outline or outside of the board outline altogether; or a misplaced component or incorrect board outline exists.

Action: Correct the bad data.

Message: Failed to free barall structure.

Cause: System error.

Action: Call P-CAD.

Message: <xxx> illegal barrier(s).

Cause: Only rectangles, circles (representing tooling holes) and lines are expected on the BRDOUT layer; different geometries than those listed above were on the BRDOUT layer.

Action: No corrective action is necessary; the geometries are ignored.

Message: Failed in end database action rc - <xxx>.

Cause: System error.

Action: Call P-CAD.

Message: Could not find start pin for net <xxx>.

Cause: System error.

Action: Call P-CAD.

Main (in extract report)

Message: Unknown route type.

Cause: System failure, or .CTL file was corrupted.

Action: Call P-CAD or restore .CTL file.

Message: Could not allocate barall structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Could not allocate new structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Could not allocate wr structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Could not find pin <n> in pin list.

Cause: System failure.

Action: Call P-CAD.

Message: Every net must be assigned a netname.

Cause: One or more nets in the database were unnamed. This is illegal because the net name is how PC-ROUTE returns the routes to the database environment.

Action: Name all the unnamed nets and rerun the extract.

Message: Pin outside of the routable area <n> <n>.

Cause: The pin specified was outside of the routable area (which is the board outline shrunk by 100 mills).

Action: Modify the board outline or pin location.

Message: Could not free barall structure.

Cause: System failure.

Action: Call P-CAD.

Message: Failed to free barall structure.

Cause: System failure.

Action: Call P-CAD.

Message: Failed to free board outline barrier structure.

Cause: System failure.

Action: Call P-CAD.

Message: Could not find the NETRULE.

Cause: System failure, or the .CTL file was corrupted.

Action: Call P-CAD.

Message: Could not find the route type.

Cause: System failure, or the .CTL file was corrupted.

Action: Call P-CAD.

Message: Could not read grid x data line.

Cause: System failure, or the .RTE file was corrupted.

Action: Call P-CAD.

Message: Could not read grid y data line.

Cause: System failure, or the .RTE file was corrupted.

Action: Call P-CAD.

Message: Failed in cf function.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in dc function.

Cause: System failure.

Action: Call P-CAD.

Message: <n> illegal barrier(s).

Cause: Any shapes other than a rectangle, filled rectangle and circle on the BAR layers are ignored.

Action: Delete the illegal barrier(s).

Message: Could not open file <xxx>.

Cause: The indicated file could not be opened.

Action: Check the disk, make sure the database is okay, try again.

Message: Could not read file <xxx>.

Cause: The indicated file could not be opened.

Action: Check the disk, make sure the database is okay, try again.

Message: Could not read rule count line.

Cause: System failure, or the .CTL file was corrupted.

Action: Call P-CAD.

Message: Dmlodall returned <n> for open file <xxx>.

Cause: The database was corrupted.

Action: Restore the database to an uncorrupted form. (Try to look at it with PC-CARDS; if unsuccessful, call P-CAD.)

Message: Failed in end database action rc = <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Dmbdsize returned <n>.

Cause: The database filename was not a valid database.

Action: Make sure the database name is a valid, uncorrupted database.

Message: Dminit returned <n>.

Cause: System error.

Action: Call P-CAD.

Message: Dmlodall returned <n> for open file <xxx>.

Cause: System error.

Action: Call P-CAD.

Message: Failed allocating component name table.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Could not open file <xxx>.

Cause: Could not open a point file named <xxx>. The disk was full or the subdirectory was full.

Action: Make space available on the disk or in the subdirectory.

Message: Failed in end database action rc = <n>.

Cause: System error.

Action: Call P-CAD.

Rtalccls

Message: Error allocating net class record.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rtallbar

Message: Failed to allocate the barrall structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Barrier(s) out of the board outline.

Cause: A rectangle on a BAR layer was outside of the board outline.

Action: Correct the barrier or the board outline.

Rtothbar

Message: Barrier(s) out of the board outline.

Cause: A rectangle on a BAR layer was outside of the board outline.

Action: Correct the barrier or the board outline.

Rtxdelred

Message: Could not allocate the allpoints structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Failed to free the memory.

Cause: System error.

Action: Call P-CAD.

Rxacute

Message: Illegal acute angle(s) on board outline.

Cause: The line describing the board outline had an acute angle, which is illegal.

Action: Redraw the board outline.

Rxbdbar

Message: Pin outside of the routable area %d %d.

Cause: A pin was within the board edge and the routable area, or outside of the board outline altogether; or a misplaced component or incorrect board outline exists.

Action Correct the bad data.

Message: Failed to free barall structure.

Cause: System failure.

Action: Call P-CAD.

Message: Could not allocate barall structure.

Cause: The memory allocation request was denied.

Action; Reduce the problem size.

Message: Failed to free board outline barrier structure.

Cause: System failure.

Action: Call P-CAD.

Message: Could not free barall structure.

Cause: System failure.

Action: Call P-CAD.

Message: <n> illegal barrier(s).

Cause: Any shapes other than a rectangle, filled rectangle, and circle on the BAR layers are ignored.

Action: Delete the illegal barrier(s).

Rxbdbtr

Message: Failed to allocate the trace structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxbdtrc

Message: Failed to allocate the prewired trace structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxbldalp

Message: Could not allocate the allpoints structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Illegal board outline(s).

Cause: Only rectangles, circles representing tooling holes, and lines are expected on the BRDOUT layer; different geometries than those listed above were on the BRDOUT layer.

Action: No corrective action is necessary; the geometries are ignored.

Message: Polygon not closed, or redundant lines (or polygons).

Cause: The line describing the board outline did not end where it began.

Action: Redraw the board outline.

Message: Failed to free the allpoint structure.

Cause: System error.

Action: Call P-CAD.

Message: Can't free the block structure.

Cause: System error.

Action: Call P-CAD.

Rxbodbar

Message: Can't free the allpoints structure.

Cause: System error.

Action: Call P-CAD.

Rxbrdout

Message: Could not find the board outline.

Cause: There was no rectangle or line on the BRDOUT layer.

Action: Draw a board outline.

Message: Illegal board outline(s).

Cause: Only rectangles, circles representing tooling holes, and lines are expected on the BRDOUT layer; different geometries than those listed above were on the BRDOUT layer.

Action: No corrective action is necessary; the geometries are ignored.

Rxchkpin

Message: Failed to allocate the prewired trace structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxchkstn

Message: Failed to allocate the prewired trace structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxchkvia

Message: Failed to allocate the prewired trace structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxclsit

Message: Failed to allocate the trace structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxconct

Message: Wire not terminate at <n> <n>.

Cause: Dangling or overlapping wires were at the specified location.

Action: Make all wires actual connections; partial routes are illegal.

Rxdaisy

Message: Wire not terminate at pin or redundant line(s) at <n> <n>.

Cause: Dangling or overlapping wires were at the specified location.

Action: Make all wires actual connections; partial routes are illegal.

Message: Failed to free pinpair structure.

Cause: System error.

Action: Call P-CAD.

Rxdelred

Message: Could not allocate the allpoints structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Failed to free the memory.

Cause: System error.

Action: Call P-CAD.

Rxdiagnl

Message: Could not allocate the allpoints structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxdsylnk

Message: Failed to allocate pinpair table.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Pin not linked in daisy chain.

Cause: System error.

Action: Call P-CAD.

Rxfdstrt

Message: Illegal board outline(s).

Cause: Only rectangles, circles representing tooling holes, and lines are expected on the BRDOUT layer; different geometries than those listed above were on the BRDOUT layer.

Action: No corrective action is necessary; the geometries are ignored.

Rxfindba

Message: Failed to allocate the trace structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Unacceptable board outline(s).

Cause: The closed polygonal line describing the board outline crossed itself.

Action: Redraw the board outline.

Rxfndalb

Message: Could not allocate the allblks structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Illegal board outline(s).

Cause: Only rectangles, circles representing tooling holes, and lines are expected on the BRDOUT layer; different geometries than those listed above were on the BRDOUT layer.

