



Master Designer

Version 8.6

What's New for MD 8.6

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means--electronic, mechanical, photocopying, recording, or otherwise--without the prior written permission of ACCEL Technologies, Inc.

ACCEL Technologies, Inc. provides this manual "as is," without warranty of any kind, either expressed or implied, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. ACCEL Technologies, Inc. may make improvements and/or changes in the product(s) and/or the program(s) described in this manual at any time and without notice.

Although ACCEL Technologies, Inc. has gone to great effort to verify the integrity of the information herein, this publication could contain technical inaccuracies or typographical errors. Changes are periodically made to the information herein. These changes will be incorporated in new editions of this publication.

Copyright © 1994-96 Altium, under exclusive rights to ACCEL Technologies, Inc. Derivative Copyright © 1996, ACCEL Technologies, Inc.

All trademarks and registered trademarks are properties of their respective companies.

Contents

Contents	3
About This Manual	5
Conventions	7
What's New 1	9
Graphic Editors.....	9
Improved Selection Mechanism	9
Improved Toggling Between Selected and Unselected Objects	11
Enhanced File Selectors.....	12
Design Manager	13
Standardizing Design Files.....	13
Acceptable Filename Characters.....	14
PCB Editor	14
Selecting Components By Name	14
Hot keys for Rotating Components	15
Hot keys for Zooming	15
New Net Attributes	15
Enhanced Ratsnest Display Performance.....	17
To set the dynamic ratsnest delay	17
Dimensions: Centered Horizontal Text	18
Deleting Unused Layers	18
To delete layers through the Delete Layer screen	19
To delete layers from the PDF File Writer screen.....	20
Updated Opcodes	22
Schematic Editor	23
Auto-Incrementing Reference Designators	23
To auto-increment reference designators	23
What's New in the Utilities 2.....	25
Design Rules Checking.....	25
Enhanced Support for Net Attributes	25
Graphics on the \$ATT Layer.....	28
UNIX Installation.....	28
Installing the Software on UNIX.....	28
Installing Master Designer Online Documentation	29
AIX Support for Online Documentation.....	29
Hardcopy	29
Scaling Apertures.....	29
CCT SPECCTRA Router Interface.....	29
PC-WinPlot	30
Plotting Multiple Plot Files (batch plotting).....	30
Defining Text Stroke Widths.....	31
Report Generator.....	32
To generate a Pin Information List.....	33
Updated Commands 3	35
To align a single component.....	37
To flip a component between the top and bottom layers ..	39
To add a net attribute	41

To change a net attribute value	41
To change a net attribute keyword.....	42
To delete a net attribute	42
Using This Command	45
To enter a component	45
Things to Remember	47
Status Area Options	48
Orthogonal Rotation Angles	48
Entering an Angle in the Response Area	49
The Component Name Box	49
Component Mirroring in the Schematic Editor	49
Selecting the Top or Bottom of a PCB	49
Automatic Component Naming	49
Auto-Incrementing Reference Designators.....	50
To auto-increment reference designators	50
Using the Picklist.....	51
Assigning Component Names	52
To fix the location of one or more components.....	55
To unfix one or more components	57
To change properties.....	59
To rotate a component	65
To restore rotated components to their original positions .	66
To control dynamic ratsnest display.....	69
New Commands 4.....	71
To set the dynamic ratsnest delay	73
To change to the old object selection method.....	75
Things to Remember	75
System Limits A	79
Filename Extensions B	83
Reserved Words C.....	87
Button Menu Trees D	91
Symbol Editor.....	93
Schematic Editor	94
Part Editor	95
PCB Editor	96
PCB Editor (con't).....	97
Command Cross Reference E.....	99
Index.....	107

About This Manual

This manual is for those of you who are upgrading to Master Designer 8.6. It explains all the new features and changes that have been made since Master Designer 8.5. By reading this manual, you'll be able to see at a glance what's new and different in Master Designer 8.6.

This manual consists of four chapters and five appendixes. A conventions page follows this section.

- | | |
|------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Chapter 1 | "What's New" describes what's new throughout Master Designer 8.6, such as an improved selection mechanism, enhanced file selectors, standard design files, new net attributes, selecting components by name, centering horizontal text, deleting unused layers, and auto-incrementing reference designators. |
| Chapter 2 | "What's New in the Utilities" describes what's new in the Master Designer 8.6 utilities, including enhanced net attribute support in DRC, AIX support for online documentation, scaling apertures, plotting multiple plot files, and a pin information list. |
| Chapter 3 | "Updated Commands" describes the commands updated for Master Designer 8.6. |
| Chapter 4 | "New Commands" describes the new menu and keyboard commands for Master Designer 8.6. |
| Appendix A | "System Limits" describes name length limits; maximums for database items such as pins, components, and nets; and ranges for other items. |
| Appendix B | "Filename Extensions" lists the P-CAD filename extensions and the tools that produce the files. |
| Appendix C | "Reserved Words" lists DOS reserved device names and P-CAD attribute keywords. |
| Appendix D | "Button Menu Tree" contains a graphical menu tree for the Button Menus. |
| Appendix E | "Command Cross Reference" contains a command cross-reference, a list of default hot-key functions, and a chart of the keys available for hot-key assignment. |

Conventions

This manual uses the following conventions:

→ or ! Connects commands. Commands following the arrow appear on submenus. For example

File!Load or *File→Load*

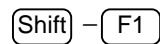
* One or more characters can occupy the asterisk's position. Also known as a wildcard.





In text, introduces a procedure that explains how to do a task.



Press the keys simultaneously. For example



The return key. Press this key after typing data in a data entry box or on a message line or to accept a default. You can click left in many P-CAD tools instead of pressing .

Enter or ↵ Indicates you need to press  after typing data.







The space bar. You can use this key to digitize a point within the drawing area.

italics Indicates variable characters.

boldface Indicates characters you enter from the keyboard; for example

Enter **sheet7** in the data entry box.

Conventions

<i>boldface italics</i>	Indicates variable characters you enter from the keyboard; for example
Click left	Enter <i>filename</i> in the data entry box. Press and release the left button (button 1) on the mouse.
Click middle	Press and release the middle button (button 2) on the mouse.
Click right	Press and release the right button (button 3) on the mouse. If you are using a two-button mouse, press  and button 2 simultaneously to clickright.
Cycle	Click left repeatedly on a cycle box until the item you want is selected. A cycle box is indicated by  .
Select	Move the cursor to an item or point and press  ,  , or click left.

P-CAD Master Designer 8.6 (MD 8.6) gives you the power and flexibility to create electronic circuit designs and PCB (printed circuit board) layouts from start to finish. With its many enhancements, however, MD 8.6 lets you complete your designs even faster than before.

Graphic Editors

Improved Selection Mechanism

With many commands, the selection mechanism now selects the object closest to the cursor on the active layer that is within snap tolerance. This makes it easier for you to predict which object will be selected first. The affected commands are:

- *Change Layer→Object*
- *Change Layer→Objects*
- *Copy→Object*
- *Copy→Objects*
- *Delete→Object*
- *Delete→Objects*
- *Delete→Trace*
- *Edit→Move Vertex*
- *Edit→Delete Vertex*
- *Edit→Move Segment*
- *Edit→Delete Segment*
- *Edit→Move Via*
- *Edit→Delete Via*
- *Edit→Segment Layer*
- *Edit→Delete Trace*
- *Edit→Trace Width*

- *Move→Object*
- *Move→Objects*
- *Query→Trace*
- *Query→Net*
- *Query→Object*
- *Query→Via*
- *Query→Properties*
- *Rotate→Object*
- *Rotate→Objects*

For those objects that Master Designer does not apply snap tolerance to, such as components, attributes, or polygons, Master Designer lets you select these objects when the cursor is within their boundaries.

If more than one object is within snap tolerance and at the same distance from the cursor, Master Designer prioritizes which objects to select. The order of priority is (from highest to lowest):

- objects on the active layer
- objects on the ABL layer
- objects on any layer set to ON
- components
- vias, in the following order of priority
 - objects on the active layer
 - objects on the ABL layer
 - objects on any layer set to ON
- nets, in the following order of priority
 - objects on the active layer
 - objects on the ABL layer
 - objects on any layer set to ON
- draw objects, in the following order of priority
 - objects on the active layer
 - objects on the ABL layer
 - objects on any layer set to ON

You can use the new */osel/* keyboard command to switch between the new selection method and to the selection method used in previous versions of Master Designer.

If your existing Master Designer macros require the old selection method, you can also use the */osel/* command in your existing Master Designer macros to switch between the old and new selection methods, as in the example below:

```
MACRO 850 -400
Opcode 192 0
Text osel
Text 1
B1 850 -450
B1 900 -750
Opcode 192 0
Text osel
Text 0
Opcode 192 0
Text mend
END_MACRO
```

In this example, the statements

```
Text osel
Text 1
```

activate the old selection method by setting *osel* to 1. Conversely, the statements

```
Text osel
Text 0
```

reactivate the new selection method by setting *osel* to 0.

Improved Toggling Between Selected and Unselected Objects

MD 8.6 features improved selection abilities for any command that selects groups of objects or windows of objects. Using any of these commands, you can now deselect any selected object. The first time you click on an object, MD 8.6 adds the object to the list of selected objects and indicates this by highlighting the object. The second time you click on it, Master Designer deselects it and removes it from the selection group. Deselected objects are unhighlighted as they are removed from the selection group. The affected commands are:

- *Change Layer→Objects*
- *Copy→Objects*
- *Delete→Objects*

- *Move→Objects*
- *Rotate→Objects*

If you're selecting a window of objects, Master Designer prompts you for another point after you select the window coordinates. To select or deselect an object, press the **Ctrl** key and click on the desired object. Repeat this process until you have selected or deselected all desired objects. Once you click on a point without pressing the **Ctrl** key, Master Designer accepts this as the requested point. The affected commands are:

- *Copy→Window*
- *Move→Window*
- *Rotate→Window*

If you use the *Delete→Window* command, then after you select the window coordinates, Master Designer displays the prompt:

OK to Delete? Yes, No or Modify

At this message, select *Yes*, *No*, or *Modify*. If you select *Yes*, Master Designer deletes all objects in the window. If you select *No*, Master Designer exits the command. If you select *Modify*, you can select or deselect more objects, then click Mouse Button 2 to redisplay the prompt.

If you use the *Chg. Layer→Window* command, then after you select the window coordinates, Master Designer displays the prompt:

OK to Change? Yes, No or Modify

At this message, select *Yes*, *No*, or *Modify*. If you select *Yes*, Master Designer changes all the layers of objects in the window to the active layer. If you select *No*, Master Designer exits the command. If you select *Modify*, you can select or deselect more objects, then click Mouse Button 2 to redisplay the prompt.

Enhanced File Selectors

In Master Designer 8.6, all file selectors and picklists let you change directories when prompting you for a filename. You can now either load a file from the current directory, or switch to another directory and load a file from there. The enhanced file selectors and picklists make it easier for you to load designs from different directories. This feature applies to all commands that use file selectors, such as *Environment→Attach Padstacks*, *Environment→Edit Aperture Table→Load Aperture Table*, and *File→Load*.

Design Manager

Standardizing Design Files

To support corporate design standards, the Design Manager lets you specify a single location for the following design files:

- special symbol files (.ssf)
- aperture table files (.apr)
- design rules files (.rul)
- check pass files (.pas)
- padstack files (.ps)
- NC drill table files (.tbl)

The Design Maintenance Screen, shown in Figure 1-1 now includes a System Directory field which lets you specify a System Directory where these design files reside. Master Designer adds this information to the pcd.cfg file. When you run any command that accesses any of the files listed above, such as *Environment→Attach Padstacks*, Master Designer automatically checks the System Directory for the appropriate file. If the file is not in the system directory, Master Designer checks the current design directory.

The System Directory field is a global option that applies to all design directories under which you run Master Designer. This is unlike the cross-reference file (.fil), which can be different for each design directory. When you change the System Directory field, it applies to all design directories.

Note: In the case of *Environment→Attach Padstacks*, Master Designer looks for padstack files in the same directory as the special symbol file, unless the special symbol file contains a path pointing to a different directory along with the padstack file name.

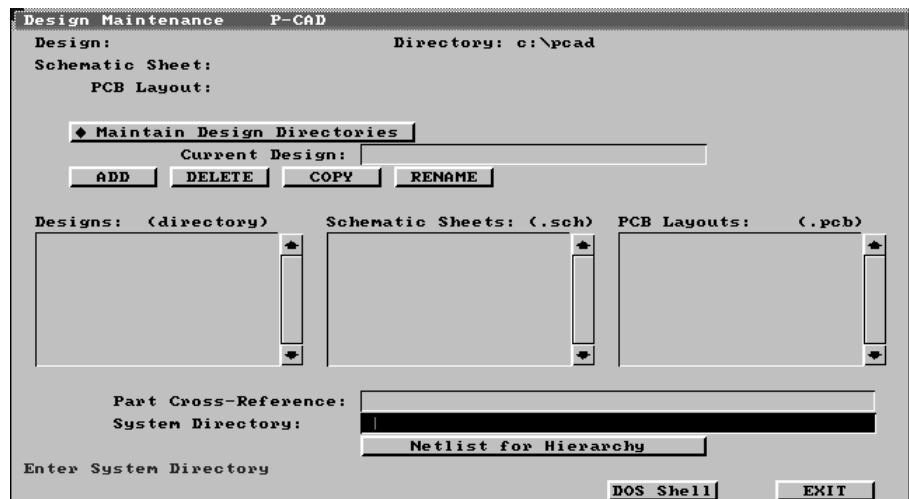


Figure 1-1: Design Maintenance Screen

Also, each time you create a new design, the Design Manager automatically copies the caps.cfg and cards.cfg files from the directory specified in the PCAD_CFG environment variable (either the system directory or the current design directory) to the new design directory.

Acceptable Filename Characters

Master Designer lets you use a wide range of characters for file names, part names, part numbers, and pin names and numbers. Table 1-1 lists the acceptable characters for file names.

Table 1-1. Acceptable Characters for Filenames

Alphanumeric	Acceptable Characters
File names	0-9 A-Z _ -
Part number	0-9 A-Z all characters except blank
Pin/net name	0-9 A-Z _ - ' ~
Pin number	0-9 A-Z

The characters : and . act as DOS functions and therefore can't be used in file names.

PCB Editor

Selecting Components By Name

More commands in Master Designer 8.6 let you select a component by name, just as the *Query→Component* command does, by

selecting the *ByName* button in the Status Area and entering a name. The affected commands are:

- *Rotate→Component* (both PCB and Schematic editors)
- *Chg. Layer→Component*
- *Align→Component*
- *Placement→Fix→Component*
- *Placement→UnFix→Component*

Chapter 3, "Updated Commands" explains how you can select components by name using these commands.

Hot keys for Rotating Components

Master Designer 8.6 features new hot keys for rotating components 90 degrees clockwise or counterclockwise. The hot keys for rotating components are:

- **Ctrl-R** - for rotating clockwise
- **Alt-R** - for rotating counterclockwise

These hot keys work with the *Move→Component* and *Enter→Component* commands.

Hot keys for Zooming

You can also use hot keys for zooming in and zooming out of the current cursor position. The hot keys for zooming are:

- **+** (plus) - for zooming in
- **-** (minus) - for zooming out

New Net Attributes

Master Designer 8.6 now includes new system net attributes to enhance Design Rules Checking on a net by net basis. You can enter these net attributes while in the Schematic Editor, then package them into a PCB design. Or you can just enter them directly into the PCB design in the PCB Editor. Then, when you run online DRC, the program can use these values to override the default clearances set for the nets that have net attributes attached to them. You can also use the net attributes in CCT SPECCTRA Router.

Table 1-2: New System Net Attributes

Net Attribute	Description
CLEARANCE	Overrides the general DRC (trace-to-trace, pad-to-trace, trace-to-board edge, and pad-to-board edge) spacing.
PAD-TO-EDGE	Overrides the pad-to-board edge DRC spacing.
PAD-TO-TRACE	Overrides the pad-to-trace DRC spacing.
TRACE-TO-EDGE	Overrides the trace-to-board edge DRC spacing.
TRACE-TO-TRACE	Overrides the trace-to-trace DRC spacing.
VIATYPE	Specifies the via type to be used during the <i>Enter→Wire</i> command.

