



# 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](http://www.altium.com)

# Table of Contents

## chapter 1 Introducing P-CAD Schematic

P-CAD Schematic Features .....	1
About the User's Guide .....	2

## chapter 2 Installation and Setup

System Requirements .....	3
Recommended System .....	3
Minimum System .....	3
Installing P-CAD Products .....	4

## chapter 3 P-CAD Schematic Basics

About the User Interface .....	5
P-CAD Schematic Window .....	5
Menu Bar .....	6
Toolbars .....	7
P-CAD Schematic Workspace .....	10
Prompt and Status Lines .....	11
Other Window Elements .....	13
Shortcut Menu Commands .....	13
Configuration Options .....	15
Showing or Hiding DataTips .....	15
Selecting a Default File Viewer .....	16
Setting the Current Zoom Factor .....	17
Setting the Autopan Percent Display .....	17
Enabling Orthogonal Modes .....	17
Specifying a Net Increment Value .....	17
Choosing an ECO Format .....	17
Enabling the AutoSave Feature .....	17
Compressing Binary Designs .....	18
Enabling DDE Hotlinks .....	18
Using a Snappy Cursor .....	18
Schematic Sheets .....	19
Adding a Sheet .....	19
Reordering Sheets .....	19
Defining the Current Sheet .....	19
Renaming a Sheet .....	20

Deleting a Sheet.....	20
Tracking Nets Across Sheets .....	20
Opening Design Files .....	21
Opening Schematic Files (*.sch) .....	21
Opening Multisheet Designs.....	21
Opening a Tango Schematic Series II File .....	21
Opening an ASCII File.....	22

## **chapter 4 Schematic Design Tutorials**

Tutorial 1 - Setting Up a Schematic Design .....	23
Setting the Workspace Size .....	23
Setting Up a Title Sheet.....	24
Choosing a Unit Measurement Scale .....	28
Defining Color Preferences .....	28
Setting Up Miscellaneous Display Options .....	29
Adding a Sheet.....	29
Setting Up the Grid Spacing .....	30
Zooming In and Out.....	31
Tutorial 2 - Working with Schematic Objects .....	32
Object Placement .....	32
Selecting Objects.....	46
Modifying Objects.....	50
Working with Your Design .....	58
Tutorial 3 - Verifying a Schematic Design.....	63
Design Verification.....	63
Setting up the ERC Options.....	63
Viewing any Errors in the Design.....	65
Fixing Errors in the Design .....	66
Tutorial 4 - Generating Reports and Netlists.....	69
Generating a Report.....	69
Generating Netlists .....	70
Tutorial 5 - Printing a Schematic Design.....	72
Setting Up a Printer or Plotter.....	72
Setting Up Page Setup Options.....	73
Setting Up Print Options .....	74
Previewing a Print Job.....	75
Generating Printouts and Printing Sheets .....	76

## **chapter 5 Working with Objects**

Placing Objects.....	78
Placing an Object .....	78
Selecting Objects.....	78
Selecting an Object .....	78
Selecting Multiple Objects .....	79
Selecting Part of an Object.....	79
Selecting Collocated Objects.....	80
Selecting All Objects.....	80

Selecting Highlighted Objects .....	80
Selecting a Net .....	81
Selecting Contiguous Net Objects .....	81
Block Selecting Objects .....	81
Defining Object Selection Preferences .....	81
Setting the CTRL/SHIFT Behavior .....	81
Specifying a Selection Point .....	82
Choosing a Selection Color .....	82
Block Selecting a Group of Objects .....	82
Block Selecting Objects .....	83
Defining Block Selection Criteria .....	83
Moving Objects .....	83
Moving an Object .....	84
Moving Objects During Placement .....	84
Moving Parts by RefDes .....	84
Rotating and Flipping Objects .....	86
Rotating Objects During Placement .....	86
Rotating Objects in a Design .....	86
Flipping Objects .....	87
Aligning Objects .....	87
Aligning Parts Horizontally or Vertically .....	87
Aligning Parts To Grid .....	87
Using Orthogonal Modes .....	88
Enabling Orthogonal Modes .....	88
Switching Between Orthogonal Modes .....	89
Switching Between Mode Pairs .....	89
Unwinding Segments .....	89
Resizing an Object .....	89
Cutting and Copying Objects .....	89
Cutting Objects from a Design .....	90
Cutting Objects From Nets .....	90
Copying Objects to the Clipboard .....	91
Copying Objects to a File .....	91
Copying Objects within the Same Design .....	91
Copying Nets .....	91
Copying a Matrix .....	92
Pasting Objects .....	92
Pasting Clipboard Objects .....	93
Pasting From a File .....	93
Splitting a Net .....	93
Object Properties .....	94
Opening a Properties Dialog .....	94
Editing Nets .....	94
Deleting Objects from Nets .....	94
Managing Net Connections .....	95
Using the Edit Nets Dialog .....	95

## chapter 6 Documentation Tools

Title Sheets.....	98
Design Borders.....	98
Zones .....	99
Title Blocks .....	99
Using the Global Title Sheet.....	100
Using a Standard Title Sheet.....	100
Creating a Custom Title Sheet.....	101
Using a Custom Title Sheet.....	102
Revision Blocks .....	103
Fields and Field Sets .....	103
Adding a Custom Field .....	104
Placing Fields in a Design .....	105
Changing Field Results .....	105
Adding a Field Set .....	106
Assigning Field Sets to Schematic Sheets .....	107
Sheet Connector Cross Referencing .....	107
Sheet Connector Overview.....	107
Annotating Sheet Connectors.....	108
Viewing Sheet Connector Properties.....	110
Jumping to Sheet Connectors .....	110
Net Index Tables.....	111
Placing a Net Index Table .....	111
Note Tables .....	112
Adding Notes to a Note Table.....	112
Importing Text Files as Notes .....	112
Exporting Notes to a Text File .....	112
Placing a Notes Table .....	113
Power and Ground Tables.....	113
Placing a Power Table.....	114
Revision Note Tables.....	114
Adding Revision Notes to a Table .....	115
Importing Text Files as Revision Notes .....	115
Exporting Revision Notes to a Text File.....	115
Placing a Revision Note Table.....	116
Spare Gate Tables.....	116
Placing a Spare Gate Table .....	117
Working with Tables .....	117
Modifying Tables .....	117
Updating Tables .....	118
Reporting on Schematic Designs .....	118
Reports Available in P-CAD Schematic.....	118
Attributes Report.....	118
Bill of Materials Report .....	119
Global Nets Report.....	119
Last Used RefDes Report.....	119
Library Contents Report .....	119
Parts Location Report.....	119

Parts Usage Report.....	119
Generating Reports .....	119
Customizing a Standard Report .....	119
Creating a Custom Report.....	120
Setting Format Options .....	121
Setting Data Selection Options .....	121
Setting Sort Options .....	122
Selecting Report Criteria .....	122
Generating a Report.....	123
Generating a Netlist.....	124

## **chapter 7 DDE Hotlinks**

Setting up the DDE Hotlinks Feature.....	127
About DDE Hotlinks.....	127
Enabling DDE Hotlinks.....	127
Selecting the Current Highlight Color .....	128
Using DDE Hotlinks .....	129
Highlighting Parts .....	129
Unhighlighting Parts .....	129
Highlighting and Unhighlighting Attached Nets .....	130
Highlighting Nets .....	130
Selecting All Highlighted Objects in a Design.....	131

## **chapter 8 Verifying a Schematic Design**

Configuring the ERC Feature .....	133
Setting up Checks and Severity Levels .....	133
Using ERC Error Annotation.....	135
Finding ERC Errors .....	135
Overriding Error Displays .....	136
Block Selecting Error Indicators .....	137
Overriding ERC Errors .....	137
Fixing and Deleting ERC Errors .....	137

## **chapter 9 Printing a Schematic**

Setting up Printers and Plotters.....	139
Print Features.....	139
Selecting a Printer or Plotter .....	139
Setting up Print Jobs .....	140
Defining Image Options, Image Scale, and a Print Region .....	140
Setting your Print Options .....	142
Previewing a Print Job.....	144
Generating Printouts .....	144
Override Settings.....	144
Printing the Current Window .....	144
Scaling to Fit Page .....	145

## chapter 10 Design Technology Parameters

Introducing the DTP Feature .....	147
What is DTP? .....	147
About DTP Files .....	147
Opening and Creating DTP Files .....	148
Opening a DTP File .....	148
Creating a DTP File .....	148
Adding a Group .....	150
Adding a Section .....	151
Adding an Item .....	153
Working with DTP Files .....	153
Browsing the Contents of a DTP File .....	153
Renaming a Group or Item .....	154
Deleting DTP File Information .....	154
Viewing Item Statistics .....	155
Updating a DTP Group with Design Information .....	155
Copying DTP Data to a Design .....	156
Setting up Attribute Handling for Copy Actions .....	158
Item-Specific Information About Copy Actions .....	158
Modifying Item Properties .....	159

## chapter 11 Simulating a Circuit Design

Using the P-CAD Circuit Simulator .....	164
Configuring the Schematic .....	164
Selecting simulation-ready schematic components .....	164
Setting up the Simulation Analyses .....	164
Running a Simulation .....	165
Working with Simulation Waveforms .....	166
Troubleshooting simulation problems .....	167

## chapter 12 File Commands

Using the File Commands .....	169
File New .....	169
File Open .....	169
File Close .....	170
File Save .....	170
Saving a File with ECOs .....	170
Saving an ASCII File .....	171
File Save As .....	171
Saving a Copy of a File .....	171
File Print .....	172
Override Settings Frame .....	173
Page Setup Dialog .....	173
Image Options Frame .....	174
Print Region Frame .....	174
Image Scale Frame .....	174
Print Options Dialog .....	174

File Print Setup .....	177
File Reports .....	177
Format Tab .....	179
Selection Tab .....	181
Sort Tab .....	182
File Design Info .....	183
General Tab .....	183
Fields Tab .....	184
Field Properties Dialog .....	185
Attributes Tab .....	185
Notes Tab .....	186
Revisions Tab .....	187
Statistics Tab .....	188
File Design Technology Parameters .....	188
Design Technology Parameters Dialog .....	188
File DXF In .....	190
Opening a DXF File .....	190
Items Supported for Translation .....	192
DXF In Notes .....	194
File DXF Out .....	195
Creating a DXF File .....	195
DXF Output Considerations .....	196
File PDIF In .....	197
File PDIF Out .....	198
File Exit .....	199

## chapter 13 Edit Commands

Using the Edit Commands .....	201
Edit Undo .....	201
Edit Redo .....	202
Edit Cut .....	203
Edit Copy .....	203
Edit Copy to File .....	203
Edit Paste .....	204
Paste Behavior .....	204
Paste From Clipboard .....	205
Paste From File .....	205
Paste Circuit .....	206
Using Paste Circuit .....	206
Paste Circuit From File .....	208
Edit Move By RefDes .....	209
Moving Parts by RefDes .....	209
Searching the RefDes List .....	211
Edit Properties .....	211
Right-Click to Select Properties .....	211
Double Click to Select Properties .....	212
Part Properties .....	212
Symbol Tab .....	213

Symbol Pins Tab .....	214
Component Tab.....	215
Component Pins Tab.....	216
Attributes Tab .....	218
Wire Properties.....	218
Wire Width .....	219
Text Styles Button .....	219
Net Tab.....	219
Bus Properties .....	220
Port Properties.....	221
Net Tab.....	221
Pin Properties .....	222
Pin Name and Pin Designator Properties .....	224
Line Properties .....	224
Arc Properties.....	225
Query Fields .....	225
Changing Arc Properties .....	225
Polygon Properties .....	226
Text Properties .....	226
Changing Text Properties.....	227
Attribute Properties.....	228
Component References.....	229
Field Properties .....	230
Query Fields .....	231
Editable Fields .....	231
ERC Errors Properties.....	232
Viewing ERC Errors.....	232
Replacing Component Types .....	233
Edit Delete .....	234
Deleting Objects .....	235
Deleting Objects from Nets.....	235
Edit Copy Matrix .....	236
Duplicating an Object .....	236
Copying Nets.....	237
Edit Explode Part.....	237
Edit Align Parts .....	237
Edit Select All .....	238
Edit Deselect All.....	238
Edit Highlight .....	238
Edit Unhighlight .....	238
Edit Unhighlight All.....	238
Edit Select Highlight.....	238
Edit Parts .....	239
Parts List Box .....	239
Set All/Clear All.....	239
Properties .....	239
Highlight/Unhighlight.....	240
Highlight/Unhighlight an Attached Net .....	240

Jump .....	240
Edit Nets .....	240
Attributes Dialog .....	243
Rename .....	244
View Attributes .....	245
Highlight/Unhighlight Nets .....	245
Highlight/Unhighlight Buses .....	246
Delete .....	246
Select .....	246
Edit Measure .....	246
Using Edit Measure .....	246
Edit Select .....	247
Status Line Information .....	247
Select Commands .....	247
Selecting Objects .....	248
When Objects Overlap .....	248
Moving and Copying Objects (Drag-and- Drop) .....	248
Resizing Objects .....	249
Rotating and Flipping .....	249
Edit Properties .....	249
Select Contiguous .....	249

## **chapter 14 View Commands**

Using the View Commands .....	251
View Redraw .....	251
View Extent .....	251
View Last .....	251
View All .....	252
View Center .....	252
View Zoom Out .....	252
Minus Key (-) .....	252
View Zoom Window .....	252
Zooming In with a Zoom Window .....	253
View Jump Location .....	253
Jump to a Location .....	253
View Jump Text .....	254
Jump to Text .....	254
View Descend .....	255
View Ascend .....	255
View Command Toolbar .....	256
View Placement Toolbar .....	256
View Custom Toolbar .....	256
View Prompt Line .....	256
View Status Line .....	257
View Snap to Grid .....	257

## chapter 15 Place Commands

Using the Place Commands .....	259
Place Part .....	259
Placing a Part .....	259
Power Components .....	262
Splitting a Net .....	262
Jumper Pins .....	262
Place Wire .....	263
Adding Wires to Nets .....	264
Jumper Pins .....	264
Orthogonal Modes .....	264
Place Bus .....	265
Status Line Measurements .....	266
Orthogonal Modes .....	266
Place Port .....	267
Display Characteristics .....	267
Placing Ports .....	267
Hotkeys .....	268
Placing an Inline Port .....	268
Jumper Pins .....	268
Moving Ports .....	268
Renaming Ports .....	269
Modifying a Port's Text Style .....	269
Deleting Ports .....	269
Backward Compatibility .....	269
Place Pin .....	269
Place Line .....	272
Placing a Line .....	272
Status Line Measurements .....	272
Orthogonal Modes .....	273
Place Arc .....	273
Placing an Arc .....	273
Changing Arcs .....	274
Changing an Arc Center Point .....	274
Place Polygon .....	274
Draft/Outline Display Mode .....	275
Rotate/Flip .....	275
Altering the Shape .....	275
Place Ref Point .....	275
Placing a Reference Point .....	275
Place Text .....	275
Place Text Dialog .....	276
Place Text Features .....	278
Place Attribute .....	279
Placing an Attribute .....	279
Rotate/Flip .....	280
Place Field .....	280
Placing a Field .....	281

Rotate/Flip .....	281
Place IEEE Symbol .....	282
Placing a IEEE Symbol .....	282
<b>chapter 16 Rewire Commands</b>	
Using the Rewire Command.....	285
Rewire Manual .....	285
Using the Rewire Manual Command.....	285
<b>chapter 17 Options Commands</b>	
Using the Options Commands.....	287
Options Block Selection.....	287
Select Mode Frame .....	288
Items Frame .....	288
Selection Mask Dialogs .....	288
Selection Mask Parameters .....	290
Related Topics .....	292
Options Configure .....	292
Workspace Size .....	293
Title Sheets .....	293
Units .....	293
Orthogonal Modes.....	294
Net Increment.....	294
ECOs.....	294
AutoSave.....	294
File Viewer .....	295
Zoom Factor.....	295
Autopan.....	295
Compress Binary Designs .....	295
DDE Hotlinks .....	295
Options Grids .....	295
Grid Spacing .....	296
Grid Toggle Buttons .....	296
Visible Grid Style.....	296
Relative Grid Origin.....	296
Mode .....	297
Options Display .....	297
Colors.....	297
Miscellaneous .....	299
Options Preferences.....	301
Keyboard Tab.....	302
Mouse Tab .....	302
Options Design Rules.....	303
Design Tab.....	303
Clearance Rules.....	304
Net Class Tab .....	304
Net Tab .....	307

Class to Class Tab .....	308
Options Net Classes .....	309
Options Sheets .....	311
Sheets .....	311
Buttons .....	311
Adding a Sheet .....	312
Options Current Wire .....	312
Options Current Line .....	313
Options Text Style .....	314
Add a Text Style .....	314
Delete a Text Style .....	315
Rename a Text Style .....	315
Text Style Properties .....	315
Display Stroke and Display TrueType Buttons .....	317

## chapter 18 Library Commands

Using the Library Commands .....	319
Library New .....	319
Library Alias .....	320
Creating an Alias .....	320
Library Copy .....	321
Copying Symbols/Components .....	322
Library Delete .....	323
Deleting from a Library .....	323
Library Rename .....	324
Renaming a Symbol/Component .....	324
Library Setup .....	325
Drag and Drop File Load .....	325
Setting Up a Library .....	325
Library Symbol Save As .....	326
Saving a Symbol .....	326
Library Archive Library .....	327

## chapter 19 Utils Commands

Using the Utils Commands .....	329
Utils Renumber .....	329
Renumbering Reference Designators .....	329
Renumbering Pins .....	331
Renumbering Default Pin Des .....	332
Utils Force Update .....	333
Utils ERC .....	334
Utils Find Errors .....	337
Utils Record ECOs .....	338
Types of ECOs .....	339
Utils Import ECOs .....	339
ECO Filename .....	340
Preview ECOs .....	340

Utils Export ECOs .....	340
View Pending ECOs.....	341
Save ECOs Now .....	341
Utils Generate Netlist .....	342
Utils Rename Net .....	343
Renaming a Net .....	343
Jumper Pins .....	344
Utils Resolve Hierarchy .....	344
Utils Module Wizard .....	345
Creating a New Module and its Link.....	346
Reusing an Existing Module.....	348
Utils Shortcut Directory.....	351
Utils P-CAD PCB.....	351
Utils P-CAD Library Executive.....	352
Utils P-CAD Pattern Editor .....	352
Utils P-CAD Symbol Editor .....	352
Utils InterPlace/PCS.....	352
Utils Customize .....	352
Displaying the Custom Toolbar .....	353
Executing a Custom Tool .....	354

## **chapter 20 Simulate Commands**

Simulate Run.....	355
Simulate Setup.....	355

## **chapter 21 DocTool Commands**

Using the DocTool Commands.....	357
DocTool Place Table .....	357
DocTool Titles .....	359
DocTool Notes.....	359
DocTool Update .....	359
DocTool Update All .....	359

## **chapter 22 Macro Commands**

Using the Macro Commands.....	361
Macro Setup .....	361
Setting Up a Macro.....	361
Macro Record.....	362
Macro Recording Tool .....	362
Temporary Macro Record (M Toggle Button).....	363
Macro Delete .....	364
Macro Rename .....	364
Macro Run.....	365
Running Temporary Macros.....	365
Recording Efficient Macros .....	365
Macro Features .....	366
Macro File Syntax.....	367

<b>chapter 23</b>	<b>Window Commands</b>	
	Window New Window .....	373
	Window Cascade.....	373
	Window Tile .....	373
	Window Arrange Icons.....	373
	Window 1,2,.....	374
<b>chapter 24</b>	<b>Help Commands</b>	
	P-CAD Schematic Help Topics .....	375
	How to Use Help.....	375
	Series II Commands .....	375
	About P-CAD Schematic .....	375
<b>appendix a</b>	<b>Keyboard Reference</b>	
	Schematic Keyboard Reference .....	377
<b>appendix b</b>	<b>Translating Tango-Schematic Designs</b>	
	Translation Process .....	383
	Considerations.....	384
	Design Practices Considered .....	384
<b>appendix c</b>	<b>P-CAD System Messages</b>	
	Error Messages .....	387
	Warning Messages .....	392
<b>appendix d</b>	<b>The Tango Netlist Format</b>	
	The Component Section .....	397
	The Net Section.....	398
<b>Index</b>	.....	401

# Introducing P-CAD Schematic

Congratulations on your purchase of P-CAD Schematic! P-CAD Schematic is a complete schematic design system for Microsoft® Windows 95™, Windows 98™, Windows 2000™, or Windows NT™ operating systems.

P-CAD Schematic distinguishes itself with a suite of features that make schematic creation fast and easy. It includes a powerful combination of placement, editing, and browse tools to quickly achieve superior design results.

## P-CAD Schematic Features

---

This section highlights some of the key features in P-CAD Schematic:

- Powerful placement and editing tools, object resize, multiline unwind, rewire, drag and drop move, and align.
- User-definable sheet, part, and net attributes.
- Intelligent wires and buses, which carry net information and attributes.
- Comprehensive Electrical Rules Checking (ERC) with error annotation and violations highlighted on screen.
- Support for Design Technology Parameters.
- Cross probe between P-CAD Schematic and P-CAD PCB design.
- Advanced bi-directional ECO capabilities.
- Automatic junction and bus entry placement.
- Single mouse-click (Click) access to edit and select options.
- Support for hierarchical schematic designs.
- Component library data integrated for use in both P-CAD Schematic and P-CAD PCB.
- Standard and user-defined schematic sheet sizes.

- Versatile documentation tools, which include zoned borders, custom title blocks, tables, annotated sheet connectors, and the ability to track nets between sheets.
- Extensive print and report options.

## **About the User's Guide**

---

This user's guide includes the following sections:

- *Getting Started*: This section explains what you what you need to know to get started using P-CAD Schematic.
- *Tutorials*: Tutorial chapters provide instructions on creating and navigating a simple schematic design.
- *Command Reference*: This section provides you with an extensive reference for the menu commands in P-CAD Schematic.

# 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.

## P-CAD Schematic Basics

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

- About the User Interface
- Configuration Options
- Schematic Sheets
- Opening Design Files

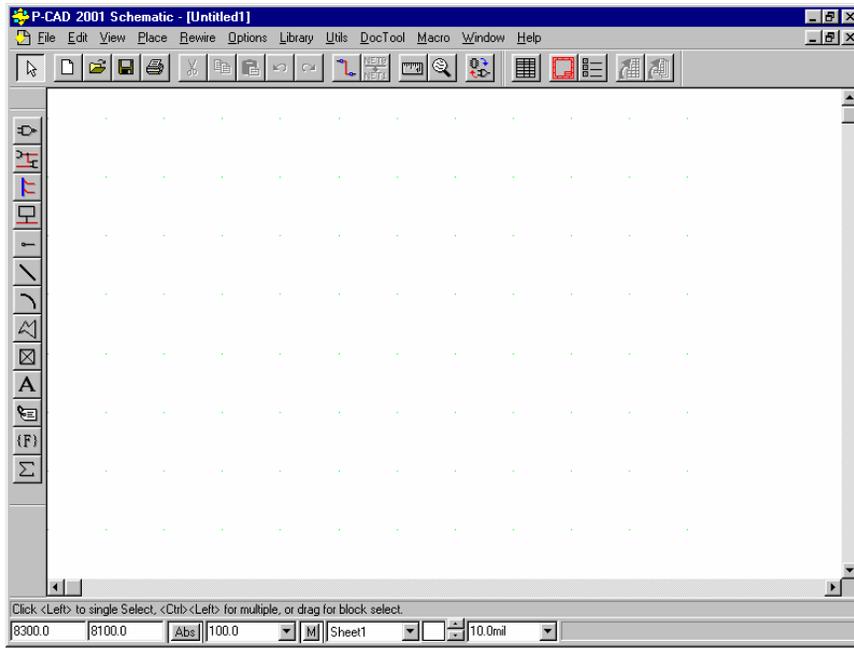
### About the User Interface

---

The features in the P-CAD Schematic user interface make it easy to create and modify schematic designs. The following topics provide you with information the elements in the P-CAD Schematic graphical user interface.

#### P-CAD Schematic Window

When you start P-CAD Schematic, a new, untitled P-CAD Schematic window appears. This window gives you a graphical view of your design and also provides you with the ability to gain access to numerous features. The following figure shows you this window:



## Menu Bar

Just below the P-CAD Schematic title bar lies the menu bar, which gives you the ability to gain access to a variety of P-CAD Schematic functions. You can choose a command from one of these menus to perform the actions associated with that command. The following figure shows you the menu bar:



If you minimize the P-CAD Schematic window, the menu bar wraps. In contrast, other window elements become truncated when you minimize the window. To learn how to minimize a window, see *Close, Minimize, and Maximize Buttons* (page 13).

## Opening a Menu

To open a menu, click the menu title or hold down the **ALT** key and press the underscored letter in the menu title. For example, press **ALT+F** to open the **File** menu.

## Choosing Menu Commands

When you open a menu, you can gain access to the functions associated with a command by clicking the command or by pressing the underscored letter in the menu and command titles. For example, press **ALT+F+O** to choose **File » Open**. When an ellipsis follows a command (e.g., **File Open...**), a dialog box appears.



