

Copyrights

Software, documentation and related materials:
Copyright © 2002 Altium Limited

This software product is copyrighted and all rights are reserved. The distribution and sale of this product are intended for the use of the original purchaser only per the terms of the License Agreement.

This document may not, in whole or part, be copied, photocopied, reproduced, translated, reduced or transferred to any electronic medium or machine-readable form without prior consent in writing from Altium Limited.

U.S. Government use, duplication or disclosure is subject to RESTRICTED RIGHTS under applicable government regulations pertaining to trade secret, commercial computer software developed at private expense, including FAR 227-14 subparagraph (g)(3)(i), Alternative III and DFAR 252.227-7013 subparagraph (c)(1)(ii).

P-CAD is a registered trademark and P-CAD Schematic, P-CAD Relay, P-CAD PCB, P-CAD ProRoute, P-CAD QuickRoute, P-CAD InterRoute, P-CAD InterRoute Gold, P-CAD Library Manager, P-CAD Library Executive, P-CAD Document Toolbox, P-CAD InterPlace, P-CAD Parametric Constraint Solver, P-CAD Signal Integrity, P-CAD Shape-Based Autorouter, P-CAD DesignFlow, P-CAD ViewCenter, Master Designer and Associate Designer are trademarks of Altium Limited. Other brand names are trademarks of their respective companies.

Altium Limited
www.altium.com

Table of Contents

chapter 1	InterPlace Introduction	
	P-CAD IPL Features.....	1
	About this Guide.....	2
	IPL within the P-CAD Product Suite	2
	P-CAD PCB Icon.....	3
chapter 2	Installation and Setup	
	System Requirements	5
	Recommended System.....	5
	Minimum System.....	5
	Installing P-CAD Products	6
chapter 3	InterPlace Basics	
	Using InterPlace.....	7
	PCB and Relay Designs in IPL.....	7
	Schematic Designs in IPL	8
	About the User Interface.....	8
	InterPlace Interface.....	8
	Menu Bar.....	9
	InterPlace Toolbars.....	9
	Prompt Line.....	11
	Status Line	11
	Data Tips.....	13
	InterPlace Views	13
	Loading, Saving and Exiting a Design.....	15
	Loading a Design	15
	Saving the Design	17
	Exiting InterPlace	17
	Right Mouse Commands.....	17
chapter 4	The Layout View	
	Working with the Layout View	21
	Viewing the Layout View	21
	Loading Designs	22
	Using Multiple Windows	24

Display Settings.....	26
Working with Components.....	27
Locating Components.....	27
Clustering Components.....	27
Locating a Specific Component.....	28
Jumping to a Location.....	28
Cross-Probing Between Applications.....	28
Placing Components.....	28
Component Placement.....	28
Fixing Components.....	29
Rotating Components.....	29
Flipping Components.....	30
Discrete Component Placement.....	30
Moving Components.....	30
Accessing Component Information.....	31
Using Net Connections.....	32
Accessing Connection Information.....	32
Setting Display Options of Specific Nets.....	33
Optimizing Net Connections.....	33
Modifying Connection Properties.....	34
Layout View Right Mouse Commands.....	34

chapter 5 The Design Manager View

Working with the Design Manager View.....	37
Accessing the Design Manager View.....	38
Accessing Design Manager Pages.....	38
Selecting Items in Design Manager.....	39
Common Design Manager View Commands and Functions.....	39
Using the Components Page.....	42
Component Information.....	42
Sorting the Components.....	43
Using the Nets Page.....	43
Net Information.....	43
Showing and Hiding Net Connections.....	44
Using the Partitions Page.....	44
Partition Information.....	44
Creating Partitions.....	45
Deleting Partitions.....	45
Renaming Partitions.....	46
Adding Components to a Partition.....	46
Removing Components from a Partition.....	46
Using the Rooms Page.....	46
Rooms Information.....	46
Adding Components to a Room.....	47
Removing Components from a Room.....	47
Using the Net Class Page.....	47
Net Class Information.....	48
Creating a Net Class.....	48

Deleting a Net Class.....	49
Renaming a Net Class	49
Adding Nets to a Net Class	49
Removing Nets from a Net Class	49
Using the Class-to-Class Page.....	49
Class-to-Class Information	50
Creating a Class-to-Class	50
Deleting a Class-to-Class	50
Design Manager Right Mouse Commands.....	51

chapter 6 Organizing and Placing Components

Organizing Design Objects.....	55
The Design Manager View	56
The Components Page	57
The Nets Page	57
The Partitions Page.....	57
The Rooms Page	59
The Net Class Page	60
The Class-to-Class Page	60
Placing Design Objects	60
Finding and Gathering Components.....	60
Placing Components	63
Discrete Placement	63
Fine-Tuning Component Placement.....	66

chapter 7 Working with PCB and Schematic

Basic Inter-Application Functions	69
Launching InterPlace.....	69
Loading a Design	69
Cross-Probing Between Applications	70
Using a Design Technology Parameters File	70
Accessing a DTP	71
DTP Operations	71
How InterPlace Updates Affect PCB and Schematic.....	72
Recording Changes with ECOs.....	72

chapter 8 The Visible Placement Area

Working with the Visible Placement Area.....	75
The VPA Toolbar	75
VPA Domain Display Options.....	76
Enabling and Disabling a VPA Domain	76
Constraint Domains.....	76
Viewing Constraint Domains	77
Constraint Visualization.....	77
Physical Constraints.....	78
Electrical Constraints.....	79
Room Constraints.....	79

chapter 9 File Commands

File Load PCB Design	81
File Load Schematic Design	82
File Close.....	82
File Update PCB.....	82
File Update Schematic.....	82
File Reports	83
Filename.....	83
Report Options	84
Page Format.....	84
Style Format	84
Lines per Page	84
Report Destination Window	84
Generate	84
File Design Technology Parameters.....	85
File Exit.....	85

chapter 10 Edit Commands

Edit Undo (Ctrl+Z).....	88
Edit Properties	88
Right Mouse Button to Select Properties.....	88
Double Click to Select Properties	88
Data Tips	88
Component Properties.....	89
Edit Move By RefDes.....	90
Edit Align Components	91
Align Horizontally or Vertically	91
Align to Grid.....	91
Edit Fix Components	92
Edit Unfix Components	92
Edit Swap Components	92
Edit Rotate To.....	92
Edit Place On.....	92
Edit Cluster By.....	93
Edit Select All	93
Edit Deselect All.....	93
Edit Highlight	93
Edit Unhighlight	94
Edit Unhighlight All.....	94
Edit Measure	94
Edit Select	95
Select Actions.....	95
Select Commands	95
Selecting Objects.....	95
When Objects Overlap	96
Moving Objects.....	96
Rotating and Flipping.....	96

Viewing Properties	97
Right Mouse Button Commands	97
Edit Discrete Placement	97

chapter 11 View Commands

View Redraw	100
View Extent	100
View Last	100
View All	100
View Center	101
C Key	101
View Zoom In	101
Plus Key (+)	101
View Zoom Out	101
Minus Key (-)	102
View Zoom Window	102
Zoom through a Window	102
View Jump Location	102
Jump to a Location	103
View Command Toolbar	103
View Prompt Line	103
View Status Line	104
View Snap to Grid	104
View Layout View Options Toolbar	104
View VPA Toolbar	104
View Design Manager	104

chapter 12 Options Commands

Options Block Selection	105
Items	106
Selection Mask Parameters	107
Layers	107
Select Mode Box	107
Selecting and Modifying	108
Options Configure	108
Options Grids	110
Mode	110
Visible Grid Style	110
Relative Grid Origin	110
Grid Spacing: Uniform/Nonuniform	111
Grid Toggle Button (or G key)	111
Options Display	111
Options Preferences	114
Keyboard Tab	115
Mouse Tab	116
Options Design Rules	116
Design Tab	117

Layer Tab	117
Rooms Tab.....	118
Net Class Tab.....	118
Net Tab.....	119
Class to Class Tab	120

chapter 13 Window Commands

Window New Window	121
Window Cascade.....	121
Window Tile	121
Window Arrange Icons.....	122
Selecting a Window	122

chapter 14 Help Commands

Help P-CAD InterPlace Help Topics	123
Help How to Use Help	123
Help About P-CAD InterPlace.....	123

Appendix A Keyboard Reference

InterPlace Keyboard Reference.....	126
------------------------------------	-----

Index	129
--------------------	-----

InterPlace Introduction

Congratulations on your purchase of P-CAD InterPlace. P-CAD InterPlace (IPL) is a productivity-enhancing tool that integrates seamlessly into the P-CAD suite of circuit board design products.

InterPlace excels at organizing the variety of objects in a design. Components can be arranged in logical or physical groups and easily gathered together in preparation for placement on the circuit board. In addition, InterPlace incorporates the design rules, or constraints, used to ensure successful placement, routing, or manufacture of a printed circuit board.

Today's engineers face the challenge of designing faster circuits of increasing complexity on smaller boards. In addition, these boards are to be manufactured with minimum time and cost. InterPlace provides you with powerful visualization and placement tools, which help you to quickly avoid constraint violations or resolve constraint conflicts.

P-CAD InterPlace is an advanced placement tool for the Microsoft® Windows 95™, Windows 98™, Windows 2000™, or Windows NT™ operating systems. InterPlace contains a combination of features designed to significantly reduce the number of design iterations needed to produce a successful board layout.

P-CAD IPL Features

This section highlights some of the important P-CAD InterPlace features:

- New and unique ways to view and organize your designs. You can simultaneously view and manipulate the arrangement of the components and nets on the physical board layout. The Layout View of IPL graphically depicts component placement and connectivity. In addition, the Design Manager allows you to view the design contents in terms of components, nets, partitions, rooms, net classes and class-to-class rules.
- IPL is a component organization and preplacement tool. With IPL you can organize components and prepare your board layout before you begin placement. Components can be grouped in logical partitions and physical rooms and quickly clustered from diverse locations to make placement easier. You can swap components, fix them in place and rotate them around their own reference points. Net connections are optimized on-the-fly when components are moved. You can also select components and drag and drop them into other locations, even another window.

- Easy placement of discrete components. Discrete components, such as bypass caps, can be associated with a particular component with just a mouse click using the Discrete Placement function. Eliminate searching among the typically large quantities of discrete components by using the Discrete Placement function to locate them for you. You can choose to locate these components by Type or Value. IPL dynamically tracks available and placed components and displays updated counts with each placement.
- Application integration. P-CAD Schematic, P-CAD PCB and P-CAD Relay software is tightly integrated with P-CAD InterPlace, assuring the synchronization of design file updates between the applications.
- Productivity features that make design work easier. Increasing productivity is simple when you put InterPlace features to use. You can easily drag components from a variety of locations and simply drop them into the board layout. Swap components with just a couple clicks of the mouse. You can quickly fix a component in its place. IPL optimizes network connections on-the-fly when components are moved. Group components with like characteristics, like the same reference designators, type, etc., using the **Cluster By** command. Rotate components to an absolute degree using their own reference points.
- Valid Placement Area Visualization. P-CAD InterPlace provides a visible placement area for your components based on design constraints. Using the Visible Placement Area tools, you can quickly view the exact physical location on the board where a component can be placed without violating a constraint restriction. Being able to place the component knowing it will not violate a design rule reduces errors and saves correction time.

About this Guide

This manual provides information about P-CAD InterPlace. It includes the following sections:

- *Getting Started*: Chapters 2 and 3 tell you what you need to get started using P-CAD InterPlace. It provides installation instructions and walks you through the basic capabilities of P-CAD IPL and its interaction with P-CAD Schematic, PCB or Relay.
- *Using InterPlace*: Chapters 4 through 8 provide information that you need to work with P-CAD IPL. They give you details on organizing design objects in the Design Manager Layout View using rooms and partitions, and placing design components on your board layout.
- *InterPlace Reference*: Chapters 9 through 14 include an extensive command reference, covering all of P-CAD InterPlace.

IPL within the P-CAD Product Suite

The interface of InterPlace varies depending on the design file you have loaded. For instance, designs loaded from P-CAD Schematic have no board layout presentation of the design, therefore Layout View activities are not available. When no icon is displayed, the feature is available for all designs.

The icon may also be found in the chapter introduction, if all features in that chapter are design specific.

P-CAD PCB Icon



When a design is loaded from PCB, the full suite of InterPlace features is available. When a feature is exclusive to a PCB design, the PCB icon is display in the left margin.

Installation and Setup

This chapter lists the required hardware and software settings you need to install the P-CAD Suite.

System Requirements

Make sure that your PC and its software conform to the following P-CAD requirements and recommendations.

Recommended System

- Windows NT 4/2000 Professional
- PC with Pentium III Processor
- 128MB RAM (256MB for high component/net count)
- 400MB Hard Disk Space
- Desktop area 1024x768 pixels
- 32-bit Color Palette
- CD-ROM Drive
- Mouse or compatible pointing device.

Minimum System

- Windows 95/98/2000Me
- PC with Pentium 166MHz
- 64MB RAM
- 200MB Hard Disk Space (without ISO libraries)
- Desktop area 800x600 pixels
- 256 Color Palette
- CD-ROM Drive
- Mouse.

Installing P-CAD Products

For up-to-date installation information refer to the file `Readme.WRI`, located on the product CD. This file can also be found in the application program folder (`\Program Files\P-CAD 2002`) after installation. Note that the setup program on the Product CD can also be used to Repair or Remove an existing P-CAD Installation.

InterPlace Basics

This chapter introduces many of the basic features you need to know when using P-CAD InterPlace. It includes general information on such topics as:

- Using IPL from PCB, Relay, or Schematic
- IPL user interface.
- Loading, saving, and closing a design.
- Right mouse commands.

What is presented here is a high level look at InterPlace capabilities. These capabilities are detailed in later chapters.

To keep with the easy-to-use tradition of P-CAD applications, much of the basic functionality of P-CAD InterPlace is similar to P-CAD PCB. As a Windows user, you will also be familiar with many of the basic manipulations used for managing designs in IPL.

Using InterPlace

InterPlace is a versatile application that can be used to manage object organization on both a Schematic and a PCB design. For a PCB design, IPL contains additional functionality which speeds component placement such as on-the-fly net optimization, data tips, discrete placement tools and component swap.

The role of InterPlace in the P-CAD product suite is summarized in the sections below.

PCB and Relay Designs in IPL

InterPlace can be invoked from P-CAD PCB or P-CAD Relay to perform object organization, floor planning and component placement. For instance, you can quickly group components into logical groups, or partitions, using the Design Manager View. You can then place these partitioned components into a specific physical board region, or room, in the InterPlace Layout View.

Unless specified, all references to InterPlace features when a PCB design is loaded are also relevant when a Relay design is loaded.

To achieve these functions, when you launch IPL from PCB you have access to the Layout View and the Design Manager. Each of these aspects of the InterPlace interface is discussed in *About the User Interface* below.

Schematic Designs in IPL

InterPlace can be activated from P-CAD Schematic to view design data and organize design components. From Schematic, you may use IPL to define logical groupings of components and placement requirements to keep the PCB Designer informed of your design intentions.

To achieve these functions, when you launch IPL from Schematic you have access to the Design Manager. Each of these aspects of the InterPlace interface is discussed in *About the User Interface* below.

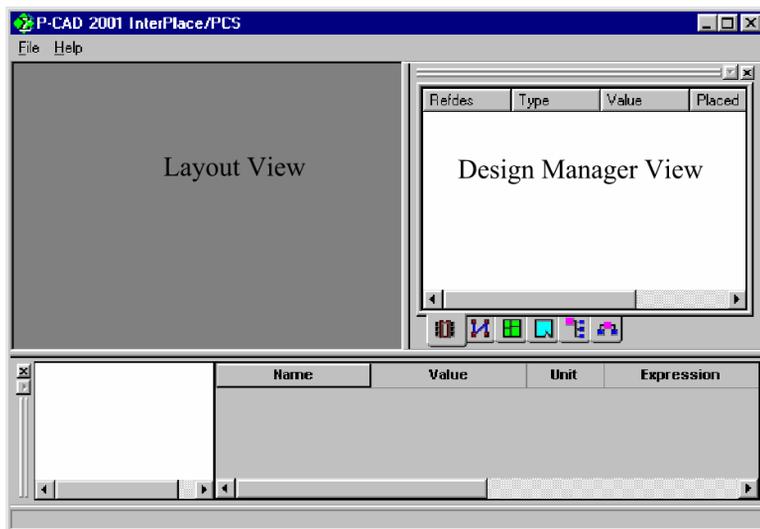
The component placement interface, the Layout View, is not available in InterPlace for a Schematic design.

About the User Interface

The P-CAD InterPlace interface follows the standard Windows format, with the addition of IPL specific controls. This section provides an overview of the user interface, including the menu bar, toolbars, and basic screen layout.

InterPlace Interface

When you first access P-CAD InterPlace, the interface displays an empty area for each of the views as shown below. Some menu functions, except **File » Load PCB Design**, **File » Load Schematic Design**, **Help** and **Exit** are grayed, indicating they are unavailable.



When you load a design file from a P-CAD application, the design information is displayed in the appropriate viewing areas. The InterPlace workspace views include:

- The Layout View
- The Design Manager

Designs loaded from P-CAD PCB, Relay or Schematic provide the design information as it pertains to the Design Manager. When a design is loaded from PCB, InterPlace also graphically displays the board in the Layout View.

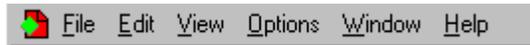
Each of these InterPlace views is summarized in the InterPlace Views section.

Menu Bar

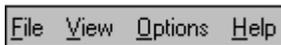
The menu bar allows you easy access to InterPlace commands and functions. The menu commands available depend on whether the design is loaded from PCB or Schematic, since the function of IPL differs for each application.

Before you load a design, the IPL menu bar contains only the **File** and **Help** menu items. From the **File** menu you can choose to load a PCB or Schematic design.

When you load a design from PCB, the menu bar of IPL appears as follows:



A design loaded from Schematic activates the abbreviated menu bar as shown below:



To activate a menu, click the menu title or press the **Alt** key in combination with the underlined letter of the menu title (e.g., **Alt+F** to open the **File** menu). When the menu appears, click a menu item, or press the underlined key, to enable a command.

- A command followed by three dots (e.g., **Open...**) opens a dialog when you choose it.
- The menu bar wraps if you reduce the width of the window. In contrast, other areas of the screen truncate with screen reduction.

InterPlace Toolbars

Toolbars provide shortcuts to commonly used commands and functions. Toolbars appear when you activate the appropriate View command. Once a toolbar is visible on your screen, you can use your mouse to drag it to a new position. It can be docked along the edges of your screen, or left floating in the middle of your display. A floating toolbar can be resized.

Tool Tips explain each of the toolbar buttons. To activate a Tool Tip, simply place the mouse over the button.

P-CAD InterPlace has one basic toolbar: the command toolbar. When a design is loaded from PCB, an additional Layout View options toolbar is available for each placement interface window.

These toolbars are detailed in the sections below.

Command Toolbar



The Command Toolbar provides shortcuts to menu commands and other system functions.

The Command Toolbar is attached to the main workspace, or Layout View. When a design has been loaded into InterPlace from PCB the toolbar shortcuts apply to the Layout View and appear as follows:



The buttons on the command toolbar perform the following functions:

	Edit Select
	File Update PCB/Schematic
	Edit Undo
	Edit Redo
	Edit Measure
	View Zoom
	Edit Discrete Placement

When a design has been loaded into IPL from Schematic the toolbar shortcuts apply only to the **File » Update Schematic** command.

Layout View Options Toolbar



The Layout View Options toolbar is available only when the design is loaded into InterPlace from PCB. These commands apply to specific Layout View windows. If you have multiple Layout View windows open on a design, each window has its own toolbar. Actions from this toolbar impact only the design in its specific window.

The Layout View Options toolbar provides board layer control and activates the Visible Placement Area, and appears as follows in the Layout View workspace:



Layout View Board Control: The buttons that appear on the Layout View Board Control options toolbar allow you to choose the current layer and enable or disable the background display of the

opposite layer. The buttons are: Top Current, Top Only, Bottom Current, and Bottom Only. They are grouped in pairs and only one button in one pair can be active at any time.

The first button in either pair enables the current layer, either the Top or Bottom. Once the current layer is selected, the second button in the pair enables or disables the display of the opposite layer. For example, the toolbar displayed above indicates that the Top layer is the current one, and the display of the bottom layer is enabled.



Visible Placement Area: The buttons that appear on the Layout View VPA toolbar are used to enable the display of an area on the board where a selected component can be placed without violating existing constraints.

The Visible Placement Areas are divided into three domains: Physical, Electrical and Room. Within each domain certain constraints, such as clearances and maximum net lengths, are used to calculate and visually display an error-free location in which to place the selected component.

Each constraint domain is enabled by clicking the appropriate button, and disabled by clicking it again.

Prompt Line



The prompt line lies below the work area and is the first line of the prompt/status line area, extending the complete width of the InterPlace display (the second line being the status line). It can be displayed or hidden using the **View » Prompt Line** command. When the prompt line has not been enabled, the area is empty.

Click <Left> to single Select, <Ctrl><Left> for multiple, or drag for block select.

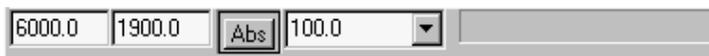
When you first enable a command or process, the prompt line displays a prompt message that provides useful instructions on what to do when a certain tool or item is selected.

Status Line



The status line area, located at the bottom of the screen, can be displayed or hidden using the **View » Status Line** command. A check mark next to the status line indicates that the status line is activated.

The status line is available when a design is loaded into InterPlace from PCB. The status line includes additional functionality for the placement interface windows: the X and Y coordinates and the grid toggle buttons. The status line of IPL is pictured below:



Status Information Area

Displays information relevant to the action you are currently performing.



Comp U12 selected.

The information area displays the following types of data:

- Identifies selected objects either specifically (reference designator) or generally (number of items selected).
- The delta X and delta Y measurements of objects being moved.

X and Y Coordinates

These two values in the lower-left corner of the status line show you the position of the cursor as you move it over the Layout View workspace. The display lets you enter the exact X and Y coordinates of specific points on the screen.

Keyboard shortcuts listed below are defaults; you can change them using the **Options » Preferences** command.

If the Select tool is enabled, these boxes act as a shortcut for the View Jump Location command:

1. Press the **J** key to get focus on the X coordinate edit box.
2. Type an X coordinate value.
3. Press the **Tab** key to move to the Y coordinate box.
4. Type a Y coordinate value.
5. Press **Enter**. The cursor moves to the new X, Y coordinate, scrolling the workspace if necessary.

The coordinate values express either millimeters or mils, depending on the settings in the *Options Configure* dialog. The decimal point for mm is set at hundredths and for mils at tenths. Negative numbers appear in relative grid mode only when the relative origin point is somewhere other than the lower-left (absolute) origin.

Grid Toggle Buttons

The **Grid Toggle** button and the grid combo box beside it allow you to easily switch between grid settings and add new grid settings. The toggle buttons switch between absolute grid  and relative grid . Your absolute and relative grid values can be changed from the combo box. The **A** key also toggles between absolute and relative grids.

Absolute grid always uses the lower-left corner of the workspace as the origin point (X and Y are both zero). Relative grid allows you to specify any point as an origin point. You can even make the origin point for relative the same as for absolute, thereby making the toggle button a combination of grid sizes rather than absolute vs. relative.

Use the drop down combo box to select a new grid and to add new grids to your design.



100mil 

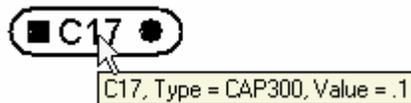
To add a new grid, type a new value and press **Enter**. The new grid becomes the current grid. The **G** key scrolls forward through the list of grid settings. **Shift+G** scrolls back through the list.

Data Tips



Data tips are available in the Layout View when the design is loaded into InterPlace from PCB. When you place the mouse over a room, a component, a component pad, or a connection, a data tip appears which contains information about the underlying object.

A component data tip is shown in the diagram below:



For rooms, the data tip contains the following room properties:

- Room Name
- Placement Side

For components, the data tip contains the following component properties:

- Reference Designator
- Component Type
- Value

For component pads, the data tip contains the following component pad properties:

- Component Reference Designator
- Component Pad Number
- Net Name

For connections, a data tip contains the following connection properties:

- Net Name
- Reference Designators of Connected Components
- Pad Numbers of Connection Endpoints

You can turn off the display of the properties data tips in the *Options Display* dialog.

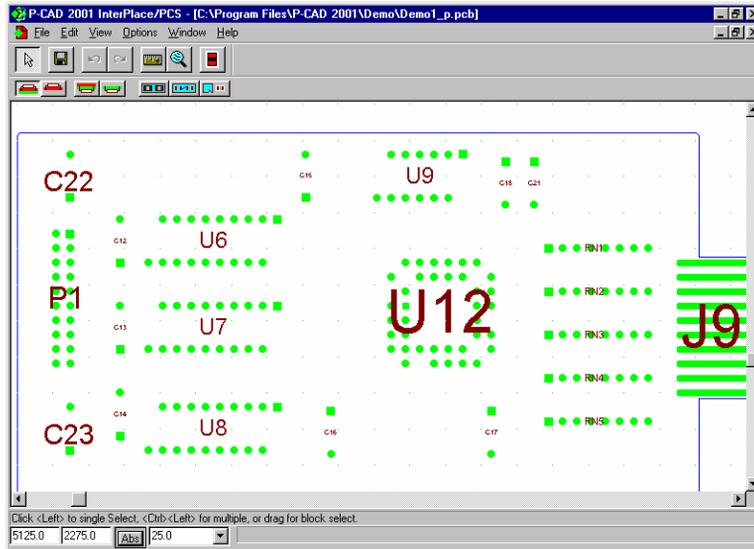
InterPlace Views

Each view in InterPlace specializes in a unique design function. The Layout View provides the graphical representation of the PCB design and the workspace to physically arrange components on the board outline. The Design Manager View is used to organize design objects and provide many different aspects of the design's data.

The Layout View



When a design is loaded into InterPlace from PCB, you can reconfigure the board's components in a Layout View window. The Layout View window shown below displays the board layout of a design:



In the IPL Layout View window, the board outline, components, and other design objects that influence component placement are displayed. Only the Top and/or Bottom layers are visible.