The *Edit→Net Attribute* and *Query→Properties* commands can be used to edit or check the values of the net attributes. The commands will check that their values are valid.

The Net Attributes dialog box features a list box containing keywords for all of the system net attributes. To use any of these net attributes, select the desired keyword from the keyword list box, or you can just enter a net attribute keyword and value. If you select a net attribute keyword from the keyword list box, Master Designer copies the keyword to the Keyword edit box and prompts you for a new value, as shown in Figure 1-2:

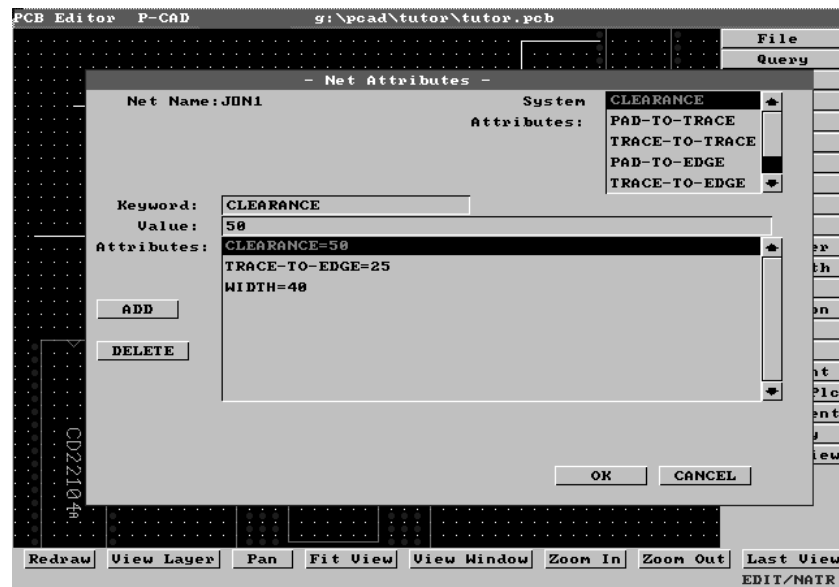


Figure 1-2: Net Attributes dialog box

With any net in your design, you can use any of the net attributes separately or in any combination with each other.

Also, when you run DRC, system net attributes overrule DRC's own default values, in the following order of importance.

1. net attributes TRACE-TO-TRACE, PAD-TO-TRACE, TRACE-TO-EDGE, and PAD-TO-EDGE, all of which have equal priority and override the CLEARANCE net attribute
2. CLEARANCE net attribute, which overrides the default DRC spacing values
3. default DRC spacing values

See the Design Rules Checking section of Chapter 2 for more details about how DRC supports the new net attributes.

Enhanced Ratsnest Display Performance

Master Designer 8.6 features an improved ratsnest display. In addition to turning dynamic ratsnest display on and off, you can now use the `/dly` command to configure the time delay for the ratsnest rubberband. This feature is useful because it lets you adjust the ratsnest display to better suit how you want the system to perform.

To set the dynamic ratsnest delay

1. Enter `/dly`. The system prompts

Dynamic Ratsnest Delay=n. New value=

2. Enter a number between .01 and 1 (in seconds). Or you can press **↵** to accept the current value.

Entering a number between .01 and 1 sets the dynamic ratsnest delay (in seconds) at that number.

The */dly* keyboard command applies to these commands:

- *Enter→Wire*
- *Edit→Enter Wire*
- *Move→Component*

Dimensions: Centered Horizontal Text

Master Designer 8.6 now displays and plots horizontal text that is centered between vertical dimension lines. To do this, the system calculates the space between vertical dimension lines and places the dimension text in the center of that space.

Deleting Unused Layers

Master Designer 8.6 lets you delete unused layers while running PDF Writer or the new Delete Unused Layers program in the PCB and Schematic Tools. Now you can have Master Designer automatically delete unused layers for you. Both the Schematic Tools module and PCB Tools module now feature a Delete Layers button. Clicking this button displays the Delete Layer screen, shown in Figure 1-3:

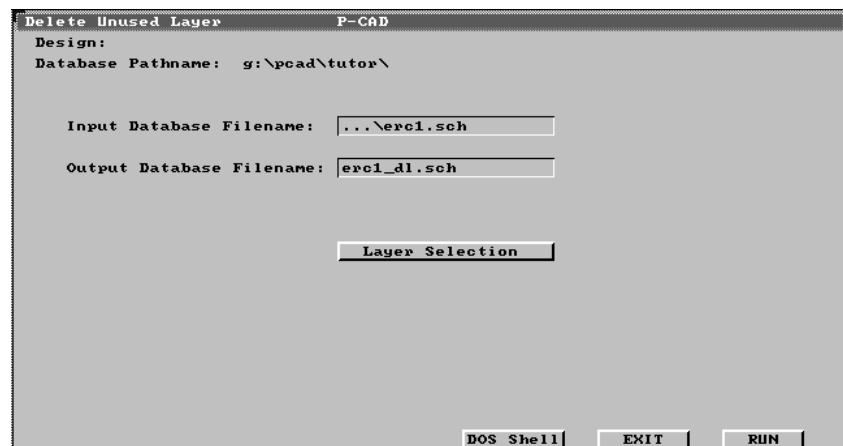


Figure 1-3: Delete Unused Layer screen

This screen prompts you for the following information:

- the input database, which contains the layers you want to delete
- the output database, which is new filename for the modified database
- the layers to delete

The screen also has four buttons:

- Layer Selection, which displays the Layer Selection pop-up window
- RUN, which runs the Delete Layer program
- Exit
- DOS Shell, which takes you to the DOS prompt

To delete layers through the Delete Layer screen

1. In the PCB Tools module, select *Delete Layer*. The Delete Layer screen appears.
2. In the Input Database field, enter the name of the database containing the layers you want to delete.
3. In the Output Database field, enter the name of the new database. By default, Master Designer adds a “_dl” to the filename before the extension. If the filename is too long, the system truncates it.

You must specify both an input and output database filename before selecting your layers. Otherwise, Master Designer displays a warning message.

4. Click the *Layer Selection* button. A pop-up window appears, displaying all the layers in the PCB or Schematic design. Figure 1-4 displays the Layer Selection pop-up window.

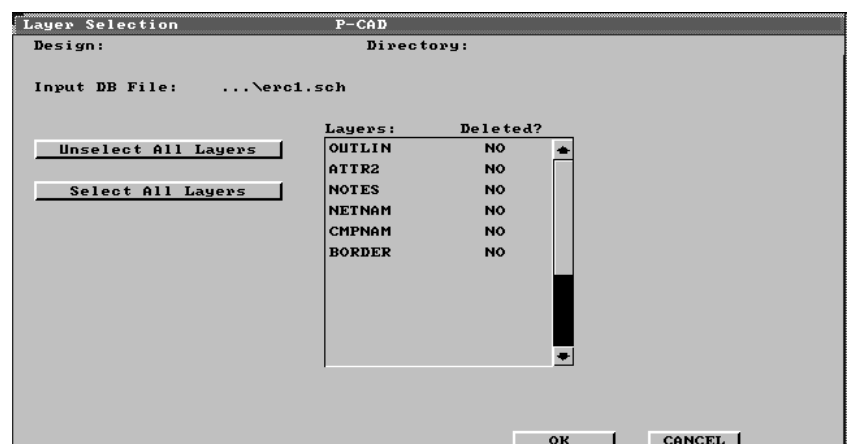


Figure 1-4: Layer Selection pop-up window

5. Select the layers to you want to delete.

or

Click *CANCEL* if you don't want to delete any layers. Clicking *CANCEL* returns you to the Delete Layer screen.

6. Select *RUN*. Master Designer edits the input database, generates a PDIF file, deletes the unused layers, and saves the PDIF file as a new database, using the name specified in the Output Database field. The input database is unchanged.

You must click *Layer Selection* and select layers to delete. Otherwise, when you select *RUN*, Master Designer displays a warning message indicating layers have not been selected.

7. Select *EXIT* when done.

You can also delete layers from within the PDIF File Writer screen. The difference with this process is that Master Designer creates a PDIF file and deletes the specified layers from this file, but does not regenerate a Master Designer database under a new name. You must use the PDIF File Reader to do this.

The PDIF File Writer screen features an option for deleting unused layers, as shown in Figure 1-5:

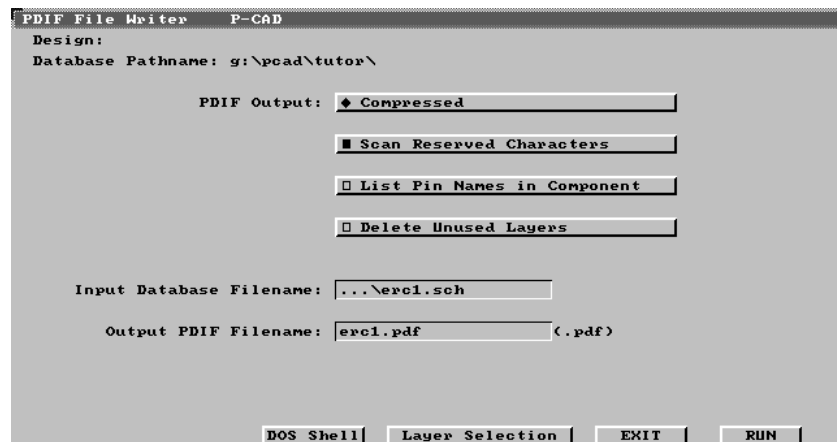


Figure 1-5: PDIF File Writer Screen

To delete layers from the PDIF File Writer screen

1. In the PCB Tools module, select PDIF File Writer. The PDIF File Writer screen appears.
2. Check *Delete Unused Layers*. Then click the *Layer Selection* button. A pop-up window appears, displaying all the layers in the PCB or Schematic design.
3. Select the layers to you want to delete.

or

Click *CANCEL* if you don't want to delete any layers. Clicking *CANCEL* returns you to the PDIF File Writer screen and deselects the *Delete Unused Layers* option.

4. Select *RUN*. Master Designer generates a PDIF file, deletes the unused layers, and saves the modified PDIF file. The original database is unchanged.

You must click *Layer Selection* and select layers to delete. Otherwise, when you select *RUN*, Master Designer displays a warning message indicating layers have not been selected.

5. Select *EXIT* when done.

Note: In a Schematic database, you can delete any unused layer, including autoswitching layers. An autoswitching layer is a layer to which Master Designer automatically stores certain data, then makes the active layer. For example, if you enter a pin name, Master Designer automatically enters the pin name on the PINNAM layer and makes the PINNAM layer the active layer. If you delete an autoswitching layer in the Schematic Editor, Master Designer stores any data meant for that layer on the current active layer. For example, if you delete the PINNAM layer and have the layer ATTR as the active layer, the system puts the pin name on the ATTR layer.

Updated Opcodes

In macros, several status area options and Master Designer commands are represented by numbers called opcodes. The list of opcodes has been updated to reflect the changes in Master Designer 8.6. Table 1-3 lists the opcodes and their meanings.

Table 1-3: Updated Opcode List

<i>Opcode</i>	<i>Operation</i>
51	Set net name.
55	Select <i>ByName</i> option.
RCW 58	Rotate component clockwise.
RCCW 59	Rotate component counterclockwise.
60	Set component name.
62	Select component orientation (toggle).
63	Select component mirror (toggle).
65	Select <i>ByName</i> when using <i>Enter→Replace Component</i> .
66	Copy pin name.
110	Select show grid toggle.
111	Select snap to grid toggle.
112	Select XY option.
115	Select snaptopin (toggle).
120	Set pin type.
121	Set pin logical equivalence.
127	Set attribute option when using <i>Enter→Replace Component</i> .
129	Text scale edit field for <i>Enter→Component</i> .
134	Component scale edit field for <i>Enter→Component</i> .
140	Set text size.
141	Set text orientation (toggle).
142	Select text mirror (toggle).
143	Mask components (toggle).
144	Mask wires (toggle).
145	Select text horizontal justification (toggle).
146	Select text vertical justification (toggle).
150	Select <i>After/Before/Split</i> (toggle when using <i>Crit. Path→Add Gate</i>).
151	Select <i>Crit. Path→Add Gate</i> path name.
152	Select <i>On/Off</i> (toggle when using <i>Display→Path/Group Visibility</i>).
153	Select <i>Path/Group</i> (toggle when using <i>Display→Path/Group Visibility</i>).
155	Select <i>ENTER</i> button for any screen.
156	Select <i>LOAD</i> button for any screen.
157	Select <i>CANCEL</i> button for any screen (<i>File→Load</i> , <i>Enter→Component</i> , message area).
158	Select <i>PICKLIST</i> button in status area.
159	Select <i>ADD</i> button for any screen.
160	Select <i>DELETE</i> button for any screen.

<i>Opcode</i>	<i>Operation</i>
161	Select <i>OK</i> button for any screen (response area).
167	Select a file in Picklist screen.
179	Edit the current file in the Picklist screen.
183	Select <i>EXIT</i> button for any screen.
184	Select the <i>Add Layer</i> button in <i>View Layer</i> .
187	Select line style (toggle).
190	Press Esc .
192	/ (initiate keyboard command).
196	Press Backspace (display last message).
197	Press X to toggle cursor crosshair display.
198	Resume execution from file.
199	Interrupt command file.
ZINC 208	Zoom in at current cursor position.
ZOTC 209	Zoom out at current cursor position.
219	Dynamic ratsnest display delay.
227	Pan with Shift -left mouse button.
274	Old selection method.

Schematic Editor

Auto-Incrementing Reference Designators

Master Designer 8.6 can now automatically assign reference designator and section numbers to symbols as you enter them in the Schematic Editor. The system also remembers the last reference designator and section number entered for each symbol. This feature is useful because it saves you from having to manually specify the reference designator and section numbers after entering the symbol. You also don't have to worry about remembering the last reference designator and section numbers entered. Master Designer increments them automatically.

To auto-increment reference designators

1. Select *Enter→Component*. The system prompts
Enter component name:
2. Enter the name of the desired component, or select *PICKLIST* to choose the component from the picklist. Remember that you can change directories if the desired component is in a different directory.
3. Click *ENTER*.

After you enter the part name in the response area or choose a part from the picklist, the system displays the component outline. Master Designer also displays a *Refd* checkbox in the status area and prompts

Select location to place *filename*. (Orientation OK?)

4. Change the status area options as needed.
5. Click the *Refd* checkbox. A button marked *unnamed* appears next to the *Refd* checkbox. This is the Reference Designator button, which you use to assign a reference designator and section number to a component.

If the design already has a component of the same type with a reference designator and section name assigned, then the Reference Designator button displays the next available reference designator and section for that component type (for example, U2/B, U3/C).

6. Click the Reference Designator button. The system prompts

Component name

7. Enter a valid reference designator and section number (for example U1/A). The name you entered now appears on the Reference Designator button. You can click this button again to change the reference designator before placing the component.
8. Select the location of the component. Master Designer places the component and displays its reference designator and pin number information.
9. Repeat step 8 for each component you want to place. As you place each component, the system increments the Reference Designator button, indicating the reference designator and section of the component you're about to place.
10. Click *CANCEL* or select another command when done.

What's New in the Utilities

2

This chapter describes the enhancements to following MD 8.6 utilities

- Design Rules Checking
- UNIX Installation
- Hardcopy
- CCT SPECCTRA Router Interface
- PC-WinPlot
- Report Generator

Design Rules Checking

Enhanced Support for Net Attributes

Master Designer 8.6 lets you override default DRC spacing values with net attributes assigned to nets or a single net in the PCB design file. When you run DRC, it prioritizes net attributes over the values specified in the design rules file, in the following order of importance.

1. net attributes TRACE-TO-EDGE, TRACE-TO-TRACE, PAD-TO-TRACE, and PAD-TO-EDGE, all of which have equal priority and override the CLEARANCE net attribute
2. CLEARANCE net attribute, which overrides default DRC spacing values
3. default DRC spacing values

The following sample scenarios show how DRC checks your design without and with net attributes.