## Command Toolbar

The buttons in the *Command* toolbar, act as shortcuts for frequently used menu commands. To show or hide this toolbar, choose **View » Command Toolbar**. When a check mark appears next to this menu command, the following *Command* toolbar is visible:



The following table describes each button in the *Command* toolbar:

Click this button	As a shortcut for choosing this command	Click this button	As a shortcut for choosing this command
	Edit » Select		Edit » Paste
	File » New		Edit » Undo
	File » Open		Rewire » Manual
	File » Save		Utils » Rename Net
	File » Print		Edit » Measure
	Edit » Cut		View » Zoom Window
	Edit » Copy		Utils » Record ECOs

## Placement Toolbar

The buttons in the *Placement* toolbar act as shortcuts for frequently used **Place** menu commands. To show or hide this toolbar, choose **View » Placement Toolbar**. When a check mark appears next to this menu command, the following *Placement* toolbar is visible:



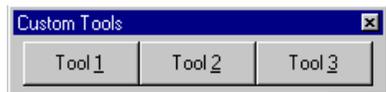
The following table describes each button in the *Placement* toolbar:

Click this button	As a shortcut for choosing this command	Click this button	As a shortcut for choosing this command
	Place » Part		Place » Polygon
	Place » Wire		Place » Point
	Place » Bus		Place » Text
	Place » Port		Place » Attribute
	Place » Pin		Place » Field
	Place » Line		Place » IEEE Symbol
	Place » Arc		

### Custom Toolbar

You can add your own buttons to the custom toolbar, which provides you with access to frequently used software programs or other documents. For example, you can add a button that gives you quick access to Notepad or Internet Explorer. For instructions, see *Utils Customize* (page 352).

Once you've added buttons to the *Custom* toolbar, you can show or hide it in the P-CAD Schematic window by choosing **View » Custom Toolbar**. When a check mark appears next to this menu command, the following *Custom Tools* toolbar is visible:



### DocTool Toolbar

The buttons in the *DocTool* toolbar act as shortcuts for frequently used **DocTool** menu commands. To show or hide this toolbar, choose **View » DocTool Toolbar**. When a check mark appears next to this menu command, the following *DocTool* toolbar is visible:

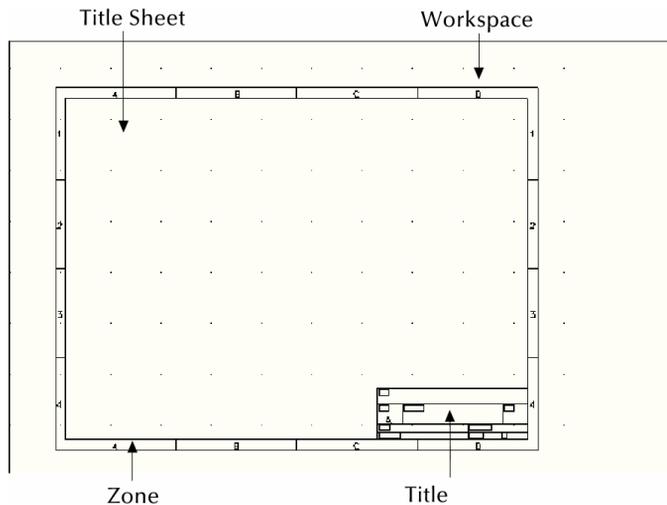


The following table describes each button in the *DocTool* toolbar:

Click this button	As a shortcut for choosing this command	Click this button	As a shortcut for choosing this command
	<b>DocTool » Place Table</b>		<b>DocTool » Update</b>
	<b>DocTool » Titles</b>		<b>DocTool » Update All</b>
	<b>DocTool » Notes</b>		

## P-CAD Schematic Workspace

The workspace is the logical design area within the P-CAD Schematic window. When you start P-CAD Schematic, the full extent of the workspace is shown. As shown in the following figure, the edges of the grid display determine the edges of the P-CAD Schematic workspace.



When you start P-CAD Schematic, the workspace contains a single schematic sheet. A schematic design can require multiple sheets, to show the connection of the parts on the board. To learn about sheets, see *Schematic Sheets* (page 19).

On the schematic sheet, a title sheet displays by default. As shown in the previous figure, a title sheet contains a design border, zones, and a title block by default. To learn about title sheets, see *Title Sheets* (page 98).

## Prompt and Status Lines

Just below the P-CAD Schematic workspace lays the prompt line and status line. To show or hide the prompt line or status line, choose **View » Prompt Line** or **View » Status Line**.

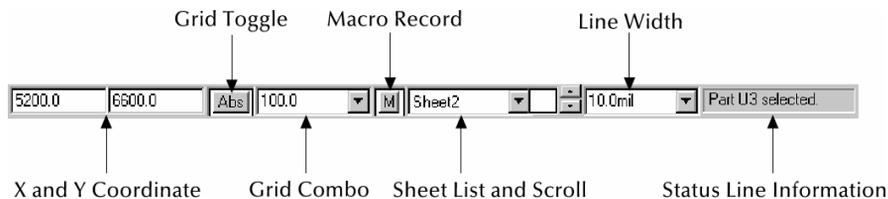
### Prompt Line

The Prompt Line displays various messages that instruct you as to what actions you can perform when a particular tool is enabled.

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

### Status Line

Just below the prompt line lies the Status Line, which contains various features that give you more control over you P-CAD Schematic workspace.



The following sections describe each element of the P-CAD Schematic status line.

#### X and Y Coordinate Boxes

The X and Y Coordinate boxes on the status line display the position of the mouse pointer as you move it over the workspace. To move the pointer to a specific location on your workspace, type an X and Y co-ordinate in the boxes.

#### Grid Toggle Button

Click the **Grid Toggle** button to switch between your absolute and relative grid settings. When you click this button, the button's label changes to indicate whether you can switch to your absolute or relative grid settings.

- Click **Abs** to switch to absolute grid mode. When active, the button background is white.
- Click **Rel** to switch to relative grid mode. When active, this button background is colored.

#### Grid Combo Box

Use the Grid Combo box to select a new grid and to add grids to your design. To add a grid, type a value in the box and press **ENTER**. The new grid becomes the current grid.

As a shortcut for selecting items from the box, you can press the following keys:

- Press **G** to scroll forward through the list.
- Press **SHIFT+G** to scroll back through the list.

### Macro Record Button

To create and record macros on the fly, click the **M** button on the status line. When you click this button, the button background changes to colored, and the macro recording process begins. After you record a series of actions, click the **M** button again to stop recording.

When you record a macro with this button, P-CAD Schematic names the macro `Sch_default.mac` and saves the file in the `Macro` folder in your PCAD installation directory by default. To learn more about this feature, see *Macro Record* (page 362).

### Sheet List Box and Scroll Buttons

Use the Sheet List and Scroll box as a shortcut for choosing **Options » Sheets**. To make a sheet the current sheet in view, select a sheet from the list. To scroll through all enabled sheets, click the **arrow** buttons on the scroll box.

As a shortcut, you can also press the **L** key to scroll through the sheets in descending order. To scroll through the sheets in ascending order, click the **down arrow** or press **SHIFT+L**.

### Line Width List

The Line Width combo box shows a list of available line widths. Because this is a combo box, you can select an existing line width or type a custom width in the box.

As a shortcut for selecting a line width from the box, you can press the **W** key to scroll forward through the list. Or, press **SHIFT+W** to scroll back through the list.

You can also set line widths by choosing **Options » Current Line**. For more information, see *Options Current Line* (page 313).

### Status Line Information Area

The Status Line information area displays information relevant to the action you are currently performing. The status line shows the following types of data:

- Identifies selected objects either specifically (type or reference designator) or generally (number of items selected).
- The delta X and delta Y measurements of objects being moved or line segments being stretched.
- The net name of selected nets.
- Results of the **Edit » Measure** command.
- The next part being placed while placing parts.

## Other Window Elements

As with most Windows-based products, the P-CAD Schematic window contains other elements:

### Close, Minimize, and Maximize Buttons

Every P-CAD Schematic window has three buttons in the upper-right corner. You can click one of the following buttons to perform one of these actions:

- Click  to close the window and quit the P-CAD Schematic program.
- Click  to minimize the window.
- Click  to maximize the window.

### Scroll Bars

Two scroll bars surround the P-CAD Schematic workspace. You can click the scroll arrows to move the workspace up and down, or to the left and right. You can also drag the scroll box to move through the workspace.



## Shortcut Menu Commands

When you right-click an object that has been placed in a design, a shortcut menu appears. This menu contains commands that can be performed on the selected object(s). The following table describes these commands:

Choose this shortcut menu command	To perform this action
<b>Add to Net</b>	A shortcut to connect a part's sub-selected pin to an existing net.
<b>Add Vertex</b>	A shortcut to add a vertex to a single selected line/line segment.
<b>Align</b>	A shortcut for <b>Edit » Align Parts</b> . Parts can be aligned around a selection point either horizontally or vertically, and as an option, equally spacing the parts. For details, see <i>Edit Align Parts (page 237)</i> .
<b>Ascend</b>	A shortcut for <b>View » Ascend</b> . Ascends the hierarchy of the design from the selected link to the corresponding module. For details, see <i>View Ascend (page 255)</i> .
<b>Copy Matrix</b>	A shortcut for <b>Edit » Copy Matrix</b> . Duplicates selected objects based on parameters you choose. For details, see <i>Edit Copy Matrix (page 236)</i> .
<b>Copy</b>	A shortcut for <b>Edit » Copy</b> . Allows you to copy the object to the clipboard.

Choose this shortcut menu command	To perform this action
<b>Cut</b>	A shortcut for <b>Edit » Cut</b> . Allows you to cut the object to the clipboard. For information, see <i>Edit Cut</i> (page 203).
<b>Delete</b>	A shortcut for <b>Edit » Delete</b> . This command deletes selected objects. For details, see <i>Edit Delete</i> (page 234).
<b>Descend</b>	A shortcut for <b>View » Descend</b> . Descends the hierarchy of the design from the selected module to the corresponding link. For information, see <i>View Descend</i> (page 255).
<b>Edit Nets</b>	A shortcut for <b>Edit » Nets</b> . This command brings up the <i>Edit Nets</i> dialog with the nets containing the selected objects highlighted in the <i>Net List</i> box. See <i>Edit Nets</i> (page 240) for more information.
<b>Explode</b>	A shortcut for <b>Edit » Explode Part</b> . This command allows you to convert a symbol back to its basic primitives, creating a collection of editable graphic objects. See <i>Edit Explode Part</i> (page 237) for details.
<b>Highlight</b>	A shortcut for <b>Edit » Highlight</b> . This command highlights the selected objects in the current highlight color set in the <i>Colors</i> tab of the <i>Options Display</i> dialog. For information, see <i>Options Display</i> (page 297). It also highlights the corresponding items (for nets and components) in P-CAD PCB if the <i>DDE Hotlinks</i> check box in the <i>Options Configure</i> dialog is selected. For information, see <i>Options Configure</i> (page 292).
<b>Highlight Attached Nets</b>	Highlights nets attached to the selected objects in the current highlight color. For information, see <i>Options Display</i> (page 297).
<b>Net Info</b>	Opens a read-only dialog containing information about the selected net.
<b>Properties</b>	A shortcut for <b>Edit » Properties</b> . Depending on the object(s) selected, the appropriate <i>Edit Properties</i> dialog appears. For information, see <i>Edit Properties</i> (page 211).
<b>Remove from Net</b>	Disconnects a part's sub-selected pin from the net.
<b>Select Contiguous</b>	Selects all ports and wires in the design visually connected to the selected item.
<b>Select Net</b>	Selects all items in the net to which the selected item is connected.
<b>Selection Point</b>	Relocates a selection point for the selected object or objects.
<b>Unhighlight</b>	A shortcut for <b>Edit » Unhighlight</b> . This command removes the highlighting from the selected objects. For information, see <i>Edit Unhighlight</i> (page 238).

Choose this shortcut menu command	To perform this action
<b>Unhighlight Attached Nets</b>	Removes the highlighting from nets attached to the selected objects.

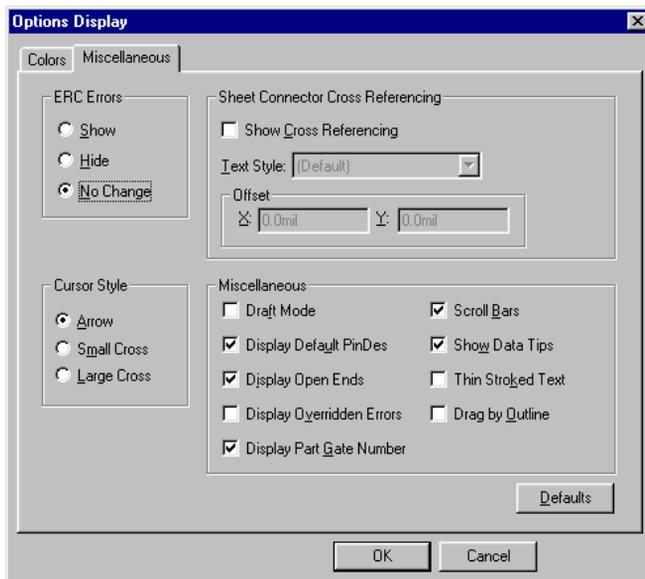
## Configuration Options

With P-CAD Schematic, you can use various features and tools only after you set them up properly. The topics in this section show you how to set up the following configuration options for P-CAD Schematic.

### Showing or Hiding DataTips

A Data Tip shows context-sensitive information about buses, pins, symbols, and wires. To show or hide DataTips in your workspace, follow these steps:

1. Choose **Options » Display** to open the *Options Display* dialog.
2. Click the **Miscellaneous** tab.



3. Choose one of these options:
  - To show DataTips, select the **Show DataTips** check box.
  - To hide DataTips, clear the **Show DataTips** check box.

The Large Cross cursor style and the **View » Snap to Grid** command do not support the DataTips feature. Enabling the Large Cross cursor style clears the **Show DataTips** check box and makes the feature unavailable. Enabling the **View » Snap to Grid** command disables the Show DataTips feature.

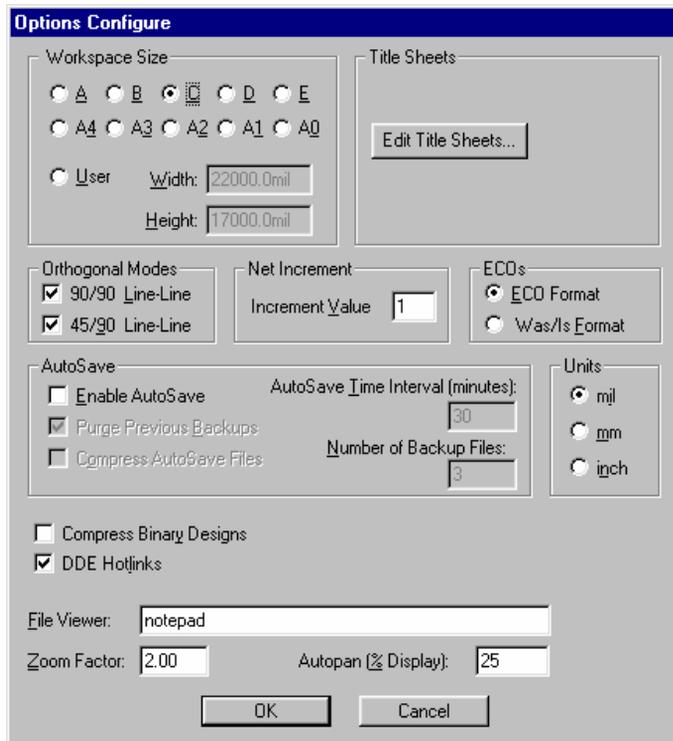
- Click **OK** to close the *Options Display* dialog.

When the DataTips feature is turned on, you can view a Data Tip by placing the mouse cursor over an object in your design.

## Selecting a Default File Viewer

In P-CAD Schematic, the default file viewer is used for viewing reports; log files, error reports, etc. To select a file viewer, do the following:

- Choose **Options » Configure** to open the *Options Configure* dialog.



- Type the program name in the **File Viewer** box. For example, type: Notepad

If the program you want to use as the default file viewer is in a directory that is not included in your system's path statement, type the pathname in the File Viewer box.

3. Click **OK** to close the *Options Configure* dialog.

## Setting the Current Zoom Factor

The current zoom factor is a value that determines the apparent enlargement of objects when you zoom in or zoom out on a design. To set the current zoom factor, follow these steps:

1. Choose **Options » Configure** to open the *Options Configure* dialog.
2. Enter a value in the Zoom Factor box.

You must enter a zoom factor that is greater than 1.00 (i.e., 1.01 or greater).

3. Click **OK** to close the *Options Configure* dialog.

To learn how to zoom in and out on a design, see *View Commands* (page 251).

## Setting the Autopan Percent Display

The autopan percent display controls the process of panning across your design. For example, an autopan of 25% moves anything at the edge of the screen 25% closer to the center of the screen.

To set the autopan percent display, follow these steps from the *Options Configure* dialog:

1. Choose **Options » Configure** to open the *Options Configure* dialog.
2. Enter a number in the Autopan% Display box.
3. Click **OK** to close the *Options Configure* dialog.

## Enabling Orthogonal Modes

You can use orthogonal modes when you place horizontal and vertical segments for wires, buses and lines. In P-CAD Schematic, the 90/90 and 45/90 modes are enabled by default. To learn more about orthogonal modes, see *Aligning Objects* (page 87).

## Specifying a Net Increment Value

A net increment value defines the step value for incrementing net names when you use **CTRL/Drag Copy** and **Edit » Copy Matrix** commands. A value of 1 appears in the Net Increment Value box by default. To learn more about the increment value, see *Options Configure* (page 292).

## Choosing an ECO Format

You use the controls in the ECO Format frame to choose a format for ECO files. In P-CAD Schematic, the **ECO Format** button is selected by default.

## Enabling the AutoSave Feature

The AutoSave feature creates backup files of your current design at intervals you define. To enable the AutoSave feature, follow these steps:

1. Choose **Options » Configure** to open the *Options Configure* dialog.

2. Select the **Enable AutoSave** check box.
3. Notice that the **Purge Previous Backups** check box is selected by default. This option causes all backups saved from the previous design session to be deleted when you begin a new design session
4. To turn the AutoSave feature off, clear the **Enable Auto Save** check box.
5. Click **OK** to close the *Options Configure* dialog.

When enabled, AutoSave won't be performed during auto-routing or if a tool is busy.

## Compressing Binary Designs

Typically, compressed files take up less storage space. To automatically compress a binary file when you save it, follow these steps:

1. Choose **Options » Configure** to open the *Options Configure* dialog.
2. Select the **Compress Binary Designs** check box.
3. Click **OK** to close the *Options Configure* dialog.

## Enabling DDE Hotlinks

The DDE hotlinks feature gives you the ability to create a connection between two related design files created in separate P-CAD programs. With this feature, you can explore the relationships between designs when you run the following programs:

- P-CAD Schematic and P-CAD PCB/Relay
- P-CAD Schematic and P-CAD PCB Viewer.

With DDE Hotlinks, you can update information in one design and the feature automatically updates the same information in a related design. For example, you could apply the current highlight color to a part in a Schematic design and the DDE Hotlinks feature updates the related part in P-CAD PCB. Or, when you highlight a net in a PCB design, the DDE Hotlinks feature highlights the corresponding net in a related Schematic design.

To enable this feature, choose **Options » Configure** and then select the **DDE Hotlinks** check box.

## Using a Snappy Cursor

Choose **View » Snap to Grid** to turn the snappy cursor on and off. When a check mark precedes the command, a snappy cursor moves from grid point to grid point. When the command is not checked, a free-floating cursor moves freely between grid points.

A snappy cursor can create a predictable point of reference when you move and rotate objects or measure distances. A free-floating cursor can enhance your ability to select items in the workspace, such as when you want to select a line that does not run true along grid points.

The **View » Snap to Grid** command does not support the DataTips feature. When **View » Snap to Grid** is a checked command, the DataTips feature is disabled by default.

Your current **View » Snap to Grid** setting (on or off) is saved in the `Sch.ini` file when you quit the program.

## Schematic Sheets

---

A schematic design can be a single sheet or multi-sheet design. When you start P-CAD Schematic, a new, untitled design window contains a single sheet. In this section, you'll find instructions that show you how to perform the following tasks:

### Adding a Sheet

To add a sheet to a design, do the following:

1. Choose **Options » Sheets** to open the *Options Sheets* dialog.
2. Type a name for the sheet in the Sheet Name box. For example, type: `Sheet2`

When you add a sheet, you must give it a unique name.

3. Click **Add**. The new sheet name appears in the Sheets list.

When you add a sheet to a design, P-CAD Schematic automatically assigns the sheet a sequential sheet number, which represents the order in which the sheet appears in the Sheets List.

### Reordering Sheets

To reorder the sheets in a design, follow these steps:

1. Choose **Options » Sheets** to open the *Options Sheets* dialog.
2. Select the sheet you want to move from the Sheets list.
3. Click **Move Up** or **Move Down**.

When you move the sheet to a different position in the Sheets list, the sheet number adjusts accordingly.

### Defining the Current Sheet

To define the current sheet, follow these steps:

1. Choose **Options » Sheets** to open the *Options Sheets* dialog.
2. Select the sheet that you want to define as the current sheet from the Sheets list.
3. Click **Current**. The asterisk (\*) moves to the sheet you selected, to indicate that it is now the current sheet.

The current sheet is the sheet that is actively in view. As a shortcut, you can also define the current sheet by selecting a sheet from the Sheets list in the status line. For details, see *Status Line* (page 11).

## Renaming a Sheet

To change a sheet name, follow these steps:

1. Choose **Options » Sheets** to open the *Options Sheets* dialog.
2. Select the sheet that you want rename from the Sheets list.
3. Type a new name for the sheet in the Sheet Name box.
4. Click **Modify**. The sheet name changes in the Sheets list.

## Deleting a Sheet

To delete the current sheet, follow these steps:

1. Choose **Options » Sheets** to open the *Options Sheets* dialog.
2. Select the sheet that you want to delete from the Sheets list.
3. Click **Delete**. The sheet name no longer appears in the Sheets list.

## Tracking Nets Across Sheets

P-CAD Schematic includes a combination of features with which you can track nets between sheets of the design. To print a net tracked between the sheets in a schematic design, follow these steps:

1. Choose **Edit » Nets** to open the *Edit Nets* dialog.
2. Select the desired net in the Net Names box. The sheets on which the net appears are automatically selected in the Sheets box.
3. Click **Highlight** to highlight the selected net.
4. Click **Print Sheets** to open the *File Print* dialog.
5. In the *File Print* dialog, click **Print Options** to open the *Print Options* dialog.
6. Select the **Print Highlighted Wires/Buses as Haloed** check box. This option distinguishes these highlighted nets in your print output.
7. Click **OK** to return to the *File Print* dialog.
8. Select the **Tile Sheets** check box to consolidate all of the net's sheets onto a single printed page.
9. To view the tiled sheets of the tracked nets on the screen, click **Print Preview**.
10. To send the tiled sheets of the tracked nets directly to the printer, click **Generate Output**.

## Opening Design Files

---

You can open files that have been saved as Schematic Files (\*.sch), Tango Series II ASCII Files (\*.s01), and P-CAD CFG Files (\*.cfg). As a shortcut for opening files with the **File » Open** command, you can also use one of the following methods:

- Use a drag-and-drop operation to open a Schematic File (\*.sch). To do so, drag a file from the Windows™ Explorer or File Manager and drop it on the P-CAD Schematic window.
- To open a recently used file, click one of the file names that appear below the **Exit** command in the **File** menu.

When you open files that were created in the Schematic Version 3.05 or earlier, P-CAD Schematic merges any global and local attributes in the file.

### Opening Schematic Files (\*.sch)

In P-CAD Schematic, design files saved in the Binary or ASCII format are given the \*.sch file name extension by default. To open a \*.sch file, follow these steps:

1. Choose **File » Open** to display the *Open* dialog.
2. Select **Schematic Files (\*.sch)** from the Files of Type list.
3. Select the file that you want to open.
4. Click **Open**. Schematic opens the .sch file. As the file loads, several messages appear to indicate the systems progress.

If an error or warning condition occurs when you open a file, P-CAD Schematic creates a log file <Design-name.log. A message box also prompts you to view this file. When you click **OK**, the log file appears in the Notepad by default. For details about possible error or warning conditions, see *Appendix C, P-CAD System Messages*.

### Opening Multisheet Designs

If you're opening a Design.cfg file, Schematic translates all the PDIF files into a single multi-sheet design. Schematic also preserves the name of the \*.cfg file you loaded. In the design, each sheet has its own window. To switch between sheet windows, select the desired sheet from the Sheet Display combo box on the status line.

### Opening a Tango Schematic Series II File

The **File » Open** command can read schematics in Tango Series II ASCII format. In Tango Series II, each sheet in a design is saved as a separate file. When you open the first sheet file in a Series II design, a dialog appears asking you if you want all subsequent sheets in the Series II design converted into a single design file.

Before loading a Tango Series II file, you must do the following:

1. Create a corresponding archived library containing all of the symbols found in your P-CAD Schematic designs.
2. Translate the archived Tango Series II library into a component/symbol library by choosing the **Library » Translate** command in P-CAD 2002 Library Executive.

Some cleanup may be necessary due to design complexity and interpretation differences due to ambiguous design practices.

## Opening an ASCII File

P-CAD Schematic supports the import and export of ASCII files, which contain complete design information. This is useful to archive and restore designs in a readable format.

After a file is saved in ASCII format, other utilities may be used to perform these alterations:

- Translation of design data from P-CAD Schematic to other design tools, and vice versa.
- Automated modification of schematic designs with custom designs.
- Interactive manual user modification of design data with text editing tools.

Errors and warnings generated while opening an ASCII file are written to a file named `<design-name.log`, which automatically appears in the Notepad.

## Schematic Design Tutorials

This chapter contains the following five tutorials that will take you through a Schematic design session from setup to printing:

- Setting up a Schematic Design
- Working with Schematic Objects
- Verifying a Schematic Design
- Generating Schematic Reports and Netlists
- Printing a Schematic Design.