Several windows can be open simultaneously. In these windows, you can view multiple aspects of the same design. For example if components are moved in one window, their new location is reflected in the other open design windows. Components can even be moved between windows. You can easily switch between windows by clicking on the window tabs at the bottom left of your workspace.

For more information about loading a design and using the Layout View of InterPlace, refer to *The Layout View*, (page 21).

The Design Manager

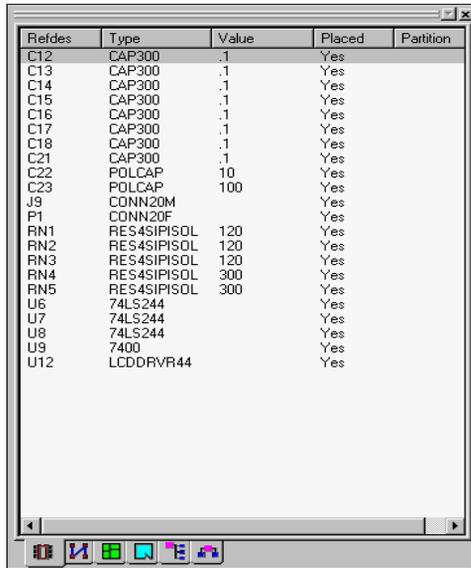
The Design Manager View provides a simple interface for organizing groups of design components and affords easy access to information about the nets, net classes and class-to-classes in your design.

With InterPlace, you can divide the design into smaller logical blocks, or partitions, and organize the components within these divisions. You can also manage physical regions on the PCB board layout, called rooms, where the partitions can be assigned.

The Design Manager view may be enabled or disabled using the **View » Design Manager** command. When visible, the view can be disabled by clicking the appropriate button at the top right corner.

The Design Manager can be resized by positioning the cursor on the edge of the line you want to move, then dragging it to the right or left and releasing the button at the desired location. Additionally, if you double-click on the gray lines at the top of the view, the view is undocked and floats in the middle of your display. You can then use your mouse to drag it to a new position.

When enabled, the view appears in your workspace as shown below:



Refdes	Type	Value	Placed	Partition
C12	CAP300	.1	Yes	
C13	CAP300	.1	Yes	
C14	CAP300	.1	Yes	
C15	CAP300	.1	Yes	
C16	CAP300	.1	Yes	
C17	CAP300	.1	Yes	
C18	CAP300	.1	Yes	
C21	CAP300	.1	Yes	
C22	POLCAP	10	Yes	
C23	POLCAP	100	Yes	
J9	CONN20M		Yes	
P1	CONN20F		Yes	
RN1	RES4SIPISOL	120	Yes	
RN2	RES4SIPISOL	120	Yes	
RN3	RES4SIPISOL	120	Yes	
RN4	RES4SIPISOL	300	Yes	
RN5	RES4SIPISOL	300	Yes	
U6	74LS244		Yes	
U7	74LS244		Yes	
U8	74LS244		Yes	
U9	7400		Yes	
U12	LCDDRVR44		Yes	

For more information about using Design Manager, refer to *The Design Manager View* (page 37).

Loading, Saving and Exiting a Design

This section details basic manipulations within InterPlace, including loading and updating a design, as well as exiting the application.

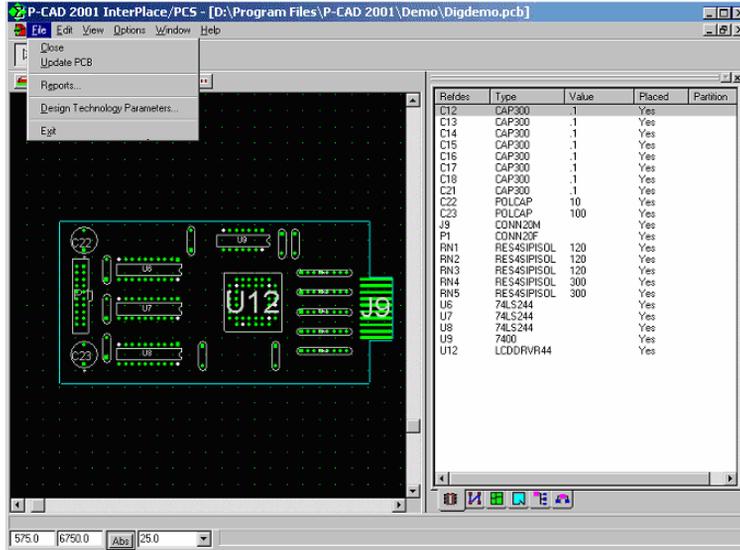
Loading a Design

To load a design into P-CAD InterPlace, the design must first be opened in P-CAD PCB, P-CAD Relay, or P-CAD Schematic.

Once the design is open and active in the originating P-CAD application, choose the **Utils » P-CAD Design Constraint Manager** command. The **Design Constraint Manager** command is used to launch InterPlace and/or Parametric Constraint Solver, depending on which product licenses you have purchased. You can also launch InterPlace from your Windows desktop.

From the InterPlace **File** menu choose the **Load PCB Design** or the **Load Schematic Design** command to begin the design loading process.

After a design has been loaded, the **File » Load PCB Design** and **File » Load Schematic Design** commands are changed to **File » Update PCB** or **File » Update Schematic**, depending on the application from which the design was loaded.



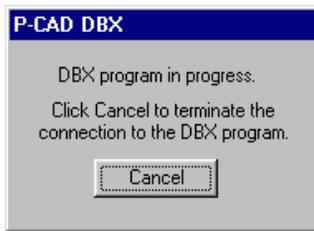
The design is loaded automatically and the InterPlace workspace displays the appropriate viewing areas. The Design Manager View is always displayed. If the design is loaded from PCB, the Layout View appears and contains design objects required for accurate component placement.

Only designs with a complete board outline may be loaded into IPL from PCB or Relay. Extraneous line segments, such as hand drawn dimensions using the line tool, can be moved off the board layer (e.g., to assembly layer) so the design may be loaded.

While a design is loaded, the originating P-CAD application is locked so that conflicting changes to the design or the design constraints cannot be made. Refer to *P-CAD Application Lock* for details.

P-CAD Application Lock

When P-CAD InterPlace is running, the P-CAD application from which the design originated is locked. The lock prevents conflicting changes to the PCB or Schematic design. The following warning is displayed over the PCB or Schematic workspace while the design is loaded into InterPlace.



Cancel terminates the DBX (P-CAD Database Exchange programmer's interface) connection between the applications.

Design information is communicated between the originating application and InterPlace via DBX. If you wish to update the design with changes made in InterPlace, do not click **Cancel**. To unlock the design, use either the **File » Update PCB** or **File » Update Schematic** command to save changes to the original design. Then, close the design or exit InterPlace.

Saving the Design

If you would like to continue working on the current design, use the **File » Update PCB** or **File » Update Schematic** commands or click the **Update PCB** or **Update Schematic** buttons to save the changes to the original PCB or Schematic design file. With any of these commands, the file remains open so you can continue working on it in P-CAD IPL. The current design file name and location are unchanged by this command.

If you have finished working with the current design, choose **File » Close**. This command closes all InterPlace views. The design is unlocked in the originating P-CAD application, permitting changes again in the current PCB or Schematic design editor.

Exiting InterPlace

Choose the **File » Exit** command to exit P-CAD InterPlace.

If an open design has been modified since the last save, you are prompted whether you want to save the changes.

The design files are unlocked in the originating P-CAD application, permitting changes again in the PCB or Schematic design editor.

The program writes information to the `.ini` file when you choose **Exit**. This information, which will apply to subsequent InterPlace sessions, consists of parameters and settings such as values set in Options Configure, etc.

Right Mouse Commands

When you select an object and click the right mouse button, a pop-up menu appears providing shortcuts to common commands performed on selected objects. This menu changes depending on the object you select and the view from which they are selected.

The following table summarizes the commands that appear on the pop-up menu along with the P-CAD design in which it is active.

Command	Description	PCB	SCH
Add Selected Components	Adds the selected components to a partition.	Yes	Yes
Add To	Adds selected components to a room.	Yes	
Cluster By	Provides a way to group components in a common area based on a criteria chosen from the pull right list of selection options.	Yes	
Delete	Deletes a selected partition or class-to-class.	Yes	Yes
Delete Net Class	Deletes a selected net class.	Yes	Yes
Hide Connections	Makes the net connections invisible for a selected net.	Yes	
Highlight	Highlights the nets in a Class-to-Class when the connections are visible.	Yes	
Highlight Attached Nets	Highlights nets attached to the selected objects in the current highlight color.	Yes	
Highlight Components	Highlights the selected components in the current highlight color.	Yes	
Highlight Nets	Highlights the selected nets in the current highlight color.	Yes	
Highlight Room and Assigned Components	Highlights a selected room along with the components included in the room.	Yes	
Jump To Component	Positions the cursor on the selected component in the Layout View.	Yes	
Jump to Pin	Positions the cursor on the selected net pin in the Layout View.	Yes	

Command	Description	PCB	SCH
Move To	Moves selected components to a partition, or selected nets to a Net Class.	Yes	Yes
New Class to Class	Adds a new class-to-class to the design.	Yes	Yes
New Net Class	Adds a new net class to the design.	Yes	Yes
New Partition	Creates a new partition.	Yes	Yes
Remove Component	Removes selected components from a partition.	Yes	Yes
Remove Nets	Removes nets from a net class	Yes	Yes
Rename	Renames a partition.	Yes	Yes
Rename Net Class	Renames a selected net class.	Yes	Yes
Show Connections	Makes the net connections visible for a selected net.	Yes	
Show Connections on Drag	Makes the net connections visible for a selected net when components in the net are moved.	Yes	
Select Component	Selects an individual component, or the components assigned to a room or partition.	Yes	
Unhighlight	Removes highlighting from the nets in a Class-to-Class when the connections are visible and highlighted.	Yes	
Unhighlight Attached Nets	Removes highlighting from the nets attached to the selected objects when they are visible and highlighted.	Yes	
Unhighlight Components	Removes highlighting from highlighted components.	Yes	
Unhighlight Nets	Removes highlighting from selected, highlighted nets.	Yes	

Command	Description	PCB	SCH
Unhighlight Room and Assigned Components	Removes highlighting from a highlighted room and its assigned components.	Yes	

The Layout View



In this chapter, the InterPlace Layout View is detailed. Manipulations of the design objects are covered, including controlling design layers, placing and moving components, viewing connections and turning on a Visible Placement Area.

The Layout View is only available if a design has been loaded from PCB or Relay. InterPlace interacts with P-CAD PCB to obtain the most current design information and displays only the information relevant to component placement. Changes made to the design in IPL can later be transferred back to update the PCB design file.

Working with the Layout View

The Layout View in InterPlace displays the graphic presentation of the design. The board outline, components and net connections appear in this area.

Because InterPlace works with design information relevant to component placement, only certain elements of design objects are loaded such as the board outline, connections, components, keepouts, pads and mounting holes and rooms.

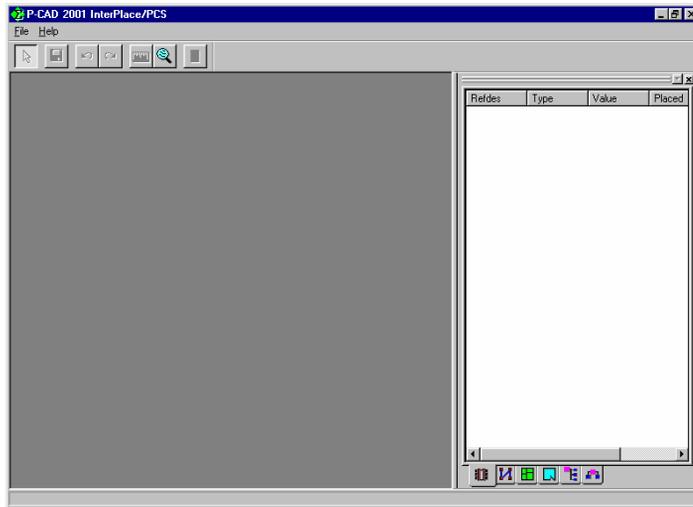
You can display the design in multiple Layout View windows simultaneously. Each open Layout View window's functions can be controlled individually. You can toggle back and forth between the windows, and arrange them in the workspace so they are all displayed at once.

With the Options Display dialog you can control familiar settings such as the background, highlight and selection colors used in the design's display. In InterPlace, the board outline and power pin colors can also be changed. You can modify the cursor style and enable or disable the display of information in the data tips. The font size for the reference designator text can be specified as well.

More information on the *Options Display* dialog can be found in *Options Commands*, (page 105).

Viewing the Layout View

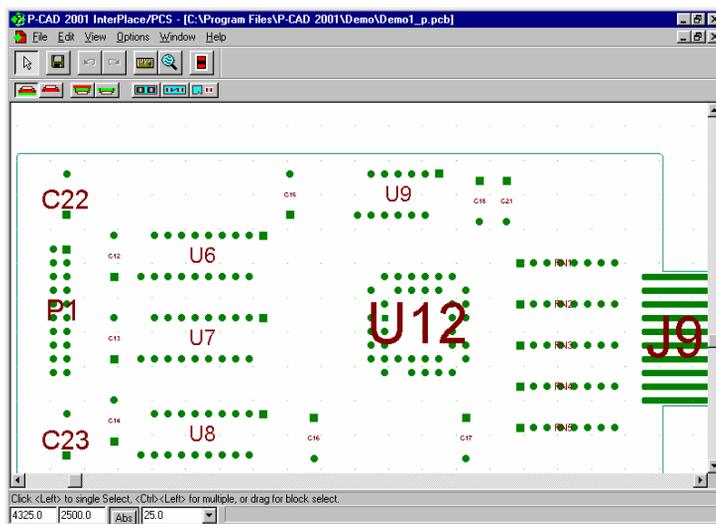
The Layout View of InterPlace is displayed in the workspace along with the Design Manager. Before a design has been loaded the IPL workspace appears as shown below:



The gray area at the left side of the workspace is the Layout View with no design loaded. The right side of the workspace contains the Design Manager view, which is detailed in *The Design Manager View*, (page 37).

Loading Designs

A design is loaded into InterPlace using the **File » Load PCB/Schematic Design** command(s). When the design is loaded from PCB or Relay, the graphic representation of the information pertinent to component placement appears in the Layout View. A loaded design might appear as follows:



The design file data imported into InterPlace is detailed in *Design Objects*.

Design Objects

When loaded into InterPlace, only objects that impact component placement are visible. Visible objects are listed below:

- Board outline
- Connections
- Components
- Keepouts
- Pads or Mounting Holes
- Rooms

The board outline, rooms, keepouts, pads, and mounting holes provide visible indications of where to place a component. These objects can not be selected. Only components may be selected and moved.

Polygon keepouts placed on the Top or Bottom (or All) layer, are transferred into InterPlace. Line keepouts are not transferred.

You may indicate a cut away portion in the interior of the board as surrounded by a contiguous line in PCB. If there exists more than one complete outline, InterPlace interprets each as a board outline. Because a design can have only one board outline, the design will not be loaded into IPL. To indicate the cut away board area, use instead a polygonal keepout on all board layers.

All design objects must be loaded with the original design. New components or new rooms, for example, cannot be added to the InterPlace workspace after the design has been loaded.

The table below shows the display characteristics of the design objects loaded into the Layout View:

Data Loaded	Display Characteristics
Board Outline	Lines and arcs only.
Keepouts	Only Poly Keepouts defined as top, bottom and all layer are displayed. Poly Keepouts appear in the color(s) selected in the IPL Options Display dialog.
Mounting Holes	Holes are displayed in a close approximation of their original shape.
Free Pads	Free pads are displayed in a rounded, rectangular shape.

Data Loaded	Display Characteristics
Rooms	Rooms are transparent with boundaries that use the color(s) selected in the IPL Options Display dialog. Rooms appear on a layer according to their PlacementSide attribute value (top, bottom or both).
Layers	Only Top and Bottom layers are displayed.
Grids	Grids mirror the settings from the PCB design. New grids can be added, but are not exported back to PCB.
Components	<p>Lines, arcs and pads are imported and displayed in their original size. The RefDes and object outlines are displayed in the color(s) selected in the IPL Options Display dialog and include selected, fixed, highlight and layer colors.</p> <p>All components of a single type are represented as a single instance in IPL. Silk screen edits done in PCB may not be displayed in IPL.</p>

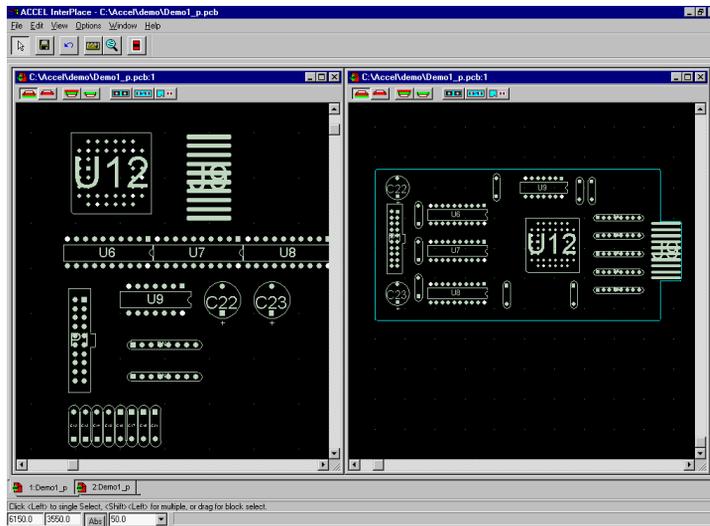
Using Multiple Windows

P-CAD InterPlace allows you to open multiple windows on the same design to view different parts of it simultaneously. To open multiple windows, choose the **Window » New Window** command.

Opening multiple windows of a design can be used to view the Top and Bottom layers of the same design. Or, multiple windows may be used to work on two different regions of the board simultaneously. Also, you may want to zoom in on a group of unplaced components in one window; while in the other window, you focus on the board region where this component group will be placed. A component can be selected, dragged across the window boundaries, and placed on the board in another window.

Each window you open creates a Layout View window inside the main application window in which you can edit the design. As is standard in Windows, you can change the length and width of each window in relation to the other windows running on your screen. You can also activate or deactivate other screen areas such as the Design Manager, toolbars and status/prompt line areas, which alternately decreases or increases the available space in the application window.

An InterPlace workspace with multiple views of the same design is shown below:



You can easily switch between windows by clicking on the window tabs at the bottom left of your workspace:



Board Layer Controls

Each window in P-CAD InterPlace has a set of toolbar buttons for the board's layer control as shown below:



These Layout View options toolbar buttons perform the following functions in the order they appear on the toolbar:

- **Top Current:** Displays components placed on the Top layer in the chosen layer color.
- **Top Only:** Hides the components placed on the bottom layer when the **Top Current** button is depressed.
- **Bottom Current:** Displays components placed on the Bottom layer in the chosen layer color.
- **Bottom Only:** Hides the components placed on the Top layer when the **Bottom Current** button is depressed.

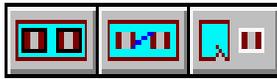
In InterPlace, only the Top and Bottom layers of any design can be displayed. One layer can be current at a time. When a layer is current, the components on that layer may be selected using the **Edit » Select** tool.

To set the current layer, click the corresponding **Top Current** or **Bottom Current** button. To view only the components on the Top layer, press the **Top Only** button. To display only the components on the Bottom layer press the **Bottom Only** button.

With multiple windows open you can have both the Top and Bottom layers current simultaneously. You can then select and move components from the Top layer to the Bottom layer and vice versa.

Visible Placement Area Controls

Each window in P-CAD InterPlace has a set of toolbar buttons for the individual Visible Placement domains as shown below:



The three Visible Placement Areas are Physical, Electrical and Room. When one or more of the placement areas is enabled, InterPlace displays an error-free area on the board where a selected component may be placed.

The Visible Placement Area, (page 75) describes in detail how to use this powerful placement tool.

Display Settings

As in P-CAD PCB, you have the option to choose how objects are displayed in your IPL board design. Among the familiar settings you can choose are:

- Colors for Layers, Pads, Components, Reference Designators, Keepouts, Background, Highlights, Selection, Connects, 1x and 10x Grids, Fixed Components, All Layer Keepouts and Power Pins.
- Cursor Style, which can be Arrow, Small, or Large Cross.
- Miscellaneous settings in which you can choose to display scroll bars, a component's reference designator and draft mode display.

InterPlace provides additional viewing options such as:

- A different display color for your board outline and connected power pins.
- A unique color for the Visible Placement Area.
- Turning on or off the display of information provided in a data tip.
- Setting the exact font size of the component reference designator.

Refer to *Options Commands, (page 105)*, for all the information on options settings.

Working with Components

One of the InterPlace specialties is component manipulation. You can move, place, swap, and fix components, rearranging their layout on the board to satisfy the design constraints.

When a PCB design is initially loaded into the P-CAD InterPlace Layout View, the components are located in their original position. These components may be easily moved in IPL. They can be dragged and dropped anywhere on the workspace – and even across windows!

All components must be loaded from the original design. New components cannot be added to the InterPlace workspace.

Organizing and Placing Components, (page 55), has complete details on using the organization and placement functions and tools provided by InterPlace.

Locating Components

In order to begin placing or moving components in the Layout View, you first need to locate them. InterPlace provides the ability to cluster a group of components into a common area, locate a specific component and jump directly to a desired location. In addition, you can highlight components and nets across the applications under certain conditions.

Additional information on highlighting across applications follows in *Cross-Probing Between Applications, (page 28)*.

Clustering Components

Placing components onto the board is much easier if the components you need are first grouped together in a common work area. From that grouping of components you can quickly find a particular one and place it in the desired position on the board.

The **Cluster By** command in the **Edit** menu offers the following options by which you can gather components into a common area:

- By Reference Designator
- By Type
- By Package size
- By Pin Count
- By ComponentHeight Attribute
- By PlacementSide Attribute

Refer to *Edit Commands, (page 87)*, for more information on each of the clustering options, or *Organizing and Placing Components, (page 55)*, which provides details on using the cluster option.

Locating a Specific Component

With the **Edit Move By RefDes** command, you can select a component by its reference designator. Once identified, the component is highlighted and the cursor becomes an X. You can click the left mouse button anywhere in the workspace and the component is placed in that position.

From the *Edit Properties* dialog, you can view or modify the properties of the selected component. For information, refer to *Accessing Component Information*, (page 31).

Refer to *Edit Commands*, (page 87), for more information on the **Edit » Move By RefDes** command.

Jumping to a Location

If you know the X and Y location coordinates of a component, or you want to quickly position your cursor in a particular spot on the board, you can use the **View » Jump Location** command.

Once the desired location coordinates are entered in the *View Jump Location* dialog, and you click **OK**, your cursor is positioned at that point on the board. You can also use the **J** key to enter those coordinates on the status line, as you do in PCB.

View Commands, (page 99), has complete information on the **View » Jump Location** command.

Cross-Probing Between Applications

When you enable the **DDE Hotlinks** option in the *Options Configure* dialogs of P-CAD Schematic and P-CAD InterPlace, it is possible to highlight a component or net in Schematic and see the corresponding component or net highlighted in InterPlace as well.

Working with PCB and Schematic, (page 69) has the step-by-step instructions on setting up the cross probing between applications.

Placing Components

The design components imported to InterPlace from the P-CAD PCB design can be manipulated in many ways. Components can be placed onto the board using the drag and drop method, the **Place On** command and the Discrete Placement tool. Once placed in their proper board position, they can be moved, fixed, rotated, flipped and swapped.

Organizing and Placing Components, (page 55), has detailed instructions on how to use the placement functions and tools.

Component Placement

When the board design is loaded into P-CAD InterPlace, the components are in their original position on the workspace. This may be within or outside of the board outline (i.e., placed or unplaced).

To place a selected component(s) in the Layout View of IPL, depress the left mouse button, drag the component(s) to the new location and release the button. If the component is released within the board outline, it is placed.

You may, for example, want to move an unplaced component onto the board to place it. Or, you may want to move a placed component to a region outside of the board outline so that you can rearrange other, placed components.

A selected component or group of components can be forced onto either the Top or Bottom layer using the **Edit » Place On** command. If you have selected a group of components, some residing on the Top layer and others on the Bottom, the **Place On** command places them all on the layer of choice.

You can also move components in IPL using the familiar commands: **Edit » Move By RefDes** and **Edit » Align Components**. Refer to *Edit Commands*, (page 87), for details.

Fixing Components

A fixed component in InterPlace may not change its location. For example, the following commands do not operate on a fixed component:

- Move (dragging a selected component)
- Move By Refdes
- Rotate
- Flip
- Align Component

A fixed component is displayed in the Fixed color selected using the *Options Display* dialog.

To fix a component that has been placed on the board outline you only need to select it and choose the **Fix Component** command from the **Edit** or right mouse menus. You can Unfix the component(s) in the same way, choosing the **Unfix Component** command instead.

Rotating Components

As in PCB, you can rotate selected components 90 degrees by pressing **R** (or **Shift+R**). In InterPlace, you can also rotate all selected components to an absolute angle. For example, you can choose to rotate all selected components to 90 degrees. To do so, choose the desired degree from the options listed when the **Edit » Rotate To** command is chosen.

The selected components are all rotated to the specified angle, regardless of their initial orientation. A single, selected component rotates around its own reference point. When a group of components is selected, and the **R** key pressed, the group rotates around a reference point placed, by default, in the middle of the group.

You can change the reference point around which a group of components rotates by placing a selection point at the desired location inside the selected group. To do so, select the group of components and choose **Selection Point** from the right mouse menu. Position the cursor at the desired location inside the block of components and click the left mouse button. The components will now be rotated around the new reference point.

Flipping Components

As in PCB, you can flip selected components to the opposite board side by pressing the **F** key. In InterPlace, you can also flip all selected components to a particular board side. For example, you can choose to flip all selected components to the Bottom layer. To flip the selected components choose the **Edit » Place On** command, and then the desired layer (Top or Bottom).

The selected components are all flipped to the specified layer, regardless of their initial flipped value.

Discrete Component Placement

With a design that uses large numbers of discrete components, like bypass caps, a designer can spend a lot of time finding and placing these components. The Discrete Placement tool helps you reduce the time expended placing these components. With just a few clicks of the mouse you can rapidly position a large number of discrete components near their associated components, eliminating the need to find each of them and drag them into place.