- **Scenario 1: No Net Attributes:** In this scenario, DRC uses the default DRC spacing values specified in the design rules file.
- **Scenario 2: CLEARANCE net attribute specified:** In this scenario, DRC uses the CLEARANCE net attribute value to override the DRC spacing values in the design rules file.
- **Scenario 3: TRACE-TO-EDGE, or TRACE-TO-TRACE, or PAD-TO-TRACE, or PAD-TO-EDGE net attribute**

specified: In this scenario, each net attribute overrides a specific DRC design rule spacing value as it relates to the system net attributes. All other values set by the CLEARANCE net attribute are still valid.

- **Scenario 4: TRACE-TO-EDGE and CLEARANCE net attributes specified:** In this scenario, TRACE-TO-EDGE overrides the trace-to-board edge spacing value, and CLEARANCE overrides the other DRC design rule spacing values, except for trace-to-board edge.

The Design Rules error report now includes a NET ATTRIBUTE ERRORS section, formerly the WIDTH ATTRIBUTE ERRORS section. This section lists the violations for all defined net attributes, as shown in the sample report below:

```
*****
* Program : PC-DRC VERSION 8.6 *
* Date : Aug 15 1996 12:38:35 PM *
* Database : x.pcb *
*****
Net ATTRIBUTES:
-----
Net Name
-----

NET1          WIDTH = 15
SIG1          TRACE-TO-TRACE = 40
              PAD-TO-TRACE = 10
              VIATYPE = 10

SIG2          WIDTH = 20
              TRACE-TO-TRACE = 50

SIG3          WIDTH = 50
              VIATYPE = 51

*****
* Program : PC-DRC VERSION 8.6 *
* Date : Aug 15 1996 *
* Database : x.pcb *
* Pass : pass_1 *
*****
LAYERS GROUPED:
-----
COMP
PADCOM
FLGCON

DESIGN RULES SET:
-----
Pad size..... 60
Nonround pad size..... 60
Via size..... 50
Trace width..... 12
Pad to pad spacing..... 10
Pad to trace spacing..... 10
Trace to trace spacing..... 10
Pad to board edge spacing..... 125
Trace to board edge spacing..... 125

INNER PLANE CONNECTIVITY:
-----
APERTURE VALUES      NETS
-----
1
2
3
4
10                     NET1 GND
```

Figure 2-1. Sample Design Rules Error Report

	TYPE	VALUE	ITEM	X1	Y1	X2	Y2
1	SIZE	7	TRACE	1000	3533	940	3593
2	SIZE	7	TRACE	940	3593	760	3593
3	SIZE	8	TRACE	760	3593	700	3533
4	SIZE	7	TRACE	700	3533	700	2833
5	SPACE	0	TRACE	680	993	660	1013
			PAD	670	1003	730	1063
6	SPACE	0	TRACE	680	993	660	1013
			PAD	670	1003	730	1063
7	SPACE	6	TRACE	660	1013	660	1133
			PAD	670	1003	730	1063
8	SPACE	6	TRACE	660	1013	660	1133
			PAD	670	1003	730	1063

NET ATTRIBUTE ERRORS:

	NET NAME	VALUE	X1	Y1	X2	Y2
WIDTH attribute errors:						
1	NET1	12.00	-440.00	-130.00	-400.00	-130.00

TRACE-TO-TRACE attribute errors:

No trace to trace attribute errors were found on this pass.

PAD-TO-TRACE attribute errors:

No pad to trace attribute errors were found on this pass.

PAD-TO-EDGE attribute errors:

No pad to edge attribute errors were found on this pass.

TRACE-TO-EDGE attribute errors:

No trace to edge attribute errors were found on this pass.

VIATYPE attribute errors:

No viatype attribute errors were found on this pass.

MESSAGES:

1. Layers not found:
PADVIA
2. No special symbol found - (uncommitted pin).
Location: 4688, 2410 Type: 6
3. No special symbol found - (committed pin).
Location: 21872, 24527 Type: 20

STATISTICS:

Number of pads processed	851
Number of vias processed	4
Number of traces processed	50
Number of other shapes processed	2
Number of pad size errors	0
Number of via size errors	0
Number of VIATYPE attribute errors	0
Number of trace width errors	4
Number of WIDTH attribute errors	1
Number of pad to pad errors	0
Number of pad to trace or shape errors	4
Number of PAD-TO-TRACE attribute errors	0
Number of trace to trace or shape errors	0
Number of TRACE-TO-TRACE attribute errors	0
Number of pad to board edge errors	0
Number of PAD-TO-EDGE attribute errors	0
Number of trace or shape to board edge errors	0
Number of TRACE-TO-EDGE attribute errors	0
Total number of entities processed	907
Total number of size violations	4
Total number of spacing violations	4
Total number of violations	8

Time finished: 12:45:03PM

Figure 2-1. Sample Design Rules Error Report (cont'd)

In this report, only the WIDTH attribute has any errors, which are noted in the NET ATTRIBUTE ERROR section, and listed in the STATISTICS section. DRC also indicates if there are no net attribute errors. Also, when reporting any VIATYPE net attribute errors, DRC reports only the first location of each VIATYPE net

attribute error for each via type value that is different from the value defined by the VIATYPE net attribute.

Note: In order to run DRC correctly, you must have an aperture table defined and attached to your database. You must also have a padstack attached to your database to run DRC correctly. For more information on how to run DRC correctly, see *PCB Tools, Volume 2*, Chapters 5 and 6.

Graphics on the \$ATT Layer

DRC marks any net attribute violations by highlighting them in magenta and adding graphics to the \$ATT layer. The net attribute graphics on the \$ATT layer are represented differently, depending on the type of violation, but they are the same size as the objects in the database. The list below indicates how DRC represents each type of violation.

- VIATYPE - solid boundary of the shape
- PAD-TO-TRACE - a dashed-outline of a circle and a dashed line
- TRACE-TO-TRACE - two dashed lines
- TRACE-TO-EDGE - a dashed line and a solid line representing the board edge
- PAD-TO-EDGE - a dashed boundary of a pad shape and a solid line representing the board edge

If there are no violations, DRC displays a message indicating it found no violations.

UNIX Installation

Installing the Software on UNIX

You can now install Master Designer 8.6 for UNIX from a CD-ROM. Instead of the current, two-step process of copying an install script and running it to install Master Designer from tape, you run an install program directly from the CD-ROM. This installation program extracts the tar files from the CD, lets you select the programs to install, and installs them to your UNIX system. See the *Installation Guide - UNIX Systems* for more details.

Installing Master Designer Online Documentation

Master Designer 8.6 now features a Master Index for the entire online documentation set. This file resides on the CD-ROM in the online documentation directory. You can view it using Adobe Acrobat Reader.

AIX Support for Online Documentation

Master Designer 8.6 now includes the latest Adobe Acrobat Reader (version 2.1) for AIX, which lets AIX users view the online documentation.

Hardcopy

Scaling Apertures

Hardcopy now lets you scale apertures when generating pen plots from your Master Designer plot file. To do this, use the "Plot" format in *Hardcopy* and select the Plot Scale field on the Hardcopy screen. The *Hardcopy* screen specifies the ratio of the hardcopy plot image size to the original database image size and aperture size, which Master Designer reads from the aperture table file.

Note: This feature applies to HP pen-plotters and Postscript printers only. To scale apertures using any other plotter, use PC-WinPlot.

This feature does NOT apply to photoplotting. That is, you cannot set the plot scale for photoplots. If you try this, everything in the photoplot scales except the apertures.

CCT SPECCTRA Router Interface

Master Designer's CCT SPECCTRA Interface now supports net attributes and uses them in the SPECCTRA router. With this support, the router can use net attributes to override general spacing design rule values. When you send a design with net attributes to the SPECCTRA router, SPECCTRA writes the appropriate net attribute information to the .do file.

For example, suppose a design has a net SIG20 with a net attribute TRACE-TO-TRACE=60. The TRACE-TO-TRACE net attribute

overrides the wire-to-wire spacing value. Therefore, SPECCTRA adds the line

```
rule net SIG20 (clear 60(type wire_wire))
```

to the design's .do file.

Note: Master Designer now uses the .do file extension instead of .pdo for the DO filename extension.

The table below lists the supported net attributes:

Net Attribute	Description
CLEARANCE	Overrides the general DRC (trace-to-trace, pad-to-trace, trace-to-board edge, and pad-to-board edge) spacing.
PAD-TO-EDGE	Overrides the pad-to-board edge DRC spacing.
PAD-TO-TRACE	Overrides the pad-to-trace DRC spacing.
TRACE-TO-EDGE	Overrides the trace-to-board edge DRC spacing.
TRACE-TO-TRACE	Overrides the trace-to-trace DRC spacing.

The CCT SPECCTRA Interface does not support the VIATYPE net attribute.

PC-WinPlot

Plotting Multiple Plot Files (batch plotting)

PC-WinPlot now lets you select and plot multiple plot files in a single session. You need only configure PC-WinPlot once, select the desired plot file(s), then select your output device (printer or plotter). You can also deselect plot files from the list before plotting. To plot multiple plot files with PC-WinPlot, you must

- configure PC-WinPlot
- select the plot files
- select the output device

You can select a set of plot files at the Open dialog box by using the standard Windows selection key combinations. The selection set can be either a single file, a group of files, or all files in the file list. The table below explains how to select files for batch plotting.

Use this mouse-key combination	To select
Left mouse button	a single file
Shift -Left mouse button	all files between the last selected (highlighted) file and the file you click on
Ctrl -Left mouse button	add single files to or subtract single files from the selection set

After selecting your plot file(s), click **OK** to continue. You can always select **File Open** again to see which plot files you selected.

Before batch plotting, you should check the files you wish to plot to make sure they will meet the desired configuration, especially your scale setting. If a selected plot file doesn't conform to the printer setup, PC-WinPlot notifies you of this discrepancy and skips to the next plot file. Suppose that you select the Limit to One Page option at the Configuration dialog box, then set the Plot Scale to 3.5 for the first of a series of plot files. If the third plot file can only be fitted to a page up to a scale of 2.0, PC-WinPlot skips this file and moves on to the next plot file. However, this does not apply if you set the Fit to Page option. In this case, PC-WinPlot automatically sets the scale so that each plot is either enlarged or reduced to fit on exactly one page.

When you're batch plotting, PC-WinPlot displays a message in the Status Bar indicating that the program is in batch plot mode. See the PC-WinPlot Online Help for more details.

Defining Text Stroke Widths

PC-WinPlot also lets you define the text stroke width for zero-width text lines. You can specify a value between 0 and 100 mils. See the PC-WinPlot Online Help for more details.

Report Generator

Master Designer now lets you generate a Pin Information List for your PCB design. The Pin Information List can be run either from the command line or in the Report Generator in the PCB Tools module. The Pin Information List has the file extension .pin.

The report includes the following information:

- the names and reference designators of all components
- the pins attached to that component
- the pin number
- the X-Y location of the pins
- the net connected to each pin

If a pin has no nets attached to it, the NETNAME field has a "?". The figure below shows an example of a Pin Information List.

```

%*****
%
% Program : PC-PIN   VERSION 8.6
% Date   : Aug 15 1996
% Time   : 10:16:33 AM
% File In : inter.pcb
% File Out : inter.pin
% Format  : PIN INFORMATION LIST
%
%*****
%
%
%
% COMPONENT/
% REF-DES
%
% PIN      LOCX      LOCY      NETNAME
%
%
% 74LS00:U23
%      1      100.00      200.00      NETNAME1
%      2      200.00      200.00      NETNAME2
%
% 74LS00:U50
%      1      100.00      200.00      NETNAME11
%      2      200.00      200.00      NETNAME22
%
% 74LS00:U60
%      1      500.00      500.00      NETNAME3
%      2      600.00      600.00      NETNAME4
%      3      700.00      700.00      NETNAME5

```

Figure 2-2. Pin Information List

To generate a Pin Information List

- At the DOS prompt, enter

PCPIN [*options*] <*PCB filename*> [<*output pinlist*>]

where

PCPIN executes the program.

options is one of the options listed below.

- ◇ *-h* displays the help message, which indicates how to use the program. The *-h* is optional.

PCB filename specifies the PCB design used to generate the Pin Information List.

output pinlist specifies the output filename for the Pin Information List (.pin).

As an example, the command line

PCPIN test1.pcb test1.pin

runs the Pin Information List program, generates a report of all pins in the PCB design test1.pcb, and saves the report to the file test1.pin.

Updated Commands

3

This chapter describes the updated commands for MD 8.6. Menu commands are discussed first, followed by keyboard commands. The updated commands are:

- *Align→Component*
- *Change Layer→Component*
- *Edit→Net Attr*
- *Enter→Component*
- *Placement→Fix→Component*
- *Placement→UnFix→Component*
- *Rotate→Component*
- *Query→Properties*
- *Rotate→Component*
- */dyn*

PCB Editor

Align**Component**

Aligns one component with a reference point.

Using This Command

To align a single component

1. Select *Align→Component*. The response area prompts

Select component.

2. Select the component.

or

Select *ByName* in the status area and enter the component name in the response area.

The system highlights the component and the response area prompts

Select alignment point.

If the component you want to select doesn't exist, Master Designer displays a warning message and prompts you again to select a component.

3. Cycle to *Horiz* or *Vert* in the status area.
4. Select the point with which you want to align the component. The system aligns the origin of the component with the alignment point.
5. Select another component.

or

Select another non-nested command to exit *Align→Component*.

Things to Remember

The alignment commands move objects along one axis at a time.

You cannot select a fixed component. If you do, the system displays the message

Component is fixed

In this case, select another component that isn't fixed.

Wires attached to aligned components won't rubberband, and the nets become disjointed.

Align

Component

To align components in a column, cycle to *Horiz* in the status area, and select an alignment point to the right or left of the column.

To align components in a row, cycle to *Vert* in the status area, and select an alignment point above or below the row.

The system aligns the origin of the component (usually pin 1) with the alignment point you select.

If alignment results in overlapping components, use *Move→Component* to make corrections, or use *Align→Undo*.

PCB Editor

Chg. Layer

Component

Flips a component between the paired top and bottom graphics layers of the board.

Using This Command

To successfully flip a component between layers, the system requires correct layer pairing. Refer to “Things to Remember”.

To flip a component between the top and bottom layers

1. Select *Chg. Layer→Component*. The response area prompts
Select a component.
2. Select a component.
or
Select *ByName* in the status area and enter the component name in the response area.
3. The system highlights the component, flips it to the other side of the board, and prompts you to select another component.

If the component you want to select doesn't exist, Master Designer displays a warning message and prompts you again to select a component.

If you attempt to flip a through-hole component without first correctly pairing the layers, the response area displays
Comp can't be flipped
4. Select another component to flip.
or
Select a non-nested command to exit *Chg. Layer→Component*. Refer to Things to Remember (next section) for correct layer pairing.

Things to Remember

Chg. Layer→Component applies only to SMT components or through-hole components whose layers are paired correctly in the PCB design file.

Use *Environment→Assign Layer Pairs* to verify that the component's graphics have been assigned to an appropriate layer pair. Refer to the *Environment→Assign Layer Pairs* description or the PCB Tools manual for a list of appropriate layers.

Chg. Layer

Component

You cannot select a fixed component. If you do, the system displays the message

Component is fixed

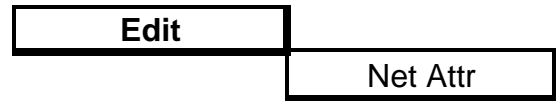
In this case, select another component that isn't fixed, or select *Placement→Unfix→Component*.

Three restrictions apply to *Chg. Layer→Component*:

- Graphics layers for the top and bottom of the board must first be paired with the *Environment→Assign Layer Pairs*. At least one pair of graphics layers in the part must be paired. For example, if you want to flip an SMD, the SLKTOP and SLKBOT layers are paired. If you want to flip a through-hole component, you need to pair one of the component's graphics layer with a layer on the bottom of the board. For example, you can pair a through-hole component's SLKSCR layer with the SLKBOT layer.
- No wires can connect to the component at the time you flip it.
- The system doesn't flip components between stitchable layers. Stitchable layers are layers reserved for traces.

For more information on mapping layer pairs, see *Environment→Assign Layer Pairs*.

PCB Editor
Schematic Editor



Lets you add, modify, and delete net attributes.