Action: No corrective action is necessary; the geometries are ignored.

Rxfreetr

Message: Can't free the trace structure.

Cause: System error.

Action: Call P-CAD.

Rxgetncl

Message: Couldn't find netclass section of strategy.

Cause: System failure.

Action: Call P-CAD.

Rxgetnw

Message: Error allocating netclass netlist.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Error freeing netclass record.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing netlist.

Cause: System failure.

Action: Call P-CAD.

Rxgetrul

Message: Failed to allocate wire rule table.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Couldn't find wirerule section of strategy.

Cause: System failure.

Action: Call P-CAD.

Rxgetwr

Message: Could not allocate wr structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Could not find the NETRULE.

Cause: A system error or control file (<filename>.CTL) was corrupted.

Action: Call P-CAD.

Rxlnkpin

Message: Failed to allocate set structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxlnksub

Message: Illegal connection(s) for minimum spanning tree.

Cause: The T connection exists in the prerouted wires and min span was specified in the routing parameters.

Action: Correct the prerouted wires or routing parameters.

Message: Wire not terminate at pin or redundant line(s) at <n> <n>.

Cause: Dangling or overlapping wires were at a specified location.

Action: Make all wires actual connections; partial routes are illegal.

Message: Failed to allocate set structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxmstend

Message: Failed to allocate the prewired trace structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxnetpin

Message: Could not find pin <n> in pin list.

Cause: System failure.

Action: Call P-CAD.

Rxnwallc

Message: Could not allocate nw structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxordpin

Message: Failed to allocate map structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Can't free the prewired structure.

Cause: System error.

Action: Call P-CAD.

Message: Failed to free map structure.

Cause: System error.

Action: Call P-CAD.

Rxoutpnt

Message: Failed to free point record.

Cause: System error.

Action: Call P-CAD.

Rxpwpnt

Message: Could not allocate PNT structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rxrdctl

Message: Could not open file <xxx>.

Cause: The file <xxx> (created by edit strategy) could not be opened; the file was deleted.

Action: Reedit the parameters.

Message: Strategy file <xxx> is not up to date.

Cause: A strategy file <xxx> was created with an old version of software and is not compatible.

Action: Specify a strategy file that is up-to-date, or convert the old one by editing it. Rerun the extract.

Message: Could not read rule count line.

Cause: System failure.

Action: Call P-CAD.

Message: Could not read gridx data line.

Cause: System failure.

Action: Call P-CAD.

Message: Could not read gridy data line.

Cause: System failure.

Action: Call P-CAD.

Message: Could not find the route type.

Cause: System failure.

Action: Call P-CAD.

Message: Unknown route type.

Cause: System failure.

Action: Call P-CAD.

Message: Could not find the shrnkval.

Cause: System failure.

Action: Call P-CAD.

Message: Could not find the horzlayr.

Cause: System failure.

Action: Call P-CAD.

Rxredunt

Message: Redundant line at <n> <n>.

Cause: Overlapping wires were at the specified location.

Action: Remove the overlapping wires.

Rxschpin

Message: Wire not terminate at <n> <n>.

Cause: Dangling wire was at the specified location.

Action: Make all wires actual connections; partial routes are illegal.

Rxstmst

Message: Failed to allocate set structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Failed to free set structure.

Cause: System error.

Action: Call P-CAD.

Rxtmstdy

Message: Illegal T for minimum spanning tree.

Cause: The T connections exist in the prerouted wires, and minimum span was specified in the routing parameters.

Action: Correct the prerouted wires or routing parameters.

Message: Illegal T connection(s) for daisy chain.

Cause: The T connections exist in the prerouted wires, and daisy chain was specified in the routing parameters.

Action: Correct the prerouted wires or routing parameters.

Rxwrtnet

Message: Every net must be assigned a netname.

Cause: One or more nets in the database are unnamed.

Action: Name all unnamed nets and rerun the extract.

Rtgetwr

Message: Could not allocate wr structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Could not find the NETRULE.

Cause: A system error or control file (<filename>.CTL) was corrupted.

Action: Call P-CAD.

Main (in return report)

Message: Failed in ending database access.

Cause: System failure.

Action: Call P-CAD.

Cmwir2bd

Message: Error in dmadnpic <n>.

Cause: System error.

Action: Call P-CAD.

Rrzapcnt

Message: Failed in dmdlnpic for net <n>.

Cause: System error.

Action: Call P-CAD.

Message: Failed in getting all nets.

Cause: System error.

Action: Call P-CAD.

Message: Failed in getting all pictures for net <n>.

Cause: System error.

Action: Call P-CAD.

Rtadpntr

Message: Failed to allocate pntrecd.

Cause: The memory allocation request was denied; the problem was too large.

Action: Reduce the problem size. It would be very unusual to get through the router and fail in the return due to problem size. Please call P-CAD if this occurs.

Rtbldlay

Message: Failed to allocate layer table for <n> layers.

Cause: The memory allocation request was denied; the problem was too large.

Action: Reduce the problem size. It would be very unusual to get through the router and fail in the return due to problem size. Please call P-CAD if this occurs.

Message: Failed to find/allocate layer name <xxx>.

Cause: The layer name <xxx> is one of the layers to which data is supposed to be returned and it is not in the database.

Action: Create the required layer through PC-CARDS, and restart the routing process (including extract).

Rtdelpts

Message: Failed to release pntrecd.

Cause: System error.

Action: Call P-CAD.

Rtgetpnp

Message: Read error on pnt stat <n>.

Cause: A read error was encountered reading route solution file <xxx>; the file was corrupted, system error.

Action: Restart router. (You do not need to reextract data.)

Rtgtdbst

Message: Could not open file <xxx>.

Cause: PC-CARDS database was destroyed.

Action; Restore PC-CARDS database.

Message: Could not read file <xxx>.

Cause: PC-CARDS database was destroyed.

Action: Restore PC-CARDS database.

Rtmtchdb

Message: Failed trying to open problem file <xxx>.

Cause: The file <xxx> could not be opened:

1. The file did not exist (probably deleted since route);

or

2. The file was not in correct format.

Action: Check the status report (<filename>.REP) for errors. Make sure you are processing a correct design. (Most likely, if the file was not intentionally deleted, the router was not run.)

Message: Unexpected end of file looking for dbname and stamp.

Cause: A read error was encountered when trying to read the first record of the route. The route solution file was corrupted. PC-ROUTE was abruptly interrupted before the first checkpoint.

Action: Check the status report (<filename>.REP) for errors.

Message: Mismatch between route data and input database.

Cause: When the extract is run, data is kept as to what database is being used and when the database was last updated. The returner ensures that there is an exact match on both data fields. This message means that there was a mismatch trying to return data to a database other than the one from which the data was extracted. The database was updated since the data was extracted.

Action: Return to the correct database. Restart the entire routing pass.

Message: Requested database is <xxx> with a dbstamp of.

Cause: This data explains the mismatch message.

Action: Return to the correct database. Restart the entire routing pass.

Message: Route data was from database <xxx> with dbstamp of.

Cause: This data explains the mismatch message.

Action: Return to the correct database. Restart the entire routing pass.

Rtnumlay

Message: Failed trying to open problem file <xxx>.

Cause: The file <xxx> could not be opened:

1. The file did not exist (it was probably deleted since route);

or

2. The file was not in correct format.

Action: Check the status report (<filename>.REP) for errors. Make sure you are processing a correct design. (Most likely, if the file was not intentionally deleted, the router was not run.)

Message: Failed to find board record.

Cause: A specific, required record was missing from a file.

Action: A file was corrupted or a system failure occurred. Restore the uncorrupted file, or restart the routing process from the beginning.

Rtopendb

Message: Dminit returned <n>.

Cause: System error.

Action: Call P-CAD.

Message: Dmlodall returned <n> for open file <xxx>.

Cause: System error.

Action: Call P-CAD.

Message: Dmdbsize returned <n>.

Cause: An invalid database file exists.

Action: Make sure the database filename is a valid database file.

Rtreadct

Message: Failed trying to open strategy file <xxx>.