The Discrete Placement tool finds the discrete components you want to place by component Type or Value. It permits you to specify how many are to be placed with each mouse click. It keeps track of the number of the components that have been placed, and how many are still available. You can also designate whether they should be placed on the same or opposite side of the current layer.

Each time you click the left mouse button over a placed component, InterPlace positions the specified number of discrete components near that component.

Organizing and Placing Components, (page 55), provides complete instructions on using the Discrete Placement tool.

Moving Components

Designing an efficient, functional board means moving components and connections a number of times to obtain the optimal design. InterPlace lets you move components around a single Layout View or from one Layout View to another. You can also drag components from a list in the Design Manager View and drop them into a Layout View. In addition, the locations of two selected components can be swapped.

Organizing and Placing Components, (page 55), provides more information on getting components into their most efficient position in the board design.

Moving Components Across Windows

When you have multiple design windows open, you can select one or more components in one window, drag them across the window boundary, and place them in another window. Refer to *Using Multiple Windows* for details.

Moving Components from the Design Manager

You can move one or more components into the Layout View from the Design Manager Components, Partitions or Rooms pages. Simply select the component(s) name(s) in the Design Manager tree, drag them to the desired location in the board outline and drop them!

Swapping Components

You can interchange the location of two components in the Layout View. Simply select two components, and from the **Edit** or right mouse menu choose the **Swap Component** command.

The selected components exchange positions, including placement layer, while maintaining their individual rotations.

Aligning Components

Components can be aligned around a selection reference point either horizontally or vertically, and as an option, they can be equally spaced. If a number of components are off-grid, they can be aligned back onto the grid.

InterPlace, like PCB, helps to quickly and accurately align components in the locations where they are most likely to satisfy the design constraints. Selected components are aligned using the **Align Components** command from the Edit or right mouse menus.

Fixed components cannot be aligned and the **Align Components** command is undoable.

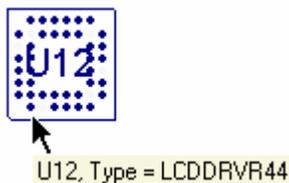
Refer to *Edit Commands*, (page 87), for additional instructions on using the **Align Components** command.

Accessing Component Information

Component Properties include basic information about the component, including reference designator, type, attributes, etc. These properties can be viewed in a data tip or by displaying the component properties dialogs.

Component Data Tips

Use the data tip feature of InterPlace to get a quick look at some basic component properties. When you place the cursor over a component or component pad, a data tip appears which contains information about the underlying object, as shown below for a component:



For a component, the data tip displays the reference designator, component type and value. The data tip for a component pad shows its component reference designator, component pad number and net name. The room name and placement side are displayed in a room data tip.

Make sure you are not overlying a connection line if you wish to view a component, pad or room data tip. You can turn off the display of the properties data tips in the *Options Display* dialog.

Component Properties

Select a component(s) and choose **Edit » Properties** to view the selected component's properties. You can also choose the right mouse **Properties** command. The *Component Properties* dialog is displayed and details component characteristics, including design constraints or attributes assigned to that component.

Refer to *Edit Properties, (page 88)*, for information.

Component properties can be modified using P-CAD PCB or P-CAD Relay.

Using Net Connections

Connections between components can be viewed in P-CAD InterPlace to visualize routability and net length requirements.

By viewing connections you can envision the board's routing channels, spacing components appropriately. Also, you can distribute the density of connection lines to minimize routing congestion. When the Visible Placement Area Electrical domain is enabled, and a maximum net length constraint applied, InterPlace shows you where the component can be placed without violating the necessary net length.

In InterPlace, net lengths may be optimized as a component is moved using automatic net optimization. You can choose the component's placement location knowing in advance the configuration of connection lines. Refer to *Optimizing Net Connections, (page 33)* for more information.

By default, net connections are hidden in InterPlace. The visibility of these connections can be enabled in a variety of ways. For more information, see *Setting Display Options of Specific Nets, (page 33)*.

Accessing Connection Information

When you place the mouse over a connection in the current layer, a data tip appears which contains information about the underlying object. For connections, a data tip contains the following connection properties:

- Net Name
- Reference Designators of Connected Components
- Pad Numbers of Connection Endpoints

A connection data tip is shown below:



Similar data tips are available for components, component pads and rooms. Connection data tips take precedence over component data tips, but not component pad data tips.

You can turn off the display of the data tips, in the *Options Display* dialog.

Setting Display Options of Specific Nets

InterPlace provides the ability to show, show only on drag or hide the net connections in your design. When the net connections are visible, you can highlight and unhighlight them.

To choose the desired net connection display, you must first select the nets in the Design Manager view. Then, from the right mouse menu you can select the kind of display you want to see:

- Show Connections
- Show Connections on Drag
- Hide Connections

To highlight a net with visible connections, select the net and choose **Highlight** from the right mouse menu. You can remove the highlighting by choosing the **Unhighlight** command in the right mouse menu.

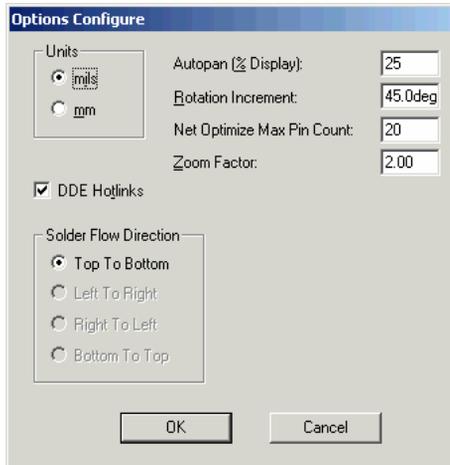
For more information, see *The Design Manager View*, (page 37).

Optimizing Net Connections

Net connections may be automatically optimized as the components are moved to new locations. This shortens the length of the connections between net nodes. Optimization rearranges connections logically in order to produce shortened or more direct routings, or dispersed routings, which ease congestion in certain board areas.

You may not want to optimize nets with a large number of nodes, such as power or ground nets. You can specify a default value for the maximum pin count of a net if it is to be automatically optimized. Nets with equal or fewer nodes are automatically optimized; nets with more nodes are not optimized.

To specify this default value, choose **Options » Configure**:



Enter the desired value for the Net Optimize Max Pin Count.

You can override this default value by assigning an Optimize attribute to specific nets. For example, you may wish to optimize only one of the four nets in the design, which have more than 20 nodes. Alternatively, you may choose not to optimize a particular net despite its small number of nodes.

Attributes can be assigned to nets in PCB prior to loading the design into InterPlace. See your *PCB User's Guide* for more information.

Modifying Connection Properties

You can modify the attributes or design constraints of a net using PCB. See your *PCB User's Guide* for details.

Layout View Right Mouse Commands

When you select an object and click the **right mouse button**, a pop-up menu appears providing shortcuts to common commands performed on selected objects.

The following list summarizes the commands, which appear on the pop-up menu in the Layout View.

- **Align:** A shortcut for **Edit » Align Components**. Components can be aligned around a selection reference point either horizontally or vertically, and as an option, equally spacing the parts.
- **Cluster By:** This command provides a way to group components in a common area based on a criteria chosen from the pull right list of selection options.
- **Fix Components:** A shortcut to **Edit » Fix Components**. This command fixes the selected components so they may not be moved, rotated, or flipped.

- **Highlight:** A shortcut to **Edit » Highlight**. This command highlights the selected objects the current highlight color.
- **Highlight Attached Nets:** This command highlights nets attached to the selected objects in the current highlight color.

This also highlights the corresponding component in Schematic if the **DDE Hotlinks** check box in the *Options Configure* dialog is checked.
- **Place On:** This command places a selected component(s) on the layer (top or bottom) chosen from the pull right list.
- **Properties:** A shortcut to **Edit » Properties**. Depending on the object(s) selected, the appropriate *Edit Properties* dialog appears.
- **Rotate To:** This command displays an additional menu from which you can choose an absolute degree of rotation for a selected component(s).
- **Selection Point:** This command allows you to relocate a selection reference point for the selected object or objects.
- **Unfix Components:** A shortcut to **Edit » Unfix Components**. This command unfixes the selected components so they may once again be moved, rotated, or flipped.
- **Unhighlight:** A shortcut to **Edit » Unhighlight**. This command removes the highlighting from selected objects.
- **Unhighlight Attached Nets:** This command removes the highlighting from nets attached to the selected objects.

The Design Manager View

Design floor planning is an integral element of the Design Manager View. With InterPlace, you can divide the design into smaller logical blocks or partitions. You can also manage physical regions on the board layout, called rooms, where these partitions can be assigned. The Design Manager View provides a simple interface for organizing design components into these logical and physical divisions.

In addition to organizing the design components into logical and physical regions, the Design Manager provides easy access to information about the nets, net classes and class-to-class rules in your design.

In this chapter, each method of organizing components, nets and net classes is discussed, as well as how you can use them to expedite the design and production of an efficient and cost effective circuit board.

Working with the Design Manager View

The Design Manager View of InterPlace displays the design information in either a tree-like structure or a simple list. You can view design data from a number of perspectives: components, nets, partitions, rooms, a net class and a class-to-class, each of which is available in a Design Manager View page.

In the Components page you can rearrange the display of the details shown there by sorting the list of components using any of the column headings. You can create partitions in the partitions page, and add components to or remove components from the partitions. If you created a room in the PCB design, you can add or remove components in those rooms. From the Nets page you can highlight existing nets and show their connections, and the nets can be organized into net classes and class-to-class groupings in their respective pages.

In addition to viewing and organizing your design data, the Design Manager also provides easy access to other design features. You can move components from the Design Manager pages and cluster a group of components into a common work area in the Layout View workspace for a PCB design.

In the next sections you will learn how to access the Design Manager View, move around in each of the pages, and employ its powerful functionality to expedite your design efforts.

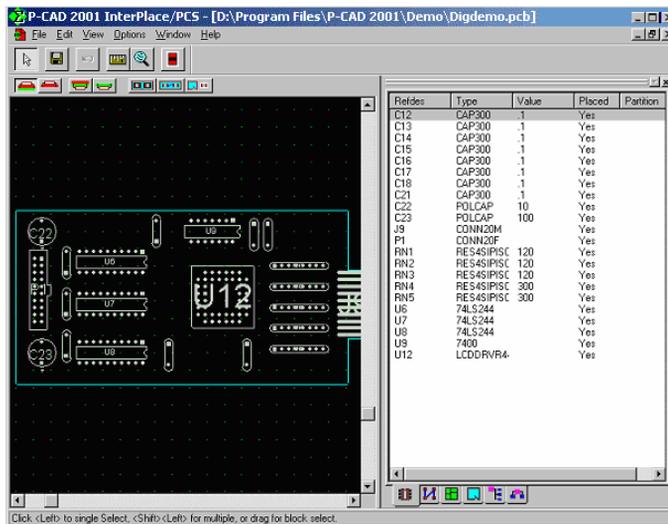
Accessing the Design Manager View

When you launch the InterPlace application, the Design Manager View appears if it was displayed at the time you last exited the application. IPL stores the position and size of the Design Manager View from your last session in the .ini file and reconstructs those settings in the workspace for the new session.

Before a design is loaded, the Design Manager View is empty. Once a design is loaded using the **File » Load PCB** or **Schematic Design** commands, the Design Manager View displays the design data in the relevant pages.

You can choose to view or hide the Design Manager View using the **View » Design Manager** command.

The Design Manager View is displayed in the right side of the InterPlace workspace when IPL is launched, as displayed below for a PCB design:



Accessing Design Manager Pages

To access any of the pages in the Design Manager View, simply click the appropriate tab at the bottom of the view. The data presented in the six different views of the design data are shown below with their associated icon:



Components Page



Nets Page

	Partitions Page
	Rooms Page
	Net Class Page
	Class-to-Class Page

On each page, except the Components Page, is a tree structure or hierarchy, containing all of the information pertinent to that page. You can view the page's contents at various levels of detail by expanding or collapsing the branches of the tree.

Groupings containing collapsed levels are shown with a  sign. To expand the grouping, simply click the . Expanded groupings are shown with a  sign. To collapse the grouping, simply click the  sign. You can also use the  and  keys on a selected group to collapse or expand groupings, respectively.

Selecting Items in Design Manager

In order to perform functions on the data items in your design, you must first select one or more items in a page. For instance, you may want to select components to add to a room or partition. You might also want to create a new net class or class-to-class from selected nets and net classes.

Items can be selected using any of these methods:

- **To Select a Single Item:** Position the cursor on the item and click the left mouse button.
- **To Select a Contiguous Block of Items:** Position the cursor on the first item in the block and click the left mouse button, press the **Shift** key and click on the last item in the block. You can also select an item, then press the **Shift+End** keys to select all the items that follow to the end of the list.
- **To Select Multiple Individual Items:** Press the **Ctrl** key while clicking the left mouse button on each item you want to select.

You can right mouse click on a selected item(s) in the tree to access its pop-up command menu. Refer to *Design Manager Right Mouse Commands*, (page 51), for the complete list of right mouse commands.

Common Design Manager View Commands and Functions



There are commands and functions that are common to more than one of the pages in the Design Manager View. For instance, you can cluster components from all of the pages and highlight components from the Components, Partitions and Rooms pages.

Selecting Components across Views

When a design is loaded from PCB, you can select components from the Components, Partition and Rooms pages, and make them “selected” in the Layout View using the **Select Components** command in the right mouse menu.

Making components selected in the Layout View is additive. Each time you apply the **Select Components** command to a selected component from the Components, Partition or Rooms pages, it becomes selected in addition to any other components already selected in the Layout View. When all the desired components are selected in the Layout View, you can move them as a block around the workspace.

Dragging a Component from Design Manager

You can place a component into the Layout View from the Design Manager Components, Partitions or Rooms pages. Simply select the component(s) in the Design Manager page, press the left mouse button, drag the object to the desired location in the board outline in the Layout View, and release the mouse button.

Clustering Components

You can cluster a group of selected components into a common work area in the Layout View using the **Cluster By** command. A number of components grouped by Type, for instance, gives you easier access to the components for quicker placement on the board outline.

In addition to clustering selected components as a group from the Components, Partitions and Rooms pages, you can select nets and net classes from their pages, or net classes from the class-to-class page, and cluster the components they contain.

Selected components are physically arranged in the workspace based on the chosen clustering option. Clustering options are listed below:

- By Reference Designator
- By Type
- By Package Size
- By Pin Count
- By ComponentHeight Attribute
- By PlacementSide Attribute

Organizing and Placing Components, (page 55), provides detailed information on using the clustering tool.

Highlighting Objects

There are a variety of ways and a number of objects that can be highlighted or unhighlighted using the appropriate commands. The following table shows where highlighting options can be initiated, and which objects can be highlighted or unhighlighted.

Initiation Point	Objects Highlighted or Unhighlighted
Components Page	Components Attached Nets
Nets Page	Nets
Partitions Page	Components Attached Nets
Rooms Page	Components Attached Nets Room and Assigned Components
Net Class Page	Nets
Class-to-Class Page	Net Classes within a Class-to-Class. In this case the two net classes within the class-to-class are each highlighted in a different color.

Once the objects are selected, and the appropriate highlighting command chosen, the Layout View displays the object(s) in the highlight color designated in the *Options Display* dialog.

Net connections must first be made visible in order for them to be highlighted. Using the Nets Page describes how to make a net visible.

Refer to *Options Commands*, (page 105), for more information on the *Options Display* dialog where the highlighting color is selected.

Unhighlighting Objects

Highlighted objects can be unhighlighted using the **Unhighlight** command in the right mouse menu. Each of the highlight commands shown in the table above has a complementary unhighlight command.

Jumping to Components and Pins

InterPlace makes it possible to quickly find a specific component. When you select a component in the Components, Partitions or Rooms pages and choose the **Jump to Component** command, IPL finds that component's location in the Layout View.

In addition to finding a specific component, you can also let InterPlace locate a specific pin in a network. To find a pin, simply click the  button next to the desired net in the Nets page to display the pins in that net, select the pin you want to locate, and then the **Jump to Pin** command.

In both cases, the cursor is placed over the object in the Layout View. If the object is on the current layer and the data tips have been enabled in the *Options Display* dialog, the relevant data tip information is displayed as well.

Using the Components Page

The Components Page lists all the components in the design, along with some pertinent information about each of them, including the Type, Value, Placed indicator and Partition name.

You can sort the components by clicking any of the column headers.



Components can be dragged into the Layout View for placement on the PCB board and you can cluster a group of them into a common work area from the Design Manager View. You can select a component in the Design Manager View and jump to its location in the Layout View using the **Jump to Component** command. Components and their attached nets can be highlighted and unhighlighted as described in Common Design Manager View Command and Functions.

Component Information

For each component in the design, the following information is displayed:

- Reference Designator or component identifier.
- Type of component.
- Value assigned to the component.
- Placed indicates whether the component is or isn't located on the board outline.
- Partition displays the name of the partition the components is assigned to, if any.

The Components Page appears as follows:

Refdes	Type	Value	Placed	Partition
C12	CAP300	.1	Yes	
C13	CAP300	.1	Yes	
C14	CAP300	.1	Yes	
C15	CAP300	.1	Yes	
C16	CAP300	.1	Yes	
C17	CAP300	.1	Yes	
C18	CAP300	.1	Yes	
C21	CAP300	.1	Yes	
C22	POLCAP	10	Yes	
C23	POLCAP	100	Yes	
J9	CONN20M		Yes	
P1	CONN20F		Yes	
RN1	RES4SIPISDL	120	Yes	
RN2	RES4SIPISDL	120	Yes	
RN3	RES4SIPISDL	120	Yes	
RN4	RES4SIPISDL	300	Yes	
RN5	RES4SIPISDL	300	Yes	
U6	74LS244		Yes	
U7	74LS244		Yes	
U8	74LS244		Yes	
U9	7400		Yes	
U12	LCDDRVR44		Yes	

Sorting the Components

You can sort the list of components for easier viewing by simply clicking on one of the column headings. For instance, if you want to see which components have been placed on the board, or which ones are available to be placed, click the **Placed** column heading and the unplaced components float to the top of the list.

Using the Nets Page



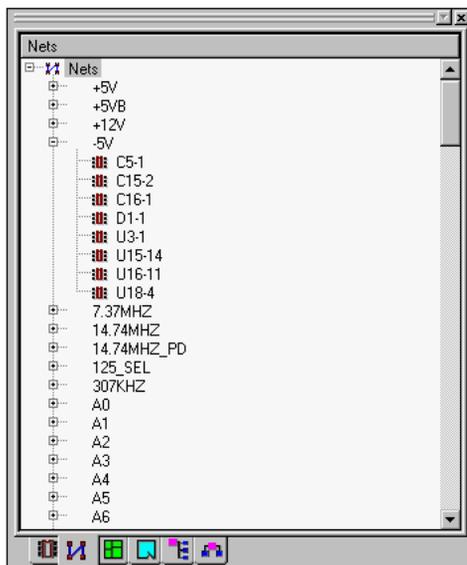
The Nets Page of the Design Manager View displays a list of the nets in the design, and their associated pins. From the Nets Page, you can highlight a net, show or hide the connections, jump to a particular node or cluster the components in a net into a common work area.

Net Information

The Nets page displays a list of the nets resident in the loaded design. Nodes associated with each net are listed for designs loaded from PCB.

A Schematic design loaded into InterPlace displays only the list of nets in the design. No other functionality, such as highlighting or showing connections, is available, since there is no Layout View in which to display the results of these commands.

The Nets page of the Design Manager View is displayed as follows:



Showing and Hiding Net Connections



Component placement or movement on the board is sometimes simpler if the net connections are not visible. The display of the nets in your design is controlled using the **Show Connections** and **Hide Connections** commands in the right mouse menu. Another option available for displaying nets is the **Show Connections on Drag** command, where the nets are only visible when components are being moved.

To turn on the display of one or more nets, simply select the desired net(s) and choose the **Show Connections** or **Show Connections on Drag** commands from the right mouse menu. You can also enable the net connections display by selecting the net category in the Nets Page or a net class category in the Net Class Page and then the appropriate **Show Connections** commands in the right mouse menu.

To hide the connections display, choose the **Hide Connections** command in the right mouse menu.

Using the Partitions Page



InterPlace provides a way to assemble components logically in a group referred to as a partition. Using a partition to group components is a great way to start organizing your design's components. For instance, you can establish partitions with like properties, such as the analog or digital components, or partitions that contain components destined for a particular layer.

Once the partitions are established, it is easy to highlight the entire partition of components, or their attached nets for a PCB design. You can cluster the components in a partition for easy placement in the Layout View, or drag individual components from the partition and drop them in the Layout View.

Partitions names can be used to describe the way the components are organized for easy identification. For instance, you may have a partition containing the analog components and name it AnalogPart, or for components located on a Schematic Sheet number one you can create a partition called Sheet1Part.

Organizing and Placing Components, (page 55), has more information on using partitions to help organize designs.

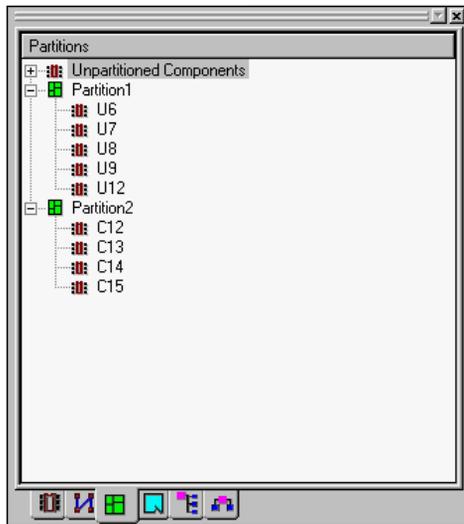
Partition Information

The Partitions page of the Design Manager View displays the list of components not assigned to a partition, provides the ability to create and delete partitions, and add or remove components from the partitions.

Components can belong to only one partition at any time.

When you first enter the Partitions page of the Design Manager View, and no partitions have been created, the components are grouped in the *Unpartitioned Components* list and are not visible. Click the  button next to *Unpartitioned Components* to display the list of components.

The *Partitions Page* appears as follows:



Creating Partitions

Components in a design with no partitions are, by default, unpartitioned. A new partition is created using the **New Partition** command in the right mouse menu. The partition name can be changed.

To Create a Partition

1. Select the `Unpartitioned Components` folder or an existing partition.
2. Choose the **New Partition** command from the right mouse menu. The partition is given a default name of `PartitionX`, where X is a number incremented with each added partition.

Deleting Partitions

Any partition you create can be deleted. If a deleted partition has components assigned to it, those components are placed back into the `Unpartitioned Components` list and the associated Partition name is deleted from the `Components Page`.

To Delete A Partition

1. Select an existing partition.
2. From the right mouse menu, choose **Delete** or press the **Delete** key. If the partition has assigned components, the components are placed back in the `Unpartitioned Components` list and the partition is deleted.

The `Unpartitioned Components` folder cannot be modified or deleted.

Renaming Partitions

The partition can be renamed. Partition name changes are also reflected in the Partition column of the *Components Page*.

To Rename a Partition

1. Select an existing partition.
2. From the right mouse menu choose the **Rename** command. You may also double click on the partition name to put the field in edit mode.
3. Enter the new partition name and press **Enter**.

Adding Components to a Partition

When a design is loaded into InterPlace, the components not assigned to a partition are part of the Unpartitioned Components list. Selected components can be moved to a newly established partition using the **Move To** command or by dragging them from the Unpartitioned list into the target partition.

Removing Components from a Partition

When you remove components from a partition they are placed back into the Unpartitioned Components list. Selected components are removed by dragging them from a partition into the Unpartitioned Components list or another partition by using the **Remove Components** command in the right mouse menu.

Using the Rooms Page



InterPlace allows you to group components into a physical area on the board known as a Room. As an example, you may want the analog components in your design to be physically located at the front of the board and the digital components in the back. The *Rooms* page provides an easy way to organize these components in the PCB design.

Rooms must be placed on the board in PCB before the design is loaded into InterPlace. A room cannot be created in InterPlace. See your *PCB User's Guide* for instructions on creating rooms.

When components have been assigned to a room, it is easy to graphically highlight the entire group of components, or their attached nets. You can cluster the components assigned to a Room for easy placement in the Layout View, or drag individual components from the room's component list and drop them in the Layout View.

Refer to *Organizing and Placing Components*, (page 55), for more information on the use of Rooms.

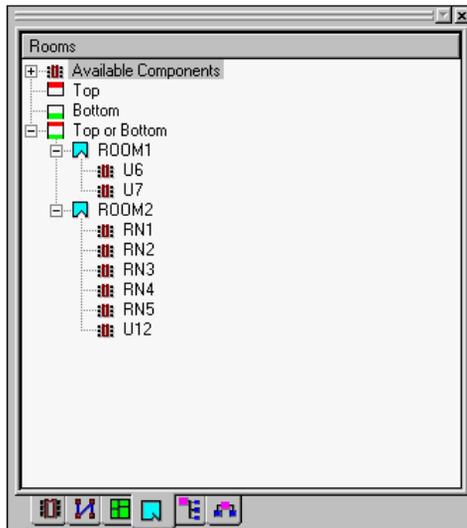
Rooms Information

When you enter the *Rooms* page of the Design Manager View, all of the design components are listed in the *Available Components* list and are not visible until you click the  button.

Components can be assigned to more than one Room, therefore, the list of Available Components does not change even after a component has been assigned to a room. All components remain available for room assignment at all times.

In addition to the Available Components folder, there is a Top, Bottom and Top or Bottom folder. Existing Rooms are displayed in their appropriate folder, depending on the layer placement.

The Rooms Page, expanded to show the available rooms, appears as follows:



Adding Components to a Room

When a design is loaded into InterPlace, the components available to be assigned to rooms are listed in the Available Components list. Selected components can be added to a room using the **Add To** command or by dragging them from the Available Components list into the target room.

Components can be included in more than one room, therefore the list of Available Components does not change when components are added to or deleted from a room. Instead of being moved, component information is copied from the Available Components list into the target Room.

Removing Components from a Room

Selected components can be removed from a room only by using the **Remove Components** command in the right mouse menu.

Using the Net Class Page



Nets that are part of a design loaded into InterPlace can be organized into Net Classes using the Net Class page. Grouping nets into a Net Class saves time and effort when you want to apply specific design constraints to a number of nets. With the nets organized in a Net Class,

constraint(s) can be applied once to the Net Class, instead of repeating the constraint for each of the individual nets.