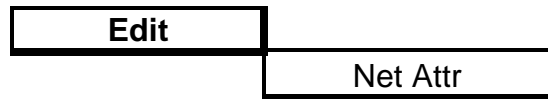
Using This Command

To add a net attribute

1. Select *Edit*→*Net Attr*. The system prompts
Select a net
2. Select the desired net.
or
Select *ByName* in the status area and enter the net name in the response area. The Net Attributes dialog box appears.
3. Click *ADD*. The system highlights the Keyword field.
4. Type the attribute keyword in the keyword field, then press Enter or Tab. The system highlights the Value field.
or
Select the keyword from the system keywords list box. The system copies the selected keyword to the Keyword edit box and prompts you for a value.
5. Type the attribute value, then press Enter or Tab. The system adds the new attribute to the list.
6. Click *OK* to save your changes, or click *CANCEL* to abort.

To change a net attribute value

1. Select *Edit*→*Net Attr*. The system prompts
Select a net
2. Select the desired net.
or
Select *ByName* in the status area and enter the net name in the response area. The Net Attributes dialog box appears.
3. Scroll to the desired attribute from the Attributes list box. The system displays the selected attribute keyword and its value in the Keyword and Value fields, respectively.
4. Click on the Value field and enter the new value.
or



Select the keyword from the system keywords list box. The system copies the selected keyword to the Keyword edit box and prompts you for a value.

5. Repeat steps 1 through 4 to edit another attribute value.
6. Click *OK* to save your changes or click *CANCEL* to abort.

To change a net attribute keyword

1. Select *Edit→Net Attr*. The system prompts
Select a net
2. Select the desired net.
or
Select *ByName* in the status area and enter the net name in the response area. The Net Attributes dialog box appears.
3. Scroll to the desired attribute from the Attributes list box. The system displays the selected attribute keyword and its value in the Keyword and Value fields, respectively.
4. Click on the Keyword field and enter the new value.
5. Repeat steps 1 through 4 to edit another attribute keyword.
6. Click *OK* to save your changes or click *CANCEL* to abort.

To delete a net attribute

1. Select *Edit→Net Attr*. The system prompts
Select a net
2. Select the desired net. The Net Attributes dialog box appears.
or
Select *ByName* in the status area and enter the net name in the response area. The Net Attributes dialog box appears.
3. Scroll to the desired attribute from the Attributes list box and select it. The system displays the selected attribute keyword and its value in the Keyword and Value fields, respectively.
4. Click *DELETE*. The system deletes the selected attribute.
5. Click *OK* to save your changes, or click *CANCEL* to abort.

Edit**Net Attr**

Things to Remember

The *Edit→Net Attr* command checks the values of all supported system net attributes which include WIDTH, CLEARANCE, TRACE-TO-TRACE, PAD-TO-TRACE, TRACE-TO-EDGE, PAD-TO-EDGE, and VIATYPE.

If you change the value of a system attribute, Master Designer checks the new value to see if it's valid. If the value is either invalid or out of range, the system displays the warning message

Attribute out of range. Enter anyway? Yes, No

If you select *No*, the system does not enter or change anything. If you select *Yes*, the system changes the attribute value.

Schematic Editor
PCB Editor

Enter

Component

Enters components in a database

Using This Command

To enter a component

1. Select *Enter→Component*. The response area prompts
Enter component name.
2. Enter the component filename.

or

Select *PICKLIST*.

To select a component in the Picklist window

- 1) Select a library or path from the *Libraries* list box.
- 2) Select a symbol or part from the *Symbol* or *Part* list box.
- 3) Select *ENTER* to enter the component in the design.

or

Select *CANCEL* to exit the *Picklist* window without selecting a component.

Note: You can double click left on the component name to enter it into the design, instead of selecting the name, then selecting *ENTER*.

If the selected part contains layers that aren't in the database, a decision box prompts

New layers may be added. Continue?

Yes No

Select *Yes* to add the component. The system adds only layers that contain data.

or

Select *No* to return to the prompt in step 1.

Enter

Component

Note: If Display Verbose Warnings on the Editor Configuration screen is disabled, the message

New layers may be added. Continue?

doesn't appear.

After you enter the part name in the response area or choose a part from the picklist, the system displays the component outline.

The response area prompts

Select location to place *filename*. (Orientation OK?)

3. Change status area options as needed.

To rotate the component

Select the *Deg* cycle for one of four orthogonal angles.

or

Select *Angle* in the status area and enter an angle in the response area. The system allows positive or negative integers. (In the Schematic Editor, only multiples of 45 degrees are accepted.)

Note: The *Angle* option overrides the *Deg* cycle option.

4. Select a location for the component.
5. Continue to select locations if you require multiple instances of the component.

or

Click middle to end the current operation and return to the step 1 prompt. If you used the Picklist, you're returned to the Picklist window.

6. Select CANCEL or another command to exit
Enter→*Component*.

Enter

Component

To change the name of a component or add a name to an unnamed component during entry

- 1) Select the unlabeled Component Name box in the status area or press **F4** before you position the component. The response area prompts

Component name:

Note: In the Schematic Editor, the component name you assign with *Enter→Component* in step [1] isn't a reference designator. To assign a reference designator interactively in the Schematic Editor, use *Enter→Ref. Des. & Section*.

- 2) Enter a new name such as the device name. The system returns to the component location prompt in step 2. For important information on component names in the Schematic Editor and the PCB Editor, refer to Things To Remember.
- 3) Select a location for the component.

If the component was created with units of measure different from the units of measure in the schematic database, a decision box prompts

Converting comp to Metric/English unit.

Continue? Yes No

- Select *Yes* to convert the units of measurement to the units used in the schematic database.

or

Select *No* to return to the prompt for a component name.

Things to Remember

A component is an entity created in the Symbol Editor or Part Editor. It may or may not be archived in a library. If you don't know how to make a part library available to the current design or how to find the part, refer to the *Library Manager* manual.

Enter

Component

To ensure consistency, the system permits only one version of a component in a database. Each time you enter a component, the system checks to make sure its origin and pin information are the same as the origin and pin information for all other components in the design with the same filename.

If you update a library symbol or part and want to enter it in a database that has previous versions of the component, use *Enter→Replace Component*.

The default filename extension for components is .sym in the Schematic Editor and .prt in the PCB Editor. When you're entering components in a design, you don't have to type the extension if you're using the default.

When you are setting up dimensions, you can use *Enter→Component* to search component library files for arrowheads, providing that these files reside in the specified search path. See the *Library Manager* manual for details on setting up your search paths.

Status Area Options

The status area options for *Enter→Component* let you select:

- orthogonal degrees of rotation (*Deg* cycle box)
- an *Angle* option for entering degrees of rotation
- an unlabeled Component Name box
- the *Mirror* check box (Schematic Editor)
- the *Top* or *Bottom* of the board (PCB Editor)
- the *AutoNm* check box (PCB Editor)
- the *Refd* check box (Schematic Editor)

Orthogonal Rotation Angles

The setting of the orthogonal rotation angle stays the same until you change it or select another command.

Enter

Component

Entering an Angle in the Response Area

The *Angle* option lets you specify any angle of component rotation. The *Angle* option overrides the orthogonal degrees of rotation. The entered angle stays the same until you end the command or select another angle.

The Component Name Box

The unlabeled Component Name box displays *unnamed* while you enter a component. You can select this box and enter a component name in the response area. The system doesn't display the name until you execute *Name→Component*.

In the Schematic Editor

Component Mirroring in the Schematic Editor

When you fill the Mirror check box in the Schematic Editor, the system mirrors the pin arrangement of the component along the X-axis. The Y-axis value for the pin locations doesn't change with mirroring. Mirror doesn't change the appearance of the symbol name.

In the PCB Editor

Selecting the Top or Bottom of a PCB

A status area cycle in the PCB Editor allows you to choose the *Top* or *Bottom* of the board for component placement.

Note: To place a component on the bottom of the board, the component's graphic (nonstitchable) layers must be paired for the top and bottom of the board. See *Environment→Assign Layer Pairs*.

Automatic Component Naming

As you enter components in the PCB layout, you can automatically define a reference designator naming pattern by enabling the AutoNm check box in the status area before placing the component. For each subsequent component entered, the number in the pattern increases by one. For example, if you use R1 as the first reference designator, the next components entered would be named R2, R3, R4 and so on.

Enter	Component
-------	-----------

The Component Name box in the status area shows the next available name in the sequence as long as the AutoNm check box is filled.

If you use *Name→Reseq. Ref. Des.* or *Name→Reseq. Window* commands (or if you use the AutoNm check box when entering components) anytime during the current editor session, the last name used by any of these commands sets the pattern for the next reference designator in the sequence, whenever you use these commands to automatically name components.

You can override the existing pattern anytime by selecting the Component Name box in the status area and entering a new name pattern.

You can disable the automatic naming feature at any time during component entry by emptying the AutoNm check box.

Auto-Incrementing Reference Designators

When you click the Refd check box, you can automatically assign reference designator and section numbers to symbols as you enter them in the Schematic Editor. The system also remembers the last reference designator and section number entered for each symbol.

To auto-increment reference designators

1. Select *Enter→Component*. The system prompts

Enter component name:

2. Enter the name of the desired component, or select *PICKLIST* to choose the component from the picklist. Remember that you can change directories if the desired component is in a different directory.
3. Click *ENTER*.

After you enter the part name in the response area or choose a part from the picklist, the system displays the component outline.

Enter

Component

Master Designer also displays a *Refd* checkbox in the status area and prompts

Select location to place *filename*. (Orientation OK?)

4. Change the status area options as needed.
5. Click the *Refd* checkbox. A button marked *unnamed* appears next to the *Refd* checkbox. This is the Reference Designator button, which you use to assign a reference designator and section number to a component.

If the design already has a component of the same type with a reference designator and section name assigned, then the Reference Designator button displays the next available reference designator and section for that component type (for example, U2/B, U3/C).

6. Click the Reference Designator button. The system prompts

Component name

7. Enter a valid reference designator and section number (for example U1/A). The name you entered now appears on the Reference Designator button. You can click this button again to change the reference designator before placing the component.
8. Select the location of the component. Master Designer places the component and displays its reference designator and pin number information.
9. Repeat step 8 for each component you want to place. As you place each component, the system increments the Reference Designator button, indicating the reference designator and section of the component you're about to place.
10. Click *CANCEL* or select another command when done.

Using the Picklist

The Symbol and Part list boxes show the symbols or parts available for the library highlighted in the Libraries list box. You can select a component by entering its filename in the Symbol or Part data entry box, or by selecting it in the list box.

Enter

Component

The Libraries list box shows the libraries made available by the *Library Manager*. You can either select a library from the Libraries list box or add a component library in the Current Library data entry box. After you enter the entire library path in the Current Library box, the Libraries list box displays the component library name. After you exit the editor, the system adds the new paths or libraries to the *Set Libs. and Search Paths* list in the *Library Manager*.

The Search Pattern data entry box lets you control the scope of component filenames displayed in the Symbol or Part list boxes. The default pattern is **.prt* in the PCB Editor and **.sym* in the Schematic Editor, which displays all symbols or parts. For example, to display only the low-power Schottky devices in the TTL library, you would enter *??ls*.prt* or *??ls** in the Search Pattern data entry box.

In the Schematic Editor

In the Schematic Editor *Picklist* window you can

- Switch between directories.
- Select the component library.
- Select the symbol or part file.
- Enter a filter to narrow the list of displayed filenames.
- Scale the size of the selected symbol up or down.
- Scale the size of the symbol's associated text up or down.

If a symbol or part is not available in the current library, you can switch between different directory and load the desired symbol or part from that directory. This makes it easier for you to load designs or borrow symbols and parts from other designs.

The scaling factor for the symbol or its text is 1% through 10,000%. The default is 100%. If you change the scale, the system returns to the default setting after you've entered the component in a schematic.

In the PCB Editor

In the PCB Editor *Picklist* window you can

- select the component library.
- select the part filename.
- enter a filter to narrow the list of displayed files.

Assigning Component Names

The following two sections explain the meaning of the user-assigned names in the Schematic and PCB Editors.

Enter

Component

In the Schematic Editor

The system automatically assigns a unique name to each instance of a component entered in a schematic. A typical system-assigned name could be UC00030004. The first four numbers represent the sheet number in the design, the last four numbers are the component number or instance. Use *Query→Component* to see the system-assigned name.

You may want to override the system-assigned name. The most common reason for overriding the system-assigned name in the Schematic Editor is to identify gate sections to a logic simulator. You can assign a name while you enter the component with *Enter→Component*, or assign it with *Name→Component* after you enter a component. This assignment is NOT the same as assigning a reference designator and section. To assign a reference designator and section, use *Enter→Ref. Des. & Section*.

You can use *Query→Component* to display a component's reference designator and section in the status area.

When entering components, the pin outline is visible as a visual aid in placing the components in the design. If the symbol or part contains more than 175 pins, only a box outline is shown.

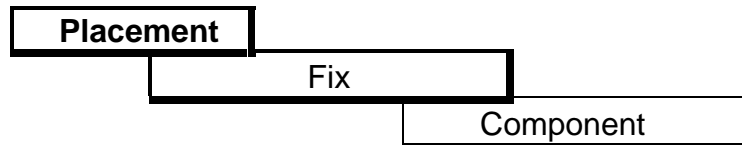
In the PCB Editor

If you used the *Package Schematic* to create the PCB database, the system has already assigned names to the components. In the PCB context, these component names are either system-assigned reference designators or the reference designators you assigned with *Enter→Ref. Des. & Section* in the Schematic Editor.

- You can use *Query→Component* to display a component's reference designator in the status area.
- You can change a component name with *Name→Reseq. Ref. Des.* or *Name→Reseq. Window*.
- When entering components, the pin outline is visible as a visual aid in placing the components in the design. If the symbol or part contains more than 175 pins, only a box outline is shown.

If you didn't use the *Schematic Packager* to create the PCB database, you can assign a component name when you interactively enter the component with *Enter→Component* or by using *Name→Component*.

PCB Editor



Fixes the location of a component to prevent the system from relocating it during automatic placement.

Using This Command

To fix the location of one or more components

1. Select *Placement→Fix→Component*. The system highlights all components currently fixed in the database. The response area prompts

Select a component.
2. Select each component you want to fix.

or

Select *ByName* in the status area and enter the component name in the response area.
3. The system highlights and displays each fixed component in the graphics area.

If the component you want to select doesn't exist, Master Designer displays a warning message and prompts you again to select a component.
4. Select another component.

or

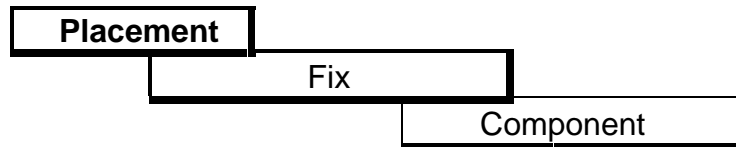
Select a non-nested command to exit
Placement→Fix→Component.

Things to Remember

Placement→Fix→Component affects automatic placement and automatic swapping. This command doesn't prevent you from using any of the move or rotate commands to move or rotate fixed components.

You can't swap a component on a fixed component. *Swap→Gate* and *Swap→Pin* give you the option to swap gates or pins of a fixed component in case each pin on the part is a separate gate.

Improve Plc.→Component and *Improve Plc.→Gate* don't swap fixed components or gates.



Always fix connectors before you use *Placement→Automatic Placement* or *Placement→Improve Placement*. The system uses connectors as reference components during automatic placement; it considers a component with a component ID type in the range of 12000-12999 to be a connector. In addition, you need to fix any other parts that are restricted because of mechanical fitting, heat emission, radio interference, or other critical conditions.

To unfix a component, select *Placement→UnFix→Component*.

PCB Editor

Placement

Unfix

Component

Unfixes a component previously fixed with *Placement→Fix→Component*.

Using This Command

To unfix one or more components

1. Select *Placement→UnFix→Component*. The system highlights all fixed components. The response area prompts
Select a component
2. Select one of the highlighted components.
or
Select *ByName* in the status area and enter the component name in the response area.
3. The system unhighlights the display and unfixes the component.
If the component you want to select doesn't exist, Master Designer displays a warning message and prompts you again to select a component.
4. The system unfixes the component and returns you to the prompt in step 1.
5. Select another component.
or
Select a non-nested command to exit
Placement→UnFix→Component.