Cause: Strategy file does not exist.

Action: Start the returner again with a valid strategy file.

Rtreadnt

Message: Failed trying to open problem file <xxx>.

Cause: The file <xxx> could not be opened:

1. The file did not exist (it was probably deleted since route);

or

2. The file was not in correct format.

Action: Check the status report (<filename>.REP) for errors. Make sure you are processing a correct design. (Most likely, if the file was not intentionally deleted, the router was not run.)

Rtreadsb

Message: Failed trying to open problem file <xxx>.

Cause: The file <xxx> could not be opened:

1. The file did not exist (it was probably deleted since route);

or

2. The file was not in correct format.

Action: Check the status report (<filename>.REP) for errors. Make sure you are processing a correct design. (Most likely, if the file was not intentionally deleted, the router was not run.)

Rtroutrt

Message: Failed allocating <n> pntrecs.

Cause: The memory allocation request was denied. The problem was too large and this route had too many vertices.

Action: Reduce the problem size. It would be very unusual to get through the router and fail in the return due to problem size. Please call P-CAD if this occurs.

Message: Failed to open points file <xxx>.

Cause: The file could not be opened with route solutions in it. The route solution file (<filename>.RTS) was deleted since route completed.

Action: Start the router again without selecting the restart option.

Message: Failed to read header for point file <xxx>.

Cause: A read error was encountered when reading the route solution file <xxx>. The file was corrupted, a system error occurred.

Action: Restart the router (you do not have to reextract data).

Message: Failed to get currency on <xxx>.

Cause: Could not get the net <xxx> current in the database. A system error occurred.

Action: Call P-CAD.

Rtsavedb

Message: Failed to save database <xxx> status <n>.

Cause: System error.

Action: Call P-CAD.

Message: Disk is probably full.

Cause: Insufficient disk space.

Action: Delete files from the disk.

Main (in router report)

Message: Cannot restart a completed route.

Cause: If a route completed without interruption, it cannot be restarted.

Action: No corrective action is necessary.

Message: Must have nine arguments.

Cause: System failure.

Action: Call P-CAD.

Message: Third argument must be NEW or RESTART, it was <xxx>.

Cause: System failure.

Action: Call P-CAD.

Message: Fourth argument must be DBGOFF or DBGON, it was <xxx>.

Cause: System failure.

Action: Call P-CAD.

Message: Fifth argument must be WINNO or WINYES, it was <xxx>.

Cause: System failure.

Action: Call P-CAD.

Rtacclin

Message: Failed to mark map <n> <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtacpbgd

Message: Failed in getting data for <n> <n> <n>.

Cause: System error.

Action: Call P-CAD.

Rtadcin

Message: Segment table allocation failed.

Cause: Most likely a system failure, or the board outline was outrageously oversized (about the size of a football field).

Action: Check the board outline; if okay, call P-CAD.

Message: Map segment <n> allocation failed.

Cause: The problem was too large.

Action: Reduce the problem size.

Message: Last map segment allocation failed.

Cause: The problem was too large.

Action: Reduce the problem size.

Rtaddest

Message: Failed calling rtpencvt <n> <n>.

Action: Trace back message for a failure.

Message: Failed to get data from density map <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed to put data back to density map <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtadlreg

Message: Failed to allocate strrlist.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtadpntr

Message: Could not get wire rule <n> current for subnet <n>.

Cause: System error.

Action: Call P-CAD.

Message: Failed writing point to points file.

Cause: System error.

Action: Call P-CAD.

Message: Could not get wire rvk <n> current for subnet <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Read error on last pnt write <n>.

Cause: System error.

Action: Call P-CAD.

Rtalcin

Message: Map segment <n> allocation failed.

Cause: The problem was too large.

Action: Reduce the problem size.

Message: Last map segment allocation failed.

Cause: The problem was too large.

Action: Reduce the problem size.

Message: Segment table allocation failed.

Cause: Most likely a system failure, or the board outline was outrageously oversized (about the size of a football field).

Action: Check the board outline. If okay, call P-CAD.

Message: Could not allocate grid masks for mapid <n>.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Message: Rowpointer table allocation failed.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtalcmen

Message: Unexpected end of problem size file, expected <xxx> record.

Cause: A corrupted problem file or system error exists.

Action: Restore the uncorrupted file and restart the routing process from the beginning.

Message: Failed to allocate <xxx> table <n> entries.

Cause: The memory allocation request was denied. The problem size was too large.

Action: Reduce the problem size.

Message: Failed trying to open problem file <xxx>.

Cause:

1. The file did not exist (it was probably deleted since the last route);
2. The file format was incorrect.

Action: Check the status report (<filename>.REP) for errors. Make sure you are processing a correct design. Most likely, if the file was not intentionally deleted, the router was not run.

Message: Unexpected end of problem size file; expected first record.

Cause: No data was in the (<filename>.REP) file; extract failed.

Action: Check the status report (<filename>.REP) for errors.

Message: Unexpected end of problem size file, expected second record.

Cause: Insufficient data was in the (<filename>.REP) file; extract failed.

Action: Check the status report (<filename>.REP) for errors.

Message: Failed to allocate pin table <n> entries.

Cause: There was insufficient memory for <n> pins to be allocated:

1. If <n> was 0, no pins were extracted.

2. If <n> was very large, too many pins were extracted.

3. The host system did not have enough memory.

Action:

1. Check the database for existence of pins (there must be some).

2. Check the database for duplicate components or erroneous pin descriptions;

   or

   If the number of pins is legitimate, the problem is too big for the amount of memory on the system.

3. Get more memory.

Message: Failed to allocate sub table <n> entries.

Cause: There was insufficient memory for the <n> subnets to be allocated:

1. If <n> was 0, no subnets were extracted.

2. If <n> was very large, too many subnets were extracted.

3. The host system did not have enough memory.

Action:

1. Check the database for existence of nets (there must be some).

2. Check database for unusual net descriptions.

   or

   If the number of pins is legitimate, the problem is too big for the amount of memory on the system.

3. Get more memory.

Message: Failed to allocate bus table <n> entries.

Cause: There was insufficient memory for <n> buses to be allocated:

1. If <n> was very large, too many pins were extracted.

2. The host system did not have enough memory.

Action:

1. Check the database for extraneous bus descriptions;

   or

   If the number of pins is legitimate, the problem is too big for the amount of memory on the system.

2. Get more memory.

Message: Failed to allocate wire rule table <n> entries.

Cause: There was insufficient memory for <n> wire rules to be allocated:

1. If <n> was 0, no wire rules were created for routing strategy.

2. If <n> was very large, too many wire rules were created for routing strategy.

3. The host system did not have enough memory.

Action:

1. Check the routing strategy for existence of wire rules (there must be some).

2. Check routing strategy for extraneous wire rule descriptions;

   or

   If the number of wire rules is legitimate, the problem is too big for the amount of memory on the system.

3. Get more memory.

Message: Failed to allocate padstack table <n> entries.

Cause: There was insufficient memory for <n> pad blocking rules to be allocated:

1. If <n> was 0, no pad blocking rules were defined.
2. If <n> was very large, too many pad blocking rules were created for the routing strategy.
3. The host system did not have enough memory.

Action:

1. Check the routing strategy for existence of pad blocking rules (there must be some).
2. Check the routing strategy for extraneous pad blocking rule descriptions;

   or

   If the number of pad blocking rules is legitimate, the problem is too big for the amount of memory on the system.
3. Get more memory.

Rtallbar

Message: Barrier(s) out of the board outline.

Cause: A rectangle on a BAR layer was outside of the board outline.

Action: Correct the barrier or the board outline.

Message: Failed to allocate the barall structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rtbldbar

Message: Failed to convert <na> coord <nb> from <xxx>.

Cause: A barrier was outside of the routable area. The value of <n a> is 0 - left x of barrier.

1 - lower y or barrier
2 - right x or barrier
3 - upper y of barrier

<nb> is the value of the coordinate; <xxx> is the description of the barrier. The barrier was described outside of the board outline (shrunk by 100 mils).

Action: Correct the barrier or expand the board outline description.