### Tutorial 1 - Setting Up a Schematic Design

---

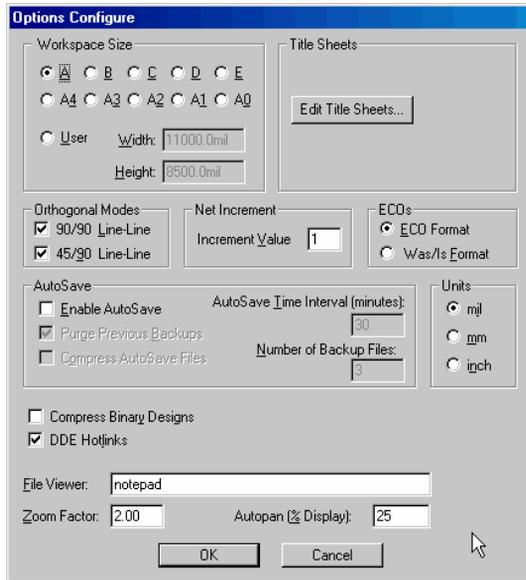
P-CAD Schematic provides you with a number of workspace options. This tutorial will show you how to configure various options for the P-CAD Schematic workspace, such as workspace size, title sheets, display options, measurements and grids. This tutorial uses some demonstration files that are supplied with P-CAD.

For more information about the topics in this tutorial, press **F1** for online Help, or refer to the relevant chapters in this *User's Guide*.

#### Setting the Workspace Size

The workspace is the logical design area within the P-CAD Schematic window. To choose a size for the workspace:

1. Start P-CAD Schematic. An empty workspace named `Untitled1` is displayed.
2. Choose **Options » Configure** to open the *Options Configure* dialog.



- In the Workspace Size frame, choose **A**.

The A-size workspace is the smallest of the predefined sheet sizes. For the design in this tutorial, an A-size drawing is adequate. However, if this becomes too small, open the *Options Configure* dialog and choose another size. You can switch to a smaller sheet at any time, as long as all of the objects in your schematic fit within the area.

- Click **OK** to close the *Options Configure* dialog.

## Setting Up a Title Sheet

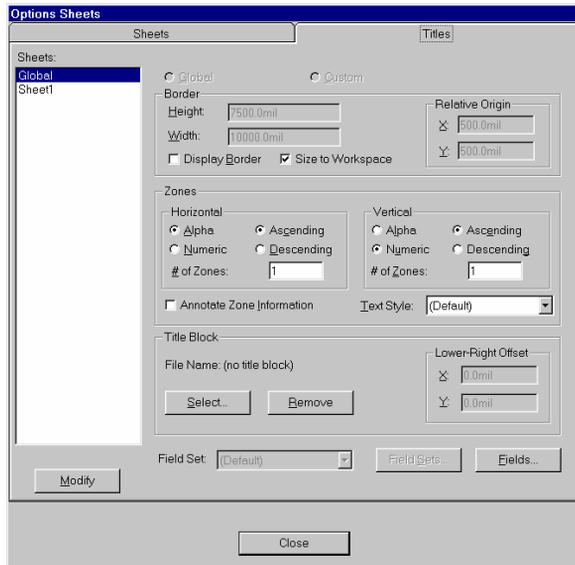
A title sheet can consist of a design border, zones and a title block. We will set up an A-sized global title sheet for all the sheets you will create within a design session. You can also select different title sheets for different sheets in a design by choosing the **Custom** sheet option. This option will override the global default title sheet for each selected sheet.

Standard title sheets (`filename.ttl`) are supplied with P-CAD Schematic in the `Titles` folder of your P-CAD installation directory. The title blocks supplied can only be changed, e.g. if you want another text size or font, by modifying the original `.ttl` file. You may wish to create your own title sheets later by modifying one of these files and saving it as a new `.ttl` file.

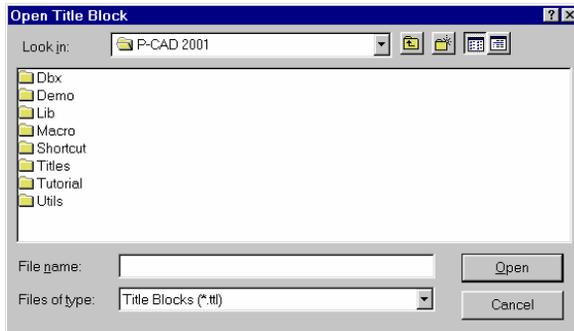
For more information about title sheets, refer to *Options Commands*, (page 287).

To set a global title sheet that will be automatically applied to all new sheets in your design:

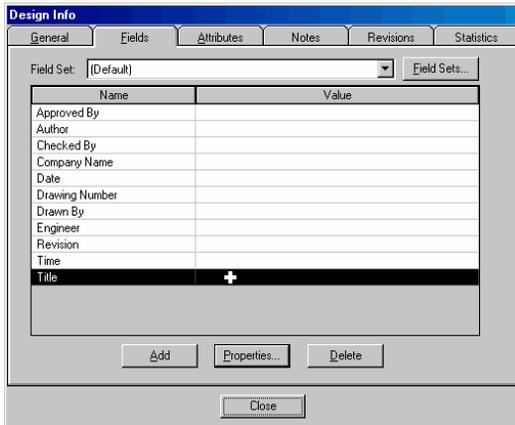
1. Open the *Options Sheets* dialog using one of the following methods:
  - Choose **Options » Configure** to open the *Options Configure* dialog. In the Title Sheets frame, click **Edit Title Sheets**.
  - Choose **Options » Sheets**. Then, click the **Titles** tab.
  - Choose **DocTool » Titles**.



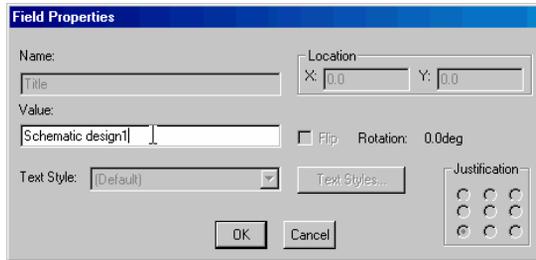
2. Select **Global** from the Sheets list box.
3. Select **Display Border**. Notice that the border dimensions have automatically adjusted to allow 500mil (.5 inch) between the edge of your workspace and the edge of the border.
4. Set up the zones that display in the border as reference marks. Type 4 in both # of Zones boxes to create four zones in both horizontal and vertical directions and select **Annotate Zone Information**.
5. Now we can select a title block that has been supplied with P-CAD with text fields already inserted for your design information. Click on **Select** in the Title Block frame to choose the filename of an existing title block. The *Open Title Block* dialog displays.



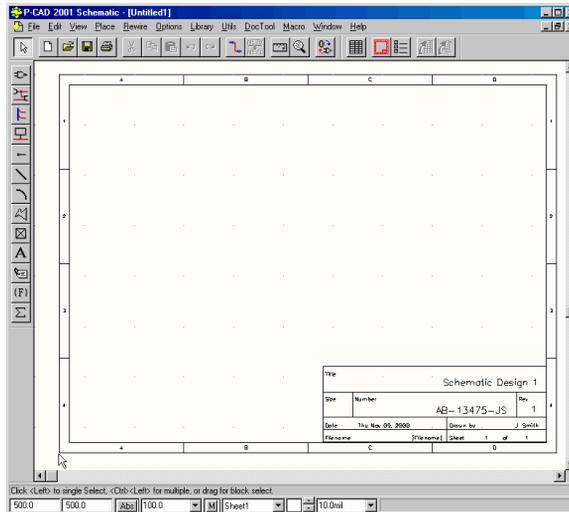
6. Navigate to the P-CAD Titles directory. Select ADT\_AB.ttl and click **Open** to return to the *Options Sheets* dialog. The file name ADT\_AB.ttl appears in the Title Block frame.
7. Within the title block we selected in the step above, there are design information fields that display the values you enter in their appropriate place. When corresponding design information has been entered for a field, a field result appears in place of the field code, e.g. Schematic Design 1 replaces {Title}. To change the field codes into relevant design information in the title block, click **Fields** to display the *Design Info* dialog.



8. Select a field name you wish to include in the title block, e.g. Title, click on **Properties** and enter the value in the *Field Properties* dialog, e.g. Schematic Design 1, and click **OK**.



9. Repeat the step above for all required fields, e.g. Drawing Number, Revision and Drawn By, and click on **Close** to complete and return to the *Options Sheets* dialog.
10. Click **Modify** to apply the changes and then **Close**. If you return to the *Options Configure* dialog, click **OK** to close this dialog. The title sheet, border, and title block, with the design information entered, now appear in the workspace, as shown below.



You can change the design information at any time by choosing **File » Design Info** and clicking the **Fields** tab to display the *Fields Properties* dialog.

11. Now we have set up our title sheet, let's save the design file by choosing **File » Save**. In the Save As dialog that appears, name your schematic *Design1.sch* and save it in the folder of your choice.

If the last saved title block is displaying when opening a new file and you do not wish to use it, remove the title block file from the **Title** tab of the *Options Sheets* dialog by clicking **Remove**, then **Modify** and **Close**. Close all open design files until a blank untitled workspace is left, exit P-CAD and then restart P-CAD Schematic before choosing **File » New**.

Next we will set up the other workspace options that will be saved with this file.

## Choosing a Unit Measurement Scale

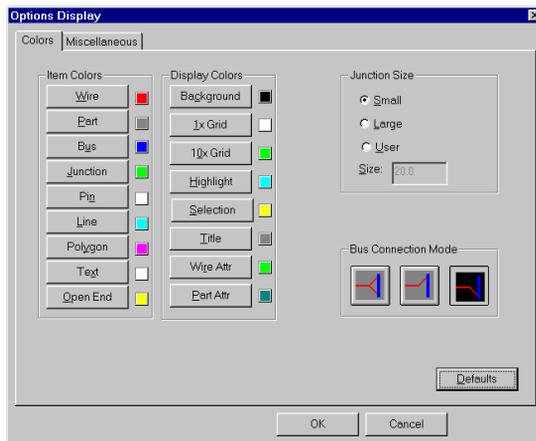
In P-CAD Schematic, you can display units in mils, mm, or inches. To choose a unit measurement scale:

1. Choose **Options » Configure** to open the *Options Configure* dialog.
2. In the Units frame, choose **mm** or **inch**. Notice that the Width and Height values in the Workspace Size frame change as P-CAD Schematic's 32-bit database converts all data to the measurement scale you choose.
3. Choose **mil** in the Units frame and click **OK** to close the *Options Configure* dialog.

## Defining Color Preferences

You can define a background color, distinguishing colors for different objects and more using the options in the *Options Display* dialog.

1. Choose **Options » Display** to open the *Options Display* dialog and click the **Colors** tab.



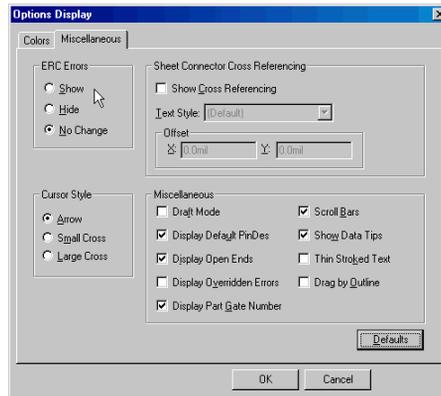
2. Click a button in the Item Colors or Display Colors frame to open a color palette.
3. Click the color swatch in the palette that you want displayed and you are returned to the *Options Display* dialog. The color you chose displays next to the Item or Display button.
4. From this **Colors** tab you can also change the junction size and bus connection mode. For this tutorial, we will use the default settings, so click **Defaults** and leave the *Options Display* dialog open.



## Setting Up Miscellaneous Display Options

To set up miscellaneous display options, follow these steps from the *Options Display* dialog.

1. Click the **Miscellaneous** tab.

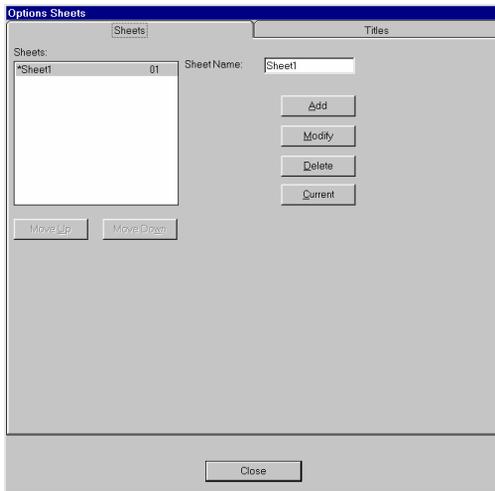


2. Select the following check boxes in the Miscellaneous frame:
  - **Display Default PinDes**
  - **Display Open Ends**
  - **Display Part Gate Number**
  - **Scroll Bars**
  - **Show Data Tips.**
3. To show the ERC error indicators in your design, select **Show** in the ERC Errors frame.
4. Choose **Arrow** in the Cursor Style frame and click **OK** to close the dialog.

## Adding a Sheet

A design will always have one sheet open. You can add additional sheets at any time during a design session. To add a sheet to a design:

1. Choose **Options » Sheets** to open the *Options Sheets* dialog and click on the **Sheets** tab.



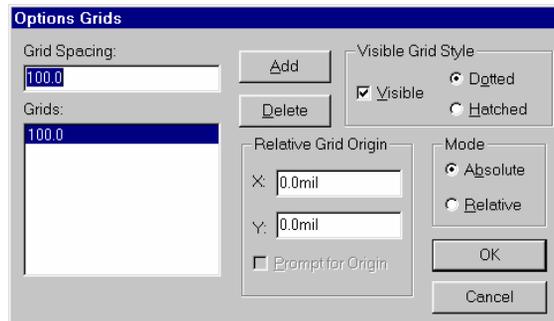
2. Type **Sheet2** in the Sheet Name box and click **Add**. Sheet2 appears in the Sheets list box.
3. Select **Sheet1** in the Sheets list box. If an asterisk (\*) does not appear next to Sheet1, click **Current** to define Sheet1 as the current sheet.
4. Click **Close** to exit the dialog. Notice that Sheet2 has been added to the list box on the status line at the bottom of the screen.
5. You can switch between sheets easily by selecting the required sheet name from the Sheet list box on the status line, or clicking on the empty box next to it to display the *Options Sheets* dialog and selecting the current sheet, or using the up and down arrows to scroll through the sheet list.



## Setting Up the Grid Spacing

You can set up a new grid spacing at any time during a design session. To add some new grids to your design, such as a 50 mil and a 25 mil spacing:

1. Choose **Options » Grids** to open the *Options Grids* dialog.



2. Type 50 in the Grid Spacing box and click **Add**. 50.0 is selected in the Grids list box.
3. Select **Visible** and **Dotted** in the Visible Grid Style frame. Select **Absolute** in the Mode frame and click **OK** to close the dialog. A dotted 50.0 mil grid appears in the workspace.
4. Another way to change the grid is to type the new grid spacing into the Grid Select list box on the status line. Type 25 and press **ENTER** to add the 25.0 grid spacing to the list. The 25.0 mil grid appears in the workspace.
5. To switch between the grid spacings you just created, select 50.0 from the Grid Select list box on the status line to switch to the 50.0 mil grid.
6. Choose **File » Save** to save the settings you have chosen during this tutorial.

## Zooming In and Out

You can use a number of commands to zoom in and zoom out on the schematic workspace.

1. Rest the mouse pointer over the area you wish to view. Press the **+** key to zoom in on your design. To zoom out, press the **-** key.
2. Create a zoom window by choosing **View » Zoom Window** or clicking  in the command toolbar. Draw a bounding outline around the region of the design you want to zoom in on. P-CAD Schematic zooms in on the region you select.
3. Choose **View » Extent** to view the extent of your design, that is, any objects placed.
4. Choose **View » All** to view the entire workspace, including the title border.

This concludes Tutorial 1. Please refer to Tutorial 2 to investigate working with Schematic objects.

## Tutorial 2 - Working with Schematic Objects

---

In this tutorial 'Working with Schematic Objects', we will explore how to place the objects that make up a schematic design, such as parts, wires, buses and ports. Off sheet connectors are also investigated to show you how to place and connect parts and nets across sheets. Once we have practiced some of the placement commands, we'll look at the various ways of selecting placed objects and modifying their properties and finish with some useful commands for working with nets and attributes.

For more information about the topics in this tutorial, press **F1** for online Help, or refer to the relevant chapter in this *User's Guide*.

### Object Placement

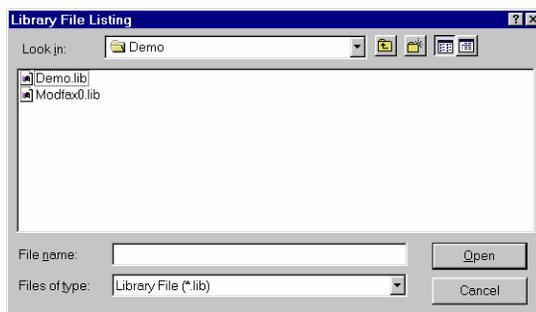
#### Placing Parts and Components

This section shows you how to place parts in your schematic design. A component contains logical and electrical data and can have one or more parts. A symbol shows a graphical representation only of the component and is used to display a part in a schematic design.

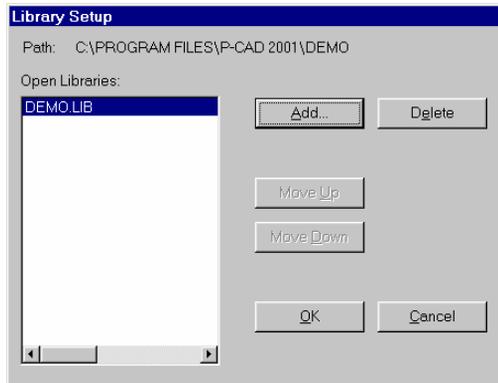
#### Opening a Library

A P-CAD library contains component and symbol information. Before you can place components or symbols in a schematic design, you must open the libraries that contain the parts you want to use. To set up a library to use:

1. Open `Design1.sch` created in tutorial 1, or create a new schematic design file by choosing **File » New**.
2. Choose **Library » Setup** to open the *Library Setup* dialog.
3. Click **Add** to open the *Library File Listing* dialog.



4. Select `Demo.lib`, located in the `Demo` folder in your P-CAD installation directory.
5. Click **Open**. The system path to the selected file appears next to the Path label and the name of the selected library appears in the Open Libraries list.



6. Click **OK** to close the *Library Setup* dialog.

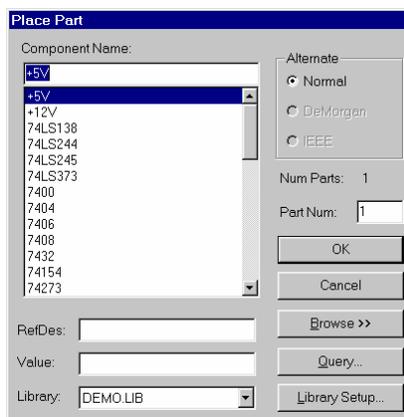
You can also use a simple drag-and-drop operation to open a library file in P-CAD Schematic. To do this, open the File Manager or Windows Explorer. Navigate to the folder that contains the file you want to open. Then, select the file, drag it to the P-CAD Schematic window, and release the mouse button to open the library.

### Placing Parts

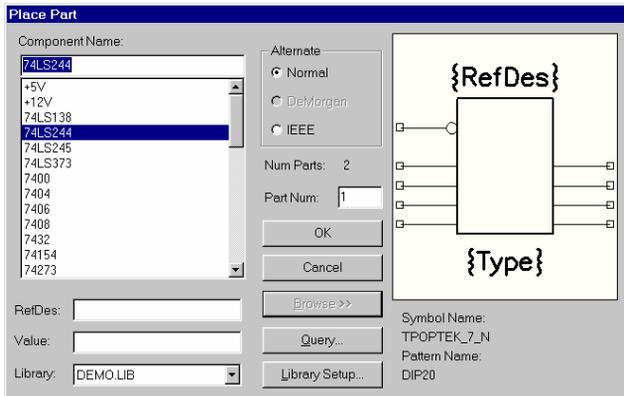
Now that the appropriate library is open, you can start placing parts.

1. Open the *Place Part* dialog using one of the following methods:
  - Choose **Place » Part**.
  - Click  in the placement toolbar.

The following figure shows you the collapsed view of the *Place Part* dialog.



- Click **Browse>>** to expand the *Place Part* dialog to display the symbol associated with the selected component.
- Select **74LS244** in the Component Name list box. As shown in the following figure, the number 2 appears next to the Num Parts label, to indicate that the component package contains two parts.



- Click **OK** to close the *Place Part* dialog. In the P-CAD Schematic workspace, a ghosted outline of the first part in the 74LS244 component package appears.
- To place the first part of the 74LS244 component package:
  - Move the mouse pointer to drag the ghosted outline of the part to the left of the workspace center.
  - As you drag the component across the workspace, the information box on the status line indicates that you will be placing the first part of the component package, RefDes U1:A.
  - Click the workspace to place the part. Notice that the information box now indicates that you will be placing RefDes U1:B, which is the second part of the component package.

A RefDes distinguishes each part in a component package and generally contains a prefix and a suffix.

The prefix typically contains a letter and number. For example, integrated circuits start with the letter U (e.g. U1), resistors (R1), capacitors (C1), connectors (J1), switches (SW1) and so on.

The suffix indicates that the part is a member of the component package. For example, the 74LS244 package contains two parts: U1:A and U1:B. The A indicates that the part is the first member of the U1 component package, and B indicates that it is the second part of the U1 package.

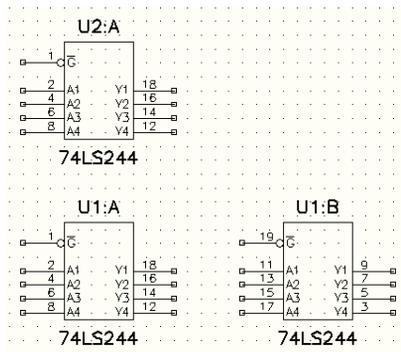
- To place the second part of the 74LS244 component package (U1:B), move the cursor to the right of U1:A. While you are positioning a part you can increase or decrease the numbering of the next part or RefDes using the following shortcut keys:

- To change the displayed part number to the next available value, press the **P** key. To select the next available reference designator, press **D**.
  - To return the RefDes to U1:B, press **SHIFT+P** until you return to the previous part number. Press **SHIFT+D** to select the previous available reference designator. Pressing these keys decrements the RefDes.
7. Click the workspace to place U1:B in your design.
  8. Right-click to quit placing the 74LS244 component package.

Notice that the **Place Part** button in the placement toolbar remains indented. This indicates that you are in placement mode. If you click the workspace, the *Place Part* dialog would appear so you could place another component type. Cancel out of part placement mode by selecting another tool.

9. Choose the Select tool using one of the following methods:
  - Press **S** or choose **Edit » Select**.
  - Click  in the command toolbar.
10. Select part **U1:A** in the schematic workspace again.
11. Hold down the **CTRL** key and drag the ghosted outline of the part so it is just above U1:A.
12. Release the mouse button to place a copy of the part. Notice that part is labeled U2:A, indicating it is the first part (A) of a new component package (U2).

Sheet 1 of your schematic design should now resemble the following illustration.



### Switching Between Sheets to Place Parts

This section shows you how to switch from Sheet1 to Sheet2, so you can place a part on Sheet2. Later in this tutorial we will wire these parts together and place sheet connectors.

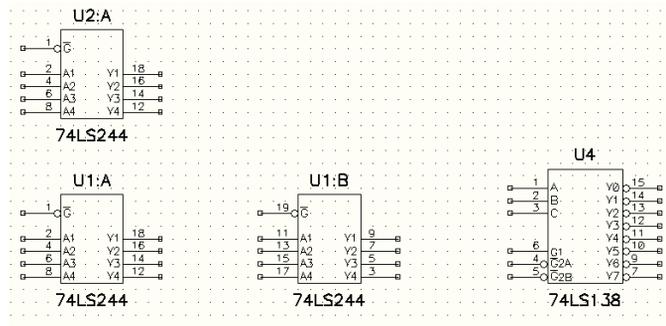
1. Select **Sheet2** from the Select Sheet list box in the status line.

2. Open the *Place Part* dialog using one of the following methods:
  - Choose **Place » Part**.
  - Click  in the placement toolbar.
3. Select **74154** from the Component list box. Notice that this component package has only one part.
4. Click **OK** to close the *Place Part* dialog. Notice that the information box on the status line indicates that you will be placing RefDes U3.
5. Place the part on the left side of Sheet2 by clicking the workspace.
6. Right-click to quit placing the 74154 component package.

Notice that the *Place Part* button in the placement toolbar remains indented. This indicates that the placement tool is active. If you clicked the workspace, the *Place Part* dialog would appear so you could select another component to place.

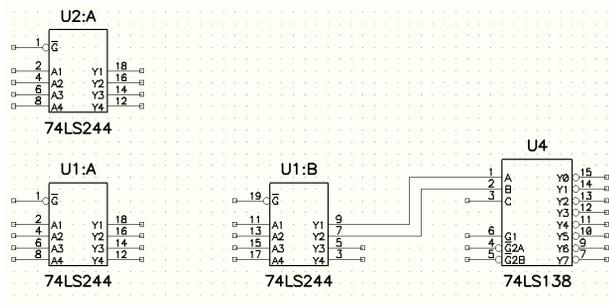
7. Select **Sheet1** from the Select Sheet list box on the status line.
8. Click the workspace to open the *Place Part* dialog.
9. Select the **74LS138** component from the Component list box.
10. Click **OK** to close the *Place Part* dialog. Notice that the information box on the status line indicates that you will be placing RefDes U4.
11. Place the U4 part to the right of U1:B by clicking the workspace.

Sheet1 of your schematic design should now resemble the following illustration.



## Placing Wires

This section shows you how to place wires in your schematic design. You will place wires in your design to connect the parts you have just placed as shown below.



1. Select the Place Wire tool using one of the following methods:
  - Choose **Place » Wire**.
  - Click  in the placement toolbar.
2. Hold down the left mouse button on U1:B pin 9 and drag the mouse pointer towards U4 pin 1. Click where you want to place vertices (corners).
3. While placing a wire, try selecting other orthogonal modes by pressing the **O** key. Also you can flip the wire by pressing the **F** key.
4. Release the mouse button when you reach U4 pin 1 to place the wire. Notice that the information box on the status line indicates that you have placed NET00000.