Things to Remember

Select *Placement→UnFix→Component* to make components available for automatic placement and swapping operations.

Symbol Editor
Schematic Editor
Part Editor
PCB Editor

Query

Properties

Lets you view and change the properties of any selectable object in your design.

Using This Command

You can view and change the properties of any selectable object in your schematic or PCB design. A cycle box on the status line lets you view properties by either selecting an object or an attribute or a net. The properties available for viewing and changing depend on what object you select. For example, where an arc has radius and line width properties, a filled circle has only a radius property.

To change properties

1. Select *Query*→*Properties*. A cycle box appears on the status area.
2. Cycle to *ByObj* (the default) to select an object. The system prompts
Select object.
If you cycle to *ByAttr*, the system prompts
Select an attribute or net.
3. Select the desired object. The system highlights the selected object and displays a dialog box showing that object's properties.
The properties dialog box has combinations of picklists, edit boxes, check boxes, or cycle boxes depending on the properties available.
If you cycle to *ByAttr* and select a net, the system displays the Net Attributes dialog box with the attributes for the selected net.
If you cycle to *ByAttr* and select an attribute, the system displays that attribute's properties in a dialog box.
4. Change the desired properties.
5. Click *OK* to accept the changes, or click *CANCEL* to discard them.

Note: If you try to select an object where there is none, Master Designer displays the error message
No object found.

Query

Properties

Things to Remember

Like the *Edit→Net Attr* command, the *Query→Properties* command checks the values of all supported net attributes which include WIDTH, CLEARANCE, TRACE-TO-TRACE, PAD-TO-TRACE, TRACE-TO-EDGE, PAD-TO-EDGE, and VIATYPE.

The tables below list the selectable objects and their included properties. Also, where possible, the tables list the acceptable values for the properties you can change. In the tables below, any property marked "label" is read-only and cannot be changed.

Table 3-1. Rectangle (unfilled)

Property	Acceptable Value/Range
Layer	pick list of ABL layers
Line Style	solid, dotted, dashed
Width	(0-255)
Point A	label (lower left coordinate)
Point B	label (upper right coordinate)

Table 3-2. Rectangle (filled)

Property	Acceptable Value/Range
Layer	pick list of ABL layers
Point A	label (lower left coordinate)
Point B	label (upper right coordinate)

Table 3-3. Text

Property	Acceptable Value/Range
Layer	pick list of ABL layers
Horizontal Justification	cycle (left, right, center)
Vertical Justification	cycle (top, center, bottom)
Text Size (Height)	edit box (2-5000)
Mirror	check box
Rotation	cycle (ortho: 0 90 180 270)
Edit Text (Content)	edit box (up to 255 characters.)
Location (insertion point)	label (XY coordinates)

Table 3-4. Line Segment

Property	Acceptable Value/Range
Layer	pick list of ABL layers
Line Style	cycle (solid, dotted, dashed)
Width	edit box (0-255)
Point A	label (segment start point)
Point B	label (segment end point)

Query

Properties

Table 3-5. Circle (unfilled)

Property	Acceptable Value/Range
Layer	pick list of ABL layers
Line Style	cycle (solid, dotted, dashed)
Width	edit box (0-255)
Radius	edit box (.01-10,000,000)
Center	label (XY coordinates)

Table 3-6. Circle (filled)

Property	Acceptable Value/Range
Layer	pick list of ABL layers
Radius	edit box (0.01-255)
Center	label (XY coordinates)

Table 3-7. Arc

Property	Acceptable Value/Range
Layer	pick list of ABL layers
Line Style	cycle (solid, dotted, dashed)
Width	edit box (0-255)
Radius	label (radius value)
Center	label (XY coordinates)
Start Angle	label (starting angle value)
Ending Angle	label (ending angle value)

Table 3-8. Polygon (PCB Editor)

Property	Acceptable Value/Range
Layer	pick list of ABL layers
Poly Fill	cycle (solid, hatched)
Aperture Width	edit box (0-250)
Net name	label (Entered Polygon)
# vertices	label
Net Attributes button	displays the Net Attributes dialog box

Table 3-9. Flash (standalone or Part Editor)

Property	Acceptable Value/Range
Layer	pick list of ABL layers only
Aperture #	edit box (1-999)
Rotation	cycle (ortho: 0 90 180 270)
Location	label (center point, XY coordinates)
Shape	label (round, square, rectangle, thermal, oval, target, polygon)
Shape Labels	
Round (diameter)	
Square	label (width)
Rectangle	label (width, height)
Thermal	label (outer diameter, inner diameter)
Oval	label (width, height)
Polygon	label (number of sides)

Query

Properties

Table 3-10. Wire, segment

Property	Acceptable Value/Range
Layer	pick list of stitchable layers
Net Name	label (if applicable; can be "unnamed")
Wire Width	edit box (0-255)
Attribute Button	leads to picklist of attributes assigned to net.
Point A	label (segment start point)
Point B	label (segment end point)

Table 3-11. Component (PCB Editor)

Property	Acceptable Value/Range
Reference Designator	edit box (Reference Designator)
Surface Mount Device	label (YES, NO)
Side	cycle (Top, Bottom)
No. of pins	label
SCAT Code	label
Fixed	checkbox
Angle	edit box (-359 to 359)
Location	label (insertion point, XY coordinates)
Attribute Button	leads to picklist of attributes assigned to component.
Part Name	label (part filename)

Table 3-12. Component (Schematic Editor)

Property	Acceptable Value/Range
Reference Des/Section	edit box
Component Name (Instance)	edit box (Component Name)
No. of pins	label
SCAT Code	label
Mirror	checkbox
Angle	cycle (ortho+45: 0 45 90 135 180 225 270 315)
Location	label (insertion point, XY coordinates)
Attribute Button	leads to picklist of attributes assigned to component.
Symbol Name	label (symbol filename)

Table 3-13. Pin (Symbol, Part, and Schematic Editors)

Property	Acceptable Value/Range
Layer	pick list of ABL layers only
Pin Name	label
Pin Number, Alphanum	label
Pin Type	edit box (PCB: 1-1000, SCH: 0-6, 16-63)
Sequence #	label (pin entered in sequence)
LEQ	edit box (0-127)
Location coordinates)	label (insertion point, XY

Query

Properties

Table 3-14. Pin (PCB Editor)

Property	Acceptable Value/Range
Layer	pick list of ABL layers only
Pin Name	label
Pin Number, Alphanum	label
Pin Type	edit box (PCB: 1-1000, SCH: 0-6, 16-63)
Sequence #	label
LEQ	edit box (0-127)
Location	label (insertion point, XY coordinates)
Padstack filename	label

Table 3-15. Attribute (Schematic and PCB Editors)

Property	Acceptable Value/Range
Layer	pick list of ABL layers only
Keyword	edit box (max 23 chars.)
Value	edit box (max 255 chars.)
Location	label (insertion point, XY coordinates)
Text Properties	(as listed above, under Text object)

Schematic Editor
PCB Editor

Rotate

Component

Specifies a *final* rotation angle for the component.

Using This Command

To rotate a component

1. Select *Rotate*→*Component*. The response area prompts

Enter angle:

2. In the Schematic Editor, enter a multiple of 45 degrees. Your entry is the final angle at which the selected component resides. Accepted entries are plus or minus 0, 45, 90, 135, 180, 225, 270, 315, 360.

or

In the PCB Editor, enter a multiple of 90 degrees. You can enter a component at any angle at 1-degree increments, ranging from -359 to 359 degrees. The system will rotate the component in 90-degree increments from the specified starting point.

The response area prompts

Select center of rotation.

3. Select the center of rotation for one component.

or

Select *ByName* in the status area and enter the component name or its reference designator in the status area. When you select *ByName* in the status area, Master Designer rotates the named component at the specified angle using the center of the component as the center of rotation.

If the component you want to select doesn't exist, Master Designer displays a warning message and prompts you again to select a component.

When you select the center of rotation, the system rotates the component about the point you select a specified number of degrees relative to the component's starting position. For example, if you first rotate a component to 45 degrees from the vertical axis, but then decide to rotate it 45 degrees more, you need to specify 90 degrees as the final angle. Degrees of rotation are measured from the starting point.

After the component rotates, the response area continues to prompt

Select center of rotation

4. Select another component to rotate to the same angle.

Rotate

Component

or

Click middle or press **Esc** once to return to the prompt in step 1.

or

Click middle twice or press **Esc** twice, then select another command.

To restore rotated components to their original positions

- Select *Rotate*→*Undo* immediately after you rotate a component to restore it to the previous position.

Things to Remember

When you try to restore components to their original positions, whether the component returns to the exact same position depends on the center of rotation you select and the original angle of rotation. If it doesn't return to the exact same position, the problem may be that the component is off-grid or you may not have selected the exact same center of rotation. You can use *Move*→*Component* to select the component's origin point and move the component back on grid.

If the rotation of the component was other than zero degrees before trying to restore its position by entering zero as the final angle, the component will still be restored to zero degrees when you enter zero for the angle of rotation.

Use *Query*→*Component* to determine the angle of rotation for a component, before or after using *Rotate*→*Component*.

The Rotate Text in Four Directions option in the *Editor Configuration* window determines one of two possible modes of component text rotation. If the Rotate Text in Four Directions check box is filled, the component text rotates with the same rotation as the component or other object. If the Rotate Text in Four Directions check box is empty, the text remains right side up when the associated object is upside down. If you change the Rotate Text in Four Directions option while the corresponding editor is open, you must close and reopen the editor before it recognizes the change in text rotation. Examples of component text are the component name, reference designator, and component attributes.

Rotate**Component**

Flash aperture rotation is distinct from component rotation. There are certain restrictions and considerations about apertures that can be rotated when you rotate a component.

- You can specify an angle of rotation for the flash aperture itself by using the Rotate field in the Aperture Table Editor. This applies only to rectangular or oval shapes, and can be used only if you are using an RS-274-X photoplotting device that supports flash rotation.
- You can also specify a rotation value for each flash instance by selecting Degree in the status area when using *Draw→Flash*. This angle of rotation will be added to any angle defined for the aperture in the aperture table.
- The flash will be rotated along with the component, regardless of the rotation angle you specify for the flash when you define the shape for the aperture. This applies if your rotated flash is in a padstack, built into the SMT component file, or if the component is placed on the top or bottom of the board.
- Thermal and target shapes do not appear rotated in your PCB database because a generic shape is used to display this type of flash shape in the PCB Editor and on a penplot. They will, however, be rotated in the actual photoplot.
- You must enable the Rotate Padstacks option on the PCB Editor Configuration screen if you want padstacks to rotate with the component when the component is rotated in the PCB database.
- If you don't have access to an RS-274-X photoplotter, you will have to define different flash aperture shapes for each type of rotated flash shape you want to use in your design. If you forget to do this, and you're using photoplotters that don't support Extended Gerber photoplotting format, you may not get accurate results.

All plots that you generate from the database show component text exactly as it appears in the drawing area.

Use whole numbers to specify an angle of rotation. Fractions are not allowed.

Rotate

Component

When you query a component, the angle of rotation is shown as a positive value; for example, -45 degrees is shown as 315 degrees.

You cannot select a fixed component. If you do, the system displays the message

Component is fixed

In this case, select another component that isn't fixed.

- In the Schematic Editor

You can only rotate a component in multiples of 45 degrees. The angle you specify is the final angle of rotation for the component.

/dyn

PCB Editor

Controls the dynamic ratsnest display.

Using This Command

To control dynamic ratsnest display

1. Enter **/dyn**. The system prompts
Dynamic Ratsnest Delay? Yes, No
2. Select *Yes*, *No*.

Entering *Yes* enables the default dynamic ratsnest delay interval set using the */dly* command. Entering *No* disables dynamic ratsnest display.

Things to Remember

You can control the dynamic ratsnest display at any time when the *Enter→Wire* or *Edit→Enter Wire* commands are active. When this feature is enabled, the dynamic ratsnest display constantly updates to show the shortest path from the cursor to the nearest unconnected segment or pin in the net. If all pins in the net are already connected, the dynamic ratsnest will point to the nearest pin in the net.

This is a nested command and can be used at any time during *Enter→Wire* without interrupting wire entry. This command also affects the *Move→Component* command.

See also *Display→Control Ratsnest* for other ways to control the ratsnest display.

This chapter describes the new commands for MD 8.6. Menu commands are discussed first, followed by keyboard commands. The new keyboard commands are:

- */dly*
- */osel*

/dly

PCB Editor

Sets the dynamic ratsnest delay (in seconds).

Using This Command**To set the dynamic ratsnest delay**

1. Enter */dly*. The system prompts
Dynamic Ratsnest Delay=n. New value=
2. Enter a number between .01 and 1 (in seconds). Or you can press ↵ to accept the current value.

Entering a number between .01 and 1 sets the dynamic ratsnest delay (in seconds) at that number.

Things to Remember

You can set a dynamic ratsnest delay without enabling dynamic ratsnest display. When you enable the dynamic ratsnest display again, the system uses the delay last set with the */dly* command.

The */dly* keyboard command applies to these menu commands:

- *Enter→Wire*
- *Edit→Enter Wire*
- *Move→Component*

/osel

Symbol Editor

Schematic Editor

Part Editor

PCB Editor

Switches between Master Designer's current selection method and that used by previous versions of Master Designer.

Using This Command

The current selection mechanism selects the object closest to the cursor on the active layer that is within snap tolerance. This makes it easier for you to predict which object will be selected first.

To change to the old object selection method

1. Enter */osel*. The system prompts
 Use old selection method?
 Yes No
2. Select *Yes* to switch to the selection method used by Master Designer versions 8.5 and earlier.
 or
 Select *No* to use the selection method used by Master Designer versions 8.6 and later.

Things to Remember

With many commands, the selection mechanism selects the object closest to the cursor on the active layer that is within snap tolerance. This makes it easier for you to predict which object will be selected first. The affected commands are:

- *Change Layer→Object*
- *Change Layer→Objects*
- *Copy→Object*
- *Copy→Objects*
- *Delete→Object*
- *Delete→Objects*
- *Delete→Trace*

- *Edit→Move Vertex*
- *Edit→Delete Vertex*
- *Edit→Move Segment*
- *Edit→Delete Segment*
- *Edit→Move Via*
- *Edit→Delete Via*
- *Edit→Segment Layer*
- *Edit→Delete Trace*
- *Edit→Trace Width*
- *Move→Object*
- *Move→Objects*
- *Query→Trace*
- *Query→Net*
- *Query→Object*
- *Query→Via*
- *Query→Properties*
- *Rotate→Object*
- *Rotate→Objects*

For those objects that Master Designer does not apply snap tolerance to, such as components, attributes, or polygons, Master Designer lets you select these objects when the cursor is within their boundaries.

If more than one object is within snap tolerance and at the same distance from the cursor, Master Designer prioritizes which objects to select. The order of priority is (from highest to lowest):

- objects on the active layer
- objects on the ABL layer
- objects on any layer set to ON
- vias
- nets
- components
- draw objects

You can use the new */osel/* keyboard command to switch between the new selection method and to the selection method used in previous versions of Master Designer.

If your existing Master Designer macros require the old selection method, you can also use the *osel/* command in your existing Master Designer macros to switch between the old and new selection methods, as in the example below:

```
MACRO 850 -400
Opcode 192 0
Text osel
Text 1
B1 850 -450
B1 900 -750
Opcode 192 0
Text osel
Text 0
Opcode 192 0
Text mend
END_MACRO
```

In this example, the statements

```
Text osel
Text 1
```

activate the old selection method by setting *osel* to 1. Conversely, the statements

```
Text osel
Text 0
```

reactivate the new selection method by setting *osel* to 0.

System Limits

A

This appendix lists the supported system limits for P-CAD Master Designer version 8.6.

Table A-1 lists name length limits. Table A-2 lists maximums. Table A-3 lists other ranges.