Message: Write error on barrier.

Cause: An I/O error occurred during a write to the file in which the barriers are kept.

1. The disk was full.

or

2. The disk was damaged.

Action:

1. Delete some files off of the disk to allow for more room. (The file size is completely dependent upon the number of barriers).

2. Use the DOS CHKDSK command (with /F option).

Rtbldbrd

Message: Failed in rtrl2gd for high <xxx> <n>.

Cause: The grid conversion failed for top or right of board outline. The grid definition was incorrect. The board was very large (>32767).

Action:

1. Check the grid definition.

2. Check the board outline.

Message: Maximum <xxx> not on grid <n>.

Cause: The top or right of the board outline was not on the grid. The top or the right of the board outline was not divisible by the grid.

Action: Modify the offending coordinate.

Rtbldbus

Message: Could not translate <xxx> coordinate <n>.

Cause: Could not translate the bus coordinate. The bus was outside of the board outline.

Action: Modify the bus or board outline to make them compatible.

Message: Bus trailer without a header ident <n>.

Cause: A file was corrupted, or an error from extract procedure exists.

Action: Check the status report (<filename>.REP) for extract errors or redo the extract. Call P-CAD if the error persists.

Message: Do not translate <x,y> coordinate <n>.

Cause: Could not translate the bus coordinate to a grid. The bus was outside of the board outline.

Action: Modify the bus or board outline to make them compatible.

Message: Illegal data construct in route description file. Extract procedure made an error.

Action: Check (<filename>.REP) for extract errors. Call P-CAD.

Rtbldctr

Message: Failed on opening route control file.

Cause: Could not open the file (<filename>.CTL); file was missing.

Action: Edit the desired routing strategy, and then rerun from the extract.

Message: Illegal route algorithm requested.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Illegal route type requested.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Illegal via type requested.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Illegal draw control parameter will use DRAWOUT.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Illegal penalty parameter will not use penalty.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Illegal net order, will use short to long.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Failed to allocate lattice structure.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Message: Illegal lattice parameter will not use lattice.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Premature EOF, expecting CONTROL SELECTION record.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing routing the strategy again. If this does not help, call P-CAD.

Message: Illegal 45 degree route parameter will use RT45NONE.

Cause: A rules interface error occurred, or the control file (<filename>.CTL) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Rtbldgrd

Message: Grid record neither grid x nor grid y.

Cause: A rules interface error occurred, or (<filename>.REP) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Failed to allocated <n> offsets for <xxx> grid.

Cause: Could not allocate memory for <n> minor offsets; if <n> is greater than 10 or 0, probably an interface problem occurred. Or, the system ran out of memory.

Action: Check the grid definition. Call P-CAD.

Message: Failed to find field <n>.

Cause: A rules interface error occurred, or (<filename>.REP) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Message: Failed to find offset <n>.

Cause: A rules interface error occurred, or (<filename>.REP) was corrupted.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Rtbldlay

Message: Failed to allocate layer table for <n> layers.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size. It would be very unusual to get through the router and fail in the return due to problem size. Please call P-CAD if this occurs.

Message: Failed to find layer name <xxx>.

Cause: Layer name <xxx> was one of the layers to which data was supposed to be returned and it was not in the database.

Action: Create the required layer through PC-CARDS and restart the routing process (including extract).

Rtbldmap

Message: Failed calling rtblksub for subnet <n>.

Action: Trace back message for a failure.

Message: Failed calling rtblksbs.

Cause: System error.

Action: Call P-CAD.

Message: Failed calling rtblkpns.

Cause: System error.

Action: Call P-CAD.

Message: Failed calling rtblkbrs.

Cause: System error.

Action: Call P-CAD.

Message: Failed calling rtblkbss.

Cause: System error.

Action: Call P-CAD.

Message: Failed calling rtlblkvi for points <n> <n>.

Action: Trace back message for a failure.

Message: No padstack <na> found for pin <nb> at <nc> <nd>.

Cause: The pin attribute <na> for the pin at location x=<nc> y=<nd> was not described in the pad blocking rules.

Action: Use the edit pad blocking rules to define the pad attribute.

Message: Failed rtinvth.

Action: Trace back message for a failure.

Message: Failed calling rtlblkvi for points <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtblkln for points <n> <n> <n> <n>.

Action: Trace back message for a failure.

Rtbldpin

Message: Could not translate <xxx> coordinate <n>.

Cause: The pin coordinate could not translated. The pin coordinate was outside the board outline.

Action: Correct the pin location or board outline.

Message: Could not allocate off grid pin structure.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtbldpnt

Message: Failed to find subnet record for id <n>.

Cause: System failure.

Action: Check the status report (<filename>.REP). If there are no correctable errors, call P-CAD.

Message: Failed to find wire rule for subnet <n>.

Cause: System failure.

Action: Check the status report (<filename>.REP). If there are no correctable errors, call P-CAD.

Message: Failed to translate <xxx> coordinate <n>.

Cause: System failure.

Action: Check the status report (<filename>.REP). If there are no correctable errors, call P-CAD.

Message: <xxx> coordinate <n> off grid, will be mapped onto grid.

Cause: Prerouted wire was not open on grid.

Action: Correct the wire coordinate.

Rtbldprb

Message: Problem description file cannot be opened.

Cause: The file (<filename>.REP) could not be opened. The extract was not run. The file was destroyed.

Action: Rerun the extract.

Message: Unexpected end of problem file.

Cause: System error.

Action: Call P-CAD.

Message: 5th and 6th lines must be grid record, found <xxx>.

Cause: System error.

Action: Call P-CAD.

Message: Failed in rtbldgrd on line.

Action: Trace back message for a failure.

Message: 7th line must be board record, it was <xxx>.

Cause: System error.

Action: Call P-CAD.

Message: Failed in Rtbldbrd 7th line was <xxx>.

Cause: System error.

Action: Call P-CAD.

Rtbldrul

Message: Could not open rules file <xxx>.

Cause: The file <xxx>, which contains rules, could not be opened. Rules definition interface failure occurred.

Action: Reedit the routing strategy.

Message: Unexpected end of problem file.

Cause: Extract failure.

Action: Check (<filename>.REP).

Message: First record of rules must be WIRE RULES, it wasn't.

Cause: System failure.

Action: Call P-CAD.

Message: No PADSTACKS record found in RULES file.

Cause: Edit pad clearances were not done for this routing strategy.

Action: Edit the padstack blocking rules.

Message: Premature EOF, expecting CONTROL SECTION record.

Cause: A rules interface error occurred.

Action: Try editing the routing strategy again. If this does not help, call P-CAD.

Rtbldsub

Message: Failed in <xxx> for subnet <n> <xxx>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in rtgtbscr for subnet <n> <xxx>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in rtgtsbcr for subnet <n> <xxx>.

Cause: System failure

Action: Call P-CAD.

Message: Illegal endpoint type.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in rtgtpncr for subnet <n> <xxx>.

Cause: System failure.

Action: Call P-CAD.

Rtbldvbr

Message: Failed to convert <n> coord <n> from <xxx>.

Cause: System failure. A boundary for a via barrier was out of the board outline.

Action: Call P-CAD.

Message: Write error on barrier.

Cause: System failure.

Action: Call P-CAD.

Rtblklin

Message: Failed calling rtlblkln for <n> <n> <n> <n>.

Action: Trace back message for a failure.

Rtblkpad

Message: Failed calling rtlblkpd for <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtvblkpd for <n> <n>.

Action: Trace back message for a failure.

Rtblksbs

Message: Failed calling rtblksub for subnet <n>.

Action: Trace back message for a failure.

Rtchkbrd

Message: Routes stopped by user.

Cause: User invoked [ESC] [ESC] sequence to abort the route.

Action: No corrective action is necessary. This is an informative message.

Rtchkmsg

Message: WARNING message #<n> have overflowed <xxx> buffer.

Cause: System failure.

Action: Call P-CAD.

Rtcloseu

Message: Error freeing file pointer.

Cause: System failure.

Action: Call P-CAD.

Rtclsbar

Message: Failed to close pnt file status <n>.