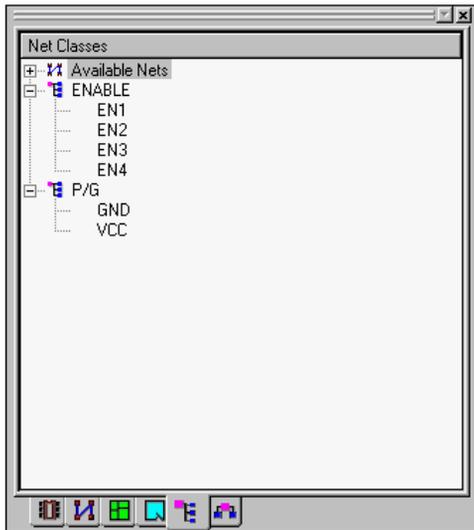
Once a Net Class is established, you can delete and rename it. You can move the nets from one Net Class to another using the drag and drop method or the **Move To** command.

The nets in a Net Class can be highlighted and unhighlighted in the Layout View for a PCB design. You can also enable or disable the constant visibility of the connections or choose to show them only when their components are moved.

Net Class Information

The Net Class page of the Design Manager View displays the design's net classes and nets. When a net has not been assigned to a Net Class, it is listed in the `Available Nets` folder. The `Available Nets` and any `Net Class` folders can be expanded for viewing the included nets by clicking the  button or pressing the  key. To collapse the lists, click the  button or press the  key.

The Net Class Page, showing the expanded list for each of the existing Net Classes, appears as follows:



Creating a Net Class

A new Net Class is created by choosing the **New Net Class** command from the right mouse menu in the Net Class page.

To Create a Net Class

1. Select the `Available Nets` folder, an existing net class or simply click in the workspace.

2. Choose the **New Net Class** command from the right mouse menu. The net class is given a default name of NetClassX, where X is a number incremented with each added net class.

Deleting a Net Class

Any Net Class in the design can also be deleted.

To Delete A Net Class

1. Select an existing net class.
2. From the right mouse menu, choose **Delete Net Class** or press the **Delete** key. If the net class has assigned nets, the nets are placed back in the *Available Nets* list and the net class is deleted.

Renaming a Net Class

The Net Class can be renamed.

To Rename a Net Class

1. Select an existing net class.
2. From the right mouse menu, choose the **Rename Net Class** command to put the field in edit mode.
3. Enter the new net class name and press **Enter**.

Adding Nets to a Net Class

Any net in the design is available to be added to a Net Class. A net can belong to only one Net Class at a time. To add a selected net to a Net Class, use the drag and drop method or the **Move To** command in the right mouse menu.

Removing Nets from a Net Class

Nets removed from a Net Class are placed back into the Available Nets list. Selected nets are removed from a Net Class by dragging them from the Net Class into the Available Nets list or by using the **Remove Nets** command in the right mouse menu.

Using the Class-to-Class Page



When you have established Net Classes in your design, you can define constraints that apply to only those unique Net Classes.

For instance, a single net in your design requires a clearance greater than any other net clearance. You can assign this one net to a net class, and all the rest to another net class. The clearance constraint between the two net classes essentially keeps the single net in the first net class an exceptional distance from the other nets in the second net class.

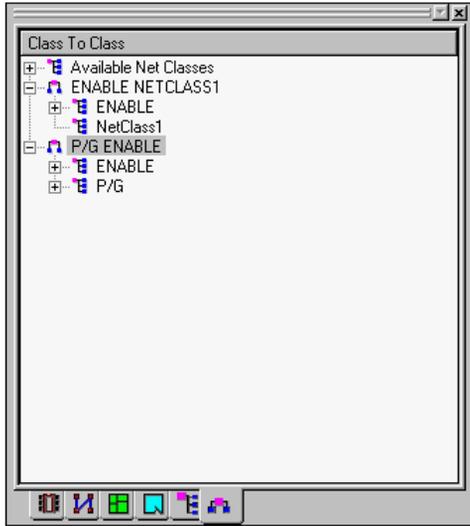
The nets in a Class-to-Class can be highlighted and unhighlighted for a PCB design. You can enable or disable the consistent visibility of the net connections or show them only when their

components are moved. Components connected by the nets in the Class-to-Class can also be grouped in the Layout View using the clustering tool.

Class-to-Class Information

The *Class-to-Class* page of the Design Manager View groups the design's net classes in the list of Available Net Classes and does not display them until you click the  button.

The *Class-to-Class* Page appears as follows:



Creating a Class-to-Class

A new Class-to-Class is created by selecting one or two net classes and then choosing the **New Class-to-Class** command from the right mouse menu.

To Create a Class-to-Class

1. Expand the Available Net Classes folder, if it is not expanded, to display the available net classes, by clicking the  button.
2. Select one or two of the Net Classes in the list of available net classes. The new Class-to-Class name is the name of the first net class with the name of the second appended to it. If you only select one net class, the new Class-to-Class name is the net class name repeated.

Deleting a Class-to-Class

You can delete any Class-to-Class in the design.

To Delete A Class-to-Class

1. Select an existing class-to-class.

- From the right mouse menu, choose **Delete** or press the **Delete** key.

Design Manager Right Mouse Commands

When you select an object and click the **right mouse button**, a pop-up menu appears providing access to the commands you can perform in the Design Manager View.

The following table summarizes the commands, which appear on the pop-up menus, and the page where the command is available.

The column headings indicate the page as follows: **C**=Components; **N**=Nets; **P**=Partition; **R**=Rooms; **NC**=Net Class; **CC**=Class-to-Class.

C	N	P	R	NC	CC	Command
		X				Add Selected Components: This command adds the selected components to a partition.
			X			Add To: This command is used to add selected components to a room.
X	X	X	X	X	X	Cluster By: This command provides a way to group components in a common area based on a criteria chosen from the pull right list of selection options.
		X			X	Delete: This command deletes a selected partition or class-to-class.
				X		Delete Net Class: This command deletes a selected net class.
	X			X	X	Hide Connections: This commands makes the net connections invisible for a selected net.
					X	Highlight: This command highlights the nets in a Class-to-Class when the connections are visible.
X		X	X			Highlight Attached Nets: This command highlights nets attached to the selected objects in the current highlight color.

C	N	P	R	NC	CC	Command
X		X	X			Highlight Components: This command highlights the selected components in the current highlight color.
	X			X		Highlight Nets: This command highlights the selected nets in the current highlight color.
			X			Highlight Room and Assigned Components: This command highlights a selected room along with the components included in the room.
X		X	X			Jump To Component: This command positions the cursor on the selected component in the Layout View.
	X					Jump to Pin: This command positions the cursor on the selected net pin in the Layout View.
		X		X		Move To: This command moves selected components to a partition, or selected nets to a Net Class.
					X	New Class to Class: This command is used to add a new class-to-class to the design.
				X		New Net Class: This command is used to add a new net class to the design.
		X				New Partition: This command creates a new partition.
		X	X			Remove Component: This command removes selected components from a partition.
				X		Remove Nets: This command removes nets from a net class.
		X				Rename: This command is used to rename a partition.

C	N	P	R	NC	CC	Command
				X		Rename Net Class: This command allows you to rename a selected net class.
	X			X	X	Show Connections: This commands makes the net connections visible for a selected net.
	X			X	X	Show Connections on Drag: This commands makes the net connections visible for a selected net when components in the net are moved.
X		X	X			Select Component: This command selects an individual component, or the components assigned to a room or partition.
					X	Unhighlight: This command unhighlights the nets in a Class-to-Class when the connections are visible and highlighted.
X		X	X			Unhighlight Attached Nets: This command removes highlighting from highlighted nets attached to the selected objects when they are visible and highlighted.
X		X	X			Unhighlight Components: This command removes highlighting from highlighted components.
	X					Unhighlight Nets: This command removes highlighting from the highlighted nets.
			X			Unhighlight Room and Assigned Components: This command removes highlighting from a highlighted room and its assigned components.

Organizing and Placing Components

Whether you are working on a PCB or a Schematic design, P-CAD InterPlace will help you organize the many objects in your design. InterPlace specializes in two of the basic areas of board design: object organization and component placement for PCB designs. A well organized board is essential to minimizing constraint violations resulting in more efficiently designed and manufactured circuit boards. Being well-organized also reduces the time it takes to get that design out the door.

The Design Manager View in InterPlace provides easy access to information on the components and nets in your design. In the Design Manager you can assign components to logical blocks or partitions and group nets into a net class and class-to-classes. For a PCB design, you can even group components into a room in preparation for placing them in a specific physical location on the board.



The Layout View, or placement interface, is available to designs loaded from PCB or Relay. Manipulation of the design objects, including controlling design layers, using rooms, moving components, and viewing connections are just a small part of the productivity enhancing tools available in InterPlace. InterPlace also provides a Visible Placement Area tool which shows an area on the board, based on constraints, where a component can be placed without incurring errors.

This chapter explains in detail how to use the organizational tools for your PCB or Schematic design objects. You will learn how to create partitions, net classes and class-to-class rules, and how to assign members to them. For PCB designs there are instructions on employing the placement functions, such as clustering, discrete placement and drag and drop across windows, which make placing objects on the board a breeze. Using the Visible Placement Area tool is introduced as well.

Organizing Design Objects

When a PCB or Schematic design is loaded into InterPlace, the components and nets are divided in the Design Manager View and displayed in one of these individual pages: Components, Nets, Partitions, Rooms, Net Class and Class-to-Class.

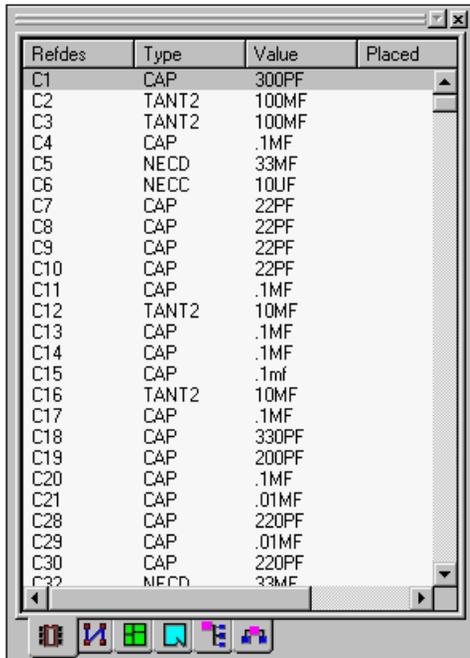
This portion of the chapter contains an explanation of the InterPlace organizational processes for the Partitions and Rooms pages, and includes step-by-step instructions on how to use them.

The *Design Manager View*, (page 37), contains an overview of all functions available in the Design Manager View, along with detailed instructions on functions unrelated to organizing design data.

The Design Manager View

Each of the design data sections, except the components page, uses a tree-like structure in which lists of objects can be expanded or contracted depending on your desired view.

The *Design Manager View* appears as follows:



Refdes	Type	Value	Placed
C1	CAP	300PF	
C2	TANT2	100MF	
C3	TANT2	100MF	
C4	CAP	.1MF	
C5	NECD	33MF	
C6	NECC	10UF	
C7	CAP	22PF	
C8	CAP	22PF	
C9	CAP	22PF	
C10	CAP	22PF	
C11	CAP	.1MF	
C12	TANT2	10MF	
C13	CAP	.1MF	
C14	CAP	.1MF	
C15	CAP	.1mf	
C16	TANT2	10MF	
C17	CAP	.1MF	
C18	CAP	330PF	
C19	CAP	200PF	
C20	CAP	.1MF	
C21	CAP	.01MF	
C28	CAP	220PF	
C29	CAP	.01MF	
C30	CAP	220PF	
C32	NECD	33MF	

Each page of data is accessible by clicking on the appropriate tab at the bottom of the view. The six pages are:



Components Page



Nets Page



Partitions Page



Rooms Page



Net Class Page



Class-to-Class Page

The Components Page



The *Components Page* is an ideal way to view the complete list of components in your design. For each component in the list the following essential information is displayed:

- **Reference Designator:** The reference designator of the component.
- **Type:** The component type.
- **Value:** The value assigned to the component.
- **Placed:** If any part of the component touches the board edge, or is located inside of the board outline, the Placed indicator displays Yes.
- **Partition:** If the component has been assigned to a partition the name of the partition is shown here.

Using the sorting capability of the Components Page you can quickly determine, for example, which components have been placed and which are assigned to a partition.

The Nets Page



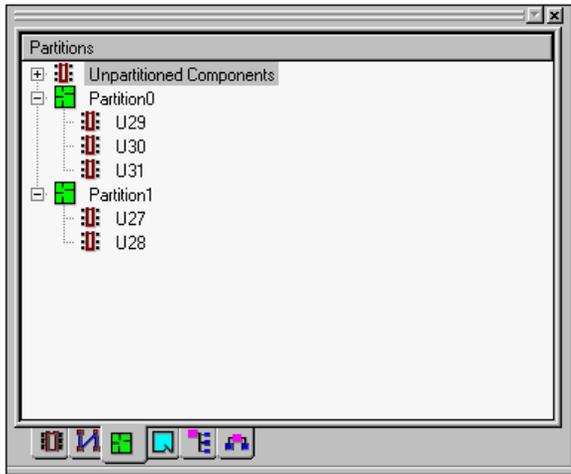
The next informational page is the Nets Page. All nets in the loaded design are present in the list of nets for both PCB and Schematic designs. For a PCB design, from the Nets Page, you can also enable the connection display as well as physically cluster the components in the selected net in the Layout View.

The Partitions Page



The Partitions Page is where a logical grouping of components, a partition, is created, and components assigned to it. For all designs you can create, rename and delete a partition. Components can be added or removed from the partition, and for PCB designs you can physically cluster components belonging to a partition in the Layout View.

The Partitions Page is shown below:



Adding a Component to a Partition

1. Display the list of Unpartitioned Components by clicking the button.
2. Select the desired components from the list.
3. Choose the **Move To** command from the right mouse menu, and then select the target partition from the right pull down list. You can also press the left mouse button and drag the selected components into the target partition.

Removing a Component from a Partition

To remove all components from a partition and delete the partition:

1. Select the partition.
2. From the right mouse menu, choose **Delete** or press the **Delete** key. If the partition has assigned components, the components are placed back in the Unpartitioned Components list and the partition is deleted.

To remove a portion of the assigned components:

1. Select the desired components.
2. Choose the **Remove Components** command from the right mouse menu, or drag them back into the Unpartitioned Components folder.

A component can belong to only one partition at any time.

Clustering Components in a Partition

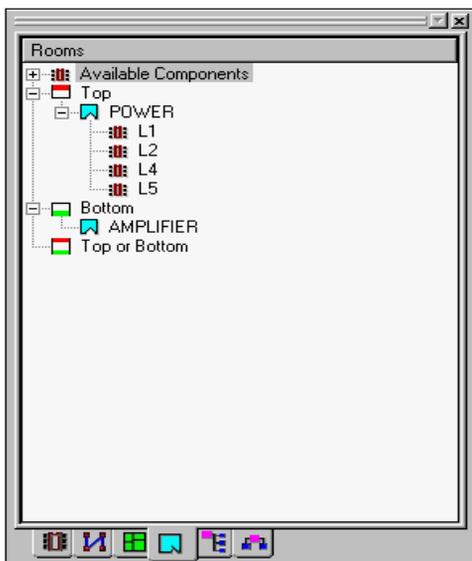
For a PCB design, components in one or more selected partitions can be retrieved from their current positions and grouped in the Layout View in a common area using the **Cluster By** command in the right mouse menu.

The Rooms Page



When a room has been placed in a PCB design, InterPlace provides a vehicle with which to organize a set of components destined for placement in the room. You may want to position the analog components in a room located in the back of the board, for instance. By grouping the analog components in a room, you can then quickly cluster and place them in one easy step.

The *Rooms Page* appears as follows:



Creating a Room

Physically drawing the boundaries of a room must be completed in PCB before the design is loaded into InterPlace. The location of the room, Top, Bottom or Top or Bottom, is determined at the time the room is placed in the design. See your *PCB User's Guide* for more information on placing a room.

Adding Components to a Room

1. Select the desired components in the list of Available Components.
2. Choose the **Add To** command from the right mouse menu and the specific room from the pull down list, or press the left mouse button and drag the selected components to their room destination.

Removing Components from a Room

1. Select the desired components in the room.
2. Choose the **Remove Components** command from the right mouse menu. The components are removed from the room.

A component can belong to any number of rooms at the same time.

Clustering Components in a Room

For a PCB design, components in one or more selected rooms can be retrieved from their current positions and grouped in the Layout View in a common area using the **Cluster By** command in the right mouse menu.

The Net Class Page



Organizing nets into one class is, again, an easy way to apply constraints once instead of individually to each net. The Net Class Page is detailed in *The Design Manager View*, (page 37).

The Class-to-Class Page



Once you have established Net Classes in your design, you can define constraints that will apply to only those unique Net Classes in a Class-to-Class. The Class-to-Class page is detailed in *The Design Manager View*, (page 37).

Placing Design Objects

Now that the components are organized in the desired partitions or rooms, and your nets are comfortably settled in their net classes and class-to-classes, you can begin to gather components in preparation for placing them on the board.

This section of the chapter details the many means provided by InterPlace to collect components into a common workspace, such as the **Cluster By** command. You will learn about the productivity enhancing tools that make placing the components easy, in particular the discrete placement tool. You will be introduced to the Visible Placement Area tool, which provides an area on the board where a component can be placed without incurring errors. In addition to gathering and placing components, there is a section on working with multiple windows, and another that describes how to fine-tune component placement.

Finding and Gathering Components

In a design with a large number of components, it becomes a monumental task to find and locate them so that they can be easily placed on the board. With InterPlace, gathering components into a common area is quick and varied.

Components can be clustered from a variety of places: rooms, partitions and nets are just a few. You can drag components from the Design Manager View and drop them into the Layout View. A specific component can be moved from one place to another just by knowing its reference designator. Finding a component is easy when you can jump right to it, or select it from the Design Manager View.

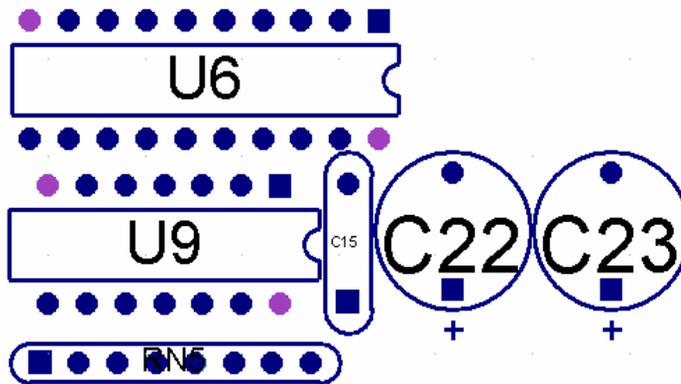
Clustering Components

The **Cluster By** command is available on every menu in the Layout View and the Design Manager View. The Cluster tool in InterPlace finds designated components and lets you place them in a chosen location arranged by the property of your choice.

The options available in the **Cluster By** command are:

- By Reference Designator
- By Type
- By Package size
- By Pin Count
- By ComponentHeight Attribute
- By PlacementSide Attribute

For all of the above options, when the components are drawn in the new location, they are arranged alphabetically by the value of the selected option. For instance, if you select a group of components having different types in the Components Page of the Design Manager View, the cluster would appear as follows:



The components are arranged alphanumerically by their type and then by their reference designator. In the above display you can see that the components U6 and U9, which have a type beginning with 7 are displayed first. The next set of components are those with a type of CAP300, POLCAP and then the one with RES4SIPICS type. From this clustering of components, each can be moved into their position on the board.

Since the **Cluster By** command appears in all the menus, you can choose to group components in many ways. The following list details the clustering possibilities in InterPlace:

- **Layout View Edit menu:** This command allows you to cluster any selected components displayed in the Layout View whether they are placed on or off the board.
- **Design Manager View Components Page:** This command clusters the components selected in the component list.
- **Design Manager View Nets Page:** This command clusters the components resident in the selected nets.

- **Design Manager View Partitions Page:** This command clusters the components assigned to the selected partitions.
- **Design Manager View Rooms Page:** This command clusters the components assigned to the selected rooms.
- **Design Manager View Net Classes Page:** This command clusters the components belonging to the nets in the selected Net Classes.
- **Design Manager View Class-to-Class Page:** This command cluster the components in the nets that belong to the net classes which comprise the selected Class-to-Class.

Drag and Drop Gathering

You can gather components in a work area using the drag and drop method of component movement. Components can even be moved across windows from the Design Manager View into the Layout View.

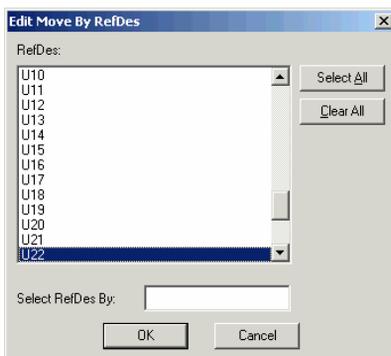
One or more selected components in the Layout View can be dragged to a new location anywhere in the Layout View.

Selected components from the Design Manager View Components, Rooms and Partitions Pages can be dragged into the Layout View and then placed on the board.

Finding a Component by Name

As part of the InterPlace gathering tools, you can employ the **Move By RefDes** command in the *Edit* menu to find and select a specific component by following these steps:

1. Choose the **Move By RefDes** command in the **Edit** menu of the Layout View which displays the *Edit Move By RefDes* dialog shown below:



2. Enter the Reference Designator in the RefDes edit box, or select one from the pull down list of all components in the design by clicking the down arrow. When you click **OK**, InterPlace locates and selects the component.
3. Move the cursor, which has changed to an X, to the desired position in the workspace and click the left mouse button. The selected component moves to the new location.

Jumping to a Component's Location

If you need to find a component and do not know the exact location, you can let InterPlace locate it for you.

To jump to a component:

1. Select a single component from any component list in the Design Manager View. This includes the Components Page, the Partitions Page and the Rooms Page.
2. Choose the **Jump to Component** command in the right mouse menu. InterPlace positions the cursor over the selected component and, if the data tips are enabled, displays the component's reference designator, type and value.

Finding Components by Selection

When components are highlighted or selected, it is easy to locate them on the board. InterPlace uses the **Select Components** command to find designated components and place them in select mode.

To use the **Select Components** command:

1. Select one or more components in the Design Manager View. The components can be selected from the Components, Partitions or Rooms Pages.
2. Choose the **Select Components** command from the right mouse menu. The components you chose in the Design Manager View are now selected in the Layout View.

Placing Components

One of the significant areas of board design in which InterPlace excels is component placement. Positioning components to maximize board real estate is crucial to efficient design and competitive manufacturing costs.

InterPlace provides placement tools that enhance design productivity. The Discrete Placement tool, for example, lets you position large numbers of discrete components with other components with just a few clicks of the mouse. The Visible Placement Area tool helps you see the exact area of the board where a component can be placed without incurring a constraint error.

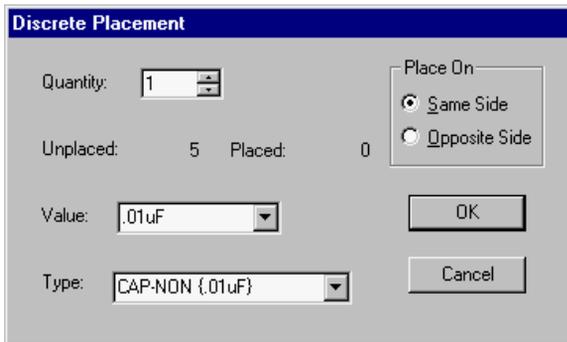
You can direct layer placement using the **Place On** command. Some of the gathering tools discussed earlier in this chapter, such as the **Cluster By** and **Select Components** commands and drag/drop and jumping, can be utilized for component placement as well.

Discrete Placement

With a design that uses large numbers of discrete components, like bypass caps, a designer can spend a lot of time finding and placing these components. The Discrete Placement tool helps you reduce the time expended placing these components. With just a few clicks of the mouse you can rapidly position a large number of discrete components near their associated components, eliminating the need to find each of them and drag them into place.

Follow these steps to rapidly associate multiple components with others in your design:

1. Select the **Discrete Placement** command in the *Edit* Menu of the Layout View. The *Discrete Placement* dialog, shown below, is displayed:



2. The Discrete Placement tool finds the discrete components you want to place by component Type or Value. Enter either a Value or Type in the respective edit boxes, or choose a Value or Type from the drop down lists associated with each.
3. Specify how many discrete components are to be placed with each mouse click by entering a quantity in the Quantity edit box. You may also click the up and down arrows to increase or decrease the quantity.
4. Enable the appropriate radio button in the Place On section to direct the component placement to be on the Same Side or the Opposite Side of the current layer.
5. Click **OK**. The cursor changes to an X and is positioned in the Layout View. The Prompt Line and Status Line, if visible, display information about the placements.
6. Position the cursor over the placed component with which you want the discrete components associated, and click the left mouse button.

Each time you click the left mouse button over the placed component, InterPlace positions the specified number of discrete components near that component.

Prompt Line: While you are in the Discrete Placement mode, the prompt line displays usage instructions as shown below:

```
Click<Left> on Component to place Discretes. Click <Right> to end. Enter <A> to increase Quantity. <Shift><A> to decrease.
```

Click the left mouse button to place discretes; the right mouse button to end placement. While you are placing you can increase the placement quantity by pressing the **A** key, and decrease the quantity with the **Shift+A** keys.

Status Line: The status line displays the choices you made in the *Discrete Placement* dialog, as shown below:

```
Type: CAP-NON (.01UF), Value: .01uF, Quantity: 1, Unplaced: 3, Side: Same
```

Included in this informational display are the Type and Value of the discrete component being placed, the Quantity that is placed each time you click the mouse button, the number of Unplaced components available for placement, and the placement Side.

If you change the placement quantity, and as the number of available components decreases with each placement, InterPlace dynamically updates this display so that you always have the most current information.

Using the Visible Placement Area Tool

The Visible Placement Area tool provides a display of the physical regions on a board where a component can be placed without breaking any rules. The VPA works in three domains: Physical, Electrical and Room. In each of the domains, if there are constraints applied, InterPlace calculates where a selected component can be placed without violating the rules for the selected constraint domain.

For more information and instructions on using the Visible Placement Area tool, see *The Visible Placement Area*, (page 75).