Table A-1. Name Length Limits

Parameter	Limit
Attribute keyword	23
Attribute value	255
Bus name (input buffer length)	79
Bus net name length	23
Component name (internal, user assigned)	23
Critical path name length	8
Device name for library member	8
External filename length (including path)	63
Footprint name length	23
Group name length	8
Layer name length	6
Net name (internal, user assigned)	23
Packaging ID (nonhomogeneous parts)	15
Pin name (internal, user assigned)	23
Pin number (alphanumeric)	7
Reference designator length	23
Reference designator prefix length	3
Schematic sheet ID length	4
Section name length	3
Symbol or part external filename, DOS	8.3
Symbol or part external filename, UNIX	255

Table A-2. Maximums

Items	Practical	Theoretical
Aliases per component	1882	
Apertures, number of, Gerber Laser Model	999	
Apertures, number of, Gerber Model 32, 33, 41	24	
Apertures, number of, Gerber RS-274-X	999	
Apertures, number of, MDA Fire 2000	999	
Apertures, size, Gerber Laser Model, RS-274-X, MDA Fire 2000	.500	
Apertures, size, Gerber Model 32, 33, 41	.240	
Association distance (placement)	1000	
Characters in an aperture macro name	12	
Characters in a text string (line)	255	
Clearance for placement	1000	
Components	32,000	(2,147,483,447)
Components per library archive	800	
Filled circle radius	255	
Gate types per package	9	
Gates per package	5000	
Grid spacing	2000	
Input buffer, aperture list, <i>Environment!</i> <i>Set Inner Plane Aperture</i>	255	
Lattice spacing (placement)	5000	
Lattices for placement	10	
Layer support	100	
LEQ codes	127	
Message character buffer, <i>/intr</i>	44	
Nets	64,000	(2,147,483,447)
Picture blocks	250,000	(2,147,483,447)
Pin count, by wire, <i>Display!Control Ratsnest</i>	512	
Pin types (parts)	1000	
Pins	250,000	(2,147,483,447)
Pins per component	5000	
Polygon aperture width	250	
Polygon clearance value	500	
Polygon + merged void vertices	10,000	
Polygon size (sq. DBUs)	10,240,000	
Polygon vertices	10,000	
Polygonal void vertices	10,000	
Programmable function keys	40	
Repeat copy count	255	
Snap tolerance value	1000	
Stitchable layers	32	
Text size (height)	5000	
Unique components	32,000	(2,147,483,447)
Via types	50	
Wire/line width	255	

Table A-3. Other Ranges

<i>Type</i>	<i>Range</i>	<i>Default</i>
Angle range, component rotation	-359 to 359	
ASCII key code range for hot keys	33-126	
Bus bits number range	128	
Cartesian coordinates range	-30,000 to 30,000	
Display range, flash size, PCB Editor	4 to 150 DBUs	(60)
Display range, pin size, PCB Editor	4 to 100 DBUs	(40)
Display range, pin size, Schematic Editor	1 to 50 DBUs	(10)
Display range, solder dot, Schematic Editor	1 to 50 DBUs	(10)
Display range, via size, PCB Editor	4 to 100 DBUs	(40)
Pin type range (symbols)	0 to 6, 16 to 63	
Range, histogram routing grid	.01 to 2,000	(50)
Range, histogram signal layers	1 to 20	(2)
Range of D-codes	10-1008	
Range of polygon aperture vertices (or sides)	3-10	
Range of ties in a thermal shape description	1-10	
Scaling factor range, components	1 to 10,000	
Scaling factor range, text	1 to 10,000	

Filename Extensions

B

Table B-1. Filename Extensions

Extension	Description	Generated By
.alt	Netlist input	User (for netlist conversion)
.am	Aperture description macro	User (for PCB aperture table editor)
.apl	Apertures used in design	<i>PCB Editor</i>
.apr	Aperture table	<i>Hardcopy</i>
.atr	Attribute	<i>Engineering Change Order</i>
.asr	Apertures used in plot file	<i>Hardcopy</i>
.atr	Attribute	<i>Engineering Change Order</i>
.bck	Back-annotation instruction file	<i>PC-PACK</i>
.bka	Back-annotation command	<i>Engineering Change Order</i>
.bnl	Annotated netlist	<i>Package Schematic</i>
.cc	CalComp plotter vector	<i>Hardcopy</i>
.cfg	Configuration	Any program run from P-CAD graphic user interface
.cir	Spice file	<i>Spice Circuit Writer</i>
.cmd	Packaging command log file	<i>Package Schematic</i>
.cmd	Command log file	<i>Schematic Editor</i> <i>PCB Editor</i> <i>Symbol Editor</i> <i>Part Editor</i>
.cm\$	Backup command log file	<i>Schematic Editor</i> <i>PCB Editor</i> <i>Symbol Editor</i> <i>Part Editor</i>
.cmp	Component list report	<i>Report Generator</i>
.cth	C. Itoh printer format	<i>Hardcopy</i>
.ctl	Autorouter control (strategy)	<i>Autorouter</i>
.dbg	Editor debugging	<i>Schematic Editor</i> <i>PCB Editor</i> <i>Symbol Editor</i> <i>Part Editor</i>
.dmp	Houston Instruments plotter	<i>Hardcopy vector</i>
.drc	Design rule check report	<i>Design Rules Check</i>
.drl	Numerically controlled drill data	<i>Drill</i>
.dxf	DXF format input/output file	<i>DXF Reader</i> <i>DXF Writer</i>
.eco	Compare/Analyze phase report	<i>Engineering Change Order</i>

Table B-1. Filename Extensions (cont'd)

Extension	Description	Generated By
.edf	EDIF format input/output file	EDIF Reader EDIF Writer
.eps	Epson format	Hardcopy
.erc	Electrical rule check report	Electrical Rules Check
.fil	Cross-reference (ASCII)	User (for Package Schematic)
.gbr	Gerber format	Hardcopy
.his	Histogram report	PCB Editor
.hp	Hewlett-Packard plotter	Hardcopy
.hpp	Hewlett-Packard LaserJet printer format	Hardcopy
.ibm	IBM printer/plotter format	Hardcopy
.icc	Interleaf (CC960) format	Hardcopy
.ins	Component insertion	Auto-Insertion
.key	Saved function key file	User (created in any graphics editor)
.lgr	Gerber laser photoplotter format	Hardcopy
.lgx	RS-274-X photoplotter format	Hardcopy
.log	Message log file (DOS)	Any program run from P-CAD graphic user interface and command line
.LOG	Message log file (UNIX)	Any program run from P-CAD graphic user interface and command line
.mac	macro file	User (created in graphics editor or text editor)
.map	DXF translation map file	DXF Reader DXF Writer
.mat	Materials list report	Report Generator
.mda	MDA Fire 2000 photoplotter format	Hardcopy
.mfg	NC drill table	Drill
.nde	Netlist report	Report Generator
.nlc	Report	Netlist Comparison
.nlt	Schematic netlist	Package Schematic
.nz	Bruning (Nicolet) Zeta plotter	Hardcopy
.oki	Okidata printer format	Hardcopy
.out	Component name change report	Engineering Change Order
.pas	Check pass file	Design Rules Check (DRC)

Table B-1. Filename Extensions (cont'd)

Extension	Description	Generated By
.pbk	Backup PCB	Engineering Change Order Autorouter
.pdf	Database (ASCII)	PDIF Writer
.pin	Pin Information List	PCB Editor
.pkg	Packaged PCB database	Package Schematic
.pkl	Packaging list report	Report Generator
.plc	Placed PCB database	User
.plt	Plot instruction file (binary)	Schematic Editor PCB Editor Symbol Editor Part Editor
.plb	PCB part library archive	Library Maintenance
.pnl	Packaged PCB netlist (binary)	Package Schematic
.ppp	plot device configuration file	Hardcopy
.prt	PCB part	Part Editor
.ps	Padstack	Part Editor
.psc	Postscript format	Hardcopy
.pty	Pin type report	Generate Reports
.rcf	Autorouter control file	Autorouter
.rep	Autorouter report	Autorouter
.rhf	Autorouter history file	Autorouter
.rpt	Part/symbol pin information table report	Component Editor
.rpt	Swap report log file	PCB Editor
.rp1	Report	Convert Netlist to PCB
.rp2	Report	Convert Netlist to PCB
.rp3	Report	Convert Netlist to PCB
.rp4	Report	Convert Netlist to PCB
.rte	Autorouter extraction	Autorouter
.rts	Autorouter solution	Autorouter
.rul	Design rules file	Design Rules Check (DRC)
.sbk	Backup schematic database	Schematic Editor
.sch	Schematic database	Schematic Editor
.sdt	Solder dot special symbol	User
.si	Source specification	User
.slb	Schematic symbol library archive	Library Maintenance
.ssf	Special symbol file	User (for PCB Editor)
.swr	Improve placement log	PCB Editor

Table B-1. Filename Extensions (cont'd)

<i>Extension</i>	<i>Description</i>	<i>Generated By</i>
.sym	Schematic symbol	<i>Symbol Editor</i>
.tbl	NC drill table	<i>Drill</i>
.upd	Update command file	<i>Engineering Change Order (ECO)</i>
.wrl	Wire list	<i>Report Generator</i>
.xnl	Expanded schematic netlist	<i>Package Schematic</i>
.xrf	Nodes cross-reference file	<i>Spice Circuit Writer</i>

Reserved Words

C

This appendix lists DOS reserved device names and attribute keywords.

DOS Reserved Device Names

Don't use the following DOS reserved device names in filename prefixes:

- aux
- com1
- com2
- com3
- com4
- con
- lpt1
- lpt2
- lpt3
- nul
- prn

Refer to your DOS manual for more information.

Attribute Keywords

Certain keywords have predefined functions in specific tools. Table C-1 lists the keywords and tools.

Table C-1. Reserved Keywords With Predefined Functions

Keyword	Tool
CARDx (where x is a number)	SPICE Circuit Writer
COMPARE=MECH	Part Editor, PCB Editor
DEVICE	EDIF Netlist Reader EDIF Netlist Writer
FP	Part Editor PCB Editor
INCLx (where x is a number)	SPICE Circuit Writer
NET	Symbol Editor Schematic Editor
PACKAGE	EDIF Netlist Reader EDIF Netlist Writer
PCERC	Electrical Rules Check
PGCONN	EDIF Netlist Reader EDIF Netlist Writer Electrical Rules Check
PINATT?	EDIF Netlist Reader EDIF Netlist Writer
PNAME	SPICE Circuit Writer
PRT	Symbol Editor Schematic Editor Package Schematic
PWGD	Symbol Editor Schematic Editor Package Schematic Component Editor
PWGD <i>i</i> (where <i>i</i> is a number)	Symbol Editor Schematic Editor Package Schematic Component Editor
REFDPRE	SPICE Circuit Writer EDIF Netlist Writer
RVALUE	Electrical Rules Check
SHEET	Symbol Editor Schematic Editor Electrical Rules Check
SNAME	SPICE Circuit Writer
S\$P	SPICE Circuit Writer
SPCC	SPICE Circuit Writer

Table C-1. Reserved Keywords With Predefined Functions (con't)

Keyword	Tool
SPGN	<i>SPICE Circuit Writer</i>
SPINIT	<i>SPICE Circuit Writer</i>
SPMI	<i>SPICE Circuit Writer</i>
SPPx (where x is a number)	<i>SPICE Circuit Writer</i>
SPTI	<i>SPICE Circuit Writer</i>
SPVC	<i>SPICE Circuit Writer</i>
TARGET	<i>Auto-Insertion</i>

Button Menu Trees

D

This appendix contains menu trees for button menus that appear in a graphics editor when you choose the *MnBtn* option in the status area.