Cause: System failure.

Action: Call P-CAD.

Rtclslsr

Message: Failed to close lsr file status <n>.

Cause: System error.

Action: Call P-CAD.

Rtclsvbr

Message: Failed to close via bar file status <n>.

Cause: System failure.

Action: Call P-CAD.

Rtclspnt

Message: Failed to close pnt file status <n>.

Cause: System error.

Action: Call P-CAD.

Rtcnvtlt

Message: <xxx> lattice value <n> is off grid.

Cause: The via lattice description referenced point was off the grid.

Action: Correct the via lattice or grid description (n.b. offset values in grid descriptions).

Message: <xxx> lattice repeat value <n> is not a multiple of grid.

Cause: The step repeat of the via lattice was not a grid multiple.

Action: Correct the via lattice or grid description.

Message: Failed in converting <xxx> of lattice from <n>.

Cause: The via lattice description referenced point was outside the routable area.

Action: Correct the via lattice description and/or board outline.

Rtcrttrg

Message: Failed to allocate strtrgpts.

Cause: Memory allocation request was denied. The problem was too large.

Action: Reduce problem size.

Rtcrtxtn

Message: Unrecognized indicator <n>.

Cause: System failure.

Action: Call P-CAD.

Rtdelpts

Message: Failed to release PNTRECD.

Cause: System error.

Action: Call P-CAD.

Rtdltarg

Message: Error freeing target point.

Cause: System error.

Action: Call P-CAD.

Rtdlunsn

Message: Failed to free strnod.

Cause: System error.

Action: Call P-CAD.

Rtdrclea

Message: Zero size area - at location <n> <n>.

Cause: System error.

Action: Call P-CAD.

Rtdrsub

Message: Failed to find wire rule <n>.

Cause: System failure.

Action: Call P-CAD.

Rtfndprb

Message: Failed calling rtprblin for <n> <n>.

Action: Trace back message for a failure.

Rtgd2reg

Message: Regcrds out of bounds <n> in <xxx>.

Cause: System failure.

Action: Call P-CAD.

Rtgetmap

Message: Failed in Xlation <n> <n> <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtgetnxt

Message: Premature EOF, expecting <xxx> record.

Cause: A rules interface error occurred.

Action: Try editing routing strategy again. If this does not help, call P-CAD.

Message: Expecting '<xxx>' record but got '<xxx>'.

Cause: A rules interface error occurred.

Action: Try editing routing strategy again. If this does not help, call P-CAD.

Rtgetpnt

Message: Read error on pnt stat <n>.

Cause: A read error was encountered reading route solution file <xxx>. The file was corrupted, system error.

Action: Restart the router (you do not have to reextract data).

Rtgoodpt

Message: Failed in gother <n> <n>.

Action: Trace back message for a failure.

Message: Failed subnet <n> due to lack of memory.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size or get more memory.

Message: Failed in crtadn for <n> <n> <n>.

Action: Trace back message for a failure.

Message: Failed subnet due to lack of memory.

Cause: The memory allocation request was denied.

Action: Manually complete the unrouted nets in PC-CARDS.

Rtgother

Message: Current route target is <n> <n>.

Cause: Informative message.

Action: No corrective action is necessary.

Message: Inputs were <n> <n> <n>.

Cause: Informative message.

Action: No corrective action is necessary.

Message: Failed in rtmrklin for subnet <n>.

Action: Trace back message for a failure.

Message: No via padstack found <n>.

Cause: Blocking rule for the via pad <n>.

Action: Use the edit pad blocking rules to define the pad attribute.

Message: Failed calling rtvblkpd for points <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtvblkln for points <n> <n> <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtvpdmax for padstack <n>.

Action: Trace back message for a failure.

Message: Failed calling rtlblkvi for points <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtlblkln for points <n> <n> <n> <n>.

Action: Trace back message for a failure.

Message: Current route target is <n> <n>.

Cause: Informative message.

Action: No corrective action is necessary.

Rtgtclsp

Message: Failed for <xxx> of subnet <n>.

Action: Trace back message for a failure.

Rtgtdbst

Message: Could not open file <xxx>.

Cause: PC-CARDS database was destroyed.

Action: Restore PC-CARDS database.

Message: Could not read file <xxx>.

Cause: PC-CARDS database was destroyed.

Action: Restore PC-CARDS database.

Rtgtnewl

Message: Illegal Direction <n>.

Cause: System error.

Action: Call P-CAD.

Rthrdblk

Message: Rthrdedg returned status = <n>.

Action: Trace back message for a failure.

Message: Rtblkbrs returned status = <n>.

Action: Trace back message for failure.

Rthrdedg

Message: Failed calling rtormap for grid <n> <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtinvsom

Message: Failed to fine line <n> x or y <n>.

Cause: System failure.

Action: Call P-CAD.

Rtladnod

Message: Failed to allocate <strlsnod>.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtlctadn

Message: Failed to allocate node at <n> <n>.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtldlnod

Message: Error freeing live node <n> <n>.

Cause: System error.

Action: Call P-CAD.

Message: Error freeing dead node <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtldlreg

Message: Can't find lsr recrd <n>.

Action: Trace message back for failure.

Message: Failed to read lsr record.

Action: Trace message back for a failure.

Rtlfree

Message: Error freeing map <n>.

Cause: System error.

Action: Call P-CAD.

Rtliner

Message: To is not a pin for subnet <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed calling rtmrklin for subnet <n> points <n> <n>.

Action: Trace back message for a failure.

Message: Targets not active for subnet <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in prbvia for <n> <n> from <n>.

Action: Trace back message for a failure.

Message: Failed calling rtmrkgrd for <n> <n>.

Action: Trace back message for a failure.

Message: Failed subnet due to lack of memory.

Cause: The memory allocation request was denied.

Action: Manually complete the unrouted nets in PC-CARDS.

Rtlinmaz

Message: Failed calling rtliner for subnet <n>.

Action: Trace back message for a failure.

Message: Failed calling rtlinprb subnet <n> to point <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtliner for subnet <n>.

Action: Trace back message for a failure.

Rtlinprb

Message: Failed calling rtprblin for <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtmrklin for subnet <n> points <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtprblin for <n> <n>.

Action: Trace back message for a failure.

Message: Failed calling rtmrklin for subnet <n> points <n> <n>.

Action: Trace back message for a failure.

Message: Failed subnet due to lack of memory.

Cause: The memory allocation request was denied.

Action: Manually complete the unrouted nets in PC-CARDS.

Rtlintwv

Message: Lintwv failed for pin on subnet <n> at <n> <n>.

Action: Trace back message for a failure.

Message: Lintwv failed for sub on subnet <n> at <n>.

Action: Trace back message for a failure.

Message: Lintwv failed for bus on subnet <n> at <n>.

Action: Trace back message for a failure.

Rtlmkbus

Message: Failed marking bus <n>.

Action: Trace back message for a failure.

Rtlmkend

Message: Failed creating new node at <n> <n> <n>.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Message: Failed marking node block map at <n> <n> <n>.

Action: Trace back message for a failure.

Rtlmkpin

Message: Failed lmkpin at <n> <n>.

Action: Trace back message for a failure.

Rtlmksub

Message: Failed allocating newnode.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Message: Failed marking node block map at <n> <n> <n>.

Action: Trace back message for a failure.

Rtlmktgt

Message: Failed allocating target point record.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtlmktrg

Message: Lmktrg failed for pin on subnet <n> at <n> <n>.

Action: Trace back message for a failure.

Message: Lmktrg failed for sub on subnet <n> at <n>.

Action: Trace back message for a failure.

Message: Lmktrg failed for bus on subnet <n> at <n>.

Action: Trace back message for a failure.

Rtlndsrt

Message: Failed in allocating sort node table for <n> entries.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtlstbar

Message: Seek error for layer <n> at position <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Read error on first bar lay <n> pos <n> stat <n>.

Cause: System failure.

Action: Call P-CAD.

Rtlstfre

Message: Error freeing net table.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing offgrid pins.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing xlat table.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing barriers.

Cause: System failure.

Action: Call P-CAD.