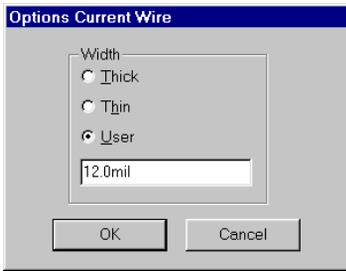
After you place a wire, the **Place Wire** button in the placement toolbar remains indented. This indicates that the wire placement tool is active. If you click the workspace, you would place another NET00000 wire segment.

5. Right-click or press **ESC** to stop placing wire segments for NET00000. The information box on the status line indicates that you will be placing NET00001.
6. Connect U1:B pin 7 with U4 pin 2. Then, right-click to stop placing NET00001. By default, P-CAD Schematic places wires with a 10.0 mil width.

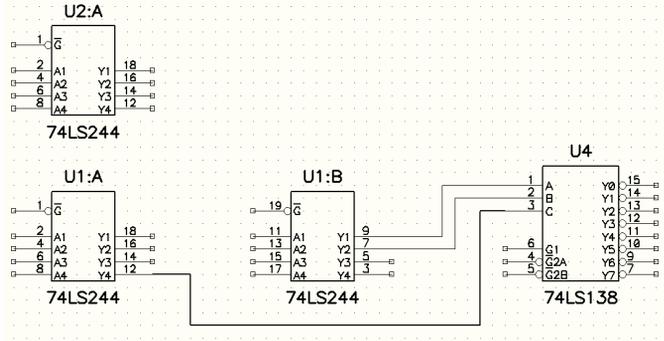
### Changing the Current Wire Width

The current wire width is the active width setting that is used when you place wires in a design. It is set for display purposes only. In P-CAD Schematic, the wire width is set to **Thin** (10.0 mil) by default. To change this setting:

1. Choose **Options » Current Wire** to open the *Options Current Wire* dialog.



2. Choose the **User** option button and type 12.0mil in the text box and click **OK**.
3. Connect U1:A pin 12 with U4 pin 3, as shown in the following figure. Then, right-click to stop placing NET00002. Sheet1 of your schematic design should now resemble the following illustration.

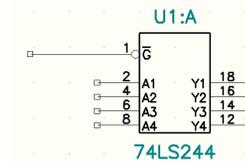


If you place a wire segment in the incorrect position, choose **Edit » Select** and then choose **Edit » Undo**. Then restart the wire placement tool by choosing **Place » Wire**. You can also add a new vertex to a single wire by selecting it, right-clicking and choosing **Add Vertex** from the menu.

### Placing an Open-ended Wire

With P-CAD Schematic, you can place a wire in a schematic design without starting the wire at a pin. To place an open-ended wire:

1. Choose **Place » Wire** or click  in the placement toolbar.
2. Click and hold down the left mouse button at the left side of the schematic workspace. Then, drag the pointer to U1:A pin 1 and release the mouse button to place the wire. A square appears on the open end of the wire, as shown.

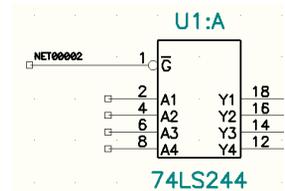


If a square does not appear on the open end of the wire, you must turn on the **Display Open Ends** feature found in the **Miscellaneous** tab of the *Options Display* dialog.

3. Press **S** as a shortcut for choosing **Edit » Select**.
4. Select the wire you just placed. Then, right-click and choose **Properties** from the shortcut menu to open the *Wire Properties* dialog.
5. Select the **Display** check box.

At this time, you could rename the wire using the options in the **Net** tab of the *Wire Properties* dialog. For this tutorial, leave the default name that appears in this field.

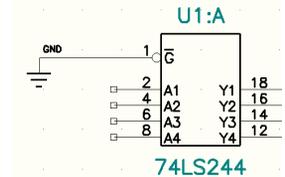
6. Click **OK** to close the *Wire Properties* dialog. Use the zoom in command (+) to see the net name on the wire.
7. Press the **SHIFT** key and select the net name that appears above the wire. When the selection box appears, release the **SHIFT** key and drag the net name to the left. Then, release the mouse button to place the net name in its new position, as shown.



### Naming Nets by Placing Power Parts

Power parts have an automatic net naming feature. When you place a power part in a schematic design and attach it to a net, the net name changes to the name defined by the power part. To automatically name a net by placing a power part in your design:

1. Open the *Place Part* dialog by choosing **Place » Part** or click  in the placement toolbar.
2. Select **GND** from the Components list box and click **OK** to close the *Place Part* dialog.
3. Click the open end of the wire to place the GND part. The open end symbol disappears, to indicate that the connection is complete.



As shown, the net name of the wire changes to GND because the GND component is a power part.

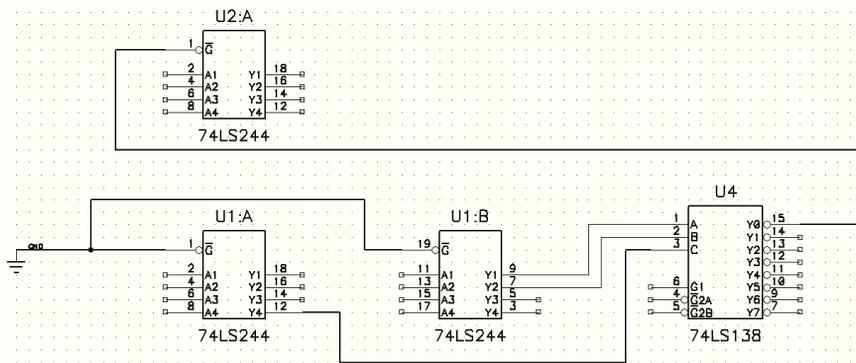
### Automatic Junction Placement

When you connect a wire to another wire, P-CAD Schematic automatically places a junction at the connection point. To see how P-CAD adds junctions:

1. Rest the mouse pointer on the ground wire you have just placed.
2. Choose **Place » Wire** and draw a connection to U1:B pin 19.

Notice that a junction is automatically placed and the new wire is added to the ground net. Junction size can be changed by choosing **Options » Display**.

3. Connect U4 pin 15 with U2:A pin 1. Draw the connection as shown in the following figure, so you can later see how to move multiple items.



4. Choose **File » Save**.

### Placing Buses

To place a data bus in your schematic design:

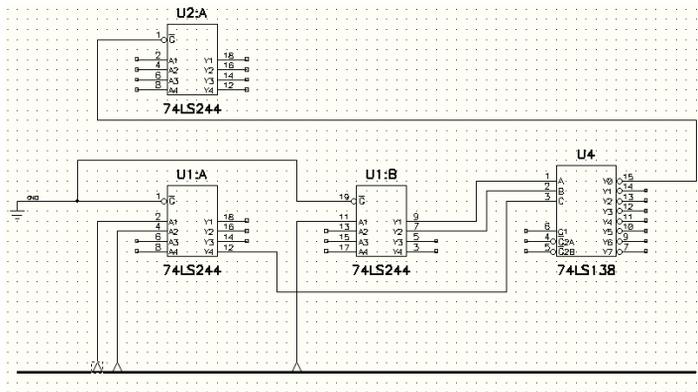
1. Select the bus placement tool by using one of the following methods:
  - Choose **Place » Bus**.
  - Click  in the placement toolbar.
2. Hold down the mouse button and drag the pointer across the workspace to draw a horizontal bus below the parts you've placed in your design.
3. Right-click to end bus placement.

### Connecting Wires to a Bus

To connect wires to the bus just placed:

1. Choose **Place » Wire** or click  in the placement toolbar.
2. Place three wires to connect the bus to the following pins: U1:A pin 2, U1:A pin 4 and U1:B pin 11.

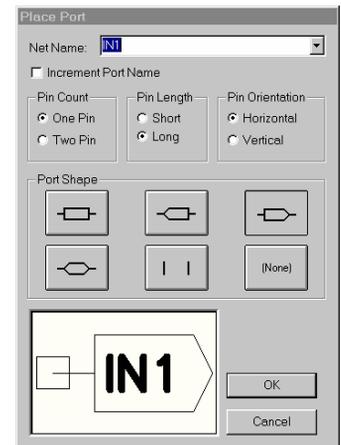
You can start or end the wires anywhere along the bus. Sheet 1 of your design should now resemble the following diagram.



## Placing Ports

Ports identify unconnected subnets of a single net and give you the ability to explicitly identify subnets on one or more sheets. Placing ports in a design also prevents unintentional net merges. To place a port:

- Open the *Place Port* dialog using one of the following methods:
  - Choose **Place » Port**.
  - Click  in the placement toolbar.
- Click in the workspace and the *Place Port* dialog displays.
- Type IN1 in the Net Names box and clear the **Increment Port Name** check box.
- Click  in the Port Shape frame. Ports with two connections are placed in-line and those with only one connection are attached to the left, right, above or below the wire. Click **OK** to close the dialog.
- Hold down the left mouse button and drag the port to the wire connecting U1:A pin 2. Release the mouse button to place it.



If you need to rotate the port during placement, press **R**. To flip the port, press **F**.

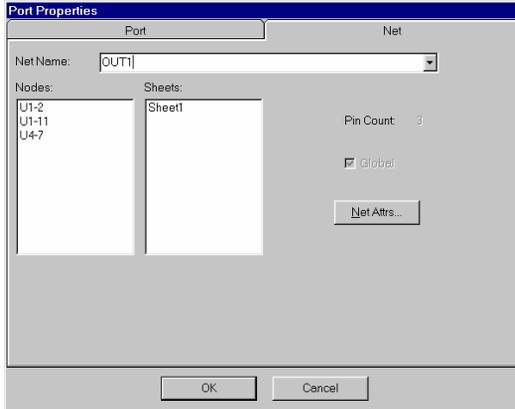
- Place another port on the wire connecting U1:B pin 11 with the bus by clicking the desired location.
- Choose another command, e.g. **S** for **Edit » Select**, to quit port placement mode.

### Placing Ports to Continue a Net on Another Sheet

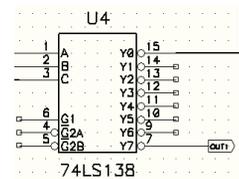
To create global nets that continue from one sheet to another, you need to create a port through which the wires of the net will pass to the next sheet. You can also then add annotated sheet connectors that show the connecting sheet's name and the zone location.

First of all, we'll create the port for a net we will name OUT1 that goes from Sheet1 to Sheet2.

1. Choose **Place » Port** or click  in the placement toolbar to open the *Place Port* dialog.
2. Click  in the Port Shape frame and select **Two-pin** in the Pin Count frame.
3. Choose **Horizontal** in the Pin Orientation frame and click **OK**.
4. Click U4 pin 7 to place the port and press **S** for **Edit » Select** to end port placement.
5. **Double-click** on the port you just placed to open the *Port Properties* dialog and click the **Net** tab.



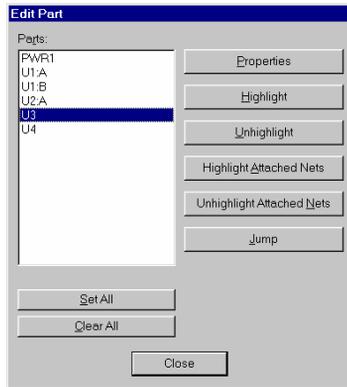
6. Rename the net by typing **OUT1** in the Net Name box and click **OK**. The new net name appears inside the port symbol.
7. Hold down the left mouse button and drag the ghosted outline of the port away from the pin. Then, release the mouse button to place the port. Notice a wire is automatically added to fill the gap, as shown.



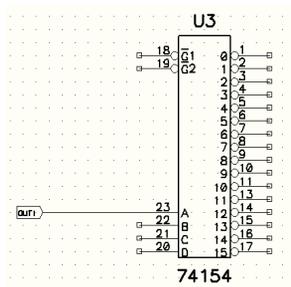
### Connecting a Net to a Part on Another Sheet Using a Port

Now we will place a port to connect the continued net OUT1 on Sheet2 to component U3. You could switch to Sheet2 by selecting this sheet from the Sheets list box on the status line.

Alternatively, you can jump to Sheet2.



1. Press **S** as a shortcut for choosing **Edit » Select** and then choose **Edit » Parts** to open the *Edit Part* dialog.
2. Select **U3** in the Parts list and click **Jump**. The mouse pointer rests on the reference point for U3 on Sheet2.
3. Choose **Place » Port** or click  in the placement toolbar. Click in the workspace to open the *Place Port* dialog. Then, define the characteristics of the port using the following specifications:
  - Type **OUT1** in the Net Name box.
  - Click  in the Port Shape frame and select **Two-pin** in the Pin Count frame.
  - Choose **Horizontal** in the Pin Orientation frame and click **OK**.
4. Place the port at U3 pin 23.
5. Press **S** as a shortcut for choosing **Edit » Select** and select and drag the port to the left, away from the pin. Sheet2 of your schematic design should now look like this.



Let's quickly check that net OUT1 does connect across the sheets. To do this:

1. Choose **Edit » Parts** to open the *Edit Parts* dialog.
2. Select **U3** from the Parts list box.

- Click **Highlight Attached Nets** and click **Close** to exit the dialog. Notice the current highlight color is applied to OUT1 on both Sheet1 and Sheet2.

### Using Annotated Sheet Connector Symbols

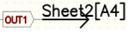
You can place an annotated sheet connector to indicate that the net continues on another sheet. Annotated sheet connectors show the connecting sheet's name and zone location using P-CAD Schematic's zone intelligence, so you must have a sheet border with zones displayed for this feature to work. Also **Show Cross Referencing** must be selected in the *Miscellaneous* tab of the *Options Display* dialog.

For information about using sheet borders with zones references, refer to P-CAD Schematic Tutorial 1 – *Setting up a Schematic Design*.

First we must place in and out sheet connector symbols that are supplied in `Demo.lib`.

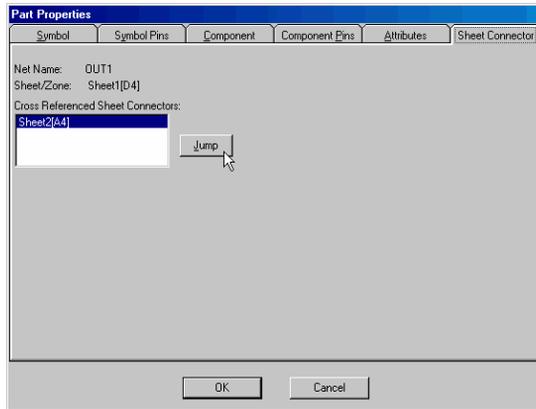
- Go to Sheet1 and zoom in around the port named OUT1. Set your grid spacing to 25mil at this stage for accurate placement.
- Choose **Place » Part** or click  in the placement toolbar to open the *Place Part* dialog.
- Select **SHEETOUT**  from the Component Name list box and place it on the spare pin at the end of the port OUT1 connected to part U4.
- Press **S** as a shortcut for choosing **Edit » Select** to exit out of part placement mode.
- Now go to Sheet2 and place a part named **SHEETIN**  on the spare pin of the port named OUT1 connected to part U3.

To display the annotated sheet connectors:

- Choose **Options » Display** and click the **Miscellaneous** tab of the *Options Display* dialog.
- Select the **Show Cross Referencing** check box in the Sheet Connector Cross Referencing frame and click **OK**. The sheet connectors display in the workspace, e.g.  which means this net continues on Sheet2 at zone location A, 2. On Sheet2, the sheet connector will display the cross reference back to Sheet1, e.g. .

When you display cross references, all sheet connectors on all sheets are annotated. If a net's location is moved or if a sheet connector is added or deleted from the net, the sheet connector cross references for all sheet connectors associated with that net are automatically updated.

You can use these annotated sheet connectors to jump to its cross reference on another sheet by double-clicking on the **SHEETIN** or **SHEETOUT** part and clicking on the **Sheet Connector** tab in the *Parts Properties* dialog.

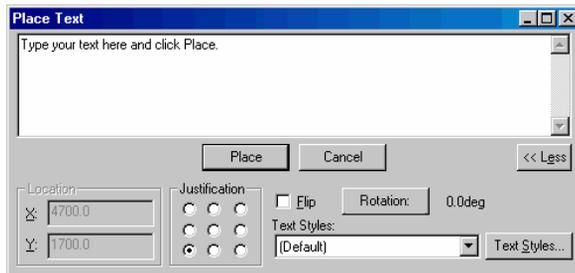


3. Select the **Cross Referenced Sheet Connector** for the sheet you wish to go to and click on **Jump**. You are taken to the selected sheet and the cursor displays on the cross-referenced sheet connector.

### Placing Text

To place text in your schematic design:

1. Choose **Place » Text** or click **A** in the placement toolbar. A positioning cursor  $\times$  appears.
2. Click on the workspace where you want the text to be positioned and the *Place Text* dialog displays. Click on **More»** to expand the dialog, if necessary.



3. Type in the text and choose a predefined style from the *Text Style* list.
  4. Click **Place** to display the text on the schematic design.
  5. Press **S** as a shortcut for choosing **Edit » Select** to exit out of text placement mode.
- For information about creating new text styles, refer to *Options Text Style*, (page 314).  
Now we have placed a few objects, we will look at the various ways of selecting them.

## Selecting Objects

This section of the tutorial shows you how to select single and multiple objects. We'll be practising on the parts and wires that you have placed earlier in this tutorial.

If the **Show DataTips** feature is enabled in the **Miscellaneous** tab of **Options » Display**, a Data Tip appears when you move the cursor over an object in your design. Data Tips show context-sensitive information about design objects and can assist you in selecting the appropriate object.

### Selecting Single Objects

To select a single object:

1. Press the **+** (plus) key to zoom in on your workspace. This makes viewing objects easier.
2. Press **S** or click the **Select** button  on the command toolbar as a shortcut for choosing **Edit » Select**.
3. Click an object to select it. The object will display in the selection color (set in **Options » Display**) with a bounding box around it.
4. To deselect the object, click in a blank space in your workspace to cancel the selection.

### Selecting Multiple Objects

To select multiple objects:

1. Select a part. Hold down the **CTRL** key and select another part (**Ctrl+Click**). Notice that both parts are selected and the bounding box includes all selected parts.
2. Click a blank space to cancel the selection of the parts.

### Subselecting Objects

You can select objects that make up a part or component. For example, you can select a pin that belongs to a component. To subselect an object:

1. Place a component in the workspace using the **Place » Component** command.
2. Press **S** or click the **Select** button on the command toolbar as a shortcut for choosing **Edit » Select**.
3. Hold down the **SHIFT** key and click on a pin that belongs to the component. Notice that only the pin is selected.
4. Now, you can perform actions on the pin. For example, you can right-click the pin and choose **Properties** from the shortcut menu to query the pin.

### Block Selecting Objects

To select a group of objects, you can perform a block select. To block select:

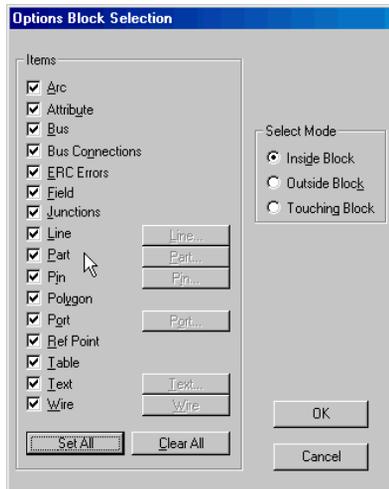
1. Hold down the mouse button and drag the cursor across your workspace to draw a bounding outline around two or three parts.

2. Release the mouse button to select the parts.
3. Deselect the parts by clicking in a blank area of the workspace.

### Block Selecting Wires

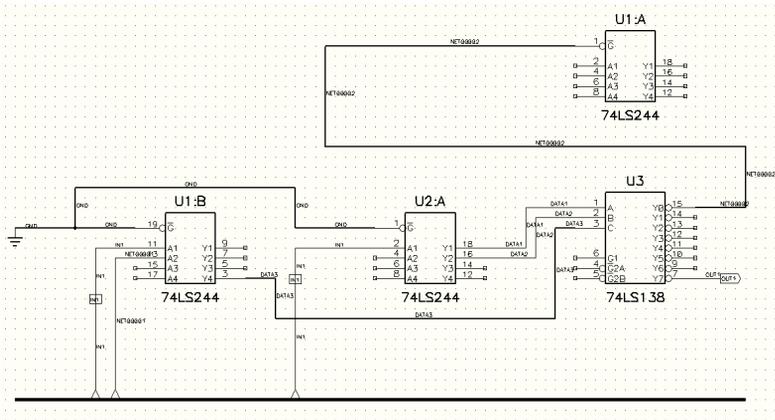
To change your block selection criteria to select all the wires in your design so we can display their net names:

1. Choose **Options » Block Selection** to open the *Options Block Selection* dialog.



2. Click **Clear All** to clear the default selection of all items. All of the item check boxes will become blank and the item buttons become shaded.
3. Click the **Wire** check box until a solid check mark appears.
4. Choose **Outside Block** in the Select Mode frame and click **OK**.
5. Draw a bounding outline in an empty area of the workspace. Release the mouse button to select all of the wires in your design.
6. Right-click and choose **Properties** from the shortcut menu to open the *Wire Properties* dialog and click the **Wire** tab.
7. Click the **Display** check box until a solid check mark appears.

At this point, your design should look like the following diagram:

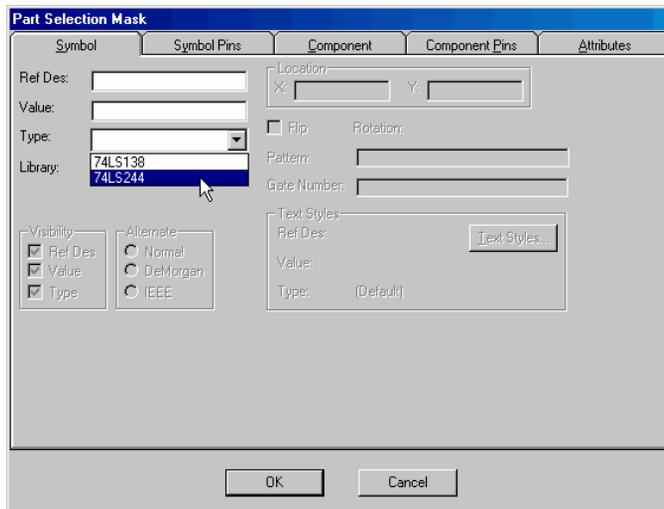


8. Click **OK** to close the dialog. Notice that net names now appear on all wires.

### Block Selecting Using a Selection Mask

You can define block selection criteria to specify which particular objects are selected when you perform a block select. This time, we will block select only the 74LS244 components, we placed earlier by using their Type to define the selection criteria.

1. Choose **Options » Block Selection**. The *Block Selection* dialog appears. The enable/check boxes have three states: checked (included); blank (excluded) or shaded (masked with additional selection criteria).
2. Click the **Clear All** button to clear the default selection of all items.
3. Click the **Parts** check box until a shaded check mark appears. When the **Parts** button becomes available, click **Parts**. The *Part Selection Mask* dialog appears.



4. Type 74LS244 in the Type text box or choose it from the Type list.
5. Click OK to save the changes and return to the *Options Block Selection* dialog.
6. Select **Inside Block** as the Select Mode and click **OK** to set your selection criteria.
7. Choose **View » Extent** to make sure the entire design is displayed in the workspace.
8. Draw a bounding outline around the entire design. When you release the mouse button, notice that only the 74LS244 parts are selected.

### Restoring the Block Selection Criteria

Remember to clear your filter when you have finished using Block Selection. To restore your block selection criteria:

1. Choose **Options » Block Selection** to open the *Options Block Selection* dialog.
2. Click **Set All** to select all of the check boxes in the Items frame. Choose **Inside Block** in the Select Mode frame and click **OK**.

### Selecting Highlighted Objects

You can select objects that have already been highlighted on screen with the highlighter feature.

1. First we will highlight a few objects by selecting them, right-clicking and choosing **Highlight** from the shortcut menu. Deselect the objects and they will display in the Highlight color set using **Options » Display** (default color is turquoise).
2. Now to select the highlighted objects again, choose **Edit » Select Highlighted**. Only the objects highlighted in step 1 will be selected.

Notice that the **Edit » Select Highlighted** command selects all highlighted objects, regardless of the highlight colors that have been applied to the objects in a design.

3. Right-click and choose **Unhighlight** from the shortcut menu.
4. Click anywhere in the workspace. Notice that the highlight color is removed from all selected objects.

To remove the highlight color from all objects, you can also choose **Edit » Unhighlight All**.

### Selecting Collocated Objects

When objects overlap so that you cannot select any one of them, you can switch between the collocated objects to select the correct one.

1. Add some objects that are collocated (in the same position) and position the cursor on top of them.
2. Click the left mouse button to toggle between objects. Information about the currently selected item displays in the status line, e.g. 'Part U1:A selected'.
3. Delete these objects by selecting them and pressing the **DELETE** key.

Now that we have gone through the different selection modes, it's time to look at what can be done with objects when they are selected.

## Modifying Objects

This section of the tutorial covers moving, rotating, resizing and changing properties of placed objects.

### Moving Objects

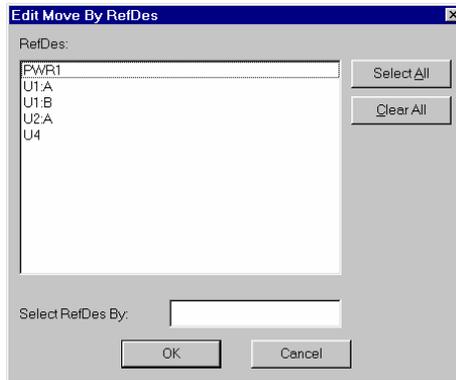
1. Select the object(s) to be moved.
2. Click on the object (or within the bounding box of several selected objects) and drag the cursor to the new location.

The following topics show you the various methods you can use to move parts in a design.