Symbol Editor

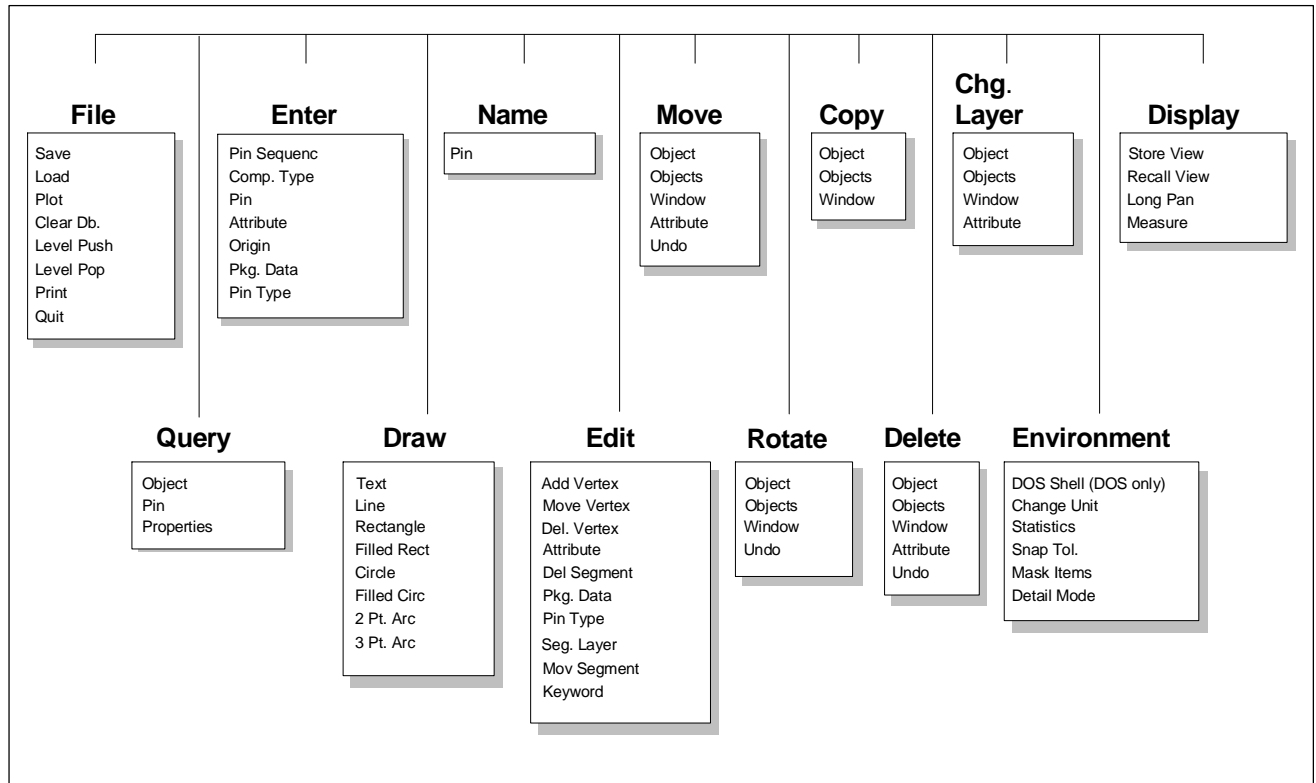


Figure D-1. The Symbol Editor Button Menu Tree

Schematic Editor

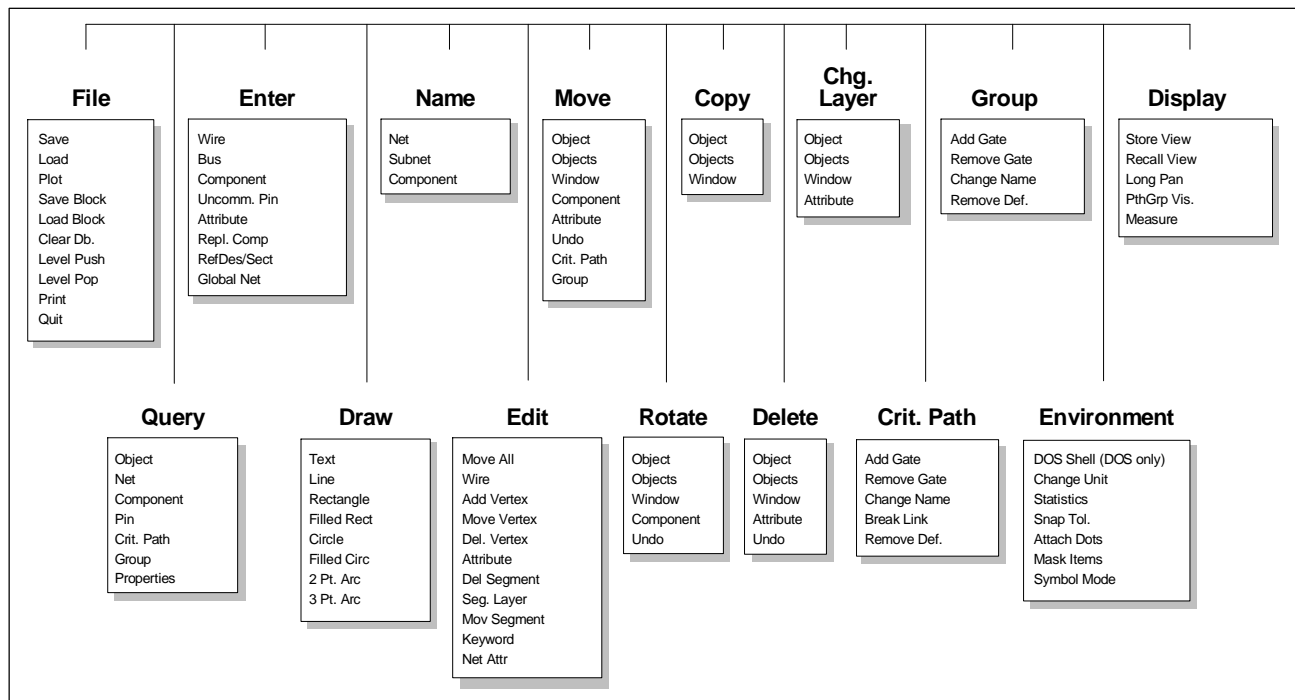


Figure D-2. The Schematic Editor Button Menu Tree

Part Editor

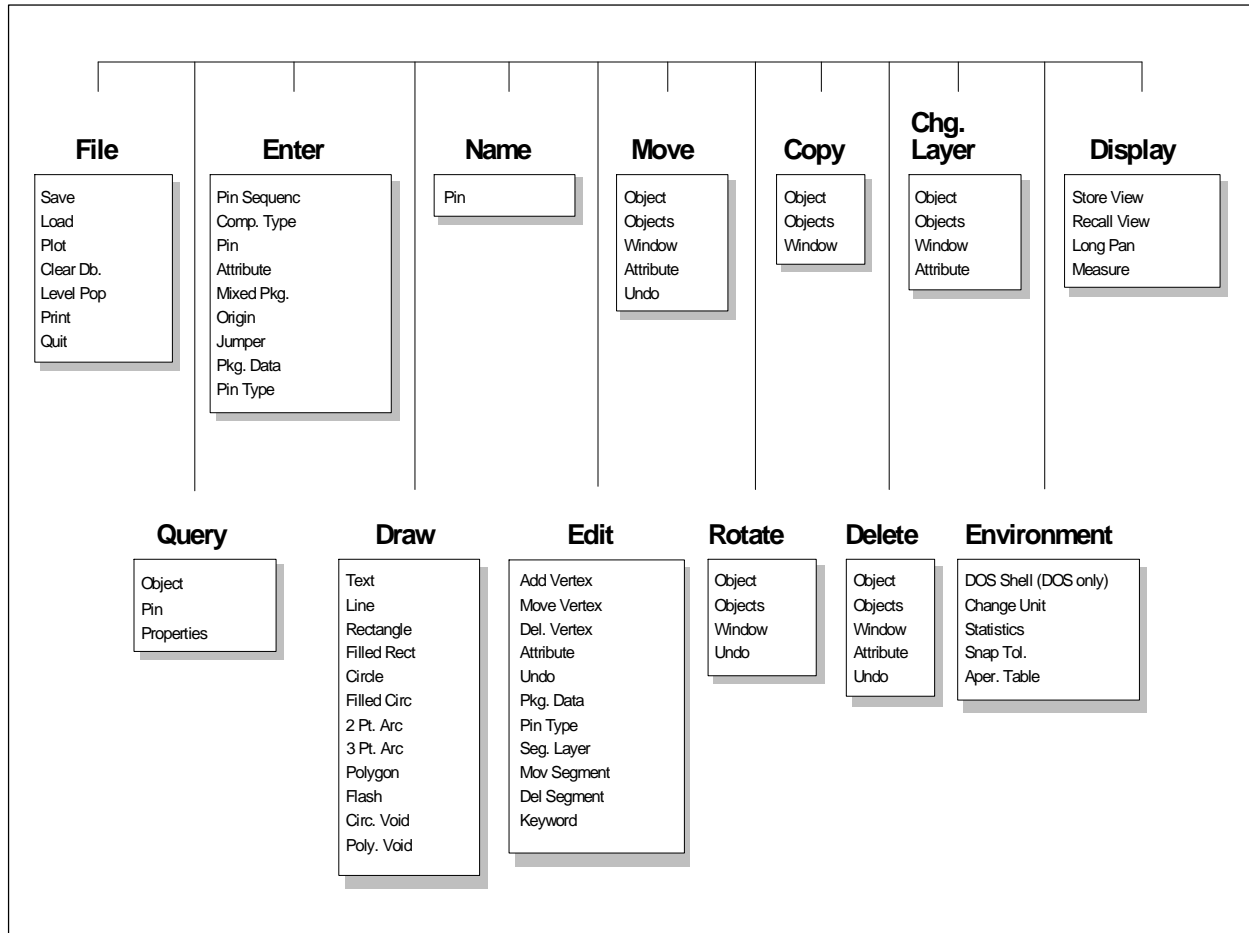


Figure D-3. The Part Editor Button Menu Tree

PCB Editor

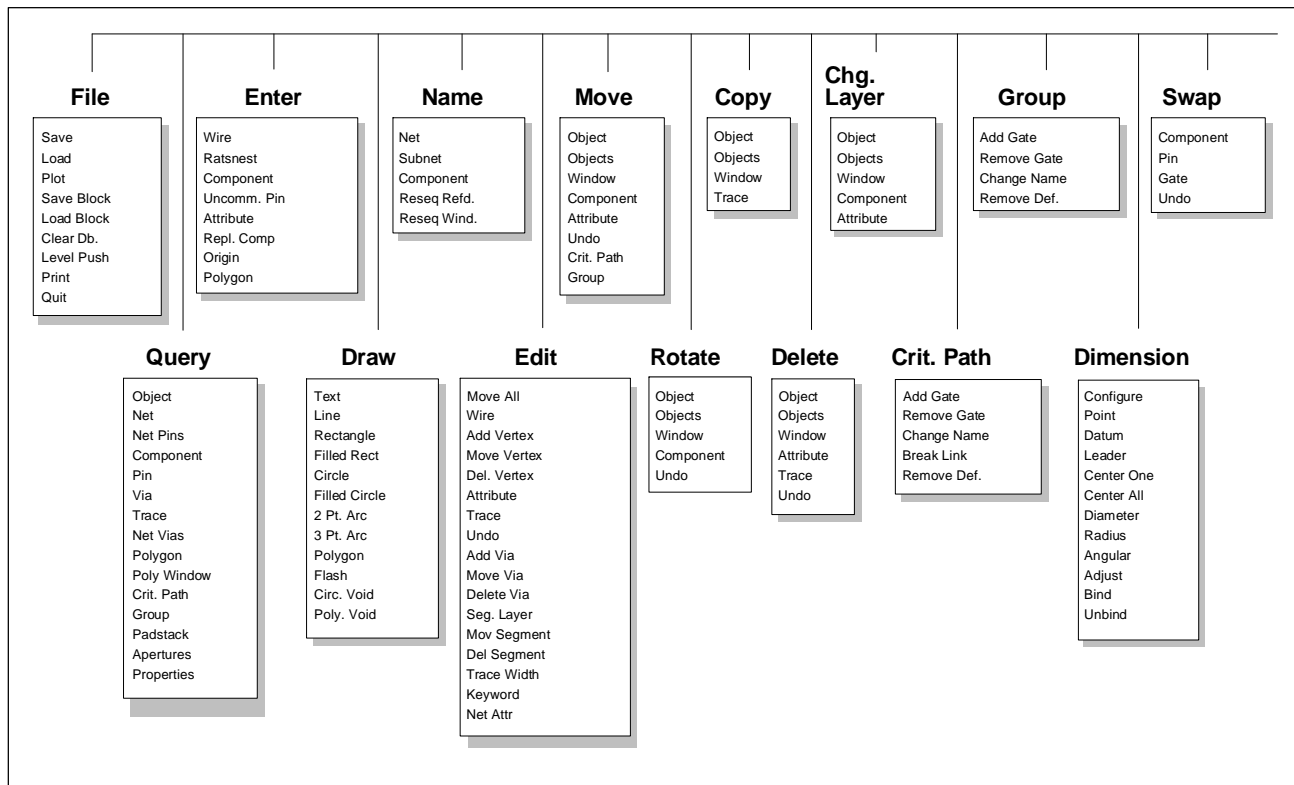


Figure D-4. The PCB Editor Button Menu Tree

PCB Editor (con't)

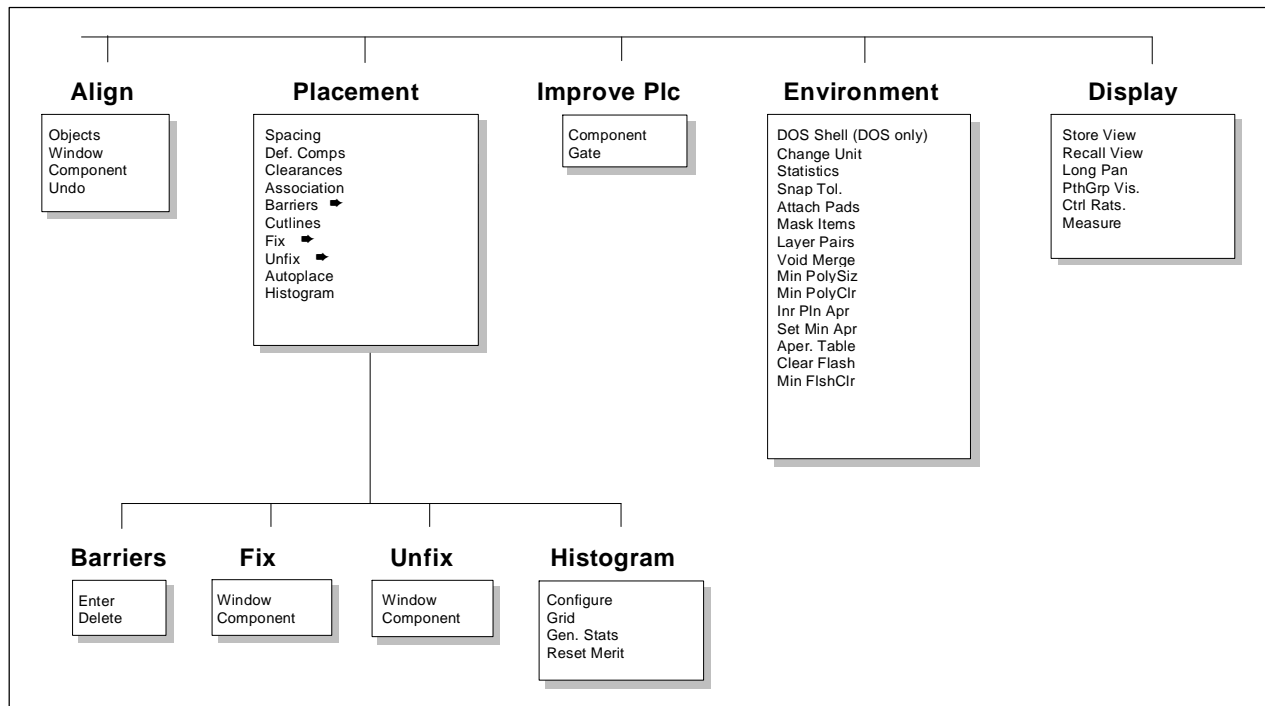


Figure D-4. The PCB Editor Menu Tree (cont'd)

Command Cross Reference

E

This appendix contains a command cross-reference, a list of default hot-key functions, and a chart of the keys available for hot-key assignment.

Table E-1 contains a complete cross-reference of MD 5.0 and MD 8.6 graphic editor commands and keyboard-equivalent commands. Refer to the *Command Reference* manual for a complete description of each command.

Commands are listed in alphabetical order by the MD 8.6 commands. Commands that appear in lower case, such as /rctl, have no equivalent menu command for the P-CAD version they're listed under. Keyboard and keyboard-equivalent commands are not case-sensitive.

Table E-1. Command Cross Reference

MD 5.0 Command	MD 8.6 Command	MD 8.6 Keyboard Equivalent
a (wire bend hot key)	a	a
a	a	/bend
n/a	/ably	/ably
ALGN	Align→Components	ALGN
ALGN>IDEN	Align→Objects	ALGN>IDEN
ALGN/UNDO	Align→Undo	ALGN/UNDO
ALGN/WIN	Align→Window	ALGN/WIN
/alyr	/alyr	/alyr
ATTR/DATR	Delete→Attribute	DEL/ATRB
c (curved wire hot key)	c	c
c	c	/wcrv
/cfil	/cfil	/cfil
n/a	Chg. Layer→Attribute	CLYR/ATRB
CLYR/COMP	Chg. Layer→Component	CLYR/COMP
FLIP	Chg. Layer→Component	CLYR/COMP
CLYR	Chg. Layer→Object	CLYR
CLYR>IDEN	Chg. Layer→Objects	CLYR>IDEN
CLYR/WIN	Chg. Layer→Window	CLYR/WIN
COPY	Copy→Object	COPY
COPY>IDEN	Copy→Objects	COPY>IDEN
COPY/TRCE	Copy→Trace	COPY/TRCE
COPY/WIN	Copy→Window	COPY/WIN
/cpos	/cpos	/cpos
CPH/TH/AG	Crit. Path→Add Gate	CPH/TH/AG
CPH/TH/UNLK	Crit. Path→Break Link	CPH/TH/UNLK
CPH/TH/RNAM	Crit. Path→Change Name	CPH/TH/RNAM
CPH/TH/RSET	Crit. Path→Remove Definition	CPH/TH/RSET
CPH/TH/UTAG	Crit. Path→Remove Gate	CPH/TH/UTAG
ATTR/DATR	Delete→Attribute	DEL/ATRB
DEL	Delete→Object	DEL
DEL>IDEN	Delete→Objects	DEL>IDEN
DEL/TRCE	Delete→Trace	DEL/TRCE
DEL/UNDO	Delete→Undo	DEL/UNDO
DEL/WIN	Delete→Window	DEL/WIN
n/a	Dimension→Adjust	ADIM/AJST

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.6 Command	MD 8.6 Keyboard Equivalent
n/a	Dimension→Angular	ADIM/ANGL
n/a	Dimension→Bind	ADIM/DGRP
n/a	Dimension→Center All	ADIM/CALL
n/a	Dimension→Center One	ADIM/CONE
n/a	Dimension→Configure	ADIM/CNFG
n/a	Dimension→Datum	ADIM/DATM
n/a	Dimension→Diameter	ADIM/DIAM
n/a	Dimension→Leader	ADIM/LEAD
n/a	Dimension→Point	ADIM/PONT
n/a	Dimension→Radius	ADIM/RADI
n/a	Dimension→Unbind	ADIM/UGRP
/rctl	Display→Control Ratsnest	RCTL
/lpan	Display→Long Pan	LPAN
/rulr	Display→Measure	RULR
/pctl	Display→Path/Group Visibility	PCTL
RCL	Display→Recall View	RCL
n/a	Display→Recall View 1	RC1
n/a	Display→Recall View 2	RC2
n/a	Display→Recall View 3	RC3
n/a	Display→Recall View 4	RC4
n/a	Display→Recall View 5	RC5
n/a	Display→Recall View 6	RC6
n/a	Display→Recall View 7	RC7
n/a	Display→Recall View 8	RC8
n/a	Display→Recall View 9	RC9
n/a	Display→Recall View 10	RC10
STO	Display→Store View	STO
n/a	Display→Store View 1	ST1
n/a	Display→Store View 2	ST2
n/a	Display→Store View 3	ST3
n/a	Display→Store View 4	ST4
n/a	Display→Store View 5	ST5
n/a	Display→Store View 6	ST6
n/a	Display→Store View 7	ST7
n/a	Display→Store View 8	ST8
n/a	Display→Store View 9	ST9
n/a	Display→Store View 10	ST10
DRAW/ARC	Draw→2-Point Arc	DRAW/ARC
DRAW/ARCP	Draw→3-Point Arc	DRAW/ARCP
DRAW/CIRC	Draw→Circle	DRAW/CIRC
DRAW/CVOD	Draw→Circular Void	DRAW/CVOD
n/a	Draw→Filled Circle	DRAW/FCIR
DRAW/FREC	Draw→Filled Rectangle	DRAW/FREC
DRAW/FLSH	Draw→Flash	DRAW/FLSH
DRAW/LINE	Draw→Line	DRAW/LINE
DRAW/POLY	Draw→Polygon	DRAW/POLY
DRAW/PVOD	Draw→Polygonal Void	DRAW/PVOD
DRAW/RECT	Draw→Rectangle	DRAW/RECT
DRAW/TEXT	Draw→Text	DRAW/TEXT
n/a	/dly	/dly
/drc	/drc	/drc
/dyn	/dyn	/dyn
EDIT/ADDV	Edit→Add Vertex	EDIT/ADDV
EDIT/AVIA	Edit→Add Via	EDIT/AVIA
ATTR/SCHG	Edit→Attribute	EDIT/ATRB
EDIT/DELS	Edit→Delete Segment	EDIT/DELS