Rtlstpnt

Message: Read error on first pnt sub <n> pos <n> stat <n>.

Cause: System failure.

Action: Call P-CAD.

Rtlstvbr

Message: Had a read error on barrier file <n>.

Cause: System failure.

Action: Call P-CAD.

Rtltrbck

Message: Failed adj resource for subnet <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing sort table.

Cause: System failure.

Action: Call P-CAD.

Rtlwrite

Message: Loose route write error for subnet <n> at <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtmadbus

Message: Bus table overflowed.

Cause: System failure.

Action: Call P-CAD.

Rtmadfil

Message: Failed to allocate space for a file.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtmadnod

Message: Failed to allocate strnod.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtmadpad

Message: Pad table overflowed.

Cause: System failure.

Action: Call P-CAD.

Rtmadpin

Message: Pin table overflowed.

Cause: You added pin blocking rules after you extracted data.

Action: Select YES for the "Extract data" option on the Routing Options Menu and rerun the router.

Rtmadsub

Message: Sub table overflowed.

Cause: You added pin blocking rules after you extracted data.

Action Select YES for the "Extract data" option on the Routing Options Menu and rerun the router.

Rtmadwir

Message: Wire table overflowed.

Cause: You added pin blocking rules after you extracted data.

Action: Select YES for the "Extract data" option on the Routing Options Menu and rerun the router.

Rtmakend

Message:	Failed subnet <n> due to lack of memory.
Cause:	The memory allocation request was denied.
Action:	Reduce the problem size.

Message:	Failed in rtcrtadn for sub <n> location <n> <n> <n>.
Action:	Trace back message for a failure.

Message:	Failed in rtmrkhom on subnet <n>.
Action:	Trace back message for a failure.

Message:	Failed subnet due to lack of memory.
Cause:	The memory allocation request was denied.
Action:	Manually complete the unrouted nets in PC-CARDS.

Rtmakvia

Message:	Failed in rtgoodpt for <n> <n> <n>.
Action:	Trace back message for a failure.

Rtmapfre

Message:	Error freeing map segment <n>.
Cause:	System failure.
Action:	Call P-CAD.

Rtmazlin

Message: Failed subnet due to lack of memory.

Cause: The memory allocation request was denied.

Action: Manually complete the unrouted nets in PC-CARDS.

Rtminvia

Message: Write error at position <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed to get pad current for pin <n> pad <n> at <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Message: NULL via padstack ptr for <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Could not unblock via at <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Could not unblock pin.

Cause: System failure.

Action: Call P-CAD.

Rtmodctr

Message: Failed to allocate pointers for layer pair <n>.

Cause: Wire clearance rules and/or pad blocking rules were misused on detailed blocking rule menus. System failure.

Action: Check for the blocking rules being defined on all layer pairs.

Rtmrkgrd

Message: Failed in putmap for <n> <n> <n>.

Action: Trace back message for a failure.

Rtmrkhom

Message: Failed in putmap for <n> <n> <n>.

Action: Trace back message for a failure.

Rtmrklin

Message: From = to, <n> <n>.

Action: Trace back message for a failure.

Message: Line neither horz or vertc from <n> <n>, to <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtmrkndp

Message: Failed in workpnt for pin <n> location <n> <n> <n>.

Action: Trace back message for a failure.

Message: Failed in rtlpdmax on padstk <n>.

Action: Trace back message for a failure.

Message: Failed in prbgrd <n> <n> <n>.

Action: Trace back message for a failure.

Message: Working pin <n> <n> <n> incr <n> as far as <n>.

Action: Trace back message for a failure.

Rtmtchdb

Message: Failed trying to open problem file <xxx>.

Cause: The file <xxx> could not be opened:

1. The file did not exist (it was probably deleted since route);

or

2. The file was not of correct format.

Action: Check the status report (<filename>.REP) for errors. Make sure you are processing a correct design (most likely, if the file was not intentionally deleted, the router was not run).

Message: Unexpected end of file looking for dbname and stamp.

Cause: A read error was encountered when trying to read the first record of route. Route solution file was corrupted. PCRTROUT was abruptly interrupted before the first checkpoint.

Action: Check the status report (<filename>.REP) for errors.

Message: Mismatch between route data and input database.

Cause: When the extract is run, data is kept regarding the database used and when the database was last updated. The returner ensures that there is an exact match on both data fields. This message means that there was a mismatch trying to return data to a database other than the one from which the data was extracted. The database was updated since the data was extracted.

Action: Return to the correct database. Restart the entire routing pass.

Message: Requested database is <xxx> with a dbstamp of.

Cause: When the extract is run, data is kept regarding the database used and when the database was last updated. The returner ensures that there is an exact match on both data fields. This message means that there was a mismatch trying to return data to a database other than the one from which the data was extracted. The database was updated since the data was extracted.

Action: Return to the correct database. Restart the entire routing pass.

Message: Route data was from database <xxx> with dbstamp of.

Cause: When the extract is run, data is kept regarding the database used and when the database was last updated. The returner ensures that there is an exact match on both data fields. This message means that there was a mismatch trying to return data to a database other than the one from which the data was extracted. The database was updated since the data was extracted.

Action: Return to the correct database. Restart the entire routing pass.

Rtmzsnap

Message: Failed calling rtliner for subnet <n>.

Action: Trace back message for a failure.

Rtndefct

Message: Failed calling rtcrtxtn subnet <n>.

Action: Trace back message for a failure.

Rtnodlin

Message: Failed subnet <n> due to lack of memory.

Cause: System error.

Action: Call P-CAD.

Message: Failed to create new node.

Action: Trace back message for a failure.

Rtnodsrt

Message: Failed in freeing sort node table.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in allocating sort node table for <n> entries.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Message: Failed new srtnod after <n> nodes.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtntvbit

Message: Failed putmap for <n> <n> on lattice <n> <n> <n> <n> <n> <n>.

Cause: The via lattice point could not be mapped into the route map. The number of repeats combined with the repeat value in via lattice record causes via locations outside of the routable area.

Action: Correct the via lattice definitions.

Message: Failed calling rtvpdmax for padstack <n>.

Action: Trace back message for a failure.

Rtnumlay

Message: Failed trying to open problem file <xxx>.

Cause: The file <xxx> could not be opened:

1. The file did not exist (it was probably deleted since route);

or

2. The file was not of correct format.

Action: Check the status report (<filename>.REP) for errors. Make sure you are processing a correct design (most likely, if the file was not intentionally deleted, the router was not run).

Message: Failed to find board record.

Cause: A specific, required record was file (<filename>.REP).

Action: The file was corrupted or system failure. Restore the uncorrupted file, or restart the routing process from the beginning.

Rtnxtbar

Message: Had a read error on barrier file <n>.

Cause: System failure.

Action: Call P-CAD.

Rtnxtpnt

Message: Had a read error on point file <n>.

Cause: System failure.

Action: Call P-CAD.

Rtnxtreg

Message: Had a read error on region file <n>.

Cause: System failure.

Action: Call P-CAD.

Rtnxtvbr

Message: Had a read error on barrier file <n>.

Cause: System failure.

Action: Call P-CAD.

Rtopendb

Message: dminit returned <n>.

Cause: System failure.

Action: Call P-CAD.

Message: dmlodall returned <n> for open file <xxx>.

Cause: The database <xxx> could not be opened by the database package. The database was corrupted.

Action: Restore the database to an uncorrupted version.

Message: Failed to open via barrier file <xxx>.

Cause: PCROUTE was not allowed to open the file PCRTVBAR.TMP.

Action:

1. Check for too many files open (16 normal maximum).

or

2. Check for a full disk.

Rtopnbar

Message: Failed to open barrier file <xxx>.

Cause: Could not open barrier file named <xxx> (intermediate file for PCRTROUT). The disk was full or a subdirectory was full.

Action: Make space available on the disk or in the subdirectory.

Message: Failed to allocate <n> entries in bar layers.

Cause: PCRTROUT was denied memory for <n> (log) integers. Number of layers requested was ridiculously high. The problem was too big.

Action: Check maximum number of layers.