Using Gathering Tools for Placement

Some of the gathering tools provided in InterPlace can also be used to place the components on the board. For all of the brief descriptions below, see the *Finding and Gathering Components* section earlier in this chapter for more details.

- You can select components in the Design Manager View and drag them into the Layout View to be dropped in position. Using Multiple Windows describes component placement between views.
- A component, once found using the **Move by RefDes** command, can be placed on the board.
- Components identified in the Layout View using the **Select Components** command in the Design Manager View, now selected, can be moved into their board destination.

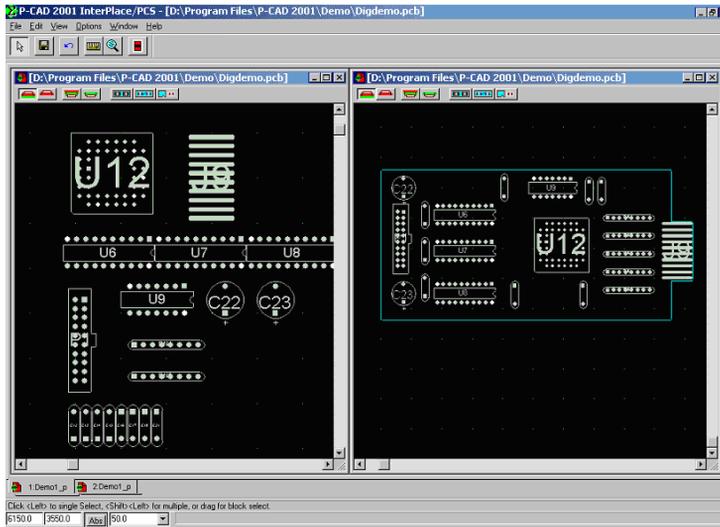
Using Multiple Windows

P-CAD InterPlace allows you to open multiple windows on the same design to view different parts of it simultaneously. To open multiple windows, choose the **Window » New Window** command.

Opening multiple windows of the same design can be used to view the Top and Bottom layers of the same design. Or, multiple windows may be used to work on two different regions of the board simultaneously. Also, you may want to zoom in on a group of unplaced components in one window; while in the other window, you focus on the board region where this component group will be placed. A component can be selected, dragged across the window boundaries, and placed on the board in the opposite window.

Each window you open creates a Layout View window inside the main application window in which you can edit the design. As is standard in Windows, you can change the length and width of each window in relation to the other windows running on your screen. You can also activate or deactivate the Design Manager View, toolbars and status/prompt line areas, which alternately decreases or increases the available space in the application window.

An InterPlace workspace with multiple views of the same design is shown below:



You can easily switch between windows by clicking on the window tabs at the bottom left of your workspace:



Fine-Tuning Component Placement

One of the InterPlace specialties is component manipulation. You can move, swap, align and fix components, rearranging their layout on the board to satisfy the design constraints.

Once components are located in their initial positions on the board, you can fix them in place if you do not want them to be moved. Components can be rotated to an absolute degree or in 90 degree increments. You can flip components from the Top to the Bottom layer, and vice versa. Two components' locations can be swapped, and you can align a group of components horizontally or vertically.

Fixing and Unfixing Components

A fixed component in InterPlace may not change its location. For example, the following commands do not operate on a fixed component:

- Move (dragging a selected component)
- Move By RefDes
- Rotate
- Flip

- Align Component

A fixed component is displayed in the Fixed color selected using the *Options Display* dialog.

To fix a component:

1. Select the component(s).
2. Choose **Fix Components** from the *Edit* menu or the right mouse pop-up menu.

To unfix a component, choose instead the **Unfix Components** command.

Rotating Components

As in PCB, you can rotate selected components 90 degrees by pressing **R** (or **Shift+R**). When a block of selected components are rotated using the **R** key, the entire block revolves around a selection point placed, by default, in the middle of the block. The selection point can be repositioned by choosing the **Selection Point** command and then clicking in the block of components at the desired location.

In InterPlace, you can also rotate all selected components to an absolute angle. For example, you can choose to rotate all selected components to 90 degrees. To do so:

1. Select a component(s).
2. Choose **Rotate To** from the **Edit** menu and then the desired rotation angle from the pull right list.



The selected components are all rotated to the specified angle, regardless of their initial orientation. A single component rotates around its own reference point. Each component in a group of selected components rotates around its own reference point when the **Rotate To** command is used.

Flipping Components

As in PCB, you can flip selected components to the opposite board side by pressing **F**. In InterPlace, you can also flip all selected components to a particular board side. For example, you can choose to flip all selected components to the Bottom layer. To do so:

1. Select a component(s).
2. Choose **Place On** from the **Edit** menu and drag the mouse to the right to select the placement layer.



The selected components are all flipped to the specified layer, regardless of their initial flipped value.

Swapping Components

You can interchange the position of two components on the board outline in the Layout View. To do so:

1. Select two placed components.
2. Choose **Swap Components** from the **Edit** menu or the right mouse pop-up menu.

The selected components exchange positions, including placement layer, while maintaining their individual rotation and constraints.

Aligning Components

Aligning components in InterPlace is identical to the same procedure in PCB. Selected components can be aligned around a selection reference point horizontally or vertically, and even equally spaced. Fixed components, of course, cannot be aligned.

Detailed instructions on using the **Align Components** command can be found in the *Edit Commands* chapter in your *PCB User's Guide*.

Working with PCB and Schematic

InterPlace enhances the design capabilities of P-CAD PCB, P-CAD Relay and P-CAD Schematic by providing easy access to design information and constraint management, and for PCB designs, an easy placement interface.

The cohesion between the originating application and InterPlace makes loading a design easy. If the applications are launched in a specified sequence, you can cross-probe between the applications to find and highlight components and nets in more than one design view.

In InterPlace, you can load a Design Technology Parameters file with constraints that have proven successful in previous designs. Placement specific constraints can then be copied into the current design.

Updates made to the design in IPL can be included in an Engineering Change Order and applied to the matching PCB or Schematic design.

This chapter describes how incorporating the capabilities of InterPlace with the P-CAD suite of applications enhances your circuit board design efforts.

Basic Inter-Application Functions

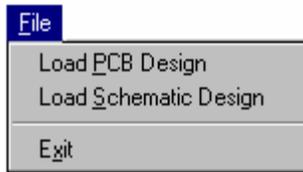
In this section you will learn how to start InterPlace, load a design and use the applications in tandem to find and highlight objects in more than one view.

Launching InterPlace

Once a design has been opened in one of the P-CAD applications (i.e., PCB, Relay or Schematic), you can launch InterPlace by choosing the **Utils » P-CAD Design Constraint Manager or Utils » P-CAD Interplace/PCS** command. The Design Constraint Manager provides access to both InterPlace and the Parametric Constraint Solver, depending on the license you have purchased. InterPlace can also be launched from your Windows desktop.

Loading a Design

When InterPlace is first launched, the application's workspace is blank. To load a design from PCB, Relay or Schematic, choose the appropriate command from the **File** menu as shown below:



When a design is open in InterPlace, the originating application is locked. Changes to the design can be made only in IPL. Once the design is closed or IPL is exited, the originating application is unlocked and can be updated.

The Layout View, (page 21), details which design objects are loaded into InterPlace and describes how they are displayed.

Cross-Probing Between Applications



When you enable the **DDE Hotlinks** option in the *Options Configure* dialogs of P-CAD Schematic and P-CAD InterPlace, it is possible to highlight a component or net in Schematic and see the corresponding component or net highlighted in InterPlace as well.

Because the Layout View is not available for designs loaded into InterPlace from Schematic, the graphic representation of the design must come from the PCB design file.

In order to establish the necessary communication link and load the design files correctly, you must follow these steps:

1. Open P-CAD Schematic.
2. Open P-CAD InterPlace using the **Utils » Design Constraint Manager** command. This establishes the link between Schematic and InterPlace. Only two applications can be linked at any time. Do not load the Schematic design into InterPlace.
3. Open P-CAD PCB and load the desired design.
4. Load the PCB design in InterPlace. InterPlace now graphically displays the design in its Layout View and locks PCB to any updates.
5. Load the Schematic design in the Schematic application. A net or component highlighted in Schematic will be highlighted in IPL as well since the two applications were linked at step 2.

When components or nets are highlighted in both Schematic and InterPlace, you can select the highlighted objects in InterPlace using the **Edit » Select Highlighted** command. Once selected, you can perform additional operations on the objects in InterPlace such as moving, fixing, etc.

If you change the highlight color of an object in the InterPlace, the same object in Schematic is automatically updated with that same highlight color.

Using a Design Technology Parameters File

When a design is loaded into InterPlace the existing constraints are included and accessible through **Options » Design Rules**. If you have a Design Technology Parameters file containing

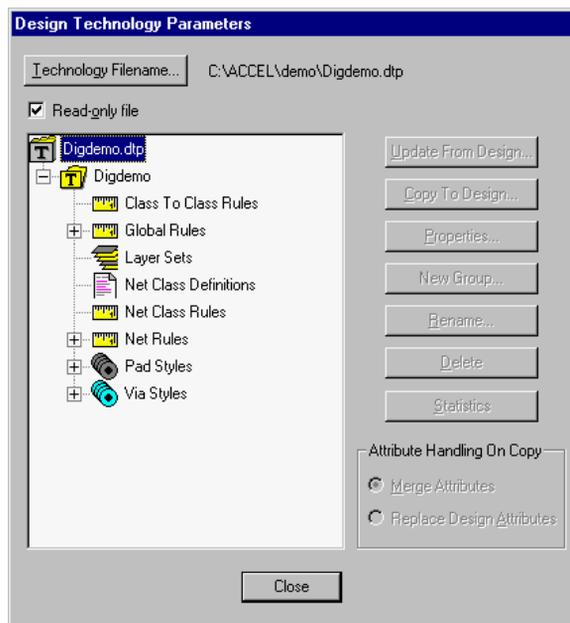
constraints that have proven successful in previous designs, you can load that file into InterPlace and update pertinent attributes in the current design.

This section describes which of the features of the Design Technology Parameters file are available to the design while in InterPlace.

For more complete information on the Design Technology Parameters file, see your *PCB User's Guide* or *Schematic User's Guide*.

Accessing a DTP

To access a Design Technology Parameters file, choose the **File » Design Technology Parameters** command. The *Design Technology Parameters* dialog, shown on the next page, appears:



DTP Operations

The following list shows which operations are available in InterPlace from the *Design Technology Parameters* dialog:

- Update From Design
- Copy To Design
- Properties
- New Section or Item
- Rename

- Delete
- Statistics

For each of the categories in the file, a particular set of operations is allowed. In all cases, you can Update From Design and Copy To Design, as well as Delete items in each category. In addition to those basic functions, the following additional operations are available in each of the categories:

- **Class-to-Class Rules:** In this category you can add a New Section or item.
- **Global Rules (Layers):** For the entire category you can add a New Section or item. For an individual layer in this category you can access the Properties and Statistics, as well as Rename an item.
- **Net Class Definitions:** In this category a New Section or item can be added.
- **Net Class Rules:** In this category a New Section or item can be added.
- **Net Rules:** A New Section or item can be added to the category. The Properties and Statistics are accessible for individual nets, which can also be Renamed.

The Attribute Handling On Copy section of the Design Technology Parameters dialog is enabled for the Copy to Design function for selected Class-to-Class Rules, Net Class Rules, Net Rules or individual nets. You may choose to Merge Attributes in the design with those in the DTP file, or Replace Design Attributes with those in the DTP file by clicking the appropriate radio button.

Refer to your *PCB User's Guide, File Commands*, for more information on how the Merge Attributes function behaves in each of the relevant categories.

How InterPlace Updates Affect PCB and Schematic

Updates made to a design in InterPlace are saved to the design file. These changes, once committed to the design file, can become part of an Engineering Change Order and subsequently transferred to a matching design in PCB or Schematic.

Recording Changes with ECOs

If you have turned on the ECO recorder in your originating application before making changes to the design in InterPlace, the changes will be recorded in the `.eco` file when the design is saved. If you are using the WAS/IS ECO format you should note that only RefDes changes are recorded, and because you cannot change the RefDes in IPL, InterPlace will make no updates to these files.

Once you exit InterPlace, you can import or export from PCB or Schematic the changes recorded in the ECO file, just as you would have done if those changes had been made in PCB or Schematic.

Make sure to turn on the ECO recorder before launching InterPlace and loading the design.

If you should forget to turn on the ECO recorder, you can still record the changes by following these steps:

1. Return to the originating application.

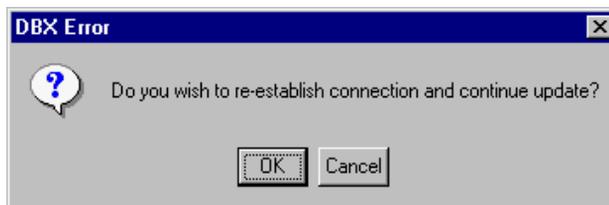
2. Terminate the link between the applications by clicking the **Cancel** button on the *P-CAD DBX* dialog, shown below:



3. Turn on the ECO recorder using the **Utils » Record ECOs** command or by clicking the  button.
4. Return to InterPlace and save the changes to the design using the **File » Update PCB/Schematic Design** command.
5. The error message dialog, shown below, informs you that the connection has been terminated. Click the **OK** button to continue.



6. In the next *DBX Error* dialog you must choose whether to re-establish the connection and continue the update process or cancel it.



7. If you wish to continue the update, click the **OK** button. P-CAD DBX is automatically re-activated and the link between the applications re-established. The changes saved to the file in IPL are also saved to the ECO recorder.

If you chose either the **File » Close** or **File » Exit** commands in step #4, and you click the **Cancel** button in the *DBX Error* dialog, your design is cleared from the InterPlace workspace and the changes you made are not recorded in the design file.

The Visible Placement Area



In addition to the basic Room tools available in the Layout View of the InterPlace, IPL also provides a visual display on the board showing the area where components can be placed without violating constraints you have assigned to a PCB design.

In combination with the basic Room tools, the Visible Placement Area enhances your ability to place components quickly, accurately and violation free. The VPA interface displays the legal region of the board where a selected component may be placed after accounting for clearance, net connection length and Room inclusion rules.

This chapter describes the basic functionality of working with the Visible Placement Area tool. You will also learn which constraints are employed by VPA for the individual constraint domains, and how to apply them during component placement.

Since only PCB designs use the graphical representation of the board layout, components and nets, this chapter applies only to designs loaded from PCB.

Working with the Visible Placement Area

Working with the Visible Placement Area is easy once you know the VPA basic functions. In this section you will learn how to enable the VPA domains using the VPA toolbar and select the color the VPA domain area uses in its display. The way a component is measured, which determines if it will fit in the constraint area, is defined here as well.

The VPA Toolbar

The VPA toolbar is attached to the main workspace in the Layout View. The VPA toolbar, shown below, activates the display of each of the placement domains.



The buttons on the VPA toolbar enable the constraint domain display as follows:



Physical Domain



Electrical Domain



Room Domain

The options in the VPA toolbar impact only its specific design window. When you have multiple design windows open in the Layout View, each window has its own set of VPA toolbar buttons. For example, you may enable the display of the Electrical Domain in one window, and the display of the Physical Domain in a second window.

VPA Domain Display Options

The Visible Placement domains can be displayed within the board outline in the color of your choice. The VPA button in the *Options Display* dialog displays the standard Windows color dialog where you can choose or design a color for the VPA display.

For more information on using the *Options Display* dialog, see *Options Commands*, (page 105).

Enabling and Disabling a VPA Domain

To enable the display of one or more constraint domains, click each desired domain button on the VPA toolbar.

When a domain is selected, its corresponding button appears pushed. Click the domain's button again to disable it.

Constraint Domains

P-CAD InterPlace specializes in managing constraints which impact the optimal layout of the components on the board. These constraints are used in the definition of the Visible Placement Area to determine and display a region on the board where components can be placed without violating the assigned rules.

InterPlace categorizes the constraints by function into domains. Each domain contains certain constraints that apply to a specific function as described below:

- **Physical:** Physical constraints specify clearances between design objects and between those objects and the board edges.
- **Electrical:** The Electrical domain provides maximum length measurements for net connections.
- **Room:** Room constraints define maximum component height and room placement.

The constraint domains, the constraints within each domain and the precedence level at which they are applied are listed in the following table:

Domain	Constraint Name	Precedence Level
Physical	BoardEdgeClearance ComponentSpacing PadToPadClearance Clearance	D D, L, C D, L, NC, N, CC D, L, NC, N, CC
Electrical	MaxNetLength	D, NC, N
Room	MaxComponentHeight Room Inclusion	D, L, R R

For the Precedence Level, the following notation has been used: **C**=Component; **CC**=Class-to-Class; **D**=Design; **L**=Layer; **NC**=Net Class; **N**=Net; and **R**=Room.

Viewing Constraint Domains

You can choose to display one or more of the Visible Placement Area domains, in any combination, within the board outline. If more than one domain is selected, the Visible Placement Area displayed satisfies all selected domain constraints simultaneously. The optimal Room location of a component also satisfies these selected constraint domains.

To view the Visible Placement Area(s) for a selected component:

1. Choose which constraint domain(s) you wish to view: Physical, Electrical or Room. Click the corresponding button(s) on the VPA toolbar: Physical Domain, Electrical Domain, or Room Domain. Any or all of the constraint domains may be viewed simultaneously.
2. Select one component. The placement regions for that component, which satisfy the selected constraint domains, are displayed in the Visible Placement Area color specified in Options Display.

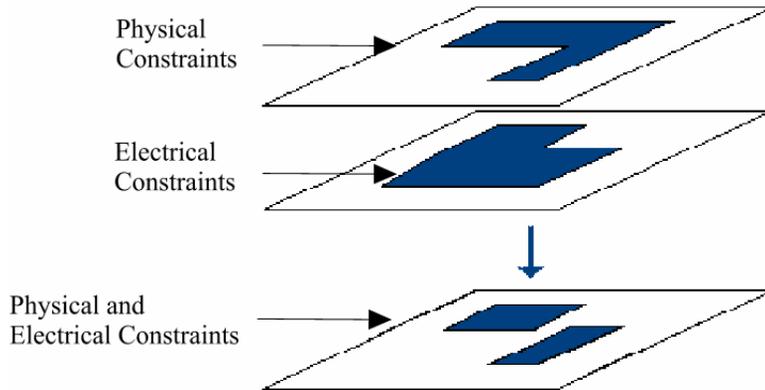
The Bounding Rectangle, which measures the area needed to place the component inside the constraint domain, combines the pads with the lines and arcs defined on the non-signal layer specified by the component attribute PackageOutlineLayer.

Constraint Visualization

With the Visible Placement Area, you have the ability to visualize the impact of design constraints on component placement.

When a component is selected for placement, the regions available for placement that satisfy the selected constraints are highlighted within the board outline. These regions can be viewed for all domains either individually or in combination. The valid placement area is displayed as a colored region on the board surface; this region is called the Visible Placement Area.

An example of combining constraint domains is shown below:



If the component is placed within the subset of the constraint areas common to both the Physical and Electrical constraints, it will satisfy the allowed placement area. Notice in the example above that to satisfy the Physical constraints or the Electrical constraints large regions of the board are available for component placement. To satisfy both the Physical and Electrical constraints simultaneously, however, the available visible placement area for valid component placement is significantly reduced.

You can view the visible placement area of a component for any constraint domain combination. The visible placement area aids in component placement. Additionally, it can help you determine which design constraints are critically limiting placement options in a board region.

Physical Constraints

The following table shows the constraints utilized by VPA in the Physical domain, along with a definition, valid entries and the units available:

Constraint Name	Description	Units
PadToPadClearance	Definition: Overrides the default Clearance by specifying a pad to pad clearance value. Valid Range: > 0	mil, inch, mm, cm
ComponentSpacing	Definition: The clearance value between components. Valid Range: > 0	mil, inch, mm, cm

Constraint Name	Description	Units
BoardEdgeClearance	Definition: The clearance value between an object and the board edge. Valid Range: > 0	mil, inch, mm, cm
Clearance	Definition: The clearance value between any net object. Valid Range: > 0	mil, inch, mm, cm

Electrical Constraints

The following table shows the constraints utilized by VPA in the Electrical domain, along with a definition, valid entries and the units available:

Constraint Name	Description	Units
MaxNetLength	Definition: The maximum length of each net connection emanating from the selected component. The length is based on the distance between pads, not the length of the entire net. PCB considers only routed copper lengths when determining the maximum allowable distance between two nodes. Valid Range: > 0	mil, inch, mm, cm

Room Constraints

The following table shows the constraints utilized by VPA in the Room domain, along with a definition, valid entries and the units available

Constraint Name	Description	Units
MaxComponentHeight	Definition: The maximum component height allowed for a specified area of the board. Valid Range: > 0	mil, inch, mm, cm
Room Inclusion	Definition: VPA checks for a room(s) assignment and, if a	N/A

Constraint Name	Description	Units
	room(s) is assigned, restricts the visible placement area to the assigned room(s).	

File Commands

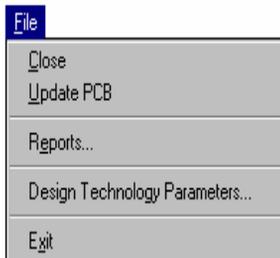


The File commands allow you to load, close, and update designs with their design constraints into P-CAD InterPlace. From the *File* menu, you can also exit the application.

The **File** menu displayed in the margin, containing the **File » Load PCB Design** and **File » Load Schematic Design** commands, is available when a design is yet to be loaded. An expanded *File* menu is available once the PCB or Schematic design has been loaded into InterPlace. See the following pages for details.

Most of the File commands cannot be undone, meaning that once an action is taken, it cannot be reversed by the **Edit » Undo** command.

File Load PCB Design



The **File » Load PCB Design** command loads an existing PCB design file into P-CAD InterPlace.

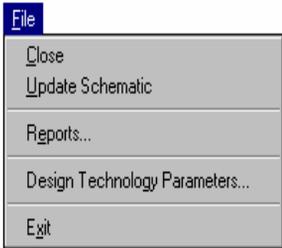
Before a design has been loaded, the File menu appears as shown in the chapter introduction. When a design is loaded, the **File** menu changes, as shown in the margin, to include the additional commands: Close, Update PCB, Reports, Design Technology Parameters and Exit.

To load the design into InterPlace, the design must be currently open in P-CAD PCB or P-CAD Relay. When you choose **Load PCB Design**, the design that is active in the P-CAD application is automatically loaded into the InterPlace

workspace. The design will be locked in the originating application while accessed through InterPlace to prevent conflicting modifications.

After loading, the board outline and components appear in a Layout View window. Refer to *The Layout View*, (page 21), for additional details. The Design Manager View organization tools become available as well. Refer to *The Design Manager View*, (page 37), for more information.

File Load Schematic Design



The **File » Load Schematic Design** command loads an existing Schematic design file into P-CAD InterPlace. If a design is already loaded, this command is unavailable.

Before a design has been loaded, the **File** menu appears as shown in the chapter introduction. When a design is loaded, the **File** menu changes, as shown in the margin, to include the additional commands: Close, Update Schematic, Reports, Design Technology Parameters and Exit.

To load the design into InterPlace, the design must be currently open in P-CAD Schematic. When you choose **Load Schematic Design**, the design that is active in the P-CAD application is automatically loaded into the InterPlace workspace. The design will be locked in the originating application while accessed through InterPlace to prevent conflicting modifications.

The organizational abilities of the Design Manager View become accessible. Refer to *The Design Manager View*, (page 37), for more information.

File Close

Closes all views for the design in InterPlace. The design is unlocked in the originating application, permitting changes again in the PCB or Schematic design editor.

If the design has been changed but not yet saved, you are asked whether or not you want to save your changes before closing.

File Update PCB

Saves the changes to the PCB or Relay design. When you select **File » Update PCB**, the file remains open so you can continue working on it in InterPlace.

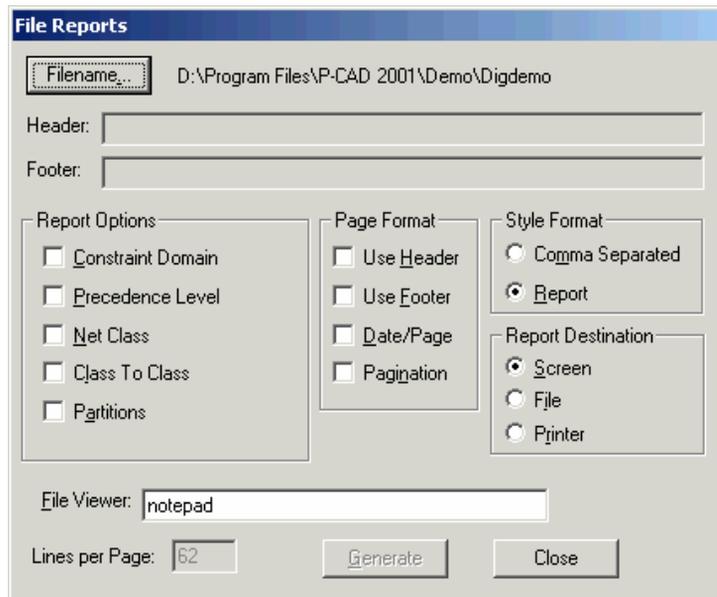
File Update Schematic

Saves the changes to the Schematic design file. When you select **File » Update Schematic**, the file remains open so you can continue working on it in InterPlace.

File Reports

Allows you to generate reports with specific output options. The selected Style Format dictates the options available for each of the styles: Comma Separated and Report. With the Report style, in addition to the Report Options, Lines per Page and Report Destination, you may select the page format and define the Header and Footer. These options are saved when you exit the program.

The dialog appears as follows:



Filename

You may specify the report or reports you wish to generate from the list of report types listed in the Report Options area. Each report type has its own, unique file extension; the filename defaults to the name of the current design.

Report File Extensions

The extensions are set as listed below and cannot be changed. The files are all saved as text files.

- constraint_domain.txt
- precedence_level.txt
- net_classes.txt
- class_to_class.txt
- partitions.txt.