### Moving Parts by RefDes

To move parts by Reference Designator:

1. Choose **Edit » Move by RefDes** to open the *Edit Move by RefDes* dialog.



2. Select **U2:A** from the RefDes list box and click **OK**.
3. Hold down the left mouse button and drag the ghosted outline of U2:A so it rests to the right of its current position.
4. Release the mouse button to place the part in its new position.

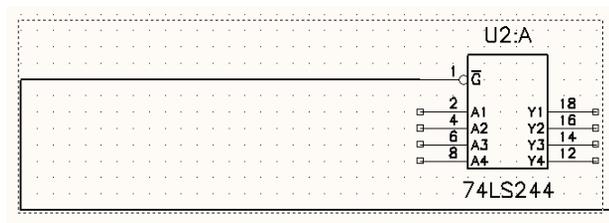
Notice that U2:A moves to the right and that the connections are maintained. In addition, *Edit Move by RefDes* dialog opens again, so you can move another part.

5. Click **Cancel** to close the *Edit Move by RefDes* dialog.

### Moving Parts by Block Selecting

To move parts using the block selection feature:

1. Draw a bounding outline around U2:A and the two wire segments closest to pin 1, as shown in the following figure.



If your bounding outline does not select one of these items, hold down the **CTRL** key and select the missing items. Then, hold down the **CTRL** button and complete step 2.

2. Drag the selected items to the right so that U2:A is above U4. Notice that the long horizontal segment shortens automatically, maintaining connectivity.

### Moving Components and Wire Segments

This section shows you how to move a component that is attached to a wire segment.

1. Select **U2:A** and move it up one or two grid points. Notice that the wire becomes non-orthogonal.
2. Restore the orthogonal by selecting the diagonal wire segment.
3. Drag the left endpoint up one or two grid points to create a 90-degree corner.

To avoid creating a diagonal, you could have selected both the wire and U2:A before moving the part upwards. To select multiple items, hold down the **CTRL** key and select the items of your choice.

### Moving Wires and Maintaining Connectivity

To see the connectivity maintained when moving wires:

1. Select a vertical wire segment in your design.
2. Drag the wire to the left and back to the right. Notice that connectivity is maintained.

Remember that when moving items, you can always undo the action by pressing the **U** key, choosing **Edit » Undo**, or by clicking the **Undo** button on the command toolbar.

### Resizing Objects

Selected objects can be resized by selecting and moving their editing handles.

1. Select an object in your design, e.g. a wire or bus.
2. Clicking on an editing handle and drag the cursor (with the outline of the object) to stretch the object.

### Rotating and Flipping Objects

Selected objects can be rotated and flipped.

1. Select an object and rotate it by 90 degrees counterclockwise by pressing the **R** key.
2. Flip the object in the X direction (i.e. about the Y axis) by pressing the **F** key.

Placed objects with connections will attempt to reconnect each time, so pressing the left mouse button while you choose **R** or **F**, will stop reconnections until you release the mouse button.

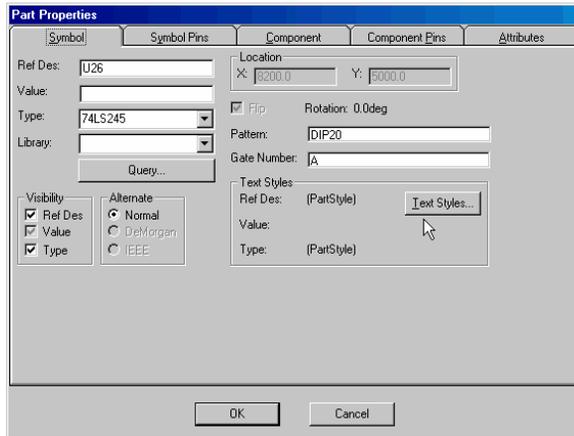
### Changing Properties

This section looks at assessing and changing an object's properties.

1. There are a couple of ways to display an object's Properties dialog:
  - Double-click on an object

- Select an object and choose **Properties** from the right-click menu
- Select an object and choose **Edit » Properties**.

A Properties dialog displays, such as the *Parts Properties* dialog below.

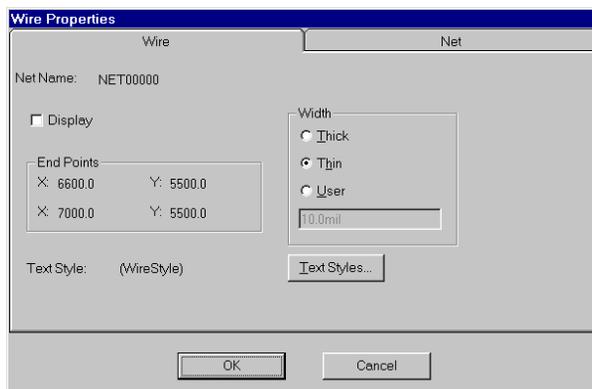


2. Make any necessary changes and click **OK**.

### Changing the Width of a Selected Wire

You can change the thickness of a selected wire by changing its properties.

1. Select the first wire segment of NET00001.
2. Right-click and choose **Properties** from the shortcut menu to open the *Wire Properties* dialog. As shown in the following figure, the **Wire** tab is selected for you by default.



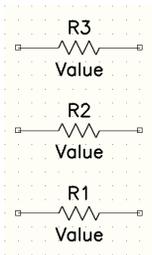
3. Choose the **Thick** option in the Width frame to change the wire thickness to 15.0 mil and click **OK**. The displayed width of the selected wire changes to 15.0 mil.

4. Select the wire segment again. Right-click and choose **Properties** from the shortcut menu to open the *Wire Properties* dialog.
5. Restore the 10.0 mil wire width by choosing the **Thin** option button in the Width frame and clicking **OK** to close the *Wire Properties* dialog.

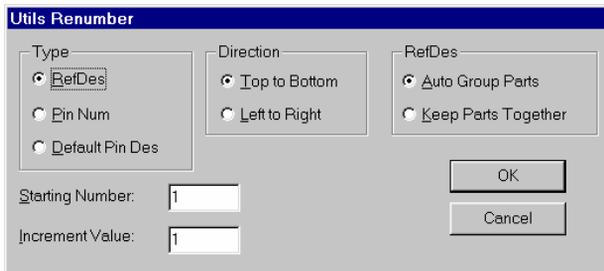
### Renumbering Reference Designators

Before you verify your design and generate output, you should renumber your reference designators (RefDes). To see how renumbering works with parts:

1. Place three RES500 resistors in the workspace, as shown below.



2. Press **S** as a shortcut for choosing **Edit » Select**.
3. Choose **Utils » Renumber** to open the *Utils Renumber* dialog.



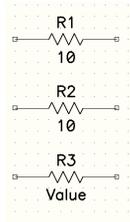
4. Choose the following settings:
  - Choose **RefDes** in the Type frame.
  - Choose **Top to Bottom** in the Direction frame.
  - Choose **Auto Group Parts** in the RefDes frame.
5. Click **OK** to close the *Utils Renumber* dialog. A warning displays a message that states this operation is undoable. Click **Yes** to continue.
6. Notice that the reference designators have been renumbered in top-to-bottom order within prefix types.

In particular, your resistors are renumbered starting with R1, and the ICs are renumbered starting with U1. Notice also, renumbering of U3 and U4 was done across sheets.

### Adding Values after Placing Parts

You can add values after you place a part. To practice this:

1. Select **R1** and **R2**.
2. Right-click and choose **Properties** from the shortcut menu to open the *Properties* dialog and click on the **Symbol** tab.
3. Type 10 in the Value box and click **OK**. Notice the field results for the value field changes to "10" for each selected resistor.



### Deleting Objects

1. Select all of the RES500 parts you placed.
2. Press **DELETE**, or right-click and choose **Delete** from the shortcut menu. The resistors are removed from your design.

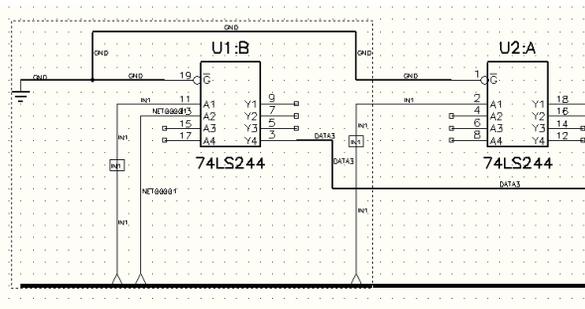
### Copying and Pasting Objects

This section investigates copying and pasting objects and circuits.

#### Copying a Group of Objects Using Drag-and-Drop

To copy the group of selected objects :

1. Block select the section of your circuit as shown below.



2. Choose **View » All**.
3. Hold down the **CTRL** key and drag the selected group of objects to an empty part of the workspace and release. Notice that you can view the exact distance of the move in the X and Y coordinates boxes on the status line.
4. Zoom in to view the copied block by pressing the **+** key.

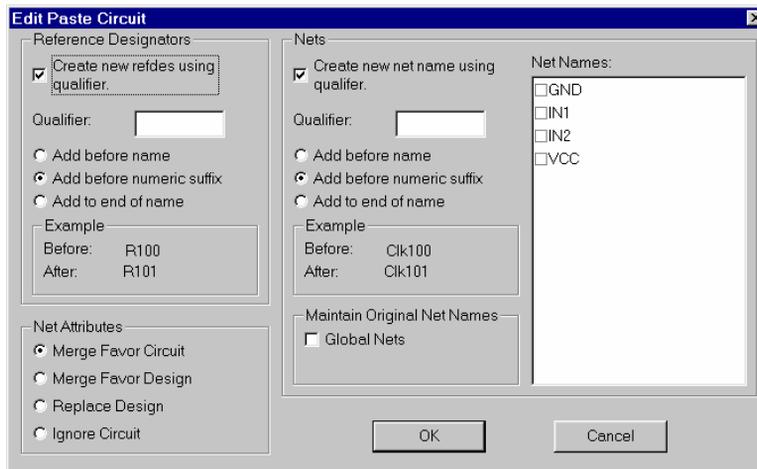
5. Probe the wires in the group of copied objects by selecting a wire and choosing **Properties** from the shortcut menu. You will notice the following about copied items:
  - Global nets maintain their original names. Global nets include nets with ports, e.g. IN1, nets attached to power parts, e.g. GND, and hidden power nets.
  - Nets that are not global are assigned unique default system names, e.g. NET00002.
  - Components are given unique reference designators.

### Copying Circuitry

Suppose a section of circuitry you have completed previously could be used in more than one area of your design. In this situation, you could copy and paste the desired data, including net names, net attributes and net information.

To copy and paste a section of circuitry:

1. Add a new sheet to the design and name it *Sheet3*.
2. Select the group of objects that you just copied using the drag-and-drop method.
3. Copy this group of parts to the clipboard using one of the following methods:
  - Choose **Edit » Copy**
  - Right-click and choose **Copy** from the shortcut menu.
4. Go to Sheet 3 and choose **View » All**.
5. Choose **Edit » Paste » Circuit** to open the *Edit Paste Circuit* dialog.



6. Select the **Create new refdes using qualifier** check box. This appends the specified characters to all the components in the copied data.

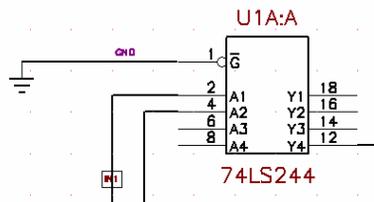
7. Type the characters to append to each RefDes in the Qualifier edit box, e.g. type `A`.
8. Choose the **Add to end of name** option to append `A` to the end of the RefDes name.  
Now you can select the nets whose names you want to modify and specify the type of change you want to apply.
9. Select the **Create new net name using qualifier** check box in the *Nets* frame and type `_A` in the Qualifier box.
10. Choose the **Add before numeric suffix** button. In the Example frame, notice that your qualifier (`_A`) will be placed after the `T` in `NET` and before the first zero in the net name (i.e., `NET_A00001`).
11. In the Maintain Original Net Names frame, select the **Global Nets** check box. This maintains the current names for all global nets and also selects all of the global nets in the Net Names list box. In this tutorial, all the nets are global. Optionally, you can choose individual nets to maintain their original names by checking the box next to each net.
12. If you were copying the circuitry from another design where net attribute values may be different than the values of the same attributes in the destination file, you must choose how to handle the merge from the options in the Net Attributes frame.

Since this copy and paste operation is within the same design and one of the options must be enabled, select the **Merge Favor Design** check box. This option keeps all of the net attribute values as they currently exist.

### Pasting the Copied Circuitry

When all selections have been made in the *Edit Paste Circuit* dialog:

1. Click the **OK** button to display the positioning cursor  $\times$  in the workspace of Sheet3.
2. Click the left mouse button and the ghosted outline of the copied items appears.
3. Drag the objects to their destined location in the middle of the sheet and release the mouse button. The copied data is placed in the workspace as shown below.



4. Notice that the RefDes and net names are modified with the qualifiers that you added. Each time you paste this same circuitry the qualifier is incremented to the next available name for each component or net name.
5. To quit using the **Paste** command, right-click, or press the **ESC** key, or choose another command.

## Working with Your Design

This section takes a closer look at some options you may need to use when working with nets and attributes. Before you start, make sure `Design1.sch` is the current design.

### Working with Nets

Now we'll have a look at some commands you may need when working with nets in a design.

#### Selecting a Subnet or an Entire Net

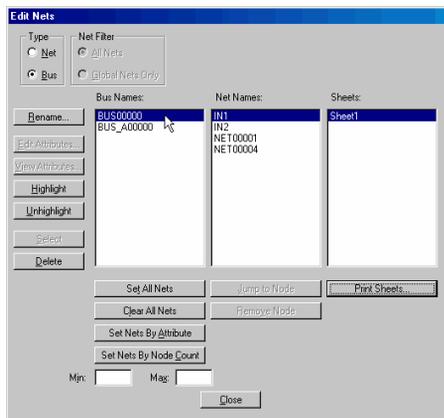
To select a subnet or an entire net:

1. Select any segment on the IN1 net, which is the net that attaches U1:B to the bus.
2. Right-click and choose **Select Contiguous** from the shortcut menu to select the physically connected wires only.
3. Right-click and choose **Select Net** from the shortcut menu. Notice now that the entire net is selected.

#### Viewing Nets in a Bus

If you ever need to recall what nets are attached to your bus, this information is readily available. To view the nets in a bus:

1. Choose **Edit » Nets** to open the *Edit Nets* dialog.



2. Select **Bus** in the Type frame.
3. Select **BUS00000** from the Bus Names list. Notice that a list of attached nets appear in the Edit Nets box and click **Close** to exit the *Edit Nets* dialog.

### Renaming Nets

This section shows you how to rename the nets you have already placed.

1. Choose **Utils » Rename Nets**. Click the workspace to open the *Utils Rename Wire/Port* dialog.

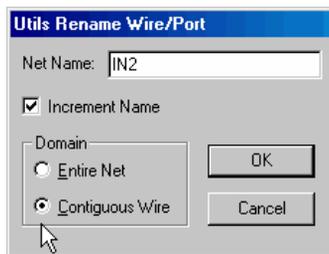


2. Type the following text in the Net Name box: `DATA0` . Select the **Increment Name** check box and click **OK** to close the dialog.
3. Click each net in your design to name them `DATA1` , `DATA2` and `DATA3` respectively.
4. Right-click or press **ESC** to stop renaming nets. The *Utils Rename Nets* tool remains active, so select another command to quit this mode, e.g. **S** for **Edit » Select**.

### Splitting a Net

To split a net, we can place parts or delete sections of a net.

1. Place a **RES500** resistor on the horizontal GND wire directly above U1:B.
2. Probe the wires on both sides. Note that the GND net is split.
3. Select the vertical **GND** wire at the junction and press **DELETE**. Notice that the GND net is split again.
4. Choose **Utils » Rename Nets** and click the workspace to open the *Utils Rename Wire/Port* dialog.



5. Type `IN2` in the Net Name box, choose **Contiguous Wire** in the Domain frame and click **OK**.
6. Click the `IN1` wire on the right side of the object group. Notice only the selected subnet is renamed.

### Working with Attributes

P-CAD Schematic provides you with predefined attributes that can be associated with nets or components or used to add extra data to the design, e.g. physical or manufacturing information.

You can also create user-defined attributes. All or selected attributes can be included in a customized Attributes report generated using **File » Reports**.

Attributes values can be simply displayed on a design using the **Place » Attribute** command. If you are creating your own symbols and require attributes such as ResDef, Type and Description of a part to be visible, you would place the attributes this way. Component attributes, such as Description or Reference to a web site, can be made visible on placed parts when added through the **Attributes** tab of the *Properties* dialog of a selected component.

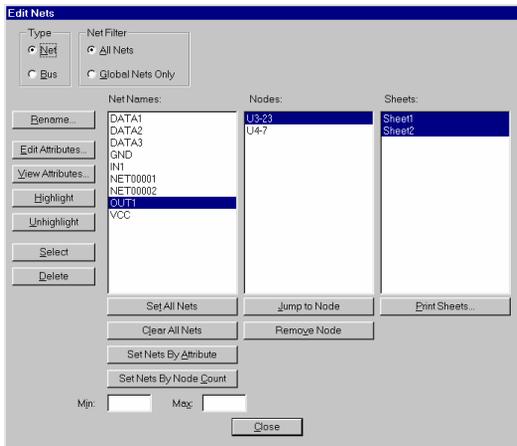
Attributes, however, are a far more powerful feature when set to define design rules. By defining the value of some attributes, e.g. Net, Clearance or Physical attributes, a design rule for each attribute is automatically created, e.g. defining the clearance rules that will apply when routing the board in P-CAD PCB. These design rules are stored with the design and can then be passed to P-CAD PCB via the netlist.

For more information about attributes, refer to *Place Commands*, (page 259).

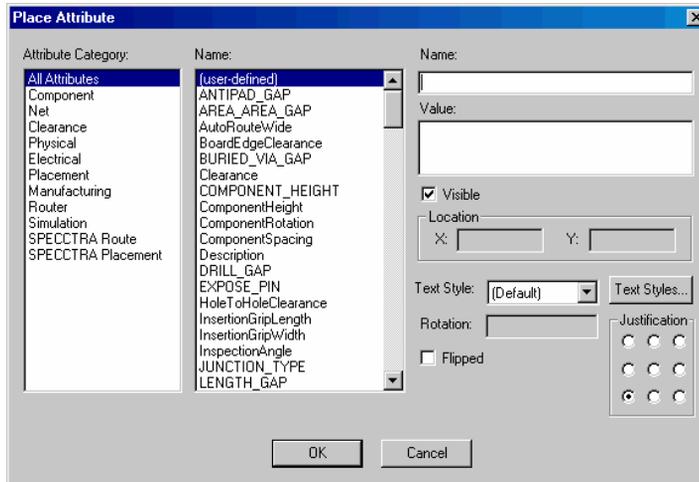
### Adding Net Attributes

In the example we will use below, a Width attribute of 20mil will be set for the net OUT1. Setting such an attribute will also create a Width design rule for this net. This design rule will automatically set the width of all tracks in OUT1 to 20 mil when you route the board in PCB using the netlist generated from this schematic design.

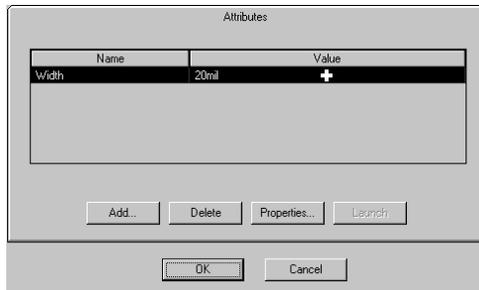
1. Choose **Edit » Nets** to open the *Edit Nets* dialog and click **OUT1** in the Net Names list box.



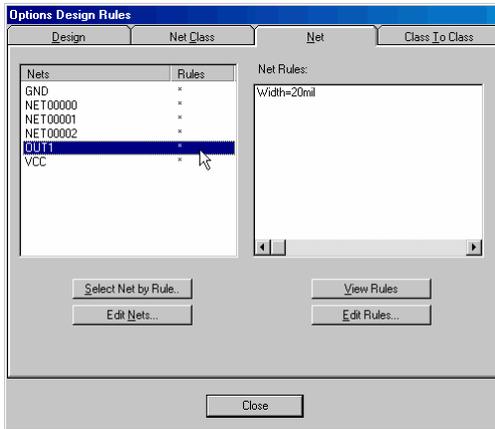
2. Click **Edit Attributes** to open the *Attributes* dialog. Attributes can be added or deleted through this dialog. Click **Add** to open the *Place Attribute* dialog.



3. Select **Net** from the Attribute Category list box. A list of attributes associated with that category appears in the Names list box.
4. Select **Width** from the Names list box and type `20.0mil` in the Value box and click **OK**. The attribute name and value appears in the *Attributes* dialog.



5. If you need to modify a selected attribute, click **Properties** from this dialog to open the *Attribute Properties* dialog again. Click **OK** until you return to the *Edit Nets* dialog and click **Close**.
6. Now to check that the design rule has been created, choose **Options » Design Rules**, click on the **Net** tab and select **OUT1**. The Width rule appears in the Net Rules frame.



7. Click **Close** and save your design file by choosing **File » Save**.

This concludes Tutorial 2. Please refer to Tutorial 3 to learn how to verify a schematic design.

## Tutorial 3 - Verifying a Schematic Design

Welcome to the P-CAD Schematic tutorial 'Verifying a Schematic Design'. During this tutorial, we will check that a design is valid according to the Electrical Rules Check (ERC). The ERC utility will find any errors in the design, such as unconnected pins or wires, or rule violations. This tutorial uses a demonstration file that is supplied with P-CAD.

For more information about the topics in this tutorial, press **F1** for online Help, or refer to the relevant chapter in this User's Guide.

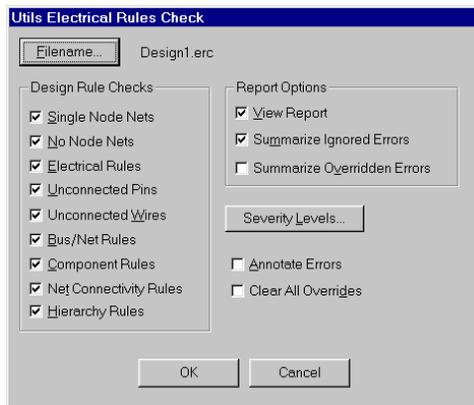
### Design Verification

P-CAD Schematic offers a range of Electrical Rules Checks (ERCs) that you can use to verify a design. This section shows you how to set up the rules you will be checking for and the level of severity, how to run the ERC report and then view the errors on screen. Finally we will have a go at fixing the errors that have been purposely included in a demonstration schematic file.

For more information about the options in Electrical Rules Check utility, refer to the Utils Commands, (page 329).

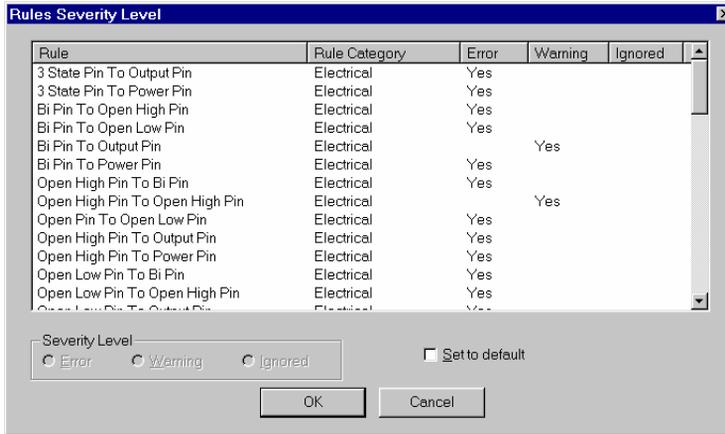
### Setting up the ERC Options

1. Open `ERC.sch` from the `Tutorial` folder of the P-CAD installation directory.
2. Choose **Utils » ERC** to open the *Utils Electrical Rules Check* dialog.



3. Select all of the check boxes in the Design Rule Checks frame.
4. Select the **View Report** check box to present an on-screen report when the checks are complete and select the **Summarize Ignored Errors** check box.
5. Select the **Annotate Errors** check box to place ERC error indicators in your design. These indicators graphically identify violation locations.

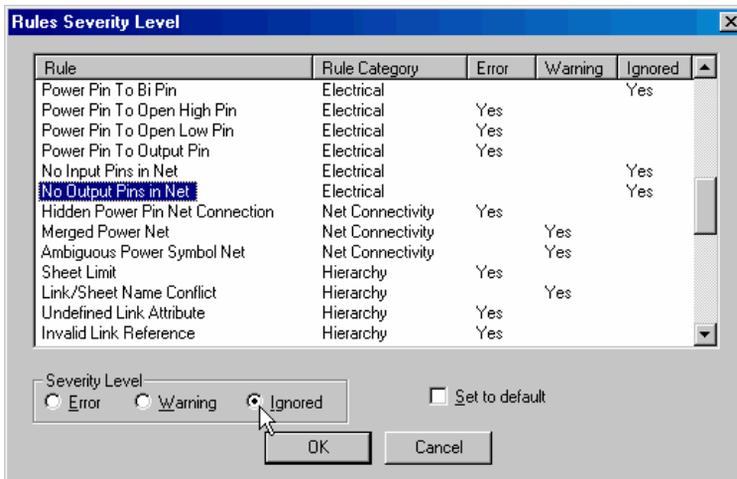
6. Click the **Severity Level** button to open the *Rules Severity Level* dialog.



Notice the severity levels for most rules are set to Error as indicated by a 'Yes' in the Error column. You can change the severity level of a rule by selecting a rule in the list and choosing one of the options – Error, Warning or Ignored – in the Severity Level frame. Only Errors will display on the design when Annotate Errors is turned on. Warnings and ignored rules will only show on the ERC report.

7. We will set a few rules to be ignored because the rigorous testing of the ERC will display errors that have been proven to be correct design choices for this schematic, e.g. pins have been correctly named as bidirectional in this case but may be problematic in a different design.