Rtopnlsr

Message: Failed to open loose route file <xxx>.

Cause: Intermediate file (<filename>) could not be opened.

Action: If disk file or directory space is full, make space available.

Message: Error writing loose route header.

Cause: Disk was full or damaged.

Action: Make space available.

Message: Failed to read header for lsr file <xxx>.

Cause: A restart was requested but restart data was not found. The route solution file has been corrupted by the route error halt.

Action: Rerun the router without selecting the restart option.

Rtopnpnt

Message: Failed to open points file <xxx>.

Cause: Could not route solution file named <xxx>. The disk was full or a subdirectory was full.

Action: Make space available on the disk or in the subdirectory.

Message: Failed to write header for point file <xxx>.

Cause: Could not write first record to point file. Disk was full.

Action: Make space available.

Message: Failed to read header for point file <xxx>.

Cause: A restart was requested when there was no data to restart from. Route solution file (<filename>.RTS) was corrupted since previous route was interrupted. Previous route did not really get started.

Action: Start the router again without selecting the restart option.

Rtopnvbr

Message: Failed to open via barrier file <xxx>.

Cause: File PCRTVBAR.TMP could not be opened.

Action: Check for a full disk or too many (more than sixteen) files opened.

Rtordrsb

Message: Failed in allocating new subnet table entries <n>.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Message: Failed in allocating subnet sort table.

Action: Trace back message for a failure.

Message: Failed to free original subnet table.

Cause: System failure.

Action: Call P-CAD.

Rtothbar

Message: Barrier(s) out of the board outline.

Cause: A rectangle on a BAR layer was outside of the board outline.

Action: Correct the board outline or the offending barrier.

Rtpinmap

Message: Failed calling pinproc for pin <n> pds <n> at <n> <n>.

Action: Trace back message for a failure.

Rtpkfrto

Message: Failed calling rtfndprb for <n> <n>.

Action: Trace back message for a failure.

Rtprbgrd

Message: Failed in getting data for <n> <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtprblin

Message: Failed in prbgrd for <n> <n> <n>.

Action: Trace back message for a failure.

Message: Inputs to prblin are <n> <n> <n> <n> <n> <n>.

Action: Trace back message for a failure.

Rtprbout

Message: Failed in <xxx> for <n> <n> <n>.

Action: Trace back message for a failure.

Message: Route box is <n> <n> <n> <n>.

Action: Trace back message for a failure.

Message: Failed subnet due to lack of memory.

Cause: The memory allocation request was denied.

Action: Manually complete the unrouted nets in PC-CARDS.

Message: Failed subnet <n> due to lack of memory.

Cause: The memory allocation request was denied. The problem was too large.

Action: Reduce the problem size.

Rtpropwv

Message: Illegal routing algorithm <n>.

Cause: System failure.

Action: Call P-CAD.

Rtprpsbn

Message: No wire rule for subnet <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed calling rtpkfrto for subnet <n>.

Action: Trace back message for a failure.

Message: Failed rtgtinpn.

Action: Trace back message for a failure.

Message: Failed calling rtwrknds for subnet <n>.

Action: Trace back message for a failure.

Message: Rtblksub failed for subnet <n>.

Cause: System failure.

Action: Call P-CAD.

Rtpstrot

Message: Failed to get wiring rule <n> for subnet <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in retrace from <n> <n> <n>.

Action: Trace back message for a failure.

Message: Failed in adpntr for <n> <n> <n>.

Action: Trace back message for a failure.

Message: Failed in Adpntr for <n> <n> <n>.

Action: Trace back message for a failure.

Rtputmap

Message: Failed in xlation <n> <n> <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtputout

Message: Rtputout could not get another file record in memory.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Message: Rtputout could not open file <xxx>.

Cause: System failure.

Action: Call P-CAD.

Rtputwrd

Message: Rtputwrd index out of range indx =<n>, max =<n>, map id =<n>.

Cause: System failure.

Action: Call P-CAD.

Rtpwpnt

Message: Could not allocate PNT structure.

Cause: The memory allocation request was denied.

Action: Reduce the problem size.

Rtreadnt

Message: Failed trying to open problem file <xxx>.

Cause: The file <xxx> could not be opened:

1. The file did not exist (it was probably deleted since route);

or

2. The file was not of correct format.

Action: Check the status report (<filename>.REP) for errors. Make sure you are processing a correct design (most likely, if the file was not intentionally deleted, the router was not run).

Rtreadsb

Message: Failed trying to open problem file <xxx>.

Cause: The file <xxx> could not be opened:

1. The file did not exist (it was probably deleted since route);

or

2. The file was not of correct format.

Action: Check the status report (<filename>. REP) for errors. Make sure you are processing a correct design (most likely, if the file was not intentionally deleted, the router was not run).

Rtrebpns

Message: Failed to get pad current for pin <n> padstk <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in rt<xxx> <xxx> for subnet <xxx>.

Action: Trace back message for a failure.

Rtretrac

Message:	Failed in Getmap for <n> <n> <n>.
Cause:	System failure.
Action:	Call P-CAD.

Message:	Illegal direction found at <n> <n> <n> direction <n>.
Cause:	System failure.
Action:	Call P-CAD.

Rtroute

Message:	It is illegal to restart with different layering.
Cause:	The predominant direct for the layers was different in the routing strategy than when the router was previously started.
Action:	Modify the routing strategy so the description is consistent.

Rtroutrt

Message:	Failed allocating <n> pntrecs.
Cause:	The memory allocation request was denied. The problem was too large and this route had too many vertices.
Action:	Reduce the problem size. It would be very unusual to get through the router and fail in the return due to problem size. Please call P-CAD if this occurs.

Message:	Failed to open points file <xxx>.
Cause:	Could not open file with route solutions in it. Route solution file (<filename>.RTS) was deleted since route completed.
Action:	Start the router again without selecting the restart option.

Message: Failed to read header for point file <xxx>.

Cause: Read error encountered while reading route solution file <xxx>. The file was corrupted, system error.

Action: Restart the router (you do not have to reextract data).

Message: Failed to get currency on <xxx>.

Cause: Could get net <xxx> current in database. System error.

Action: Call P-CAD.

Rtrpterr

Message: HELP REPORT FILE CANNOT BE ACCESSED FROM RTRPTERR.

Cause: System error.

Action: Call P-CAD.

Rtrstfil

Message: Failed to open points file <xxx>.

Cause: A restart was requested when there was no route solution file. The route solution file (<filename>.RTS) was deleted since the previous route was interrupted. The previous route did not really get started.

Action: Start the router again without selecting the restart option.

Rtrstreg

Message: Read error on lsr stat <n>.

Cause: System failure.

Action: Check the status report (<filename>.REP). If there are no correctable errors, call P-CAD.

Message: Failed to get subnet <n> current.

Cause: System failure.

Action: Call P-CAD.

Rtrstsubs

Message: Read error on pnt stat <n>.

Cause: Failed restarting router; route solution file is empty.

Action: Start the router again, but do not choose the restart option.

Message: Failed to get subnet <n> current.

Cause: System failure.

Action: Call P-CAD.

Rtrtebox

Message: Failed put maprtbox.

Cause: System error.

Action: Call P-CAD.

Rtryaccl

Message: Failed from <n> <n> <n> <xxx> incr <n> to <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Backtracking 1.

Cause: System failure.

Action: Call P-CAD.

Message: Failed from <n> <n> to <n> <n> on lay <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in gother <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtsavedb

Message: Failed to save database <xxx> status <n>.

Cause: System error.

Action: Call P-CAD.

Rtsbefct

Message: Failed calling rtndefct for from of subnet <n>.

Action: Trace back message for a failure.

Message: Failed calling rtndefct for to of subnet <n>.

Action: Trace back message for a failure.

Rtscorpn

Message: Failed in rtgetmap block x <n> y <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in rtgetmap density x <n> y <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Failed in rtputmap penalty x <n> y <n>.

Cause: System failure.

Action: Call P-CAD.

Rtsetdln

Message: Failed from <n> <n> <n> <xxx> incr <n> to <n>.

Cause: System failure.