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.6 Command	MD 8.6 Keyboard Equivalent
n/a	Edit→Delete Trace	EDIT/TRCE
EDIT/DELV	Edit→Delete Vertex	EDIT/DELV
EDIT/DVIA	Edit→Delete Via	EDIT/DVIA
n/a	Edit→Keyword	EDIT/KWD
EDIT/MOVA	Edit→Move All	EDIT/MOVA
EDIT/MOVS	Edit→Move Segment	EDIT/MOVS
EDIT/MOVV	Edit→Move Vertex	EDIT/MOVV
EDIT/MVIA	Edit→Move Via	EDIT/MVIA
n/a	Edit→Net Attr	EDIT/NATR
SCMD/EPKG	Edit→Packaging Data	EPKG
SCMD/EPNL	Edit→Packaging Data	EPNL
n/a	Edit→Pin Type	EDIT/SPAT
EDIT/LAYS	Edit→Segment Layer	EDIT/LAYS
n/a	Edit→Trace Width	EDIT/TWID
/undo	Edit→Undo Delete Segment	UNDO
EDIT/WIRE	Edit→Wire	EDIT/WIRE
ATTR/ACOM	Enter→Attribute	ENTR/ATRB
ENTR/ORG	Enter→Board Origin	ENTR/ORG
ENTR/BUSB	Enter→Bus	ENTR/BUSB
ENTR/COMP	Enter→Component	ENTR/COMP
SCMD/SCAT	Enter→Component Type	ENTR/SCAT
SCMD/SNAT	Enter→Global Net	ENTR/SNAT
SCMD/JMPR	Enter→Jumper	ENTR/JMPR
SCMD/NPKG	Enter→Nonhomogeneous Pkg	ENTR/NPKG
ENTR/ORG	Enter→Origin	ENTR/ORG
SCMD/PNLC	Enter→Packaging Data	ENTR/PNLC
SCMD/SPKG	Enter→Packaging Data	ENTR/SPKG
ENTR/PIN	Enter→Pin	ENTR/PIN
ENTR/SEQ	Enter→Pin Sequence	ENTR/SEQ
SCMD/SPAT	Enter→Pin Type	ENTR/SPAT
ENTR/POLY	Enter→Polygon	ENTR/POLY
ENTR/RATN	Enter→Ratsnest	ENTR/RATN
NAME/REFD	Enter→Ref.Des.& Section	ENTR/PNUM
SCMD/PNUM	Enter→Ref.Des.& Section	ENTR/PNUM
/repl	Enter→Replace Component	REPL
ATTR/ACOM	Enter→Sheet Number	ENTR/SHID
ENTR/UCOM	Enter→Uncommit Pin	ENTR/UCOM
ENTR/WIRE	Enter→Wire	ENTR/WIRE
SCMD/LPAR	Environment→Assign Layer Pairs	LPAR
SCMD/GSSF	Environ.→Attach Cust.Sold.Dots	GSSF
SCMD/GSSF	Environment→Attach Padstacks	GSSF
SCMD/UNIT	Environment→Change Units	UNIT
SYS/DOS	Environment→DOS Shell	DOS
DETL	Environment→Detail Mode	DMOD
SYS/STAT	Environment→Display Statistics	STAT
n/a	Environment→Edit Aperture Table	ATE
SCMD/SIPC	Environ.→Inner Plane Apertures	SIPC
/mask	Environment→Mask Items	MASK
/msk	Environment→Mask Items	MSK
SCMD/VMRG	Environ.→Merge Polygon Voids	VMRG
n/a	Environ.→Merge Voids by Layer	VMBL
n/a	Environ.→Merge Voids by Poly	VMBP
SCMD/PSIZ	Environment→Min. Polygon Size	PSIZ
SCMD/PCLR	Envir.→Polygon Wire Clearance	PCLR
n/a	Environ.→Set Minimum Aperture	MAPW
/sgat	Environment→Set Snap Tolerance	SGAT

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.6 Command	MD 8.6 Keyboard Equivalent
SYMB	Environment→Symbol Mode	SMOD
/exe	/exe	/exe
/exec	/exec	/exec
n/a	/fast	/fast
FILE/ZAP	File→Clear Database	FILE/ZAP
SYS/PLOT	File→Create Plot File	FILE/PLOT
FILE/POP	File→Level Pop	FILE/POP
LEVL/POP	File→Level Pop	FILE/POP
FILE/PUSH	File→Level Push	FILE/PUSH
LEVL/PUSH	File→Level Push	FILE/PUSH
FILE/LOAD	File→Load	FILE/LOAD
FILE/BKLD	File→Load Block	FILE/BKLD
n/a	File→Print	FILE/PRINT
SYS/QUIT	File→Quit	FILE/QUIT
FILE/SAVE	File→Save	FILE/SAVE
FILE/BKSV	File→Save Block	FILE/BKSV
/fit	Fit View	FIT
/fitv	Fit View	FITV
n/a	/grid	/grid
GRP/TAG	Group→Add Gate	GRP/TAG
GRP/RNAM	Group→Change Name	GRP/RNAM
GRP/RSET	Group→Remove Definition	GRP/RSET
GRP/UTAG	Group→Revome Gate	GRP/UTAG
IMPR/COMP	Improve Plc→Components	IMPR/COMP
IMPR/GATE	Improve Plc→Gates	IMPR/GATE
/intr	/intr	/intr
/lang	/lang	/lang
/lsty	/lsty	/lsty
/lwid	/lwid	/lwid
/lyrn	/lyrn	/lyrn
n/a	Last View	PREV
/mac	/mac	/mac
n/a	Main Menu	/main
/mend	/mend	/mend
n/a	/menu	/menu
MOVE/ATRB	Move→Attribute	MOVE/ATRB
MOVE/COMP	Move→Component	MOVE/COMP
MOVE/APTH	Move→Critical Path	MOVE/CPTH
MOVE/AGRP	Move→Group	MOVEAGRP
MOVE	Move→Object	MOVE
MOVE/IDEN	Move→Objects	MOVE/IDEN
MOVE/UNDO	Move→Undo	MOVE/UNDO
MOVE/WIN	Move→Window	MOVE/WIN
NAME/COMP	Name→Component	NAME/COMP
NAME/NET	Name→Net	NAME/NET
NAME/PIN	Name→Pin	NAME/PIN
NAME/RSEQ	Name→Reseq. Ref. Desd	NAME/RSEQ
n/a	Name→Reseq. Window	NAME/SEQW
NAME/SUBN	Name→Subnet	NAME/SUBN
/ofly	/ofly	/ofly
n/a	/only	/only
n/a	/osel	/osel
PAN	Pan	PAN
Pan Bars	Shift Pan	[SHIFT]+[Btn 1]
/pdel	/pdel	/pdel
/pend	/pend	/pend
/pkey	/pkey	/pkey

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.6 Command	MD 8.6 Keyboard Equivalent
/pkld	/pkld	/pkld
/pksv	/pksv	/pksv
/pzap	/pzap	/pzap
PLCE	Placement→Automatic Placement	PLCE
BARR/DEL	Placement→Define Barriers→Delete	BARR/DELB
BARR/DELB	Placement→Define Barriers→Delete	BARR/DELB
BARR	Placement→Define Barriers→Enter	BARR
ASSC	Placement→Define Lattices→Associate Discretes	ASSC
LATC	Placement→Define Lattices→Define Components	LATC
LATP	Placement→Define Lattices→Enter Spacing	LATP
CLR	Placement→Define Lattices→Specify Clearance	CLR
CUT	Placement→Enter Cutlines	CUT
FIX	Placement→Fix→Component	FIX
FIX/WIN	Placement→Fix→Window	FIX/WIN
HIST/CNFG	Placement→Histogram→Configure	HIST/CNFG
HIST/GRID	Placement→Histogram→Define Grid	HIST/GRID
HIST/STAT	Placement→Histogram→Generate Statistics	HIST/STAT
HIST/RSET	Placement→Histogram→Reset Merit Factor	HIST/RSET
UNFX	Placement→UnFix→Component	UNFX
UNFX/WIN	Placement→UnFix→Window	UNFX/WIN
n/a	Query→Apertures	QRY/APRS
QRY/COMP	Query→Component	QRY/COMP
QRY/APTH	Query→Critical Path	QRY/APTH
QRY/AGRP	Query→Group	QRY/AGRP
QRY/NET	Query→Net	QRY/NET
QRY/NPIN	Query→Net Pins	QRY/NPIN
n/a	Query→Net Vias	QRY/NVIA
QRY	Query→Object	QRY
QRY/PSTK	Query→Padstack	QRY/PSTK
QRY/PIN	Query→Pin	QRY/PIN
n/a	Query→Polygon	QRY/POLY
n/a	Query→Polygon Window	QRY/PWIN
n/a	Query→Properties	QRY/PROP
n/a	Query→Trace	QRY/TRCE
QRY/VIA	Query→Via	QRY/VIA
/rcmn	n/a	n/a
RCL	Recall View	RCL
REDR	Redraw	REDR
n/a	/regn	/regn
/resu	/resu	/resu
ROT/COMP	Rotate→Component	ROT/COMP
ROT	Rotate→Object	ROT
ROT>IDEN	Rotate→Objects	ROT>IDEN
ROT/UNDO	Rotate→Undo	ROT/UNDO
ROT/WIN	Rotate→Window	ROT/WIN
/stgl	/stgl	/stgl
SWAP/COMP	Swap→Component	SWAP/COMP

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.6 Command	MD 8.6 Keyboard Equivalent
SWAP/GATE	Swap→Gate	SWAP/GATE
SWAP/PIN	Swap→Pin	SWAP/PIN
SWAP/UNDO	Swap→Undo	SWAP/UNDO
n/a	/tmod	/tmod
/ucp	/ucp	/ucp
/undo	/undo	/undo
VLJR	View Layer	VLJR
VSAV	n/a	n/a
VWIN	View Window	VWIN
/wait	/wait	/wait
x (full screen cursor hot key)	x (full screen cursor hot key)	x
x	x	/xhar
ZIN	Zoom In	ZIN
ZOUT	Zoom Out	ZOUT

Table E-2 describes the default hot-key assignments for MD 8.6 commands or functions. Hot keys for subcommands will be recognized only if the main menu command has already been executed. For example, to move a wire vertex, press **E** to activate *Edit*, then **G** to execute the *Move→Vertex* command.

Table E-2. Default Hot Key Key Functions

Key	Command or Function
a	Toggle wire bend
b	Delete Vertex
c	Toggle curved wire (PCB Editor only)
d	Delete
e	Edit
f	File
g	Move Vertex
h	Delete Segment
i	Wire
j	Move Segment
k	Component
l	Load
m	Move
n	Enter
o	Objects
p	Pan
q	Query
r	Rotate
Alt+R	rotate clockwise
Ctrl+R	rotate counterclockwise
s	Save
t	Trace (PCB Editor only)
u	Undo
v	Add Vertex
w	Window
x	Cursor type (regular, vertical or diagonal cross hair)
y	Segment Layer
z, +	Zoom In
Z*, -	Zoom Out

* Note: Except for Z only lower-case keys have default functions.

Table E-3 contains a chart of the ASCII characters available for hot-key assignment in MD 8.6.

Table E-3. ASCII Key Codes Available For Hot Key Assignment

		ASCII Key Codes									
↓ Left Digit	Right Digit	0	1	2	3	4	5	6	7	8	9
3					!	"	#	\$	%	&	'
4		()	*	+	,	-	.	n/a	0	1
5		2	3	4	5	6	7	8	9	:	;
6		<	=	>	?	@	A	B	C	D	E
7		F	G	H	I	J	K	L	M	N	O
8		P	Q	R	S	T	U	V	W	X	Y
9		Z	[\]	^	-	'	a	b	c
10		d	e	f	g	h	i	j	k	l	m
11		n	o	p	q	r	s	t	u	v	w
12		x	y	z	{		}	~			

ASCII key codes range from 00 to 127. The following characters or ASCII key codes aren't available for hot-key assignment:

- 00 to 31 and 127 aren't available because they're nonprintable control characters
- 47 (/) is reserved for initiating keyboard commands
- 32 represents a blank and isn't available for user assignment
- **Shift**, **Ctrl**, **Alt**, **⌘**, **<**, and **>** can't be used to assign commands or functions to P-CAD hot keys

This appendix lists a command cross-reference, default hot key functions, and a chart of keys available for hot keys.

Index

—\$—

\$ATT Layer
 net attribute graphics, 28

—A—

Apertures
 scaling, 29
arrow, 7
asterisk, 7
attribute keywords (P-CAD)
 attribute keywords (P-CAD), 87

—B—

button menu trees, 91
ByName
 selecting components, 14

—C—

CCT SPECCTRA
 net attribute support, 29
Changing directories
 with file selectors, 12
command cross reference, 99
Components
 consistency, 48
 definition, 47
 selecting by name, 14
 updating in database, 48
cycle, 8
cycle symbol, 8

—D—

Deleting
 unused layers, 18
Design Manager
 standardizing design files, 13
Design Rules Checking
 enhanced net attribute support, 25
 net attribute graphics on \$ATT
 layer, 28
device names (DOS), 87
Dimensioning
 centered horizontal text, 18

DOS

 reserved device names, 87

—F—

File Selectors
 enhancements, 12
filename extensions, 83
Filenames
 when entering components, 47
Files
 acceptable characters for names,
 14
 standardizing in Design Manager,
 13

—H—

Hardcopy
 scaling apertures, 29
Hot keys
 rotating components, 15
 zooming, 15

—I—

Improved selection mechanism
 defined, 9
Installing
 Master Designer UNIX from CD-
 ROM, 28

—K—

keywords
 attribute, 87

—L—

Layers
 deleting unused, 18

—M—

Master index, 29
menus
 slide-off, 7
Move component
 enhanced ratsnest display, 17

—N—

Net attributes
 CCT SPECCTRA support, 29
 new for MD 8.6, 15
New net attributes
 CLEARANCE, 15
 PAD-TO-EDGE, 15
 PAD-TO-TRACE, 15
 TRACE-TO-EDGE, 15
 TRACE-TO-TRACE, 15
 VIATYPE, 15

—O—

Object selection commands
 improved selection mechanism, 9
 toggling between (un)selected
 objects, 11
Online documentation
 AIX support, 29
 master index, 29
Opcodes
 updated list, 22

—P—

P-CAD
 P-CAD
 attribute keywords, 87
PC-WinPlot
 defining text stroke widths, 31
 multiple plot files, 30
Pins
 information report, 32

Plot files
 plotting multiple files, 30

—R—

Ratsnest display
 enhanced performance, 17
Reference Designators
 auto-incrementing, 23, 50
Report Generator
 pin information report, 32
reserved words, 87
Rotating components
 hot keys, 15

—S—

Schematic Editor
 auto-incrementing reference
 designators, 23, 50
Selecting components by name, 14
Selecting objects
 improved selection mechanism, 9
 toggling between selected and
 unselected, 11
system limits, 79

—W—

wildcard, 7

—Z—

Zooming
 hot keys, 15