Select the following rules and choose the **Ignored** button in the Severity Level frame: **Bi Pin To Power Pin**, **Power Pin To Bi Pin**, **No Input Pins In Net** and **No Output Pins In Net**.



8. Click **OK** to close the *Rules Severity Level* dialog and return to the *Utils ERC* dialog.
9. Click **OK** to begin the ERC process. ERC will check all rules and will summarize those set to the Ignored severity level.
10. When the process is complete, an ERC report appears in the text editor, e.g. Notepad, as shown below. A copy is automatically saved to file with the filename defaulting to the name of the design file with an `.erc` extension.

Five errors and one warning will be reported, including a single node net, some unconnected wires and bus/net violations. An unconnected pin will be flagged as a warning.

```
P-CAD Electrical Rules Check Report
-----
D:\Program Files\P-CAD 2001\Tutorial\ERC.erc:
ERC Report Options:
-----
Single Node Nets: On
No Node Nets: On
Electrical Rules: On
Ignored Rules:
  Bi Pin To Power Pin
  Power Pin To Bi Pin
  No Input Pins in Net
  No Output Pins in Net
Unconnected Pins: On
Unconnected Wires: On
Bus/Net Rules: On
Component Rules: On
Net Connectivity Rules: On
Hierarchy Rules: On

ERC Errors:
-----
SINGLE NODE NETS:
Error 1 -- Net NET00000 is a single node net
0 warning(s) detected.
1 error(s) detected.

NO NODE NETS:
0 warning(s) detected.
0 error(s) detected.

ELECTRICAL RULES:
0 warning(s) detected.
0 error(s) detected.

UNCONNECTED PINS:
Warning 1 -- Unconnected pin J8:2-2 on sheet:Sheet1 at mils (3050.0,3000.0)
1 warning(s) detected.
0 error(s) detected.

UNCONNECTED WIRES:
Error 2 -- Unconnected wire end on sheet:Sheet1 at mils (3650.0,8700.0)
Error 3 -- Unconnected wire end on sheet:Sheet1 at mils (3350.0,8700.0)
0 warning(s) detected.
2 error(s) detected.

BUS/NET RULES:
Error 4 -- Bus ENBUSS only has one ref to net EN4
Error 5 -- Bus DATABUSS only has one ref to net EN1
0 warning(s) detected.
2 error(s) detected.

COMPONENT RULES:
0 warning(s) detected.
0 error(s) detected.

NET CONNECTIVITY RULES:
0 warning(s) detected.
0 error(s) detected.

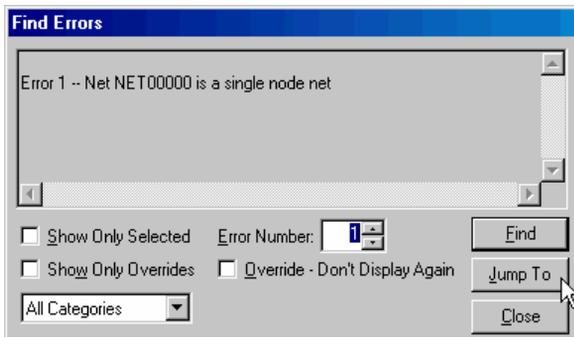
HIERARCHY RULES:
Hierarchy is simple.
Hierarchy is resolved.
0 warning(s) detected.
0 error(s) detected.

ERC Summary:
-----
Single Node:
  Errors: 1
  Warnings: 0
  Ignored Errors: 0
No Node:
  Errors: 0
  Warnings: 0
  Ignored Errors: 0
Electrical:
  Errors: 0
  Warnings: 0
  Ignored Errors: 115
Unconnected Pin:
  Errors: 0
  Warnings: 1
  Ignored Errors: 0
Unconnected Wire:
  Errors: 2
  Warnings: 0
  Ignored Errors: 0
Bus/Net:
  Errors: 2
  Warnings: 0
  Ignored Errors: 0
Component:
  Errors: 0
  Warnings: 0
  Ignored Errors: 0
Net Connectivity:
  Errors: 0
  Warnings: 0
  Ignored Errors: 0
Hierarchy:
  Errors: 0
  Warnings: 0
  Ignored Errors: 0
```

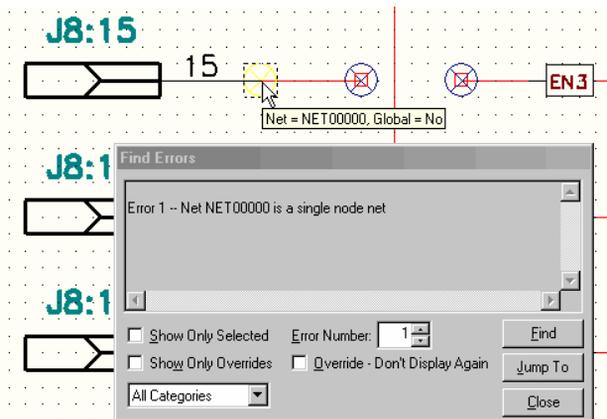
11. You can print the report by choosing **File » Print**. Keep it open for cross referencing.

### Viewing any Errors in the Design

1. Switch from the report to your design by clicking the workspace and notice that ERC error indicators  appear at error locations. Each error marker contains error information.
2. To find and view information about all the errors, choose **Utils » Find Error**. The *Find Errors* dialog displays with Error 1 information.



- In the Error Number scroll box, type another number or click the arrows to show information for all errors. The Description area shows the first error in the selected category, the error number and the reason for the error. The Categories drop-down list allows you to limit the type of errors listed, e.g. only Unconnected Wire errors.
- Select **Error Number 1** from the Error Number scroll box and click **Jump To**. You are switched to the workspace, the error indicator is selected and the cursor is placed on it.



- To view information about any other visible error indicator while in the workspace, select the indicator, right-click, and choose **Properties** from the shortcut menu to display the Find Errors dialog again. Alternatively, just double-click on an unselected error indicator.
- Note that an error indicator for an unconnected pin on part J8:2-2 (co-ordinates 3050.0, 3000.0) is not shown because it was flagged as a Warning only in the ERC severity rules. You can only view warnings in the ERC report.

## Fixing Errors in the Design

Now we will have a look at fixing the design errors found in this schematic.

### Fixing the Single Node Net and Unconnected Wire Errors

1. Select **Error Number 1** again from the Error Number scroll box of the *Find Errors* dialog. The description tells us it's a single node net error. Click **Jump To**. The cursor is placed on that selected error indicator.
2. Now double-click the second error indicator on that wire. Error 3 information will display in the *Find Errors* dialog, indicating an unconnected wire. Click back to the workspace and double-click on the next indicator to the right (Error 2) which is also an unconnected wire error. These three errors are therefore related to the break in the wire.
3. Click back on the workspace and fix the error by connecting the wire. This will resolve Errors 1, 2 and 3.
4. If you wish to delete the errors from the list as you solve them, click **Override – Don't Display Again** in the *Find Errors* dialog.

### Fixing the Bus/Net Errors

Now we will look at the remaining two errors, which are bus/net errors.

1. Click on the *Find Errors* dialog box, select **Error 4** (Bus ENBUSS only has one ref to net EN4) and **Jump To**. The clue lies in the fact that there is only one reference to net EN4. There should be a reference for the net going into the bus and another for it coming out again. This error description therefore implies that net EN4 is not properly connected to ENBUSS.
2. Find net EN4 by choosing **Edit » Nets**, selecting **EN4** from the Net Names list and click on **Jump to Node**. Now highlight net EN4 by selecting any wire in it, right-click and select **Highlight Net**. The entire net is highlighted for easy tracing.
3. View the entire net EN4 by zooming and scrolling around the workspace. You will notice that EN4 goes into ENBUSS but then is wired straight into part U9:D instead of coming out of ENBUSS first. This is the design error we will have to fix.
4. Since EN4 is individually wired directly to U9:D as well as attempting to go through the bus, one solution to this error is simply to select the wire segment that connects to ENBUSS and press **DELETE**. Alternatively, you could modify the wiring from U9:D pin 13 back into the ENBUSS.
5. Now we'll fix the last error, Error 5 – Bus DATABUSS only has one ref to net EN1. Highlight net **EN1** (see step 2 for details) to trace the net through the design. It appears that the net is connected to the wrong bus (DATABUSS).
6. Select the wire segment that goes from U6:A pin1 to DATABUSS and stretch it to ENBUSS. A new bus connector is displayed. Delete the old bus connector marker.
7. Finally, let's just check that warning of an unconnected pin that appeared on the report as 'Warning 1 -- Unconnected pin J8:2-2 on sheet:Sheet1 at mils (3050.0,3000.0)'. This appears to be simply a mistake, so delete the part by selecting it and pressing **DELETE**.

8. Choose **Utils » ERC** to rerun the ERC report and to clear the indicators of all errors that have been fixed. Check that there are no more errors in the ERC report. Choose **File » Exit** to close the ERC report and exit from the text editor.
9. Save and close the design.

The following suggestions form a basic strategy for resolving multiple errors in a design. First of all, run ERC checking only for single node nets, no node nets and unconnected pins and wires. Fix any errors and then rerun ERC to check for bus/net and net connectivity rules. Fix errors and rerun for component rules. Finally, select only the electrical and hierarchy rules (if you have a hierarchical design) and rerun ERC. With each pass, the number of related errors should diminish.

Now you have verified your design, please refer to Tutorial 4 for instructions about generating reports and netlists.

## Tutorial 4 - Generating Reports and Netlists

Welcome to the P-CAD Schematic tutorial 'Generating Schematic Reports and Netlists'. During this tutorial, we will generate reports using a demonstration file that is supplied with P-CAD. Then we will generate a netlist from this file that will include all the component and net information in the design.

For more information about the topics in this tutorial, press **F1** for online Help, or refer to the relevant chapter in this *User's Guide*.

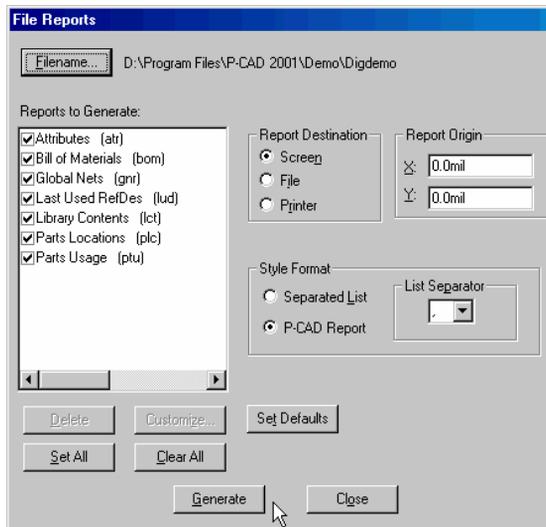
This section covers generating standard reports from your schematic design, such as a Bill of Materials (BOM) and Attributes, Global Nets and Parts Usage reports. These reports can be viewed on screen, saved in a file or sent to a printer. You can customize your reports to include format options, like headers and footers, or set data selection or sort options.

For more information about customizing your reports, refer to the *File Commands*, (page 169).

### Generating a Report

To generate a report:

1. Open the schematic design file `Digdemo.sch` from the P-CAD Demo directory.
2. Choose **File » Reports**. The *File Reports* dialog appears.



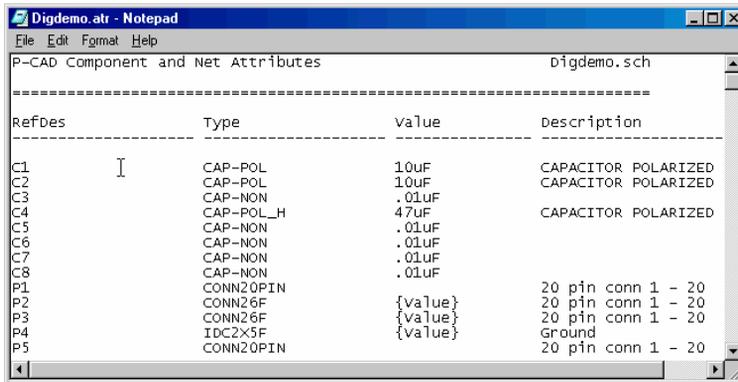
3. In the Reports to Generate list, select the check boxes that correspond to the type of reports you want to generate.
4. Click **Filename**. The *Reports File Save As* dialog appears.

5. Navigate to the directory in which you want to save the report file and type a name for the report file in the File Name box.
6. Select **Report Files (\*.\*)** from the Save As Type list and click **Save**. You return to the *File Reports* dialog.
7. In the Report Destination frame, choose one of the following buttons:
  - **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.
8. In the Style Format frame, choose one of the following buttons:
  - **Separated List** - Puts all data in character-separated format. This format can be imported into other spreadsheet and database programs.
  - **P-CAD Report** - Produces a report format with columns and spaces, etc.

If you choose Separated List, select a separator from the List Separator list.

The list separator character displayed in the box is used for both imported and exported files. The default character is your computer's regional setting.

9. Click **Generate** and the selected reports will display on the screen in your text editor, such as an Attributes report displayed in Notepad below, or will be sent straight to the printer or nominated file.

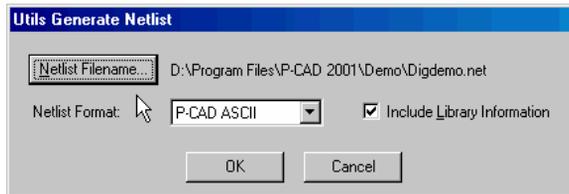


## Generating Netlists

With P-CAD Schematic, you can generate netlist that lists the components and nets in your design. Typically, a netlist is used by a printed circuit board editor, such as P-CAD PCB, to form the basis of your board design.

To generate a netlist:

1. Open the schematic design file `Digdemo.sch`, located in the P-CAD Demo directory.
2. Choose **Utils » Generate Netlist**. The following *Utils Generate Netlist* dialog appears.



3. Click **Netlist Filename**. The following *Netlist File* dialog appears.



4. Navigate to the directory in which you want to save the netlist file and type a name for the netlist file in the File name box.
5. Select *Netlist Files (\*.net)* from the Save As Type list and click **Save**. You return to the *Utils Generate Netlist* dialog.
6. In the Netlist Format list, select **P-CAD ASCII** as your netlist format.

The P-CAD ASCII format supports attribute passing. If you select this option, you may also select the **Include Library Information** check box.

7. Click **OK** to generate the netlist. The netlist is saved with the filename and in the directory you specified.

This concludes Tutorial 4. Please refer to Tutorial 5 for information about printing your schematic.

## Tutorial 5 - Printing a Schematic Design

Welcome to the P-CAD Schematic tutorial '*Printing a Schematic Design*' in which we will look at setting up print jobs. This tutorial uses a demonstration file that is supplied with P-CAD.

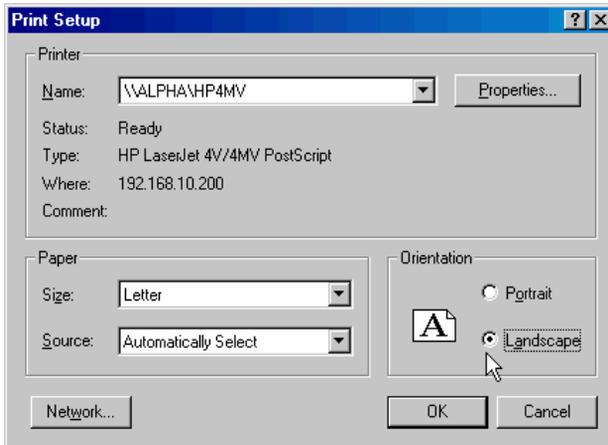
For more information about the topics in this tutorial, press **F1** for online Help, or refer to the relevant chapter in this User's Guide.

This final tutorial covers printing or plotting your schematic design. First, we'll check the printer/plotter and page setups, then set the print options, before previewing and generating a print job that goes directly to the printer.

### Setting Up a Printer or Plotter

To set up or change the printer or plotter settings on your computer:

1. Choose **File » Print Setup** to open the *Print Setup* dialog.



2. Select a print device from the Name list box. This list shows the printers and plotters that have been installed on your computer.
3. Select the paper size and source from the corresponding list boxes in the Paper frame and choose **Landscape** in the Orientation frame.
4. Click **Properties** to open the *Properties* dialog and configure other print parameters and click **OK**.

Since print parameters are device-specific, the options in the *Properties* dialog depend upon the print device you selected.

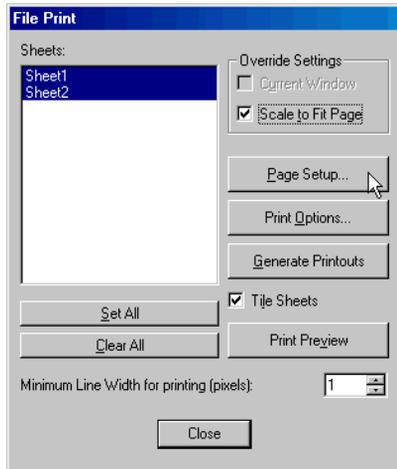
5. Click **OK** to close the *Print Setup* dialog.

## Setting Up Page Setup Options

For each sheet, you can scale your image to fit standard or custom size papers, define a print region, set horizontal and vertical offsets, show a title, and rotate the image. Individual sheets within a design can have different options set and these are saved with the design. We will print our `Digdemo.sch` file in this section.

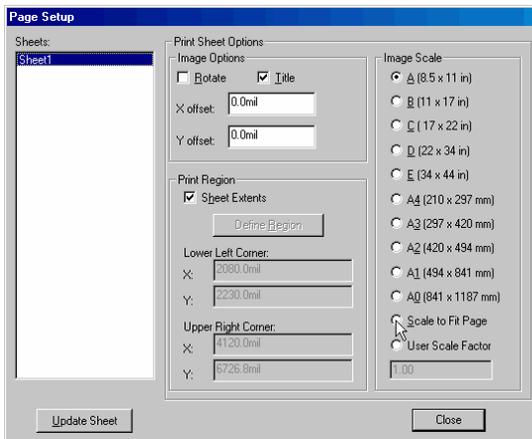
To define your page setup options:

1. Choose **File » Print** to open the *File Print* dialog.



2. Select the sheet(s) from the Sheets list to which you are applying these options and click **Tile Sheets** and **Scale to Fit Page** to display all sheets on one page.
3. Click **Page Setup** to open the *Page Setup* dialog.

For more information about options in this dialog, such as Print Region, refer to the *File Commands*, (page 169).

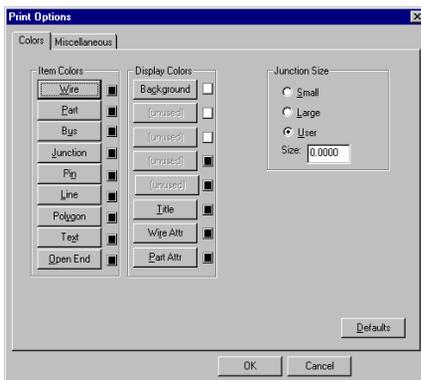


4. Select **Title** to display the title block on the printed image.
5. Click **Update Sheet** to apply your changes to the print sheet and click **Close** to return to the *File Print* dialog, where we can set the print options.

## Setting Up Print Options

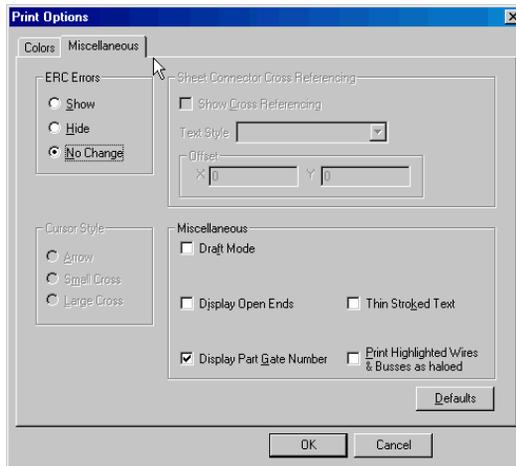
With the *Print Options* dialog, you can set up various print options, such as color and display.

1. Click **Print Options** from the *File Print* dialog to open the *Print Options* dialog and click the **Colors** tab.



2. Choose your print colors using one of these methods:
  - If you have a color printer, click a command button in the Item Colors or Display Colors frame. When the color palette appears, click a color swatch.

- If you have a black and white printer, click the **Defaults** button. This sets all color options to monochrome. We recommend that you use this setting to avoid undesirable output when your printer converts color settings to grayscales.
3. Choose **Small**, **Large** or **User** in the Junction Size frame.  
Notice that you can enter a value in the Size box only when you choose **User**. You can enter a value in inches, centimeters, etc. However, the value must be between 0 - 10 mm.
  4. Click the **Miscellaneous** tab. In this tab, you can choose to show or hide ERC error indicators and set other miscellaneous print options as shown in the following figure.

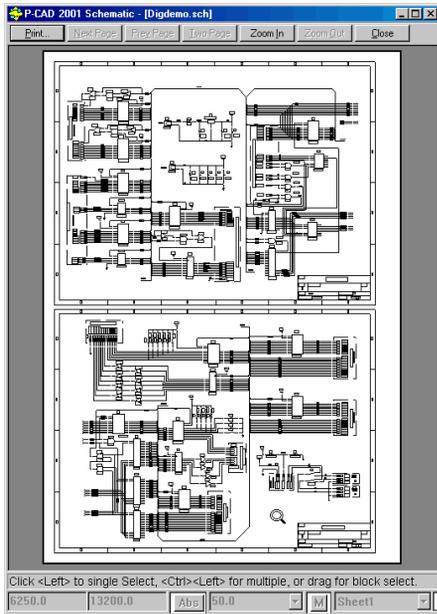


5. We'll use the default settings, so click the **Defaults** button and click **OK** to return to the *File Print* dialog.

## Previewing a Print Job

To check that the print job you have set looks right, preview the print job.

1. Click **Print Preview** in the *File Print* dialog. The specified print output appears on your screen, such as the Digdemo schematic below.



2. Click the **Zoom In** button to zoom in on the center area of the current page. Use the scroll bars to move to the desired viewing region to check that the print job is correct.
3. Click **Print** to send the current print jobs directly to the printer, or click **Close** to close the Print Preview window and return to the *File Print* dialog.

## Generating Printouts and Printing Sheets

After you have defined your page setup and print options, return to the *File Print* dialog to print the sheets.

1. Click **Set All** in the *File Print* dialog to enable all the sheets in the design.
2. Click **Generate Printouts** to print the selected sheets.

## Working with Objects

Consistent with other Windows™ compliant interfaces, the P-CAD user interface provides you with the menu choices, dialogs, options, and feedback based on what is appropriate given the current state of your design and P-CAD. The interface provides power, flexibility, and logical choices based on the object(s) currently selected and the command or process currently being invoked.

This chapter includes detailed information on such topics as:

- Placing Objects
- Selecting Objects
- Defining Object Selection Preferences
- Block Selecting a Group of Objects
- Moving Objects
- Rotating and Flipping Objects
- Aligning Objects
- Using Orthogonal Modes
- Resizing an Object
- Pasting Objects
- Object Properties
- Editing Nets

## Placing Objects

---

To place an object in a schematic design, choose a command from the **Place** menu or click a button in the *Placement* toolbar shown in the following figure:



When you choose a **Place** command, the tool remains active until you choose another command. For example, when you choose **Place » Wire**, you remain in wire placement mode until you choose another command, such as **Place » Port** or **Edit » Select**.

Each **Placement Command** operates differently. Some objects require the interaction of a dialog before the object can be placed (e.g., content of text), but the general placement characteristics are the same.

For example, when you choose **Place » Line**, P-CAD Schematic automatically places a line segment each time you click the workspace. Whereas, when you choose **Place » Pin** and click the workspace, a dialog box appears. After you set the pin characteristics using the options in the dialog, P-CAD Schematic places a new pin each time you click the workspace.

### Placing an Object

There are two methods than you can use to place an object. For quick placement, click the workspace. For more precise placement, hold down the left mouse button and drag the object into position. Then, release the mouse button to place the object. You remain in placement mode until you right-click or press **ESC**.

When you place objects in your design, P-CAD Schematic places the object at a grid point, regardless of your cursor setting. For more information, see *Options Grids* (page 295). To learn more about object placement, see *Place Commands* (page 259).

## Selecting Objects

---

This section summarizes the techniques you can use to select objects in a schematic design. Once you are familiar with these techniques, you can define various object selection preferences. For details, see *Defining Object Selection Preferences* (page 81).

### Selecting an Object

Use the single selection feature to select an object in your design. To select an object, follow these steps:

1. Choose one of these methods to enable the select tool:
  - Click  in the *Command* toolbar

- Choose **Edit » Select**
  - Press the **S** key.
2. Click an object in the workspace.

When you select an object, the object appears in the selection color set in the *Options Display* dialog. To change this color, see *Choosing a Selection Color* (page 82).

To cancel your selection, click anywhere in the workspace.

## Selecting Multiple Objects

Use the multiple selection feature when you want to select more than one object at a time. To select multiple objects, follow these steps:

1. Choose one of these methods to enable the select tool:
  - Click  in the *Command* toolbar
  - Choose **Edit » Select**
  - Press the **S** key.
2. Hold down the **CTRL** key and click the objects that you want to select.

P-CAD Schematic is set up to use the **CTRL** key for multiple selections by default. To change this behavior, see *Selecting Part of an Object* (page 79).

To cancel your selection, click anywhere in the workspace.

## Selecting Part of an Object

The subselection feature lets you select a part of an object. For example, you can select a single pin in a part. To select part of an object, follow these steps:

1. Choose one of these methods to enable the select tool:
  - Click  in the *Command* toolbar
  - Choose **Edit » Select**
  - Press the **S** key.
2. Hold down the **SHIFT** key and click the part of the object that you want to select.

P-CAD Schematic is set up to use the **SHIFT** key for subselections by default. To change this behavior, see *Selecting Part of an Object* (page 79).

Once selected, you can view and in some cases modify properties associated with the part of the object. For example, if you selected a pin, you can open a *Properties* dialog for the pin and then change the pin style. To cancel your selection, click anywhere in the workspace.