Report Options

The reports are described below:

- **Constraint Domain:** The Constraint Domain report lists the three domains pertinent to placement constraint management: Physical, Electrical, and Room. Within each domain, in their precedence order, are each item's constraint names, values, expressions and comments.
- **Precedence Level:** The Precedence Level report shows, for each precedence level and item, the constraint names, values, expressions and comments.
- **Net Class:** The Net Class report shows the nets that are assigned to each net class in the design.
- **Class-to-Class:** The Class-To-Class report shows the net classes assigned to each class-to-class along with the constraints added to the class-to-class.
- **Partitions:** The Partitions report lists the components assigned to each Partition.

Page Format

These Page Format options are enabled when the Report format is selected:

- **Use Header** and **Use Footer** include the information you specified in the header and footer dialog fields.
- **Date/Page** includes the current date and the page number.
- **Pagination** allows you to create your own pagination (lines per page). When this option is enabled print from the **DOS Print** command, not the Notepad.

Style Format

- **Comma Separated** puts all data in comma-separated format. This format can be imported into other spreadsheet and database programs.
- **Report** produces a report format with columns and spaces, etc.

Lines per Page

Lines per Page allows you to specify the number of lines per page in your report.

Report Destination Window

- **Screen** sends the output to a file and opens the file using the Notepad Utility.
- **File** sends the output to a file.
- **Printer** sends output directly to the printer without creating files.

Generate

Generates all selected reports using the options selected.

File Design Technology Parameters

Captures design data, including design constraints, and stores this data in a design technology parameters file. This data can be used in any P-CAD design. Thus this file becomes a template from which you can selectively use the captured design data in any design.

A design technology parameters file is like a storage box. You can reorganize it, add or remove items, alter some of its contents, and use what is stored inside. For a project or series of projects, you may have several items that you will have to use repeatedly for creating PCB or Schematic designs. The design technology parameters file is a perfect storage box for the following design items: class-to-class rules, global (layer) rules, layer sets, net class definitions, net class rules, net rules and pad and via styles.

Design constraints, or rules, can be transferred using a design technology parameter file between InterPlace and any PCB or Schematic design.

In P-CAD InterPlace, Design Technology Parameters can be used in a variety of ways, including:

- Applying company-standard design rules to your board layout.
- Storing design rules created or organized in InterPlace for later use in other PCB or Schematic designs. You can create or change design rules pertaining to Class-to-Class Rules, Global (layer) Rules, Net Class Definitions, Net Class Rules and Net Rules in InterPlace.

For more information, see *Using a Design Technology Parameters File*, (page 70).

File Exit

Exits the InterPlace program.

If the design has been modified since the last save, you are prompted whether you want to save the changes.

The design file is unlocked in the originating application, permitting changes again in the PCB or Schematic design editor.

The program writes information to the `.ini` file when you choose **Exit**. This information, which will apply to subsequent InterPlace sessions, consists of parameters and settings such as values set in Options Configure, etc.

Edit Commands



Edit commands allow you to make changes to objects in the InterPlace workspace. Often these editing actions can be reversed using the **Undo** command.

The **Edit** menu, shown in the left margin, exists only when a PCB design is loaded into InterPlace.

Many of the Edit commands operate only if the object is already selected. See **Edit » Select** for selection details.

Edit Undo (Ctrl+Z)



Undo reverses your last completed action to the design. You can undo component modifications in the Layout View such as move, rotate, and flip (actions performed in Select mode). Many File commands, such as **File Load Design**, **Update Design**, etc. cannot be undone. If an action cannot be undone (or there is nothing to undo), **Undo** appears grayed on the **Edit** menu.

The **Undo** button on the toolbar and the **U** key are equivalent to the **Edit » Undo** command.

Edit Properties

Displays the *Properties* dialog for the component selected in the Layout View. This dialog lets you query the selected component's properties. You must use the Select tool to select the component(s) before you can choose **Edit » Properties** from the *Edit* menu or the *right mouse* popup menu.

The *Properties* dialog that appears is specific to the component you select. If multiple components are selected, the properties displayed match in all selected components.

Right Mouse Button to Select Properties

You can also choose the **Properties** command from a pop-up menu. Select a component and click the right mouse button to bring up a pop-up menu.

See *Right Mouse Commands*, (page 17), for a complete list of right mouse commands.

Double Click to Select Properties

If the **Double Click Displays Properties** option is enabled in the *Options Preferences* dialog, you can also choose the **Properties** command by left double-clicking on a component to open its *Property* dialog.

Data Tips

A data tip contains information about the underlying object and appears when you place the mouse over the following objects:

- A component
- A component pad
- A connection
- A room

For components, the information displayed includes the reference designator, component type, and value attribute. For a component pad, this information includes the reference designator, pad number, and attached net name. For a connection, this information includes the net name, as well as the reference designators and pad numbers of the connection attachments. For rooms, the room name and placement side are displayed.

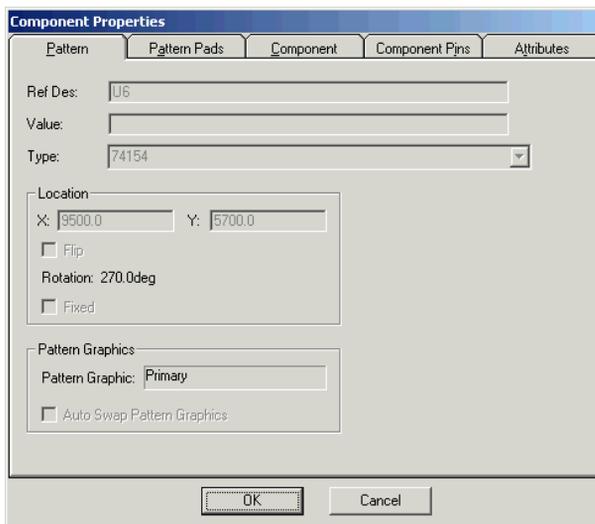
A connection data tip is shown in the diagram below:



You can turn off the display of the properties data tips in the *Options Display* dialog.

Component Properties

When you select one or more components and choose the **Edit » Properties** command, the *Component Properties* dialog appears with the **Pattern** tab selected.



The *Component Properties* dialog allows you to observe properties for the selected component.

Tabs provide access to additional information:

- **Pattern:** Displays the component pattern and its properties, including reference designator, type, value, and fixed state. The Pattern Graphic used is also listed.
- **Pattern Pads:** Lists the pin numbers of the pads, as well as their location, rotation, and pad style.
- **Component:** Displays component information on a gate-by-gate basis.
- **Component Pins:** Displays the pin information for the component, including pin number, pin designator, pin equivalence, and electrical type.
- **Attributes:** Displays the component's attributes by their name and value.

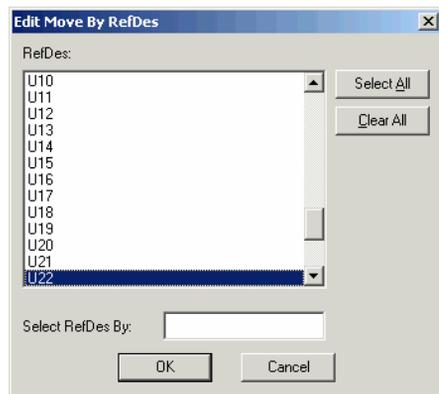
All component properties are defined in P-CAD PCB or Relay. These attributes are transferred into P-CAD InterPlace upon loading the design. Refer to your *PCB User's Guide* or online help for information about modifying component properties or attributes.

Edit Move By RefDes

This command allows you to select and move a component in the Layout View by entering its reference designator, without knowing where the component is located in the design.

This command does not work on fixed components.

When you choose this command the *Edit Move By RefDes* dialog appears:



To move a component by its reference designator:

1. Enable the **Select** tool.
2. Choose the **Edit » Move By RefDes** command.
3. Select a reference designator from, or type a value in, the RefDes drop down combo box. The list contains all reference designators in the design.
4. Click **OK**.
5. The component is selected, and the cursor appears as a crosshair shape.
6. Move the cursor to the workspace location where you want to place the component.
7. Click to move it to the new location.

You can click down to make a ghost, then drag and drop (release) to place it more accurately. To cancel ghosting (and moving) of a component, click the right mouse button.

Edit Align Components

In the Layout View, components can be aligned around a selection reference point either horizontally or vertically, and as an option, the components can be equally spaced. If a number of components are off-grid, these components can be aligned back on-grid.

The alignment of components is undoable.

This command does not work on fixed components.

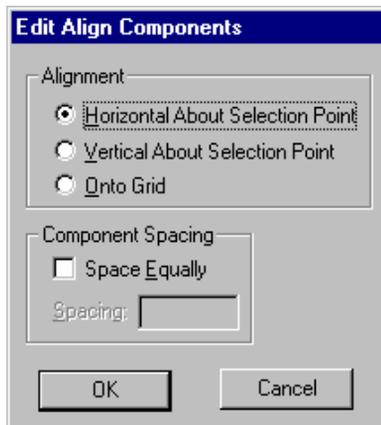
Align Horizontally or Vertically

To align a component:

1. Select the components to align.
2. Place a selection reference point using the right mouse pop-up menu. This is the point about which the components will be aligned, either horizontally or vertically.

Without selecting a selection reference point, the alignment and component spacing options are grayed.

3. Select the command **Edit » Align Components**. The *Edit Align Components* dialog appears.



4. Select either horizontal or vertical alignment.
5. To align the components with equal spacing, check the **Space Equally** check box and enter the Spacing value. The spacing value is the distance between the reference points of the components.
6. Press **OK** and the selected components will be aligned.

Align to Grid

To align a component to grid:

1. Select the component(s).
2. Select the command **Edit » Align Components**. The *Edit Align Components* dialog appears as shown above.
3. Click **OK** and the selected components are aligned to grid. Each selected off-grid component is moved to the nearest grid point.

Edit Fix Components

Fixes selected components in their position on the board outline in the Layout View. Fixed components cannot be moved, rotated, or flipped.

Edit Unfix Components

Unfixes selected components in the Layout View. Unfixed components can be moved, rotated, or flipped.

Edit Swap Components

Reverses the position of two selected components.

Edit Rotate To

Used to rotate to an absolute degree, chosen from the list of options shown below, a single component or a group of components.



The selected component(s) are rotated about their own reference point(s).

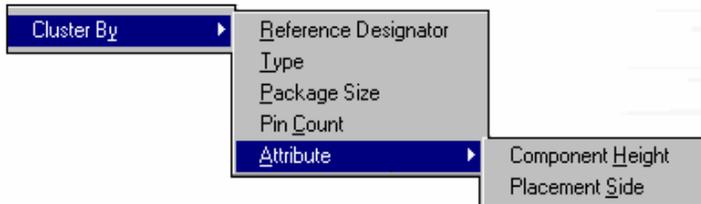
Edit Place On

Places a selected component or group of components on a chosen layer (Top or Bottom), as shown below:



Edit Cluster By

Groups components into a common area from anywhere in the design. Components may be gathered by reference designator, type, package size, pin count or the ComponentHeight and PlacementSide attributes from the Cluster By options list, shown below:



Once the cluster is selected, the cursor changes to an X. Position the cursor in the Layout View location where you want the components to be placed and click the left mouse button. The components appear clustered in the order chosen from the Cluster By options.

Edit Select All

Selects all items on all enabled layers of the Layout View and makes them available for additional commands to be performed.

Edit Deselect All

Deselects all currently selected items in the Layout View.

Edit Highlight

Highlights the selected item or items in the Layout View using the current highlight color chosen through the **Options » Display** command.

You can change the highlight color for an object you want highlighted without affecting the highlight colors of other highlighted objects. To do so, change the highlight color in the *Options Display* dialog, select the object, and choose **Edit » Highlight** from the **Edit** menu or **Highlight** from the popup menu. Repeat this process for each object you want to highlight, so that each object selected has a different highlight color.

When you choose this command, the selected items are drawn in the highlight color until they are unhighlighted. The selection color overrides the highlight color, so you won't see the highlights until the items are deselected.

You can also access this command by selecting an item or items, clicking the **right mouse button** to bring up the popup menu, and choosing **Highlight**.

Objects can be highlighted between the applications when the DDE Hotlinks option is enabled in their respective *Options Configure* dialogs. *The Layout View*, (page 21), provides detailed instructions on *Cross-Probing Between Applications*.

Edit Unhighlight

Removes the highlighting from the selected item or items in the Layout View and restores the normal object colors.

If P-CAD InterPlace and P-CAD Schematic or PCB are both running, and if the **DDE Hotlinks** check box in the *Options Configure* dialog is checked in both applications, the highlights are removed from the selected item or items in both applications.

You can also access this command by selecting an item or items, clicking the **right mouse button** to bring up the popup menu, and choosing **Unhighlight**.

Edit Unhighlight All

Select the **Unhighlight All** command to remove the highlight from all items in the Layout View and to restore the normal object colors. This command applies to all highlighted objects, regardless of whether they are selected or not.

If P-CAD InterPlace and P-CAD Schematic or PCB are both running, and if the **DDE Hotlinks** check box in the *Options Configure* dialog is checked in both applications, the highlights are removed from the selected item or items in both applications.

Edit Measure

Measures the X distance, Y distance, and total distance between two points in the placement interface. Use the ruler button on the toolbar as a shortcut for the **Edit » Measure** command.

You can measure vertical, horizontal, and diagonal distances and the results appear on the status line. The measurements are in either mils or millimeters, depending on current settings in Options Configure.

Measure is a mode, meaning that if you were, for example, selecting objects, when you use Measure you exit the select mode to go into measure mode. After you have measured, you need to restart whatever mode you were in to resume editing.

To measure the distance between points:



1. Select **Measure** from the **Edit** menu (or use the toolbar button).

2. Move the cursor to the first point of your measurement in the workspace. Click and drag to the end point of the measurement.
3. Before you release, look at the results on the status line for X distance, Y distance, and T (for total) distance (in mm or mils). When you release the button, the results disappear.

Measuring with the mouse does not snap to grid, if you have View Snap to Grid disabled.

Edit Select

Edit » Select is a tool which allows you to perform operations on design objects in the Layout View. You can select items within the Constraint Editor and the Design Manager Views without enabling the Select tool.



Select can also be enabled from the toolbar button (arrow). The **S** key may also be used as a shortcut key.

Select Actions

- single-, multiple-, block-select.
- move, resize, rotate, flip, modify, highlight, unhighlight, fix, and unfix.

Status line information

When you select an item, the status line information area identifies the item, either specifically (part reference designator) or generally (number of items selected).

Comp U12 selected.

Select Commands

Edit Align Components, Move by RefDes, Rotate To, Place On, Cluster By, Select All, Deselect All, Highlight, Unhighlight, Fix Components, Unfix Components, and Properties are operational in Select mode.

Select actions are possible only if an object is selected. For example, you cannot fix a component unless it has been selected.

Information included in this section only covers the mouse/cursor actions for Select.

Selecting Objects

To single select, click a single object. All other selected objects are deselected. You must be on the object's layer.

To multiple select, first select a single object. Then, hold down the **Ctrl** key and click on additional objects/items. The selected objects are surrounded by a selection box, which increases as you add items to the multiple selection. Click again on selected items (still using the **Ctrl** key) to deselect them individually. If you release the **Ctrl** key and click anywhere other than one of the selected

objects, all are deselected. You can select a list of items in any of the Design Manager View pages from the first selected item to the end of the list by selecting the first item and then pressing the **Shift+End** keys.

To deselect, click an empty area of the workspace to deselect all items outside the selection region or choose the **Edit » Deselect All** command.

To block select, click, hold and drag the cursor to create a selection box surrounding a block of objects, then release the button. You can add objects to the block selection individually by doing a multiple select (see above paragraph). To cancel while dragging the selection box, click the right mouse button.

When block selecting, the selection window follows the layer option and selection mode set in the **Options » Block Selection** command. Inside Block is the default setting, which means that all items completely inside the selection block will be included. If you have the Outside Block option enabled, then the selection occurs outside of the selection block. If you have the Touching Block option enabled, a block selection includes everything inside and touching the selection block.

A block selection mask can be used. Objects can be filtered or masked in a variety of ways, depending on how you set up the selection options. Use the **Options » Block Selection** command for altering or setting the selection options.

When Objects Overlap

Only objects on the current layer may be selected. If objects on the top and bottom layers overlap, you must move to the object's layer to select it. Click the **Top Current** or **Bottom Current** button on the toolbar to make that layer current.

Objects on both Top and Bottom layers may be selected using the **Options » Block Selection** or **Edit » Select All** commands.

Moving Objects

To move an object, select it, then click on the object and drag the cursor to the new location. Release to place the object. In P-CAD InterPlace, only components may be selected and moved.

If you are moving multiple objects within a selection box, click anywhere in the selection box and drag; all the selected objects in the box follow. Release the button to place the objects.

To cancel a move in progress, click the right mouse button while the left mouse button is still depressed.

Rotating and Flipping

Select a component. Rotate and flip functions work on multiple- or block-selected objects as well.

Press **R** to rotate 90 degrees counterclockwise. **Shift+R** rotates the object by the value specified in the Rotation Increment field of the *Options Configure* dialog (default is 45 degrees).

To flip, press **F** while the object is selected. The object is flipped to the opposite side of the board. The object's orientation reverses (about the Y axis) since you view the component from its back side.

When you flip a component, pad characteristics on the Top layer are swapped with corresponding characteristics on the Bottom layer.

Viewing Properties

With the Select tool enabled, choose **Edit » Properties** after an object is selected to display a Properties dialog for the selected object. Refer to *Edit Properties*, (page 88), for information on the Properties dialogs and data tips.

Right Mouse Button Commands

When you select an object and click the right mouse button, a pop-up menu appears providing shortcuts to common commands performed on selected objects.

The pop-up menu is operational in Select mode.

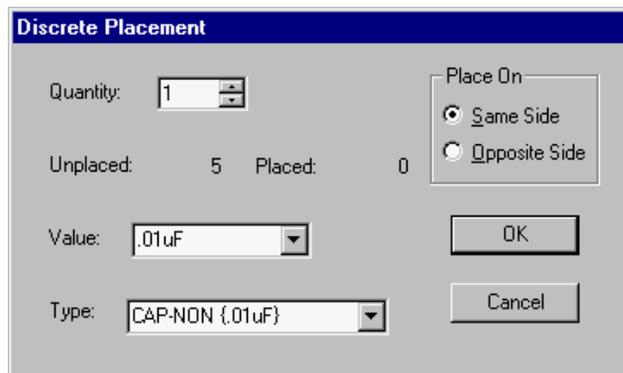
Refer to *Design Manager Right Mouse Commands*, (page 51), for a summary of the right mouse command pop-up options.

Edit Discrete Placement



The Discrete Placement command is used to associate discrete components, such as bypass caps, with particular components. These discrete components are quickly located and easily moved into place with the **Edit » Discrete Placement** command. IPL dynamically maintains a count of placed and available components.

Select the **Edit » Discrete Placement** command or click the **Discrete Placement** button on the command toolbar to display the dialog:



The options in the *Discrete Placement* dialog are:

- **Quantity:** The Quantity edit box indicates the number of components to be placed.

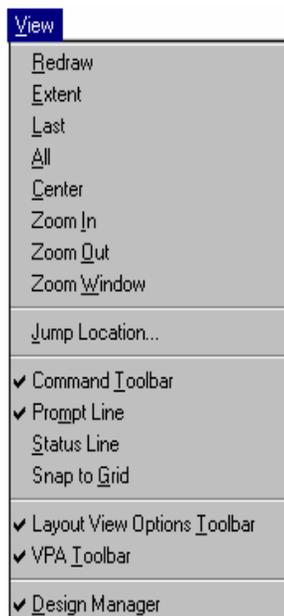
- **Unplaced/Placed:** Displays the number of components that have been placed and those that remain to be placed. These counts are dynamically updated as components are placed.
- **Value:** Type in the Value of the components to be placed or choose from the list of components that appears when you click the **down arrow** button.
- **Type:** Enter the Type of the components to be placed or choose from the list of components that appears when you click the **down arrow** button.
- **Place On:** Enable Same Side or Opposite Side by clicking the desired button.

To place components using Discrete Placement:

1. Select the **Edit » Discrete Placement** command or click the **Discrete Placement** button on the command toolbar. The *Discrete Placement* dialog appears.
2. Enter a Value or Type in the appropriate edit box, or select the desired component's Value or Type from the drop down lists. InterPlace displays how many of the selected component have been placed and how many are available to be placed.
3. Enter the number of components to be placed in the Quantity edit box. You must enter a quantity less than or equal to the number of unplaced components.
4. Choose the layer side on which to place the component.
5. Click **OK** to begin the placement. The *Discrete Placement* dialog disappears and the cursor, which is now a cross, is positioned in the Layout View.
6. Position the cursor over the component with which you want to associate the discrete component(s) and click the left mouse button. The number of components you entered in the Quantity box are placed next to that component. Continue clicking over each component where you want to place the discrete components. When you have finished placing the discrete components, click the right mouse button to terminate placement for that component.

To continue placement of another type of discrete component, simply click the **left mouse button** and the *Discrete Placement* dialog reappears.

View Commands



View commands allow you to temporarily alter your view of the workspace. In the Layout View of a loaded PCB design, you can change the view to better pinpoint locations and objects in the design. You can also enable or disable the display of the Design Manager View, toolbars, and other workspace information.

For more permanent adjustments to your display characteristics, refer to the **Options** menu.

The **View** menu of InterPlace, for a design loaded from PCB, is shown in the left margin. When a design is loaded from Schematic, the **View** menu is reduced. Commands available in the **View** menu when a Schematic design is loaded are noted in this chapter with the Schematic icon.

View Redraw

Clears everything in the Layout View window to the background color and then repaints the screen.

View Redraw is good to use when you have leftover shapes from moving objects; Redraw erases leftover graphics.

When you redraw, the items that reside on the current layer are drawn last. The current layer for the active window is indicated by its Top Current or Bottom Current toolbar buttons.

This command causes collocated top and bottom layer items to be drawn in the correct order. For example, if the current layer is Top, all Top layer SMT pads of an edge connector are drawn on top all Bottom layer SMT pads in the edge connector. Other View/Zoom commands do not necessarily draw collocated pads in the correct order.

To interrupt a redraw in progress, click the right mouse button or press **Esc**.

View Extent

Displays the extent of all objects placed in the Layout View window.

P-CAD Interplace computes and draws the window(s) so that all placed objects are visible on enabled layers. Disabled layers are ignored.

To enable a layer, click the **Top Current** or **Bottom Current** button(s) on the Layout View Options Toolbar. Disable a layer by clicking the **Disable Bottom** or **Disable Top** buttons on this same toolbar.

View Last

Redraws the previous view, if you have altered the view in any way.

There is no previous view until you run at least one View command that changes the view area. If you run **View » Last** multiple times, you toggle between the last two views.

Scrolling does not affect the previous view.

View All

Redraws the Layout View window with the entire workspace shown. The scroll bars don't appear at this zoom level.

The workspace size displayed matches that of the loaded P-CAD PCB or Relay design.

View » All is the default view when you load a PCB design into P-CAD Interplace.

View Center

Redraws the Layout View window using the cursor as the relative center point.

The cursor becomes a magnifying glass shape, signifying the zoom mode and prompting you to click in the workspace; the point where you click in the workspace becomes the center of the screen. To cancel the zoom after the magnifying glass cursor appears, click the right mouse button or press **Esc**.

The selected point may not be in the center of the screen if you are near a workspace boundary.

C Key

The shortcut key for **View » Center** is **C**. Using **C** is useful for panning across the workspace. You don't need to click the mouse button, just move the cursor to the point you want centered and press **C**. You can also use the **arrow** keys to pan your workspace if you've enabled the Autopan option in the *Options Configure* dialog.

View Zoom In

Zooms in by the magnification factor set in Options Configure. The zoom factor must be greater than 1.

When you select **View » Zoom In**, the cursor takes on the shape of a magnifying glass (the zoom cursor), prompting you to click for the center point of the zoomed area. The cursor position becomes the center of the zoomed-in area. You must re-invoke the command for every zoom action. To cancel the zoom after the zoom cursor appears, click the right mouse button or press **Esc**.

Plus Key (+)

An easier way to zoom in is to use the plus key (+) as a shortcut (the keypad plus key is also functional). Your current cursor location becomes the center of the zoom action when you press the **plus** key; you don't have to click in the workspace.

View Zoom Out

Zooms out by the magnification factor set in Options Configure. This is a temporary mode that does not affect whatever other mode you may be operating in.

When you select **View » Zoom Out**, the cursor takes on the shape of a magnifying glass (the zoom cursor), prompting you to click for the center point of the zoomed area. The cursor position becomes the center of the zoomed-out area. You must re-invoke the command for every zoom action. To cancel the zoom after the zoom cursor appears, click the right mouse button or press **Esc**.

Minus Key (-)

An easier way to zoom out is to use the minus key (-) as a shortcut (the keypad minus key is also functional). Your current cursor location becomes the center of the zoom action when you press the **minus** key; you don't have to click in the workspace.

View Zoom Window

Zooms to an area of the placement interface window that is specified by a zoom window. A zoom window is a rectangle you click and drag to create in the workspace; the window you create will fill the screen, depending on the shape of the window that you draw. This is a temporary mode that does not affect whatever other mode you may be operating in.

Zoom through a Window



There are three ways to invoke the zoom window: the **Z** key, the toolbar zoom button, or the **View » Zoom Window** command from the menu bar. In any case, after it is invoked, you must draw the zoom window in the placement interface window.

1. After you enable the zoom window function, the cursor becomes a magnifying glass shape until you click and drag to create a rectangular shape around a specified view area.
2. Release when you have completed your "window". The area you surround with the zoom window is enlarged on the screen (the window fills the screen).
3. The active tool is still active after you do the zoom window action.

You must drag the cursor to create a zoom window. If you click and release in the workspace without dragging the cursor, the program responds with a beep, and does not zoom in.

To cancel the zoom action once the zoom cursor appears, click the right mouse button or press **Esc**.

View Jump Location

Positions the cursor to a specified location (X, Y coordinates) in the placement interface window. You can also use the **J** key to enter coordinates on the status line.

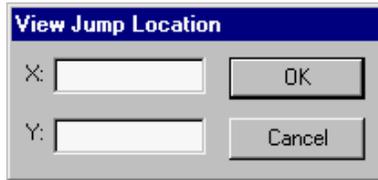
If you are zoomed in, this command pans the workspace to the specified location, attempting to center the location. Your current zoom setting is not changed by the jump location panning (except View Last will be updated). If the specified location is already visible on the screen, no panning is necessary.