Action: Call P-CAD.

Rtsftblk

Message: Failed calling rtblksbs.

Cause: System failure.

Action: Call P-CAD.

Message: Failed calling rtblkpns.

Cause: System failure.

Action: Call P-CAD.

Message: Failed calling rtblkbss.

Cause: System failure.

Action: Call P-CAD.

Rtstrcmp

Message: Failed to allocate temporary buffer.

Cause: Memory allocation failed.

Action: Reduce the problem size and rerun.

Rttblfre

Message: Error freeing pin table.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing sub table.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing bus table.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing wire rule table.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing pad table.

Cause: System failure.

Action: Call P-CAD.

Message: Error freeing net table.

Cause: System failure.

Action: Call P-CAD.

Rtunprps

Message: Failed calling rtblksub for subnet <n>.

Action: Trace back message for a failure.

Message: Failed calling rtwrknds for subnet <n>.

Action: Trace back message for a failure.

Rtwrtunr

Message: Can't connect subnet <n>.

Action: Trace back message for a failure.

Message: Failed for <xxx> of subnet <n>.

Action: Trace back message for a failure.

Message: Failed in rtgclsp for subnet <n> close to <n> <n>.

Action: Trace back message for a failure.

Message: Failed getting can't connect <xxx> for subnet <n>.

Cause: System failure.

Action: Call P-CAD.

Rtxltmap

Message: X crd is too big to translate into map <n> <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Y crd is too big to translate into map <n> <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Message: Layer is too big to translate into map <n> <n> <n>.

Cause: System failure.

Action: Call P-CAD.

Rtxtnacc

Message: Backtracking 2.

Action: Trace back message for a failure.

Rtwrknds

Message: Failed (un)blking <n> <n> to <n> <n> subnet <n>.

Action: Trace back message for a failure.

Rtwrtunr

Message: Failed for <xxx> of subnet <n>.

Action: Trace back message for a failure.

Message: Failed in rtgtclsp for subnet <n> close to <n> <n>.

Action: Trace back message for a failure.

Message: Failed getting can't connect <xxx> for subnet <n>.

Action: Trace back message for a failure.

Rt1stbar

Message: Lseek error for layer <n> at position <n>.

Cause: System error.

Action: Call P-CAD.

Message: Read error on first bar lay <n> pos <n> stat <n>.

Cause: System error.

Action: Call P-CAD.

Rt1stpnt

Message: Read error on first pnt sub <n> pos <n> stat <n>.

Cause: System error.

Action: Call P-CAD.

Rt1streg

Message: Read error on first reg sub <n> pos <n> stat <n>.

Cause: System failure.

Action: Call P-CAD.

Rt1stvbr

Message: Had a read error on barrier file <n>.

Cause: System error.

Action: Call P-CAD.

Rt45adsn

Message: Memory limits exceeded - 45 degree route.

Cause: The problem was too large.

Action: Reduce the problem size.

Rt45route

Message: Memory limits exceeded - 45 degree route.

Cause: The problem was too large.

Action: Reduce the problem size.

Rxbldalp

Message:	Polygon not closed or redundant lines (or polygons).
Cause:	The board outline was not a closed polygon.
Action:	Fix the board outline.
Message:	Could not allocate the allpoints structure.
Cause:	The memory allocation request was denied.
Action:	Reduce the problem size.
Message:	Illegal board outline(s).
Cause:	The board outline can only be one rectangle or one closed polygon. Anything else on the BRDOUT layer will be an error.
Action:	Modify the board outline.
Message:	Polygon not closed; or redundant lines.
Cause:	The line describing the board outline did not end where it began.
Action:	Redraw the board outline.
Message:	Failed to free the allpoint structure.
Cause:	System failure.
Action:	Call P-CAD.
Message:	Can't free the block structure.
Cause:	System failure.
Action:	Call P-CAD.

Setnpcon

Message: Error in calling dmowncmp.

Cause: System failure.

Action: Call P-CAD.

Updnxblk

Message: Does not start w/PEC.

Cause: System failure.

Action: Call P-CAD.

Updnxlst

Message: PEC expected.

Cause: System failure.

Action: Call P-CAD.

APPENDIX B. SAMPLE STATUS REPORT

After PC-ROUTE completes processing, a status report file (<filename>.REP) is generated. This appendix contains a sample report. It lists the start and end time of each phase and any errors encountered during processing. It also lists the subnets that could not be connected by the router, the routing completion percentage achieved, the metal (trace length) laid per layer, and the number of vias used.

```
            P-CAD ROUTE EXTRACT REPORT

    Starting Time 8/13/1986 - 10:43:35

   Report for database test.plc using strategy file test.ctl

    Ending time  8/13/1986 - 10:43:06

     P-CAD ROUTE EXTRACT REPORT SUMMARY
  No errors encountered during run

          END OF P-CAD ROUTE EXTRACT REPORT


            P-CAD AUTO-ROUTER REPORT

  Start Time 8/13/1986 10:43:11
  Data Base  test.plc
  Strategy   test.ctl

                   Layers 1 and 2

                   Routing Results

  Pass        Attempted Completed      Start     Finish

  45 degree        21        20    10:43:30  10:43:45
  1                61        56    10:43:45  10:47:45
  2                 5         1    10:47:47  10:49:02

                   Resource Usage

                              Layer 1       Layer 2

  Metal Length                  46950         42300
  Metal Length (grids)            939           846
  Via Count                        77            77

  Total Subnets   81
  Routed Subnets  77
  Completion Rate 95%
  Elapsed Time    0:07:19

  No errors encountered during run

  End Time    8/13/1986 10:50:44

          END OF P-CAD AUTO-ROUTER REPORT


            P-CAD ROUTE RETURNER REPORT

    Starting time  8/13/1986 - 10:55:01

  Unrouted subnets
  NET NAME    REFDES           FIRST PIN          REFDES            SECOND PIN
  Net UN00025           pin 4  (     0,  1100)               pin 12 ( 1300,  1200)
  Net QC'               pin 6  (     0,  1950)               pin 15 ( 1100,   350)
  Net QA                pin 3  ( 1000,  1200)               pin 7  ( 1300,   450)
  Net QB                pin 4  ( 1500,  2150)               pin 14 ( 1000,   350)
    Ending time  8/13/1986 - 10:55:42


     P-CAD ROUTE RETURNER ERROR SUMMARY
  No errors encountered during run

          END OF P-CAD ROUTE RETURNER REPORT
```

APPENDIX C. CONVERTING A CONTROL FILE

To convert a control file that was created with an earlier than 1.35 version of PC-ROUTE, follow the steps below.

1. Type the "database name" on the PC-ROUTE Main Menu.

2. Select the "routing strategy" on the PC-ROUTE Main Menu.

3. Select the "Edit routing strategy" option on the PC-ROUTE Main Menu. The Routing Strategy Menu is displayed and the system displays the following message:

   WARNING

   <filename>.CTL was created with an old version of software. It will be converted to the present version but the wiring rules must be checked to make sure that the CLEARANCES are correct.

   NOTE: Version 1.35 of PC-ROUTE allows you to specify clearances for wiring rules in DBUs instead of grids. Therefore, when PC-ROUTE converts a control file, it converts the clearances to the PCAD1 routing strategy default, 12 DBUs. You should check the clearances on the Wiring Rules Menu to make sure these are the values you want to use.

4. Press any key to continue. The Routing Strategy Menu is displayed.

5. Select the "Edit Wiring Rules" option to review the clearances for the wiring rules.

6. Enter new values in the Wiring Rules Menu (if desired). Press [Esc] to return to the Routing Strategy Menu.

7. Now you can save the routing strategy or edit other routing parameters and then save it. To save the the routing strategy, press [Esc]; and the options for saving the strategy are displayed.

8. Select the "Save under current name" or the "Save under new name" option.

   NOTE: If you don't save the routing strategy, the control file conversion will not be saved. If you try to route using an unconverted control file, PC-ROUTE will abort and display the message:

   Fatal error RXRDCTL strategy file <XXX.CTL> is not up to date.

INDEX

M

N

O

P

T

U

V

W