## Selecting Collocated Objects

When objects on different sheets overlap, you can switch between the collocated objects, to select objects that reside on different sheets. When objects overlap, do the following:

1. Choose one of these methods to enable the select tool:
  - Click  in the *Command* toolbar
  - Choose **Edit » Select**
  - Press the **S** key.
2. Hold down the **SPACEBAR** and click the left mouse button to switch between the collocated objects.

Pressing the **SPACEBAR** twice is equivalent to a mouse click (a press and release of the mouse button).

## Selecting All Objects

To select all objects in a design, follow these steps:

1. Choose one of these methods to enable the select tool:
  - Click  in the *Command* toolbar
  - Choose **Edit » Select**
  - Press the **S** key.
2. Choose **Edit » Select All**.

When you select an object, the object appears in the selection color set in the *Options Display* dialog. To change this color, see *Choosing a Selection Color* (page 82).

To cancel the selection of all objects, choose **Edit » Deselect All** or click outside the selection box.

## Selecting Highlighted Objects

If you've applied a highlight color to objects in your design, you can select all of those objects. To do this, follow these steps:

1. Choose one of these methods to enable the select tool:
  - Click  in the *Command* toolbar
  - Choose **Edit » Select**
  - Press the **S** key
2. Choose **Edit » Select Highlighted**. The highlighted objects are selected.

The highlighter features interact with DDE hotlinks feature, to enable the exchange of hotlink data between two P-CAD programs. In Schematic, hotlink data consists of highlighting and unhighlighting commands for parts and nets. In PCB, hotlink data is for components and nets.

## Selecting a Net

There are two ways you can select all items in the net to which an item is connected.

- Choose **Edit » Nets** and select a net from the list in the dialog that appears. Then, click **Select**.
- Right-click a net and choose **Select Net** from the shortcut menu.

In both cases, the complete net is selected, subject to any criteria set in *Options Block Selection*. For information, see *Defining Block Selection Criteria* (page 83).

## Selecting Contiguous Net Objects

With the select contiguous feature, you can select all contiguous net objects of a selected net. Net objects are contiguous if they are visually connected on the design. For instance, wires are contiguous if they are connected from endpoint to endpoint. If a net consists of multiple subnets, only the sub-net to which the selected item belongs is selected.

To use the select contiguous feature, right-click a wire or port, then choose **Select Contiguous** from the shortcut menu. This feature only works for one net at a time.

## Block Selecting Objects

With P-CAD Schematic, you can also use the block selection feature. For details, see *Block Selecting a Group of Objects* (page 82).

# Defining Object Selection Preferences

---

You can define various object selection preferences for P-CAD Schematic. This section provides you with information on your options. To learn about the techniques you can use to select objects, see *Selecting Objects* (page 78).

## Setting the CTRL/SHIFT Behavior

By default, P-CAD Schematic is set up to use the **CTRL** key for multiple selections and the **SHIFT** key to select part of an object. However, you can reverse this setting. To do this, follow these steps:

1. Choose **Options » Preferences** to open the *Options Preferences* dialog.
2. Click the **Mouse** tab.
3. In the Ctrl/Shift Behavior frame, choose the **Ctrl subselects; Shift extends selection** button.
4. Click **Close** in the *Options Preferences* dialog.

To learn how to extend a selection, see *Selecting Multiple Objects* (page 79). For instructions on the subselection process, see *Selecting Part of an Object* (page 79).

## Specifying a Selection Point

When you select one or more objects, a selection point appears at the closest grid point to the center of the selection box by default. However, you can change the location of this point by following these steps:

1. Select one or more objects.
2. Right-click and choose **Selection Point** from the shortcut menu.
3. Press and hold down the left mouse button. Then, drag the ghosted outline of the selection point into position. The following figure shows you a selection point.



4. Release the mouse button to set the selection point at the cursor location.

When you cut or copy an object from a design, the selection point moves to the Clipboard or block file. When you paste an object, the selection point snaps to the grid.

If the selection point is off-grid when a move operation begins, then it is automatically snapped to the nearest grid point and all the selected objects move the same relative distance.

Parts can be vertically or horizontally aligned around a selection point. To do this, place a selection point at the point of alignment, choose **Edit » Align Parts**, and then set the desired alignment options in the dialog that appears.

## Choosing a Selection Color

When you select objects in a P-CAD Schematic design, the objects appear in the selection color set in the *Options Display* dialog. To choose a selection color, follow these steps:

1. Choose **Options » Display** to open the *Options Display* dialog.
2. Click the **Colors** tab.
3. In the Display Colors frame, click **Selection**. A color palette appears.
4. Click a color in the palette. Then, click .
5. Click **OK** to close the *Options Display* dialog.

## Block Selecting a Group of Objects

---

The block selection feature gives you the ability to select a group of objects. You can configure this feature to include or exclude specific types of objects and your selection criteria can be as specific or as general as you choose.

The procedures in this section show you how to perform the following use and set up the block selection feature.

## Block Selecting Objects

To block select a group of objects, follow these steps:

1. Choose one of these methods to enable the select tool:
  - Click  in the *Command* toolbar
  - Choose **Edit » Select**
  - Press the **S** key.
2. Drag the cursor diagonally across an area of the design to draw a selection box around a region.
3. Release the mouse button. Any objects that match your block selection criteria are selected. To learn how to define block selection criteria, see *Defining Block Selection Criteria* (page 83).

To remove the selection box, click outside the box.

## Defining Block Selection Criteria

You can configure the block select feature to select all objects, objects by type, or even objects with specific parameters. To define block selection criteria, follow these steps:

1. Choose **Options » Block Selection** to open the *Options Block Selection* dialog.
2. In the *Items* frame, select the check boxes that correspond to the objects you want to block select.

Some check boxes have corresponding command buttons (e.g., *Line*, *Part*, *Pin*). To make a command button available, click a check box until a shaded check mark appears. Then, click the button to open a *Selection Mask* dialog. You use the options in these dialogs to narrow your selection criteria. For more information, see *Edit Properties* (page 211).

3. In the *Select Mode* frame, choose of the following buttons:
  - **Inside Block.** Choose this button to select all items within the selection box.
  - **Outside Block.** Choose this button to select items outside the selection box.
  - **Touching Block.** Choose this button to select all items inside or touching the selection box.
4. Click **OK** to close the *Options Block Selection* dialog.

## Moving Objects

---

There are several ways to move objects in a P-CAD Schematic design. When moving an object, any connections rubber band to maintain the established nets.

## Moving an Object

There are two basic methods for moving objects in P-CAD Schematic:

- Select the object that you want to move and drag it to a new position.
- Select an object and press the **arrow** keys.

With either method listed above, you can move one or multiple objects. To move multiple objects, draw a bounding outline around the group of objects to move. Then, drag them to a new position or press the **arrow** keys.

As described in the following sections, you can also move objects during the placement process or with the **Edit » Move by RefDes** command.

## Moving Objects During Placement

You can move parts, wires, buses, ports, points, text, attributes, fields and IEEE symbols before you place them in a design. To move an object during the placement process, follow these steps:

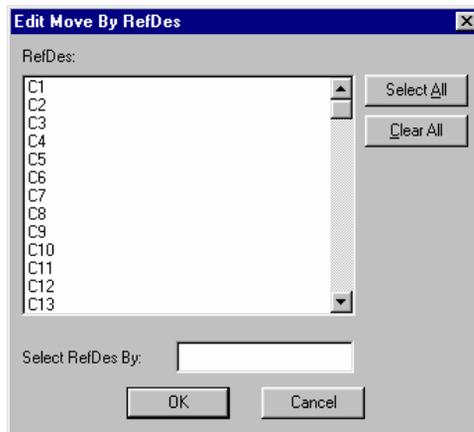
1. Choose a command from the **Place** menu. For example, choose **Place » Port**. The *Place Port* dialog appears.
2. Use the controls in the *Place Port* dialog to specify the characteristics of the port you are about to place. Then, close the dialog box.
3. Press and hold down the left mouse button. Then, drag the ghosted outline of the object to a location.
4. Release the mouse button to place the object at the cursor location.

As a shortcut for holding down the mouse button during a drag-and-drop operation, press the **ALT** key and click the workspace. When the ghosted outline of the object appears, move the object to a location and press **ALT** to release the object at the cursor location.

## Moving Parts by RefDes

To move parts by reference designator, follow these steps.

1. Choose **Edit » Move by RefDes** to open the *Edit Move by RefDes* dialog.



If you select parts in your design before you open this dialog, those parts are selected in the *RefDes* list when the dialog appears

2. Use one of the following methods to select parts from the *RefDes* list:
  - **To select one part:** Double-click a **RefDes** in the list. Or, select a **RefDes** from the list and then click **OK** or press **ENTER**.
  - **To select all parts:** Click **Select All**. Then, click **OK**.
  - **To select a range of parts:** Hold down the **SHIFT** key and click the first and last parts in the range to select. Then, click **OK** or press **ENTER**.
  - **To select various parts:** Hold down the **CTRL** key and click each part to select. Then, click **OK** or press **ENTER**.
  - **To search for parts in the *RefDes* list:** type search criteria in the *Select RefDes By* box and then press **ENTER**.

When the *Edit Move by RefDes* dialog closes, the status line shows the name of the Next *RefDes* to move. You are in placement mode.

To cancel your selection, press **ESC** or right-click. This reopens the *Edit Move by RefDes* dialog, so you can choose another part to move.

3. Choose one of these methods to place the parts:
  - For quick placement, click the location at which you want to place the part.
  - For precise placement, do the following: (1) press and hold down the left mouse button or press the **SPACEBAR**. (2) Move the ghosted outline of the part to a location. If appropriate, press **F** to flip or **R** to rotate the part. (3) Place the part by releasing the mouse button or by pressing the **SPACEBAR**.

If you selected multiple parts, the status line shows the Next RefDes to move. To skip to the next *RefDes*, press **N**. To move to the previous RefDes, press **B**.

4. Once you've moved all of the selected parts, the *Edit Move By RefDes* dialog appears. You have these options:
  - To move another part, select it from the RefDes list.
  - To close the *Edit Move by RefDes* dialog, click **Cancel** or press **ESC**.

## Rotating and Flipping Objects

---

You can rotate and flip many objects in your design. When you rotate or flip an object, P-CAD Schematic rotates the object about the selection point. For details, see *Specifying a Selection Point* (page 82).

For items with associated connections, hold down the left mouse button while rotating or flipping to prevent reconnection attempts each time you rotate or flip. Connections are changed when the left mouse button is released.

To find out which objects you can rotate or flip, see the appropriate section in *Place Commands* (page 259).

### Rotating Objects During Placement

To rotate an object during the placement process, follow these steps:

1. Choose a command from the **Place** menu. For example, choose **Place » Port**.
2. Click the workspace to place the selected object. If a dialog appears, use the options in the dialog to specify the characteristics of the object you are about to place. Then, close the dialog box.
3. Hold down the left mouse button. Then, press **R** to rotate the object 90 degrees.
4. Release the mouse button to place the object in your design.

### Rotating Objects in a Design

To rotate an object that has been placed in a design, follow these steps:

1. Choose one of these methods to enable the select tool:
  - Click  in the *Command* toolbar
  - Choose **Edit » Select**
  - Press the **S** key.
2. Select the object that you want to move. Then, press **R** to rotate the object 90 degrees.

## Flipping Objects

To flip an object that has been placed in a design, select the object that you want to flip. Then, press **F** to flip the object. When you flip an object, P-CAD Schematic flips the object about the selection point. For details, see *Specifying a Selection Point* (page 82).

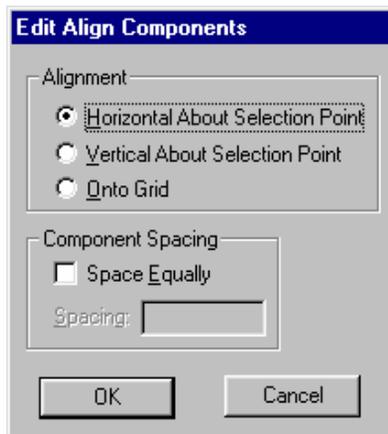
## Aligning Objects

---

### Aligning Parts Horizontally or Vertically

To align parts horizontally or vertically, follow these steps:

1. Select the parts to align. Only parts can be selected for this command to be enabled.
2. Right-click the parts and choose **Selection Point** from the shortcut menu. This is the point about which the parts will be aligned, either horizontally or vertically.
3. Place the selection reference point by clicking in the workspace. Without selecting a selection reference point, the alignment and part spacing options are shaded.
4. Choose **Edit » Align Parts**. The *Edit Align Components* dialog appears:



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

### Aligning Parts To Grid

To align parts to grid, follow these steps:

1. Select the parts to align to grid. Only parts can be selected for this command to be enabled.

2. Choose **Edit » Align Parts**. The *Edit Align Parts* dialog appears.
3. Click **OK** and the selected parts will be aligned to grid. Each selected off-grid part will be moved to the nearest grid point.

## Using Orthogonal Modes

---

During the line placement process, you can press the **O** key to switch between the enabled *orthogonal modes*. The available orthogonal modes are divided into the following *mode pairs*:

- **90/90 Line-Line Mode:** Both lines are either horizontal or vertical (displayed perpendicular to each other). For long, the first segment is always longer than the second. For short, the first segment is shorter. To switch between the two configurations, press the **F** key.



- **45/90 Line-Line Mode:** The first mode makes the first segment displayed at a 45-degree angle and the second segment is either horizontal or vertical. The second mode makes the first segment either horizontal or vertical and the second segment. To switch between the two modes, press the **F** key.



## Enabling Orthogonal Modes

To enable an orthogonal mode, follow these steps:

1. Choose **Options » Configure**. The *Options Configure* dialog appears.
2. In the Orthogonal Modes frame, select the following check boxes to enable a mode pair.
  - **90/90 Line-Line Mode**
  - **45/90 Line-Line Mode**

For placing lines, the modes are limited to line segments (no arcs).

3. Click **OK** to close the *Options Configure* dialog.

After you enable the orthogonal modes, you can switch between the enabled modes, or switch between the mode pairs. For instructions, see *Switching Between Orthogonal Modes* (page 89) and *Switching Between Mode Pairs* (page 89).

## Switching Between Orthogonal Modes

Press the **O** key to switch between the orthogonal modes. The non-orthogonal mode straight-line mode is always available, no matter which orthogonal modes are enabled.

## Switching Between Mode Pairs

Press the **F** key to switch between the mode pair of the orthogonal mode you are in. The non-orthogonal mode straight-line mode is always available, no matter which orthogonal modes are enabled.

## Unwinding Segments

You can unwind (undo a segment by pressing the **BACKSPACE** key) wires, lines, and polygons.

When you place a multiple segment object, you can press the **BACKSPACE** key to delete the previous segment. If you have finished placing the object by right clicking, then the unwind function does not work. You can undo the placement of a finished object (**Edit » Undo**), but you cannot unwind it.

## Resizing an Object

---

You can resize an object by clicking one of its handles and dragging to stretch the object. Object handles are the squares that appear when you select certain objects (e.g., arcs or polygons). Handles appear only if you select one object, and if the object can be resized.

The resize function varies among the different objects. However, some objects cannot be resized (e.g., pins).

For example, to resize an arc you click one of the endpoint resize handles and drag the endpoint to increase the sweep angle. To resize a polygon, you can grab one of its vertex handles and move it to change the polygon.



When you move a polygon handle that is on an edge between two vertices, a new vertex is created (allowing you even more reshaping). You can delete a vertex by moving it to an adjacent vertex and releasing.

Lines, wires, buses, arcs, and polygons can be resized.

## Cutting and Copying Objects

---

In P-CAD Schematic, you can cut or copy objects to the Clipboard, which is a temporary storage area on your computer that houses the information that you want to transfer between designs,

documents, etc. You can also copy objects to a Meta file or block file. For details, see *Copying Objects to a File* (page 91).

## Cutting Objects from a Design

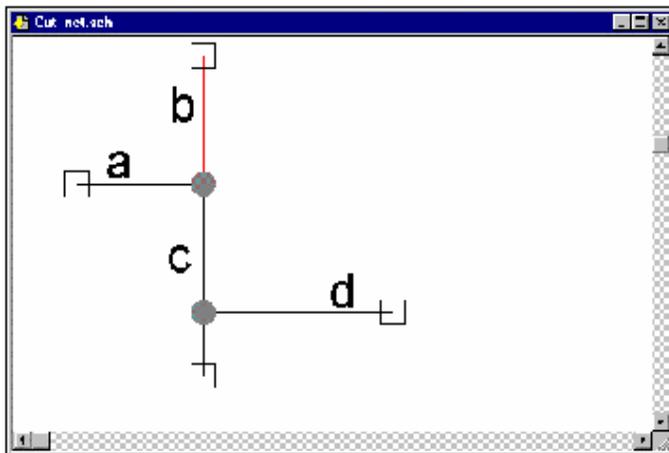
You can cut objects from a design and paste them to another location, design, or program. To cut an object from a design, follow these steps:

1. Select the objects that you want to cut.
2. Choose **Edit » Cut** or press **CTRL+X**.

The objects move to the Clipboard. When you cut objects from nets, you can get a variety of results, depending on what you cut and the makeup of the net from which you remove it. For example, deleting a wire from a net may split the net if the net is a contiguous net.

## Cutting Objects From Nets

When you choose **Edit » Cut** to remove objects from nets, the results depend on what you cut and the makeup of the net you remove it from.



- If you remove a wire from the middle of a net (**bc** in the example above):
- If the deleted wire has a system-assigned net name (for example, NET00001) then one of the nets is given a new system-assigned net name, while the other retains the original net name.
- If the deleted wire was connected to a global net, the subnet still attached to the port or power symbol retains the original net name, while the other net is renamed to a unique system-assigned net name.
- If both subnets are connected to a port or power symbol, then both subnets will retain the original name.

- If the wire was connected to a jumper pin and nothing else is connected to that jumper pin. Then all the jumpered pins are removed from the net.

If you delete a wire that isolates a pin from the rest of the net, you end up with a disconnected node that is no longer part of any net. For example, in the above figure, if you delete wire **cd**, the node **d** becomes isolated from the net.

## Copying Objects to the Clipboard

You can copy objects and move them to the Clipboard. To do this, follow these steps

1. Select one or more objects.
2. Choose **Edit » Copy** or press **CTRL+C**.

The object moves to the Clipboard. You can then paste the contents of the Clipboard into your design, to another design, or even another program.

## Copying Objects to a File

To copy objects to a block or Meta file, follow these steps:

1. Select one or more objects.
2. Choose **Edit » Copy to File**. The *Edit Copy to File* dialog appears.
3. Navigate to the directory in which you want to save the file.
4. Type a name for your file in the *File Name* box.
5. Select one of the following file formats from the *Save as Type* list:
  - Block Files (\*.blk). A file format that stores the information that you want to transfer between P-CAD Schematic designs.
  - Meta Files (\*.wmf). An image file format that can be transported between computers.
6. Click **Save**.

## Copying Objects within the Same Design

To copy objects within the same design, follow these steps:

1. Select one or more objects.
2. Hold down the **CTRL** key. Then, drag the ghosted outline of the selected objects to a position.
3. Release the mouse button to place a copy of the selected object at the cursor location.

## Copying Nets

When you copy a contiguous net, P-CAD Schematic always creates a new system-assigned net name. **CTRL/Drag** and **Copy Matrix** provide automatic net name incrementation based on increment values you set using the *Options Configure* dialog.

Nets are contiguous if they are visually connected on the design. For instance, wires are contiguous if they are connected endpoint to endpoint.

When wires are placed down, without being connected to an existing net, they are automatically given a unique net name, for example, NET00001. These system-assigned nets can subsequently be renamed, or ported to a new or existing net.

The **Wire Properties** and **Utils Rename Net** commands allow you to rename a selected net.

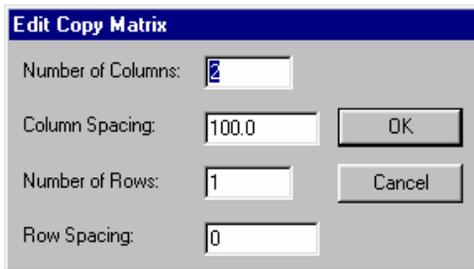
## Copying a Matrix

A matrix is an array of circuit elements. With the **Edit » Copy Matrix** command, you can duplicate one or more selected objects in both the horizontal and vertical directions.

For example, you could choose this command to select a pin and create a column of four pins, or you can select an existing column and create any number of additional columns.

To create a matrix of objects, follow these steps:

1. Select the objects that you want to duplicate.
2. Choose **Edit » Copy Matrix** to open the *Edit Copy Matrix* dialog.



3. In the Number of Columns box, specify how many duplications you want to perform horizontally. In the Column Spacing box, enter a value to determine the spacing between duplications and in which direction (positive=right, negative=left) to duplicate.
4. In the Number of Rows box, specify how many duplications you want to perform vertically. In the Row Spacing box, enter a value to determine the spacing between duplications and in which direction (positive=up, negative=down) to duplicate.

You will receive an error message if what you specify for your duplication is too large to fit in the Workspace.

5. Click **OK**. If your duplication is unsatisfactory, choose **Edit » Undo** to reverse the action.

## Pasting Objects

Once you cut or copy information to the Clipboard or a block file, you can paste the objects into a schematic design. This section summarizes the techniques you can use to paste objects in a design.

You must have enough space in the target workspace location for the items you are pasting. If the space isn't sufficient or you are too close to the edge, the data can't be pasted to the design.

## Pasting Clipboard Objects

To paste the contents of the Clipboard to a design, follow these steps:

1. Cut or copy information to the Clipboard.
2. Move the cursor to the location where you want to paste the information.
3. Choose **Edit » Paste From Clipboard** or press **CTRL+V**.

To learn how to copy information to the Clipboard, see *Copying Objects to the Clipboard* (page 91).

## Pasting From a File

To paste objects from a file to a design, follow these steps:

1. Copy information to a file. For instructions, see *Copying Objects to a File* (page 91).
2. Choose **Edit » Paste From File** or **Edit » Paste Circuit from File**. The *Edit Paste from File* dialog appears.
3. Select the file that stores the objects you want to paste in your design. Then, click **Open**. The cursor takes the shape of a crosshair cursor.
4. Click the workspace at the location at which you want to paste the contents of the file.

To learn how to copy objects to a file, see *Copying Objects to a File* (page 91).

## Splitting a Net

The naming of nets separated as a result of a cutting, moving, or pasting action depends on whether the nets were global or local. This situation may arise when a wire is deleted from a net or when a 2-pin part is inserted over a wire segment.

- If the original wire has a system-assigned net name (for example, NET00001) then one of the nets is given a new system-assigned net name, while the other retains the original net name.
- If the original wire was connected to a global net, the sub-net still attached to the port or power symbol retains the original net name, while the other net is renamed to a unique system-assigned net name.
- If both sub-nets are connected to a port or power symbol, then both sub-nets retain the original name.

Net splitting occurs if an inserted part has only two pins, and it is inserted over a single wire segment.

## Object Properties

---

With P-CAD Schematic, you can gain access to a *Properties* dialog that contains context-sensitive information about an object in a design.

When displaying the properties of multiple objects, the result depends on whether the objects are of the same type.

If the objects are the same type, but have different styles or other characteristics, then the *Properties* dialog appears, but specific information about the objects will be blank or grayed where the information differs between objects.

You can enter information in the dialog and click **OK**, and then all of the selected objects uniformly take on the characteristics (e.g., dimensions) of what you specified in the dialog. You can use the block selection mask to restrict selection. For information, see *Options Block Selection* (page 287).

Although you cannot display the properties of objects of different types, you can perform other edit commands.

### Opening a Properties Dialog

To open a *Properties* dialog, choose one of these methods:

- Select an object and choose **Edit » Properties**.
- Right-click an object and choose **Properties** from the shortcut menu.
- Double-click an object. A *Properties* dialog only displays if you've selected the **Double-Click Displays Properties** check box in the **Mouse** tab of the *Options Preferences* dialog.

## Editing Nets

---

This section discusses deleting objects from nets, managing net connections and the edit nets function.

### Deleting Objects from Nets

When you delete objects from nets, you get a variety of results depending on what you delete and the makeup of the net from which you delete the objects. The function of smart nets is to maintain certain connections when objects such as wires, parts, and net nodes are deleted. In general, the following can occur.

If you remove a wire from the middle of a net:

- If the deleted wire has a system-assigned net name (for example, NET00001) then one of the nets is given a new system-assigned net name, while the other retains the original net name.
- If the deleted wire was connected to a global net, the subnet still attached to the port or power symbol retains the original net name, while the other net is renamed to a unique system-assigned net name.

- If both sub-nets are connected to a port or power symbol, then both sub-nets will retain the original name.
- If you delete a wire that isolates a pin from the rest of the net, you end up with a disconnected node that is no longer part of any net.

## Managing Net Connections

A single symbol pin or a part's sub-selected pin can be connected to a net. To do this, right-click the pin and choose the **Add to Net** command. When the *Add to Net* dialog appears, select a pin you must supply the name of the existing net to which it will be connected.

A symbol pin connected to a net can be disconnected from the net. To do this, right-click the pin and choose the **Remove from Net** command. The pin remains in the design as a single pin.

## Using the Edit Nets Dialog

You can also use the *Edit Nets* dialog to select nets, edit net attributes and rename and delete nets and buses. A net is an electrical connection (e.g., two pins connected by a wire). Buses are a graphical representation of a bundle of one or more wires used to show multiple parallel wires on the schematic.

If you click the **Edit Attributes** or **View Attributes** button, the *Attributes* dialog appears. Within the collection, each attribute's name and value appear in the column.