The units used for the location value (mil or mm) are determined by the setting in Options Configure. Select **Options » Configure** to override the default settings.

The location is also based on the Options Grids setting, either Absolute or Relative; e.g., if your grid setting is in Relative mode, then the location is a Relative coordinate. Also, you can use negative coordinate values when in Relative mode. Refer to *Options Grids*, (page 110), for more information.

Jump to a Location

1. Choose **Jump Location** from the *View* menu.



2. When the dialog appears, specify the X and Y coordinates (in the text boxes) of where you want your cursor to be in the workspace.
3. Click **OK**.

View Command Toolbar

With this command you show or hide the command toolbar. A check mark next to the toolbar menu item indicates that the toolbar is visible. The toolbar gives you quick access to the most frequently file management and editing commands (such as the Select tool).

The *Command* toolbar for a Schematic design, which consists of only the **Update Schematic** command, is permanent and the display cannot be disabled.

The *Command* toolbar, for designs loaded from PCB, is displayed below:



Disabling the command increases the space within the applicable window.

The setting of the toolbar visibility is saved to your `.ini` file when you exit the program, and restored when you restart it.

View Prompt Line

Allows you to either show or hide the prompt line. A check mark next to the prompt line menu item indicates that the prompt line is visible.

Press and release <Left> or <Space><Space> to select a component.

The prompt line is useful in that it gives instructions on what to do in certain modes. For example, while you are in the Select mode, it tells you what to do next, depending on what action you have already taken.

The current settings for the visibility of the prompt line will be saved to your `.ini` file when you exit the program.

View Status Line

Allows you to either show or hide the status line. A check mark next to the status line menu item indicates that the status line is visible. The status line provides status information. For a design loaded from PCB, it also allows you to change grid spacing and view current placement coordinates, as shown below:



A check mark alongside the command indicates that the status line is visible. Disabling the command increases the space within the applicable window.

The setting of the status line visibility is saved to your `.ini` file when you exit the program, and restored when you restart it.

View Snap to Grid

Enables a snappy cursor, meaning that the cursor can only move from grid point to grid point within the placement interface, as opposed to a free floating cursor. The benefits of a snappy cursor are mainly a question of personal preference (e.g., you may be accustomed to a snappy cursor from other program applications). Also, a snappy cursor can create a predictable point of reference and placement when moving and rotating objects or measuring distances.

At times you may want to disable this command (and cause the cursor to float freely).

The current settings for Snap to Grid (whether it is enabled or disabled) is saved to your `.ini` file when you exit the program.

View Layout View Options Toolbar

Displays or hides the Layout View Toolbar.

View VPA Toolbar

Displays or hides the Visible Placement Toolbar.

View Design Manager

Displays the Design Manager View. In the Design Manager, you can organize design components into logical partitions or physical rooms. These rooms and partitions can be used for grouping unplaced components, as well as placing them in a common board region. For more information, see *The Design Manager View*, (page 37).

Options Commands

Options

Block Selection...
 Configure...
 Grids...
 Display...
 Preferences...
 Design Rules...

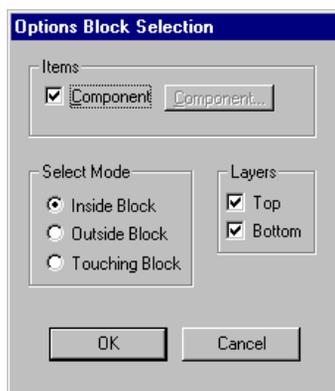
The Options commands allow you to change settings affecting many P-CAD InterPlace actions. For example, you can use the Options commands to determine block selection criteria (selection mask), set default units (mm or mils), specify grid settings, and select object display colors.

The **Options** menu, shown in the left margin, is available when a PCB design has been loaded into InterPlace. When a design is loaded from Schematic into IPL, only the **Options » Configure** command is available.

Options Block Selection

Defines selection filter(s) for block selects (see **Edit » Select**). You can configure the block select feature to select all components by type, or even objects with specific parameters. The selection possibilities can be as specific, limited, or as general as you choose.

Choose **Options » Block Selection** to display the *Options Block Selection* dialog. From this dialog you can set criteria affecting the selection of specific objects and layers.



Items

In the Items area, you can specify individual objects for inclusion in a block select. Enable the specific items you would like to include by clicking the item.

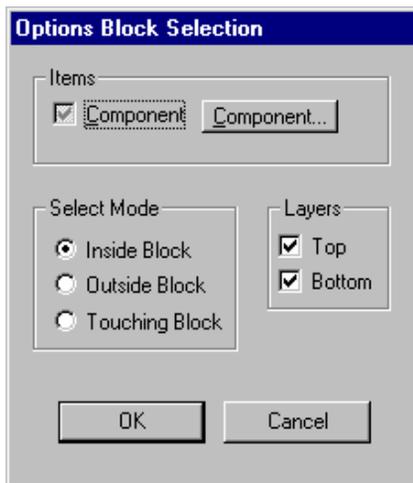
In P-CAD InterPlace, you can block select components.

Items Buttons (Selection Mask Dialog)

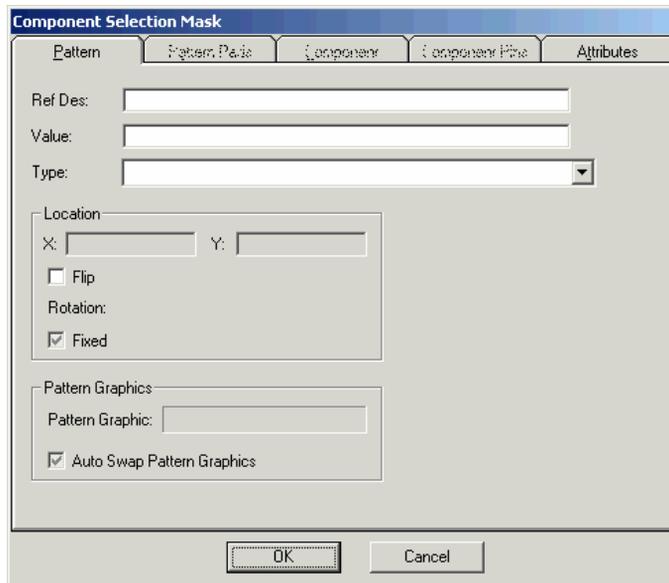
The component selection has a corresponding **Property** button . A **Property** button has three states:

- Included (checked).
- Excluded (blank).
- Masked (grayed).

This feature allows you to narrow your selection further by setting specific properties as selection. To access the property “mask”, set the item check box to its third state, the masked (grayed) state. This allows access to the item button as shown in the following illustration:



Then click the **Component** button to display the respective *Selection Mask* dialog:



In the *Selection Mask* dialog, you can specify the parameters for the components that you want as part of the selection block.

Selection Mask Parameters

You can select components by a variety of parameters, specified in the *Selection Mask* dialog. A summary of these parameters is listed below.

For Component, you can specify component type, reference designator, value, fixed position, and attributes. For example, you can include a specific component of a certain value, and the block select will include only those of that value. Type, reference designator, and value fields support wild card characters: ? to match any single character, and * to match a sequence of zero or more characters. For example, a RefDes value of U? matches all components with a two character RefDes string beginning with U.

Layers

The Layers area of the dialog allows you to select any combination of layers for the selection mask. To choose a layer, check its corresponding check box by clicking it. Checked layers will become part of the selection list.

With **Options » Block Selection** you can select items that are not located on the current layer.

Select Mode Box

Inside Block is the default setting, which means that all items completely inside the selection block will be included, according to the selection criteria you establish in this dialog.

Outside Block allows you to select outside of the block select (outside of the selecting rectangle). All of the selection criteria that you specify in the *Options Block Selection* dialog will function outside the block rather than inside.

Touching Block selects all items inside and touching the selection block. This is a more inclusive selection option than Inside Block.

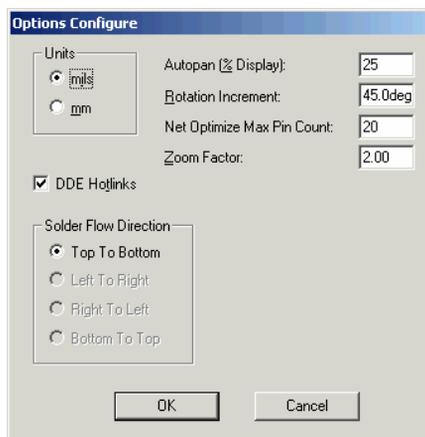
Selecting and Modifying

This is a scenario for selecting all capacitors on the top layer of your board:

1. Choose **Options » Block Selection** to display the dialog.
2. Click on the **Component** check box until the check box becomes grayed. The **Component** button then becomes ungrayed.
3. Click on the **Component** button to display the *Component Selection Mask* dialog.
4. Choose the component type CAP from the drop down Type list, or enter *CAP* in the Type box. The former option chooses only components with the exact name CAP (i.e., CAP300 is not selected). The latter option chooses all components that have the three letters CAP located anywhere in their name. Click **OK**.
5. In the Layers area, click **Bottom Layer** to uncheck its box. The Top Layer box alone is checked to allow selection on only the top layer.
6. In the Select Mode area, click the **Inside Block** radio button to enable it.
7. Do a block select of the complete design by clicking and dragging from one corner of the region to the other. Only the capacitors within the selected region will be selected.

Options Configure

When you select this command, the *Options Configure* dialog appears as shown below. This command is available when a design is loaded from Schematic, PCB or Relay.



From the *Options Configure* dialog, you can set many of the general P-CAD InterPlace parameters. A design loaded from PCB has access to all the options in the *Options Configure* dialog. A Schematic design, which lacks the graphic presentation of the design in the Layout View, has access to only the Units, File Viewer and DDE Hotlinks.

The *Options Configure* dialog contains the following options.

- Units:** You can alter your display units between mils and millimeters with this option. Dimensions are not altered, only the unit of measurement of dimension. A mil equals 0.001 inch or .0254 mm. A mm equals 0.001 meter.

This setting affects all dialogs, reports, Status line displays, etc. containing measurements. For example, setting Units to mm causes all dialogs to display measurements in millimeters. These units can be overridden in many command settings.
- Autopan:** Allows you to adjust the amount of autopanning that occurs when you move the cursor to the edge of the screen with the arrow keys. An autopan of 25 moves anything at the edge of the screen 25% nearer to the center of the screen; 50 moves fringe objects to the center, etc.
- Rotation Increment:** The rotation value you set here is activated when you select an object and use Shift+R to rotate it. Values must be between 0 and 360 degrees, and can be specified down to tenth degree resolution.

You can also use the **R** key to rotate a selected object by 90 degrees; **R** is not affected by what you specify in this dialog.
- Net Optimize Max Pin Count:** Allows you to specify the default pin count for determining whether a net is automatically optimized. If the net has more than this number of nodes, it is not optimized.
- Zoom Factor:** Allows you to adjust the amount of zoom that will occur when **View » Zoom In** or **View » Zoom Out** is enabled. A factor of 2.00 will double (or halve) the size of objects in the workspace, etc. Zoom factors must be greater than 1.00.

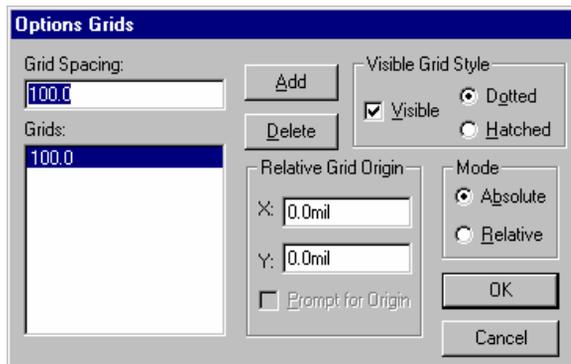
- **DDE Hotlinks:** If checked, this option enables exchange of hotlink data with P-CAD Schematic. Hotlink data consists of highlighting and unhighlighting commands for components and nets. The state of the DDE Hotlinks option is saved in the `.ini` file.

The *Layout View*, (page 21), has detailed information on how to use *Cross-Probing Between Applications*.

Options Grids

Defines the current placement grid.

When you choose **Options » Grids**, the following dialog appears:



The grid units used are determined by the **Units** setting in *Options Configure*.

Mode

You can specify Absolute when you want the grid origin point to be the lower-left corner of the workspace. You can specify Relative to allow any point as an origin point.

A typical setting is to set your Absolute mode to 100 mil grid spacing and Relative to 25 mil grid spacing. In this way, you can toggle the Grid button on the status line between 100 mil and 25 mil. (The relative grid origin should be set to Absolute 0,0 for this to work.)

Visible Grid Style

The Visible enable box allows you to display or not display grids. When you enable this option, the grid style options are available. Dotted displays pinpoint grid points, while Hatched draws lines along the grids to show grid intersections (like graph paper).

Relative Grid Origin

You can specify your X and Y relative grid origins by entering the coordinates if you have disabled the **Prompt for Origin** option.

If you switch to Relative grid in the dialog and the *Prompt for Origin* dialog is enabled, you will be prompted for the new origin point after you click **OK**. You will be prompted every time you change from Absolute to Relative Grids using the **Grid Toggle** button or the **A** key.

Grid Spacing: Uniform/Nonuniform

You can select appropriate values for grid spacing for specific modes in the Grids list box. You are not limited to using the grids in the list box; you can specify your own custom grid spacing in the Grid Spacing edit box, then click **Add** to add it to available choices in the list box. To delete a grid-spacing value, highlight it in the Grids list box and click **Delete**.

The grid defines the spacing pattern of the grid points. The grid pattern may be uniform (equal spaces between all lines) or it may be nonuniform. To specify a nonuniform grid, enter explicitly the multiple spacing values, e.g., 40-20-40.

Remember to set up your grids correctly before you place your components. It is important that you place the components on the appropriate grids (e.g., standard DIP components on 100 mil absolute).

Grid Toggle Button (or G key)



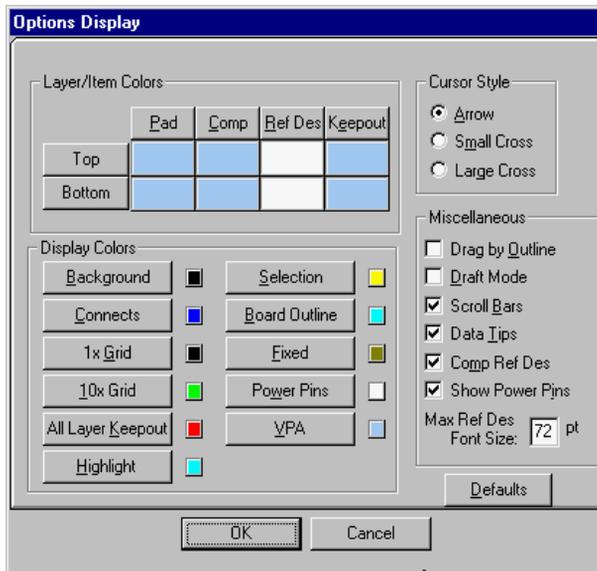
You can easily toggle between absolute and relative grid settings with the **Grid Toggle** button or **G** key. The button appears as  for absolute grid spacing and  for relative grid spacing. Your absolute and relative grids are determined by what you have set in the *Options Grids* dialog. For example, if you have Relative set to 25 mils and Absolute set to 100 mils, the Status line will read 25 mil when you are in Relative grid and 100 mil when you are in Absolute grid.



If you use the grid toggle button or the **G** key to switch to Relative, you will be prompted for the origin point if you have the **Prompt for Origin** option enabled in *Options Grids*. The crosshair cursor (the cursor displayed when the system is awaiting input) is displayed; when you click in the workspace, that becomes the relative origin point ($X=0$, $Y=0$) and the cursor returns to normal.

Options Display

Defines color preferences, cursor style, and other display-related items for P-CAD InterPlace. When you choose **Options » Display**, the *Options Display* dialog appears:



The colors you establish in the dialog are saved in your .ini file.

Each of the options is summarized below.

Layer/Item Colors

This is a color matrix with items across the top (columns) and layers along the side (rows). You can specify everything on a layer to be of one color by clicking on a layer button (e.g., Bottom). By the same process, you can make an object the same color on all layers by clicking an item button (e.g., Pad). Or you can make an object a specific color on a specific layer by clicking on the square where the object and the layer meet. Any of these choices display the *Color* dialog.

Color dialog

When you click a layer or item button, the *Color* dialog appears where you can choose a color for the layer, object, or combination. You can also display the *Color* dialog by right mouse clicking an individual layer or item. In the *Color* dialog, simply click one of the 20 colors and the color is applied.

To close the *Color* dialog without choosing a color, press the **Esc** key.

To select a custom color, click the **Custom** button on the *Color* dialog. Select a custom color from the common *Windows Color* dialog by one of the following methods:

Method 1:

1. Click on one of the predefined colors in the Basic colors section.
2. Click **OK** to select the color.

Method 2:

1. Define the custom color by either:
 - Clicking over the color palette to define the Red/Green/Blue settings. Move the arrowhead at the far right up and down to set the luminescence of the color.
 - Type in the desired Red/Green/Blue settings for the color.
2. Click the **Add to Custom Colors** button. The selected color appears in the Custom colors section.
3. Click **OK** to select the color.

The display of the custom colors in your P-CAD design depends on the color settings of your video display. When the video display is set to 256 colors, these custom colors are approximated by the nearest solid color; the Red/Green/Blue settings for the selected color are retained. When the video display is set to more than 256 colors, the custom colors are accurately displayed.

Once custom colors are selected, they appear as the chosen item or display color on the Options Display dialog or the Printer Colors dialog. These colors are saved in the `.ini` file for use in later design sessions. In the Custom colors section of the common Windows Color dialog, however, colors not selected for an item/display color are forgotten once the P-CAD application is exited.

Display Colors

These buttons determine general display colors, regardless of layer or item colors. You must be careful that your display colors do not conflict with layer or item colors that you specify; for example, if you set background color to be the same as component color, your components will not appear.

These commands also display the *Color* dialog, where you can select your display color.

Each object you select and highlight appears in the current highlight color. You can use highlight colors to highlight objects using different colors. To do so, change the highlight color, select the object, and choose the **Edit » Highlight** command. Repeat this process for each object you want to highlight, so that each object selected has a different highlight color. When you change the highlight color of one object, it does not affect the highlight color of other highlighted objects, which are not selected.

Cursor Style

Allows you to change the shape of your cursor to arrow, small cross, or large cross. The large cross stretches horizontally and vertically to the edges of the screen.

Miscellaneous

The following items appear in the *Miscellaneous* box.

Drag by Outline

Allows you to drag objects around the screen more quickly by only displaying the bounding box of the objects that are being moved. This only affects the move operations of the **Edit » Select** command and the **Cluster By** command.

Draft Mode

Displays a thin (single-pixel) outline of pads and text.

Displays segmented, outlined representations of arcs, lines, and any line segment objects. Draft Mode can help in faster redraws and also allows you to view any segment overlaps.

Draft Mode displays outline forms.

Scroll Bars

Enables or disables the display of Windows scroll bars in the active window.

Data Tips

Enables or disables the display of data tips. Data tips provide a quick view of the properties for components, component pads, and connections.

Comp RefDes

Enables or disables the display of the component reference designators. Generally, the maximum size of the text is determined by the component outline. To restrict the text size when zooming in on a component, enter the maximum point size in the Max RefDes Font Size box.

Show Power Pins

Check the Show Power Pins box to enable the display of those pins designated as power pins in the chosen Power Pins color.

Max RefDes Font Size

Allows you to enter a maximum font size in which the Reference Designator is displayed.

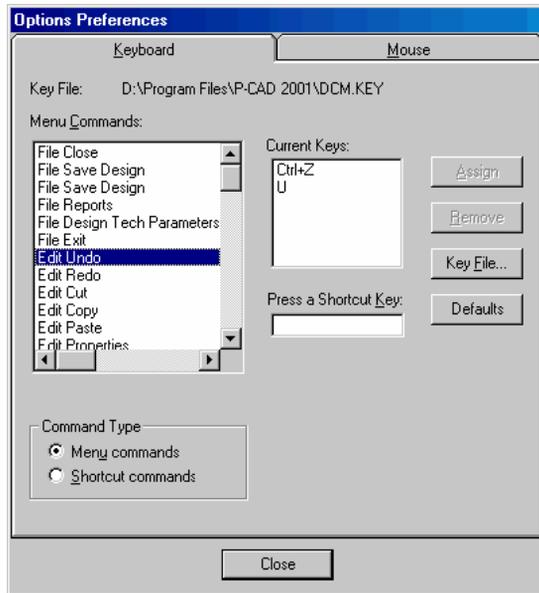
Defaults Button

The **Defaults** button on the *Options Display* dialog returns your display setup to the default scheme.

Options Preferences

Defines keyboard and mouse preferences used to set up the application. The default keyboard shortcuts are listed in Appendix B.

When you choose the **Options » Preferences** command, the dialog appears with the *Keyboard* tab selected:



Keyboard Tab

The **Keyboard** tab lets you customize key assignments for menu commands and shortcut key commands.

- **Command Type:** Choose the type of command for which you want to change shortcut key assignments.
- **Menu Commands/Shortcut Commands:** Select the command you want to add a shortcut key assignment to or from which you want to remove a shortcut key assignment.
- **Current Keys:** Displays the existing key assignments for the command you select in the Menu Commands/Shortcut Commands box.
- **Press a Shortcut Key:** Press the keys you want to assign to the selected command. You can press the **Ctrl** or **Shift** key plus any other combination of numeric or alphabetic keys and function keys.

F10 is reserved by Windows and cannot be assigned to any other command.

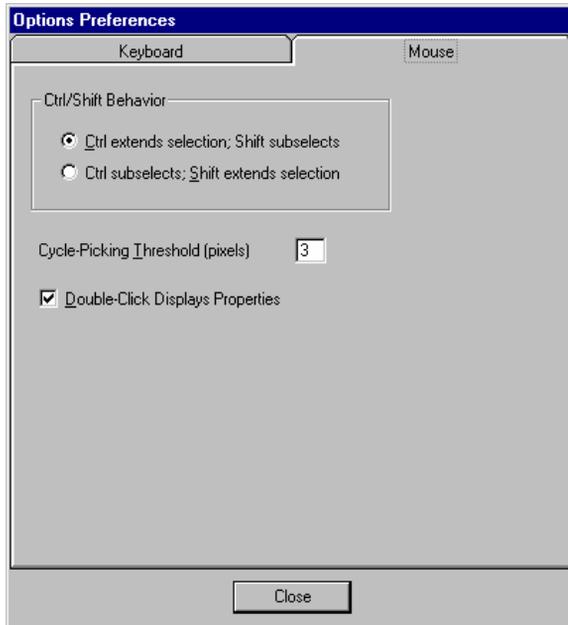
If the shortcut is currently assigned, the current assignment appears in the Current Binding field just below this box.

- **Assign:** Assigns the key appearing in the Press a Shortcut Key box to the selected command. If the shortcut is currently assigned, the current assignment disappears
- **Remove:** Removes the key you select in the Current Keys box.

- **Key File:** Allows you to select or create a key binding file to use with this application. When the *Select Key File* dialog appears, select the file you want to use. The current key file appears at the top of the dialog, and is written to the application `.ini` file.
- **Defaults:** Restores original default shortcut key assignments to all commands.

Mouse Tab

When you select the **Mouse** tab, the *Options Preferences* dialog appears as follows:



The **Mouse** tab lets you customize certain mouse behaviors.

- **Ctrl/Shift Behavior:** Allows you to choose which keys to use (**Ctrl** or **Shift**) for multiple selections.
- **Cycle-Picking Threshold:** The number of pixels you can move the mouse during cycle-picking.
- **Double-Click Displays Properties:** When enabled, this option allows you to double-click a component to bring up its Properties dialog.

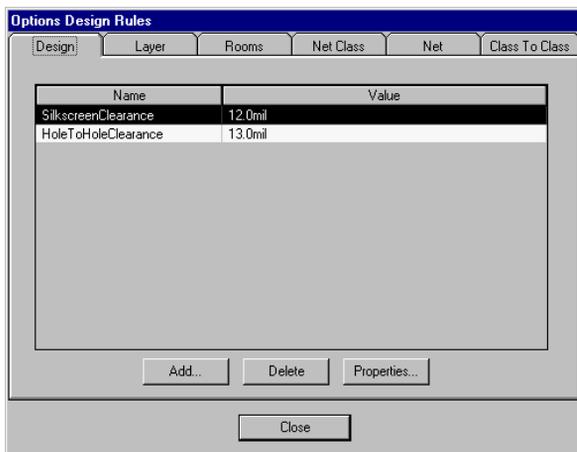
Options Design Rules

When you choose the **Options » Design Rules** command, the *Design Rules* dialog appears with the **Design** tab selected with values for Silkscreen and Hole to Hole Clearance assigned.

If you have purchased P-CAD Parametric Constraint Solver, the *Options Design Rules* dialog is unavailable because PCS provides extended access to the design rules.

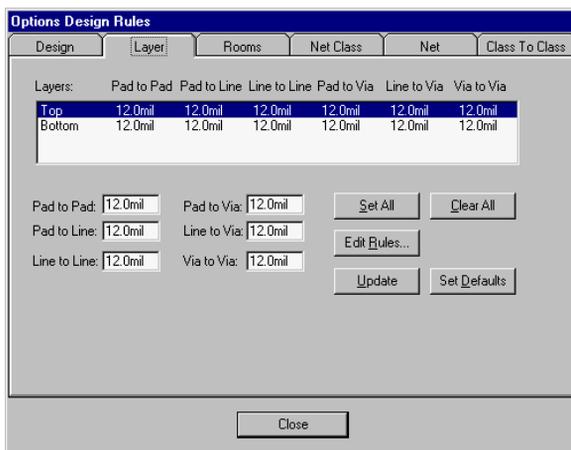
Design Tab

The **Design** tab shows global default clearance rules which apply to the entire design.



Layer Tab

The **Layer** tab of the *Options Design Rules* dialog contains the default clearance values for signal layers and displays them as shown below:



Layers are ordered according to the layer order set with the **Options » Layers** command in PCB

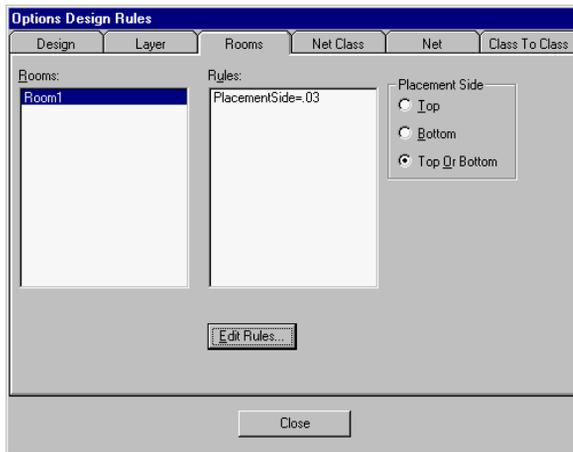
The clearance values of the layer you select appear in the Pad to Pad, Pad to Line, Line to Line, Pad to Via, Line to Via, and Via to Via boxes. If you have a variety of settings and you click on two layers that contain conflicting values, the box is blank. The value you enter in the box is applied to the selected layer(s) when you click the **Update** button.

Clearance Rules

When a clearance rule for a specific object is requested (e.g., DRC), the design rules category determines which rules apply and the order in which they are applied. *Options Commands*, in your *PCB User's Guide*, has more information on the way clearance rules are handled by DRC.

Rooms Tab

Clicking the **Rooms** tab displays all of the Rooms in the design as seen below:

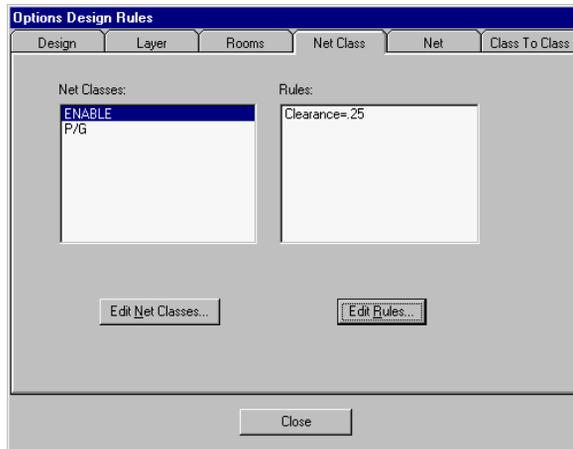


Select a **Room** from the Rooms list box and its associated Rules are displayed along with its Placement Side. If you want to add, delete or edit the rules click the **Edit Rules** button and make the appropriate changes in the *Attribute* dialog.

You can edit a rule's properties by double clicking the rule in the Rules list box.

Net Class Tab

When you click the **Net Class** tab, the dialog appears as follows:



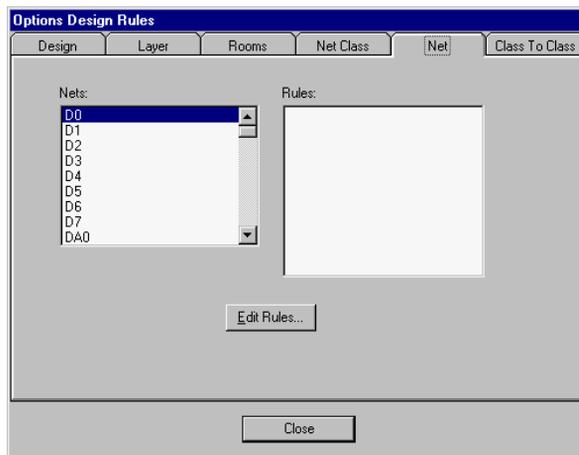
The **Net Class** tab lists all net classes and the rules associated with each.

To edit a rule's Properties, double-click the rule in the Rules list box.

To add, delete or edit rules, click the **Edit Rules** button to access the *Attributes* dialog. Refer to the *Edit Nets* command section in *Options Commands*, in your *PCB User's Guide* for more information.

Net Tab

When you click the **Net** tab, the dialog appears as follows:



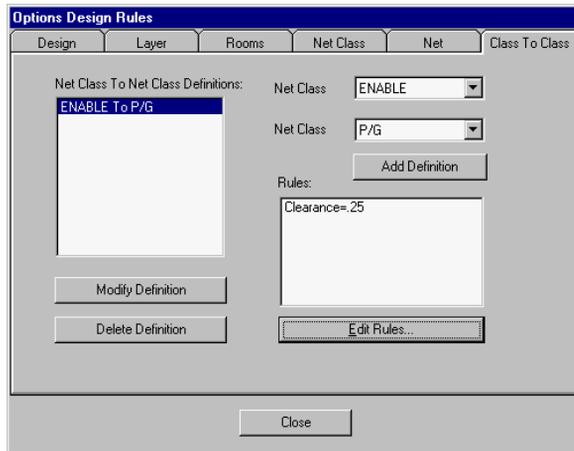
The **Net** tab allows you to specify clearance rules for a specific net in the design. The dialog lists all nets and shows the rules associated with the net you select from the *Nets* list box.

To edit a rule's Properties, double-click the rule in the Rules list box.

To add, delete or edit rules, click the **Edit Rules** button to access the *Attributes* dialog. Refer to the *Edit Nets* command section in *Options Commands*, in your *PCB User's Guide* for more information.

Class to Class Tab

When you click the **Class to Class** tab, the dialog appears as follows:



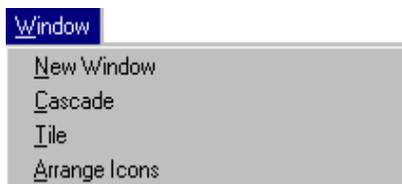
The **Class To Class** tab allows you to specify clearance rules between nets grouped into a Net Class. The dialog lists all class to class definitions and the rules associated with each.

To create a Class to Class definition, select nets classes from the two Net Class Name list boxes, and click **Add Definition**. Once you have created a definition, it can be deleted using the **Delete Definition** button and modified using the **Modify Definition** button.

To edit a rule's Properties, double-click the rule in the Rules list box.

To add, delete or edit rules, click the **Edit Rules** button to access the *Attributes* dialog. Refer to the *Edit Nets* command section in *Options Commands*, in your *PCB User's Guide* for more information.

Window Commands



Window commands allow you to manage multiple Layout View windows.

P-CAD InterPlace allows you to open multiple windows on the same design to view different parts of it at once. Because the Layout View window is not available for designs loaded from Schematic, the window commands are relevant only to designs loaded from PCB.

The window you are working in is the active window. The work that you perform in the active window is reflected in all windows of the design.

Window New Window

Allows you to open additional Layout View windows for the design. You can move independently in each window, making it easy to compare different parts of the same design. For more information, refer to *Using Multiple Windows in The Layout View*, (page 21).

A number identifying the window is added to the file name in the Title bar and at the bottom of the **Window** menu. For example: C:\InstallationDirectory\TUTORIAL\TUTOR.PCB:2.

Window Cascade

Arranges all open Layout view windows so that the window titles are visible.

All windows overlap, starting in the upper-left corner of your workspace. You can see each window's title, making it easy to switch between windows.

Window Tile

Arranges all open Layout View windows so that all windows are visible.

Windows are resized and arranged side-by-side so that all windows are visible and none overlap.

Window Arrange Icons

Arranges Layout view window icons (minimized windows) in the main application window. This command arranges these icons so that they are evenly spaced and don't overlap.

Windows can be minimized into icons by clicking **Minimize** button, the left-most of the three buttons in the upper-right corner of the window.

You can open one of these window icons by double clicking it.

Selecting a Window

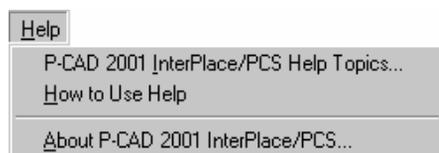
The bottom of the **Window** menu lists all open windows. To make a window active select it from this list. Design windows appear on the **Windows** menu in the order that you opened them.

You can also switch between windows using the window tabs at the bottom left of the Layout View window.

If there are more than nine windows opened, an option **More Windows...** appears. Select it to see additional windows.

To close a window, double click the control button in the upper-left corner of the window. Closing the last window is equivalent to the **File » Close** command.

Help Commands



P-CAD InterPlace provides extensive online help. The online help contains reference information, but differs from the manual in its format and accessibility.

The main advantages to online help are the availability of hypertext links between related subjects (an electronic cross-reference utility), and the keyword search function (an electronic index).

Help P-CAD InterPlace Help Topics

Displays the P-CAD InterPlace online help including the **Contents** tab, which is structured to match the order of commands as they appear in the product, and the **Index** tab, which lets you look up a specific concept or keyword.

Help How to Use Help

Connects you to the Windows help system, where instructions on how to use the help system are provided.

Help About P-CAD InterPlace

Displays a dialog that contains information such as the product version number, release date, memory used, memory available, and license number.

Keyboard Reference

This chapter is a reference of commands and functions accessed through P-CAD shortcut keys and standard Windows accelerators.

Standard Windows key combinations are functional for all of the menu commands; use the normal combination **Alt,x,y**, where x equals the underlined menu character, and y equals the underlined command character.

The InterPlace Keyboard Reference section includes general keystrokes available with InterPlace. The InterPlace Keyboard Reference section includes keystrokes specific to P-CAD InterPlace.

You can use the Options Preference dialog to change shortcut keys for commands.

InterPlace Keyboard Reference

Alt+F4 (File Exit)	Shortcut for File » Exit , which exits the P-CAD program. If the current design has been modified since the last save, you will be prompted (YES or NO) as to whether you want to save the changes to the file. The program will write information to the .ini file when you exit.
Alt + mouse click	For any click-and-drag or drag and drop operations, you can hold down the Alt key, click the left mouse button, then move or drag the object wherever you want without having to keep the mouse button depressed. Without the Alt key, you would normally have to click and drag with the button depressed while you are dragging.
arrow keys	A directional arrow key moves the cursor to the next grid point. If Ctrl+arrow , then the cursor is moved 10 grid points, which can be used to pan the screen.
Ctrl+S (File Update PCB)	Saves changes to the current design without closing it.
Ctrl+Z (Edit Undo)	A shortcut for Edit » Undo , which reverses a completed action.
Ctrl+F4 (File Close)	Closes the current design, but does not exit the InterPlace application.
Ctrl+F6 and Ctrl+Tab	Switches focus to the next window. Use Ctrl+Shift+F6 or (Ctrl+Shift+Tab) to switch focus to the previous window.
Shift+F (Edit Fix Components)	Fixes selected components in the workspace. Fixed components cannot be moved, rotated, or flipped.
Shift+U (Edit Unfix Components)	Unfixes selected components in the workspace. Unfixed components can be moved, rotated, or flipped.
Shift+End (Select to end of list)	Selects all items from the first selected item to the end of the list.
Shift+F4 (Window Tile)	Shortcut for the Windows Tile command. Windows are resized and arranged side-by-side so that all windows are visible and none overlap.
Shift+F5 (Window Cascade)	Shortcut for the Windows Cascade command. All windows overlap, starting in the upper-left corner of your Workspace. You can see each window's title, making it easy to switch between windows.
F1 (Help)	Displays context-sensitive help. If you put focus on a command or dialog and press F1 , the Help window appears containing information specific to the focus item.

Keyboard Reference

Page Down	Scrolls one page down in the workspace. Ctrl+Page Down scrolls one page right.
Page Up	Scrolls one page up in the workspace. Ctrl+Page Up scrolls one page left.
SPACEBAR	The SPACEBAR can be used in place of the left mouse button; but the action is different. To simulate a typical click and release of the mouse button, you need to press and release the SPACEBAR twice. Therefore, to simulate the click and hold mouse action, you press and release the SPACEBAR once. As the left mouse button is used in such a variety of ways throughout the P-CAD program, this SPACEBAR keystroke can become a regular part of your work.
Esc (Escape)	Terminates placement of objects with multiple segments; it also cancels a redraw in progress. It is often equivalent to the right mouse button. Esc also exits from dialogs (equaling the Close or Cancel button).
+ Plus Key (View Zoom In)	Shortcut for View » Zoom In command. The keypad plus key also works. The plus key causes a zoom in to occur at the cursor location. Refer to the View » Zoom In command documentation for more information. The plus key does not change the cursor to a zoom cursor (as do the zoom commands from the <i>View</i> menu).
- Minus Key (View Zoom Out)	Shortcut for View » Zoom Out command. The keypad minus key also works. The minus key causes a zoom out to occur at the cursor location. The minus key does not change the cursor to a zoom cursor (as do the zoom commands from the <i>View</i> menu).
A key (Grid Toggle)	Toggles between absolute and relative grid settings.
C Key (View Center)	Shortcut for View » Center command. This command allows you to center your cursor location. Place the cursor in the area of your design that you want centered and press C . Repeated uses of the C key can be used to pan across the workspace.
F Key (Flip Object)	Flips an object during place and move operations. Not all objects can be flipped. Refer to the Edit » Select command (in this Reference manual) for more information about flipping objects.
G Key (Grid Select)	This key scrolls forward through the list of grid settings. Use Shift+G to scroll back through the list.
J Key (Enter Coordinate)	Gives focus to the X coordinate box in the status line. From there, you can enter new X coordinate. Then, click Tab to give focus to the Y coordinate and enter its new value.

Keyboard Reference

L and Shift+L Keys (Change Layer)	L and Shift+L cycle through the board layers, toggling between making the Top or Bottom layer current.
R and Shift+R keys (Rotate)	R Rotates objects 90 degrees during Place and Select operations. Shift+R rotates objects by the rotation angle specified in the <i>Options Configure</i> dialog. For more information about rotating objects with Select operations refer to the Edit » Select command.
S Key (Select)	Shortcut for the Edit » Select command (also the toolbar Select button). When enabled, you are able to click on objects to highlight them for selection, then move, rotate, and otherwise modify them.
U Key (Undo)	Shortcut for the Edit » Undo command, which reverse a completed action.
X Key (Cursor Style)	Toggles between the three cursor styles: Arrow, Small Cross, and Large Cross.
Z Key (Zoom Window)	Shortcut for the View » Zoom Window command. Just press Z and then draw the zoom window; a zoom cursor (magnifying glass) will display until you draw the zoom window. Whatever you surround with the zoom window will fill the screen.

-A-

absolute
 degree rotation29
 grid toggle12, 111
 align to grid91
 aligning components31, 68, 91
 All, View command100
 alteringsee editing or modifying
 autpanning, adjusting..... 109

-B-

block selection
 select mode box107
 Block Selection, Options105
 board
 layers25
 bounding rectangle.....77
 buttons
 command toolbar103
 grid toggle12, 111
 visible placement area.....26
 VPA toolbar75

-C-

cascade.....121
 Center, View command.....101
 changingsee editing or modifying
 Class to Class clearances.....120
 class-to-class49
 creating new50
 deleting.....50
 report84
 clearance rules118
 closing a design.....17, 82
 Cluster By, Edit command.....93
 clustering components60

clustering options40, 61
 colors
 custom.....112
 display113
 display options111
 display settings.....26
 layer/item112
 command toolbar103
 Command Toolbar, View command.....103
 common commands.....39
 components23, 27
 adding to partitions.....46
 adding to rooms.....47
 align to grid91
 aligning.....31, 68
 cluster options.....27
 clustering27, 40, 60, 93
 data tips.....31
 definition of fixed29, 66
 Design Manager page.....42
 discrete placement30, 63
 drag and drop30, 62
 drag from design manager30
 dragging from Design Manager40
 finding60
 fixing29, 67
 flipping.....30, 67
 gathering.....60
 in partitions45
 information in Design Manager42
 jump location.....28
 jump to41, 63
 locating.....27, 28
 logical partitions44
 move by refdes.....62
 moving28, 30, 96
 moving across windows30

organizing.....37, 55
 pad
 data tip31
 place on (layer).....29
 placement fine-tuning.....66
 placing.....28, 55, 60, 63, 92
 placing between windows.....24, 65
 placing with VPA.....65
 properties.....31, 32, 89
 reference designator114
 removing from partition.....46
 removing from rooms.....47
 rotating.....29, 67, 92
 select command.....63
 selecting in Layout View40
 selection criteria107
 selection point.....29
 sorting in Design Manager43
 swapping.....31, 68
 unfixing.....29, 67
 unhighlighting.....41
 unpartitioned.....44
 Configure, Options command.....108
 connections.....32
 data tips.....32
 hiding44
 jump to a.....33
 optimize by pin count33
 optimizing33
 properties.....34
 show on drag44
 showing.....44
 constraint domain report.....84
 constraint domains.....76
 controlling board layers25
 coordinates, X and Y.....12
 copying..... see also pasting
 Cross Probing28, 70
 current layer.....25
 cursor
 free floating.....104
 snappy104
 cursor style113
 cursor,tracker.....12
 cut away boards.....23
 cutting..... see also deleting or pasting

-D-

data tips 13, 31, 32, 63
 displaying..... 26, 114
 DBX
 connection
 re-establishing..... 73
 Database Exchange Interface 16
 lock..... 16
 DBX errors..... 73
 DDE Hotlinks.....28, 35, 70, 94, 110
 deleting
 items in a block..... 108
 Deselect All, Edit command 93
 deselecting objects 96
 design
 clearances 117
 components
 organizing..... 37
 constraints
 domain domain 77
 PlacementGrid..... 110
 viewing 77
 visualization 77
 loading..... 69
 updates..... 72
 Design Constraint Manager
 InterPlace 69
 Parametric Constraint Solver..... 69
 Design Manager View 14, 104
 accessing..... 38
 Class-to-Class Page 49
 collapsing groups..... 39
 commands 51
 common commands..... 39
 Components Page..... 42
 expanding groups..... 39
 icons..... 38
 Net Class Page 47
 Nets Page 43
 Partitions Page 44
 Rooms Page 46
 viewing 37
 Design Rules, Options 116
 Design Technology Parameters 70
 available operations in IPL 71
 Merge Attributes 72
 Replace Design Attributes 72

Index

Design Technology Parameters, File		
command	85	
discrete components		
placement		
prompt line	64	
status line	64	
placing.....	30	
Discrete Placement, Edit command.....	97	
Display		
board outline	26	
data tips	26	
font size for refdes	26	
Options.....	26	
power pins	26	
settings.....	26	
Visible Placement Area.....	26	
VPA options	76	
Display, Options command.....	111	
domain		
design constraints	77	
domain domain.....	77	
domains		
viewing.....	77	
draft mode.....	114	
drag and drop	62, 96	
drag by outline	113	
DTP		
accessing a file	71	
-E-		
ECO		
recording changes.....	72	
ECO Files		
.ECO file.....	72	
.WAS file	72	
Edit Commands	87	
Align Components	91	
Cluster By	93	
Deselect All	93	
Discrete placement	97	
Fix Components.....	92	
Highlight.....	93	
Measure	94	
Move By RefDes.....	90	
Place on	92	
Properties	88	
Rotate To	92	
Select	95	
Select All	93	
Swap Components	92	
Undo.....	88	
Unfix Components	92	
unhighlight.....	94	
Unhighlight All	94	
editing.....	see also modifying	
items in a block	108	
electrical constraints	79	
MaxNetLength	79	
enabling layers.....	25	
Exit File command.....	85	
exiting InterPlace	17	
Extent, View command	100	
-F-		
features.....	1	
File Commands		
Close	82	
Design Technology Parameters.....	85	
Exit	85	
Load PCB Design.....	81	
Load Schematic Design	82	
Reports	83	
Update PCB	82	
Update Schematic.....	82	
Fix Components, Edit command	92	
fixed component		
definition	29, 66	
flip object key	127	
flipping	96	
components	67	
-G-		
gathering components	60	
drag and drop	62	
gathering tools.....	65	
grid		
nonuniform	111	
origin	110	
snap cursor to.....	104	
spacing	111	
style		
dotted.....	110	
hatched	110	
visible	110	
toggle		

(absolute and relative settings)	12
(absolute and relative settings)	111
uniform	111
Grid Toggle button	111
Grids	
Options	110

-H-

hardware requirements	5
Help Commands	123
About P-CAD IPL	123
How to Use Help	123
P-CAD IPL Help Topics	123
hide connections	33, 44
highlight	40
connections	33
removing	94
Highlight, Edit command	93
horizontal alignment	91
hotlinks, enabling	110
how to use help	123
hypertext links	123

-I-

icons	
arrange window icons	122
command toolbar	103
Design Manager View	38
toolbar	9
imperial units	109
importing	see also loading
INI file	17
Installation and Setup	
installing P-CAD products	6
system requirements	5
inter-application	
probing	70
inter-application functions	69
probing	28
InterPlace	
interface	8
using from Relay	7
using from Schematic	8
InterPlace	
basics	7
exit	17
launching	69
Layout View	21

placement visualization	77
using from PCB	7
items	
selecting	39

-J-

jump	
to component	41, 63
to location	28, 103
to pin	41
Jump Location, View command	102

-K-

keepouts	23
keyboard	
preferences	115
reference	126
keys, shortcut	126

-L-

Last, View command	100
launching	
Design Constraint Manager	69
InterPlace	15
layer selection mask	107
layers	25
layers selection criteria	107
laying out the board	21
Layout View	14
Board Control	10
display characteristics	23
multiple windows	24
options toolbar	10
Options Toolbar	104
switching between windows	14
viewing	21
Visible Placement Area	11
loading	see also opening
a board outline	16
a design	15
a PCB design	81
a Schematic design	82
board outline	23
components	23, 24
cut away boards	23
designs	22
free pads	23
grids	24

Index

keepouts.....23
layers24
mounting holes.....23
rooms24
locating components27, 28
location, jumping to.....102
lock
 application16, 70
 terminating73

-M-

max refdes font size114
Measure, Edit command94
menu bar9
metric units109
mode
 changing absolute and relative grid....110
 draft114
modify shortcut.....97
modifying see also editing
mouse preferences116
Move By RefDes62
moving
 components.....28, 30
 items in a block.....108

-N-

net
 class47
 adding nets to49
 creating new48
 deleting.....49
 renaming49
 report84
 classes118
 clearances119
 connections32
 optimization109
 page43
 removing from Net Classes.....49
 unhighlighting.....41
new window.....121
nonuniform grid spacing111

-O-

object selection mask106
objects
 highlighting.....40

 placing.....60
 opening see also loading
 optimization
 setting pin count.....109
 optimizing connections.....33
Options Commands
 Block Selection.....105
 Configure.....108
 Design Rules116
 Display.....111
 Defaults.....114
 Miscellaneous113
 show power pins114
 Grids110
 Preferences114
Options display
 max refdes font size114
organizing
 in Class-to-Class Page60
 in Components Page.....57
 in Net Class Page.....60
 in Nets Page57
 in Partitions Page57
 in Rooms Page59
 objects55
outline, drag by.....113

-P-

PackageOutlineLayer77
panning (View Center command).....101
panning screen, adjusting.....109
partitions
 adding components46, 58
 adding new.....45
 clustering components.....58
 deleting45
 page44
 removing components46, 58
 renaming.....46
 report.....84
P-CAD InterPlace
 features1
P-CAD IPL Help Topics, Help123
P-CAD PCB Icon3
physical constraints78
 BoardEdgeClearance.....77, 79
 Clearance.....79

Index

ComponentSpacing77, 78
PadToPadClearance77, 78
pins
 jump to.....41
Place On, Edit command.....92
placement
 domains76
 fine-tuning.....66
 interface.....21
 visible placement area.....26
 Visible Placement Area.....65
PlacementGrid constraint110
placing..... see also pasting
 components.....28, 63
 between windows24, 65
 discrete components63, 97
pop-up commands34, 51
pop-up menu97
power pins
 display settings26
 displaying114
precedence level report.....84
Preferences, Options command114
previous view.....100
prompt line.....11, 64
Prompt Line, View command.....103
properties31
 component.....89
 connections32, 34
 data tips.....88
 double-click item.....88
 right mouse menu88
Properties, Edit command.....88

-R-

redraw
 interrupting.....100
 speed114
Redraw, View command.....100
reference designator
 displaying114
 font size114
 font size display26
relative grid toggle.....12, 111
removing..... see deleting
reports
 class-to-class84

 constraint domain..... 84
 file extensions..... 83
 generation 84
 lines per page..... 84
 net class..... 84
 options 84
 output options..... 84
 page format 84
 partitions..... 84
 precedence level..... 84
 style format..... 84
 types..... 84
Reports, File command 83
right mouse commands 17, 34, 51
right-click to modify 97
room constraints 79
 MaxComponentHeight..... 79
 Room Inclusion 79
rooms 23, 46, 118
 adding components 47, 59
 clustering components..... 60
 creating..... 59
 data tip..... 31
 organizing components 59
 placing in PCB..... 46
 removing components 47, 59
rotating 96
 components 29, 67, 92
 items in a block..... 108
 rotation increment..... 109

-S-

saving
 a design 17
saving a design..... 82
scroll bars
 displaying..... 114
Select actions 95
 drag and drop 96
 moving..... 96
 overlapping objects..... 96
 rotating and flipping..... 96
Select All, Edit command 93
Select, Edit command..... 95
selecting
 block select mask..... 96
 blocks..... 96

Index

data tips.....	13
menu bar	9
placement.....	21
prompt line	11
status line.....	11
using	
IPL	
from PCB or Schematic	7
using multiple windows	24, 65
using the ECO recorder.....	72
Utils	
Record ECO	73

-V-

vertical alignment	91
View Commands	
All	100
Center	101
Command Toolbar	103
Design Manager.....	104
Extent.....	100
Jump Location	102
Last	100
Layout View Options Toolbar	104
Prompt Line.....	103
Redraw	100
Snap to Grid.....	104
Status Line	104
VPA Toolbar	104
Zoom In	101
Zoom Out.....	101
Zoom Window.....	102
viewing	
connections	32
properties	97
viewing constraint domains	77
views	
Design Manager.....	14
Layout View.....	14
visible design objects.....	23
visible placement area	77

Visible Placement Area	65, 75
connections.....	32
controls	26
display color	26
display options	76
domains.....	11, 75, 76
electrical.....	76
physical	76
room	76
electrical constraints	79
physical constraints.....	78
room constraints	79
toolbar	75
VPA	
Toolbar Display.....	104

-W-

Window	
New Window	121
Window Commands	
Arrange Icons.....	122
Cascade	121
Tile.....	121
Window, View command	102
windows	14
switching between.....	14
tabs	25
using multiple	24, 65
working with PCB.....	69
working with Schematic	69

-X-

X and Y distance, measuring.....	94
----------------------------------	----

-Z-

Zoom	
factor.....	109
In, View command	101
Out, View command	101
shortcut keys.....	127
through a window.....	102