- **Adding an Attribute:** To add an attribute, click the **Add** button to open the *Place Attribute* dialog. Add a pre-defined attribute by choosing first the Category and then the desired attribute. To add a new attribute, select {user-defined} in the Name list and enter the name for the attribute. Enter the Value for the attribute and click **OK**, and the attribute is added to the table.
- **Editing Attribute Properties:** To edit an attribute's properties, select an attribute from the table and click the **Properties** button (or double-click the attribute) to open the *Place Attribute* dialog.
- **Deleting an Attribute:** Highlight an attribute in the table and click **Delete**.
- **Launching a Reference Link:** When the special attribute Reference, whose value is a reference link, is added to the pattern, you can select the Reference attribute and click **Launch** to launch an application or web address to display a document or web site.



## Documentation Tools

The Documentation Toolbox focuses on accelerating the development of a complete, informative documentation package that details the fabrication, assembly, and testing of the printed circuit board. This chapter includes the following topics:

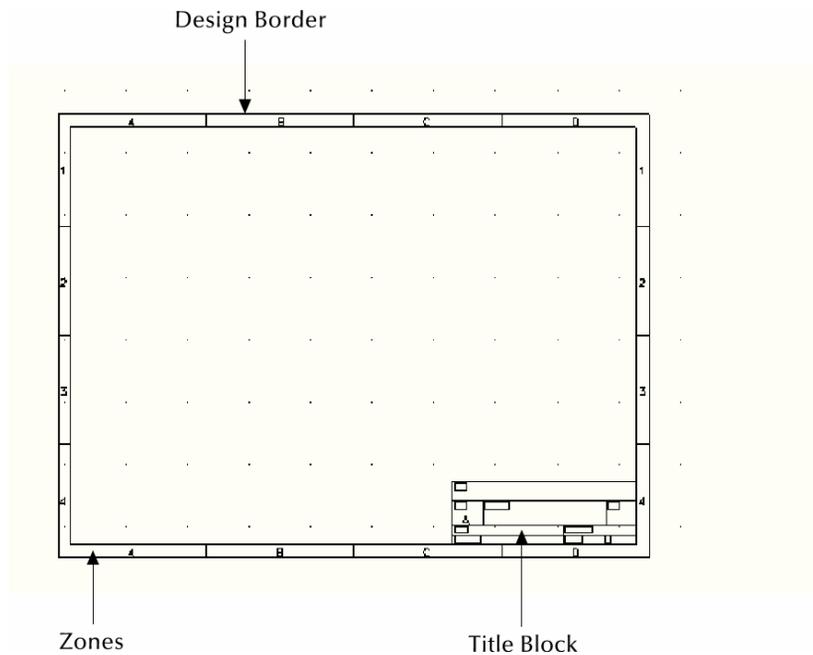
- Title Sheets
- Revision Blocks
- Fields and Field Sets
- Sheet Connector Cross Referencing
- Net Index Tables
- Note Tables
- Power and Ground Tables
- Revision Note Tables
- Spare Gate Tables
- Working with Tables
- Reporting on Schematic Designs
- Generating Reports
- Generating a Netlist

## Title Sheets

With P-CAD Schematic, you can create custom title sheets for each sheet in your schematic design or you can select one of P-CAD's standard title sheets to use on all of the sheets in your design.

- Custom title sheets give you the ability to add a design border, zones, and a title block independently, using the combination and layout that best fits your design needs. For instructions, see *Creating a Custom Title Sheet (page 101)*.
- Standard title sheets are a collection of title sheet templates that come with P-CAD Schematic. These sheets contain a design border, zones, and title block by default. For instructions, see *Using a Standard Title Sheet (page 100)*.

A title sheet can contain several items. As shown in the following figure, a title sheet can contain a design border, zones, and a title block.



Before you create a custom title sheet or use a standard sheet, you should be familiar with the basic components of a title sheet: design borders, zones, and title blocks.

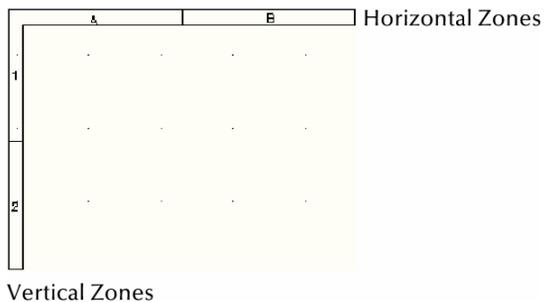
### Design Borders

Depending on your workspace size, the design border around a title sheet has a default dimension of 1/2 inch (500mils or 12.7mm) inside each edge of the workspace. The following table details the standard workspace sizes and the corresponding border dimensions in P-CAD Schematic:

Workspace Size	Dimensions (inches)	Workspace Size	Dimensions (millimeters)
A	8.5 x 11	A4	210 x 297
B	11 x 17	A3	297 x 420
C	17 x 22	A2	420 x 594
D	22 x 34	A1	594 x 841
E	34 x 44	A0	841 x 1187

### Zones

A design border and its zones are the primary components in a schematic title sheet. Like a road map, zones divide the horizontal and vertical areas of the design into various sections. The following figure shows you a portion of a design border and its zones:



The zone intelligence in P-CAD Schematic give you the ability to track nets between sheets and automatically include these net locations as annotations in your design. This feature is enabled by default, whether or not the zones are displayed in the workspace. However, to prevent overlay, it is recommended that you turn off the display of the border and zones when you use a standard title sheet.

### Title Blocks

You can place a title block on any sheet, which includes design information such as, Title, Author, Design Number and Revision fields. The following figure shows you a sample title block:

Title .....		
Size .....	Number .....	Rev .....
A .....	.....	.....
Date .....	Drawn by .....	
Filename .....	Sheet .....	of .....

The structure of a title block can be unique for each schematic sheet. P-CAD Schematic includes several schematic title block files that you can use in your design.

You can create your own title block files. In addition, you place fields within the block. These fields can also be unique to each sheet. For information, see *Fields and Field Sets (page 103)*.

In P-CAD Schematic, title block files have the \*.tbl file name extension. In P-CAD PCB, title block files have the \*.tbk file name extension.

After you are familiar with the basic components of a title, see one of the following topics to learn how to include title sheets and related components in your design:

- *Using the Global Title Sheet (page 100)*.
- *Using a Standard Title Sheet (page 100)*.
- *Creating a Custom Title Sheet (page 101)*.

## Using the Global Title Sheet

To use the global title sheet, follow these steps:

1. Open the *Options Sheets* dialog using one of the following methods:
  - Choose **Options » Sheets**.
  - Choose **DocTool » Titles**.
  - Choose **Options » Configure**. Then, click **Edit Title Sheets** in the Title Sheets frame.
2. When the *Options Sheets* dialog appears, click the **Titles** tab.
3. From the Sheets list, select the sheet to which you want to apply the standard title sheet.
4. Choose **Global**.
5. Click **Modify**.
6. Click **Close**.

Unless otherwise specified, the global title sheet appears on every sheet in your design by default.

## Using a Standard Title Sheet

A standard title sheet includes a design border and zones. Several title sheet files are available in the Titles folder, which is located in your installation directory. To use one of these title sheets in your design, follow these steps:

1. Open the *Options Sheets* dialog using one of the following methods:
  - Choose **Options » Sheets**.
  - Choose **DocTool » Titles**.
  - Choose **Options » Configure**. Then, click **Edit Title Sheets** in the Title Sheets frame.

2. When the *Options Sheets* dialog appears, click the **Titles** tab.
3. From the Sheets list, select the sheet to which you want to assign the standard title sheet.
4. In the Title Block frame, click **Select**. The *Open Title Block* dialog appears.
5. Select the title sheet file you want to use and click **Open**. Standard title sheet files are located in the `Titles` folder of the P-CAD installation directory and have the `*.ttl` file name extension.
6. (Optional) Turn off the display of the design border and its zones:
  - To turn off the display of the design border, clear the **Display Border** check box.
  - To turn off the display of the zones, clear the **Annotate Zone Information** check box.

To correspond to the zoned perimeter included on the standard title sheet, enter information in the Borders and Zones frames. However, it is recommended that you turn off the display of the design border and its zones to prevent overlay. The border selections specify the dimensions of the perimeter; the default border dimensions correctly specify the bounds of a standard title sheet. The zones selections specify the number of zones in the horizontal and vertical directions. The zone intelligence is enabled whether or not the zones are displayed and is included to annotate a sheet connector or to complete one of the tables of design data.

7. Click **Modify** to apply your changes to the selected sheet.
8. Click **Close**.

## Creating a Custom Title Sheet

To create a custom title sheet that you can later use in P-CAD Schematic designs, follow these steps:

1. Create a one-sheet design in P-CAD Schematic. For title sheets, you may place:
  - Lines
  - Arcs
  - Polygons
  - Text
  - Attributes
  - Fields
2. Choose **File » Save As**. The Save As dialog appears.
3. Navigate to the `Titles` folder in your P-CAD installation directory.
4. In the File name box, type the file name and file name extension as shown in the following example: `"title.ttl"`

When you type the file name, you must enclose the filename and file name extension in quotation marks (e.g., "TitleA.ttl").

5. Click **Save**.

You can now use the custom title sheet in your designs. For instructions on using a custom title sheet, see the following section.

## Using a Custom Title Sheet

To use a custom title sheet in a design, follow these steps:

1. Open the *Options Sheets* dialog using one of the following methods:
  - Choose **Options » Sheets**.
  - Choose **DocTool » Titles**.
  - Choose **Options » Configure**. Then, click **Edit Title Sheets** in the Title Sheets frame.
2. When the *Options Sheets* dialog appears, click the **Titles** tab.
3. In the Sheets list, select the schematic sheet on which you want to place the title sheet components.
4. Choose **Custom**. The options in the **Titles** tab become available.

If you choose **Global**, P-CAD Schematic uses the Global title sheet. The Global title sheet contains a design border, zones, and a title block by default.

5. In the Border frame, select the **Display Border** check box to show a border in the workspace.
6. In the Relative Origin frame, type a value in the Height and Width boxes. Then, type a value in the X and Y boxes.

Alternatively, select the **Size to Workspace** check box to use the default border size and location.

7. To add zones to the design border, follow these steps:
  - In the Horizontal frame, select the **Alpha** or **Numeric** button and the **Ascending** or **Descending** button.
  - In the Vertical frame, select the **Alpha** or **Numeric** button and the **Ascending** or **Descending** button.
  - Select the **Annotate Zone Information** check box.
  - Select a text style from the Text Style box.
8. If you want to add a title block, enter values in the File Name and Lower Right Offset boxes in the Title Block frame. For information, see *Title Block* (page 99).
9. (Optional) To add a title block, follow these steps:

- Click **Select**. When the *Open Title Block* dialog appears, select a \*.ttl file from the *Titles* folder in your P-CAD installation directory.
  - Select the **Field Set**, which includes the fields to place in the title block. For details on assigning a field set to a sheet, see *Fields and Field Sets* (page 103).
10. Click **Modify**. This assigns the chosen title sheet components to the selected sheets. Repeat from step 1 for other sheets if desired.
  11. When you click **Close**, the modified sheets show the border, zones, and title blocks specified.

## Revision Blocks

---

A revision block details the differences between versions of a design. In P-CAD Schematic, a revision block is placed as a Revision Note Table. For more information, see *Revision Note Tables* (page 114). The following figure shows you an example revision block, as it looks in a schematic design.

Revision Notes
A – Initial Release
B – Added C38 & C40 to provide additional by-pass capacitance for U15 & U18, per ECO # 1132

## Fields and Field Sets

---

A field acts as a placeholder for specific design information and you can insert a field anywhere in a design or title block. For example, if you insert the Title field in a design, P-CAD Schematic automatically displays the design title wherever the field is placed. After you place a field in a design, either a field code or field result appears.

P-CAD Schematic includes the following fields by default: Approved By, Author, Checked By, Company Name, Date, Drawing Number, Drawn By, Engineer, Revision, Time and Title. You can also create custom fields to include in a design.

As shown in the following illustration, a field code appears between the braces (e.g., {Drawing Number} or {Date}). A field result appears when any corresponding design information exists (e.g., *Title* (Digdemo.sch), *Size* (C), and *Rev* (5s)).

Title <b>DIGDEMO.SCH</b>		
Size <b>C</b>	Number {Drawing Number}	Rev 5a
Date {Date}		Drawn by <b>RWC</b>
Filename Digdemo.sch	Sheet <b>1</b> of <b>2</b>	

You can also group fields into field sets and then assign a set to a schematic sheet. P-CAD Schematic automatically updates the field information as field values change, based on the field set of the sheet on which they are located.

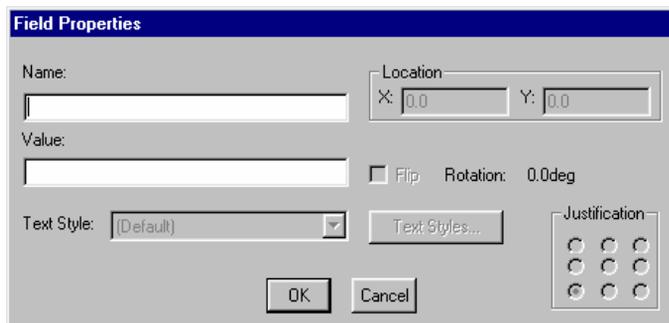
To learn how to use fields and field sets in a design, see one of the following topics:

- *Adding a Custom Field (page 104).*
- *Placing Fields in a Design (page 105).*
- *Changing Field Results (page 105).*
- *Adding a Field Set (page 105).*
- *Assigning Field Sets to Schematic Sheets (page 107).*

## Adding a Custom Field

To add a custom field to a design, follow these steps:

1. Choose **File » Design Info**. The *Design Info* dialog appears.
2. Click the **Fields** tab.
3. Click **Add**. The following *Field Properties* dialog appears.



4. Type a name for the custom field in the Name box.
5. Type a value for the custom field in the Value box.

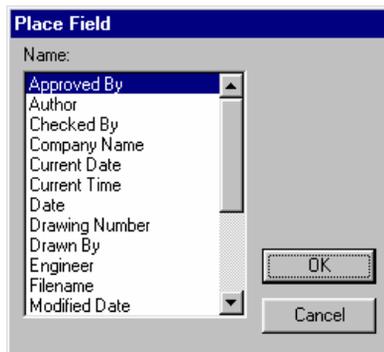
- Click **OK** to close the *Field Properties* dialog. Your custom field appears in the spreadsheet of the *Fields* tab.

To learn how to place fields in a design, see the following procedure.

## Placing Fields in a Design

To place a field in a design, follow these steps:

- Choose **Place » Field** or click the **Place Field** button in the *Placement* toolbar.
- Click the workspace. The *Place Field* dialog appears.



- Select a field from the Name list. For example, select **Approved By**.
- Click **OK** to close the *Place Field* dialog.
- Click the workspace in the location at which you want to place the field. The field code {Approved By} appears.
- To place another field, click the workspace again and repeat steps 3 - 5.
- To exit field placement mode, choose **Edit » Select** or another menu command.

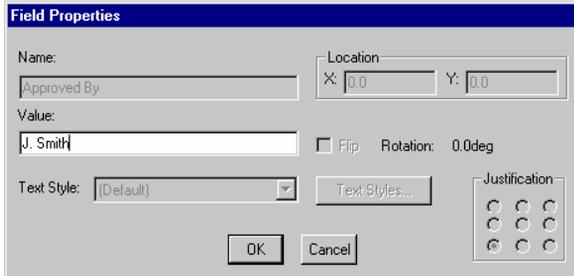
When design information exists for a field, a field result replaces the field code. For more information, see *Changing Field Results* (page 105).

## Changing Field Results

When corresponding design information exists for a field, a field result appears in place of the field code (e.g., J.Smith replaces {Approved By}). To change field results, follow these steps:

- Choose **File » Design Info**. The *Design Info* dialog appears.
- Click the **Fields** tab.
- Open the *Field Properties* dialog using one of the following methods:
  - Double-click the field name.

- Select the field name and click **Properties**.
4. When the *Field Properties* dialog appears, enter information in the *Value* box. For example, enter a name, as shown in the following figure:

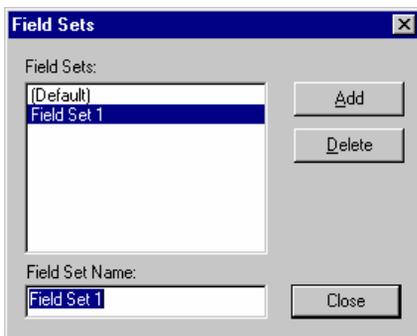


5. Click **OK** to close the *Field Properties* dialog.
6. Click **Close** in the *Design Info* dialog. The value you entered appears in the field placeholder.

## Adding a Field Set

You can group fields into distinct field sets, which can contain fields such as title, author, drawing number, etc. To create a field set, follow these steps:

1. Choose **File » Design Info**. The *File Design Info* dialog appears.
2. Click the **Fields** tab.
3. Click **Field Sets**. The *Field Sets* dialog appears.
4. Click **Add**. As shown in the following figure, the name Field Set 1 appears in the Fields Sets list and the Field Set Name box.



5. In the Field Set Name box, type a name for your new field set.
6. Click **Close** to return to the **Fields** tab of the *File Design Info* dialog. Your new field set is available in the Field Set list.

## Assigning Field Sets to Schematic Sheets

You can assign sheet-specific field sets to the sheets in a schematic design. To do this, follow these steps:

1. Open the *Options Sheets* dialog using one of the following methods:
  - Choose **Options » Sheets**.
  - Choose **DocTool » Titles**.
  - Choose **Options » Configure**. Then, click **Edit Title Sheets** in the Title Sheets frame.
2. Click the **Titles** tab.
3. Select the sheet to which you want to apply the field set from the Sheets list.
4. Select a field set from the *Field Sets* list. For instructions on adding sets to this list, see *Adding a Field Set* (page 106).
5. Click **Modify** to apply your changes to the selected sheet.
6. Click **Close**.

## Sheet Connector Cross Referencing

---

When a schematic sheet contains zones, you can cross-reference the sheet connectors. A sheet connector is annotated with the zone location of all other cross sheet connectors associated with the same net on all schematic sheets. For more information, see one of the following topics:

- *Sheet Connector Overview* (page 107)
- *Annotating Sheet Connectors* (page 108)
- *Viewing Sheet Connector Properties* (page 110)
- *Jumping to Sheet Connectors* (page 110)

### Sheet Connector Overview

To create a component to use as a sheet connector, you must save it as a Sheet Connector. Sample sheet connectors (i.e., SHEETIN, SHEETOUT) are located in the `Demo.lib` file in the `Demo` folder of your P-CAD installation directory.

A cross-referenced sheet connector is annotated with the sheet name or sheet number, as well as the alphanumeric zone location of all instances of sheet connectors belonging to the same net. A sample cross-referenced sheet connector is displayed in the following figure:



A sheet connector indicates that the attached net is continued on another sheet. It has no net intelligence, with the exception that when placed in a schematic design, the value attribute is set to the net name. You can display the net name, along with the sheet connector's zone location. For details, see *Annotating Sheet Connectors* (page 108).

If the value attribute of a sheet connector changes, it will automatically be reset to the net name when you load the design into P-CAD Schematic.

A sheet connector does not impact whether the net is local or global. For the net to be global, a sheet connector must be used in conjunction with a port. The port designates the net as global; the sheet connector provides the ability for cross sheet annotation.

These sheet connector cross-reference zone locations can be summarized in a net index table. This table can be placed on a schematic sheet for reference. For information, see *Net Index Tables* (page 111).

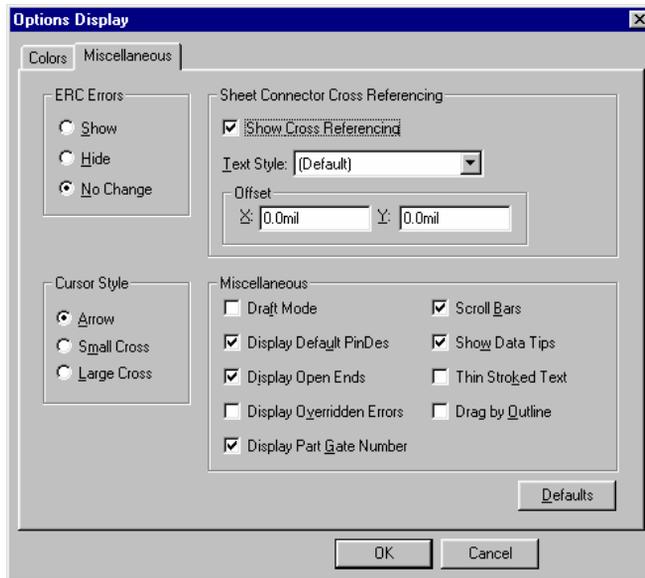
Zone locations of global nets are automatically included in the Global Nets report. For information, see *File Reports* (page 177).

## Annotating Sheet Connectors

When a net spans more than one schematic sheet, a sheet connector is a graphical indication that the net continues on another sheet. Using P-CAD Schematic's zone intelligence, you can record the sheet on which the net is continued. The sheet connector can be annotated with the connecting sheet's name and its zone location on that sheet.

To annotate a sheet connector, follow these steps:

1. Choose **Options » Display**. The *Options Display* dialog appears.
2. Click the **Miscellaneous** tab.
3. In the Sheet Connector Cross Referencing frame, select the **Show Cross Referencing** check box as shown in the following figure:



4. Select a text style for the cross-reference text from the Text Style list.
5. Specify the desired **X** and **Y Offset** from the part origin to the beginning of the cross-reference text.
6. Click **OK** to close the *Options Display* dialog.

The title sheet border must be zoned to include alphanumeric zone locations in the reference, although the zones need not be displayed.

When you choose to display cross-references, all sheet connectors on all sheets are annotated. If a net's location is moved or if a sheet connector is added or deleted from the net, the sheet connector cross-references for all sheet connectors associated with that net are automatically updated.

If updating the sheet connector cross-reference causes the annotation to fall outside the workspace, the annotation is relocated to its original offset point on the workspace. Subselect and drag the cross-reference text to move it.

Originally the cross-reference annotations for a single sheet connector are organized in a column at the offset point. You can subselect the annotations and relocate them. The relocated annotations will be automatically rescreened based on the sheet name. The resequencing order is from left to right, then from top to bottom, as shown in the following example:

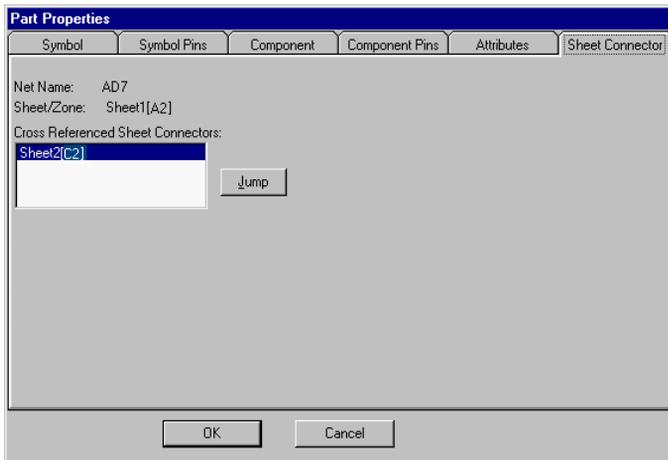
```
Sheet1 [A3] Sheet2 [C5] Sheet3 [H4]
Sheet4 [B2] Sheet5 [D1]
Sheet6 [J4]
Sheet7 [K5]
```

## Viewing Sheet Connector Properties

To view sheet connector properties, follow these steps:

1. Select a cross-referenced sheet connector.
2. Open the *Part Properties* dialog choosing one of the methods:
  - Choose **Edit » Properties**.
  - Right-click and choose **Properties** from the shortcut menu.

The *Part Properties* dialog appears, with the **Sheet Connector** tab selected:



This tab lists the net name, sheet, and zone information of the selected sheet connector, and all sheet connectors belonging to the same net on other sheets.

## Jumping to Sheet Connectors

You can jump to a sheet connector of the same net on another sheet:

1. Select a sheet connector.
2. Right-click and choose **Properties** from the shortcut menu. The *Part Properties* dialog appears.
3. Select the sheet connector to which you want to jump in the Cross-referenced Sheet Connectors list.
4. Click **Jump**. The focus of the window jumps to the selected sheet connector.

## Net Index Tables

A net index table summarizes the sheet connector cross-references in a design. This table can be placed on a schematic sheet as a reference of net locations. The following is a sample net index table:

Net Index Table	
Net Name	Sheets
+5V	PROCESSOR,DAA
+5VB	PROCESSOR,DAA
+12V	PROCESSOR
-5V	PROCESSOR,DAA
7.37MHZ	PROCESSOR[E7][18]
307KHZ	PROCESSOR[E9][18]
A0	PROCESSOR
A1	PROCESSOR
A2	PROCESSOR
A3	PROCESSOR
A4	PROCESSOR
A5	PROCESSOR
A6	PROCESSOR
A7	PROCESSOR
AD0	PROCESSOR
AD1	PROCESSOR

The sheet connectors must be cross-referenced between sheets in order to include zone information in a net index table. For details, see *Annotating Sheet Connectors (page 108)*.

When the nets are relocated or the sheet connector information of the current design is otherwise modified, the net index table should be updated to reflect the design changes. For instructions, see *Updating Tables (page 118)*.

You can also select a net index table and choose **Edit » Properties**. Through the *Properties* dialogs you can change a variety of object characteristics but not the table type or the design data contained within the table. With table properties, you can modify the table's line width, name, and text style.

A table is updated when its properties are modified.

### Placing a Net Index Table

To place a table in a design, follow these steps:

1. Choose **DocTool » Place Table**. The *Place Table* dialog appears.
2. Select **Net Index Table** from the Table Type list.

3. Type a name for the table in the Table Name box. The name of the table type appears in this box by default.
4. Select a text style for the table from the Text Style list.
5. Click **OK** to close the *Place Table* dialog.
6. Click the workspace at the location at which you want to place the table. A table cannot be placed outside the workspace.
7. (Optional) To place another table, click the workspace again. Then, repeat steps 2 - 6.
8. To quit placing tables, choose **Edit » Select** or another menu command.

## Note Tables

---

A note table contains any notes that you've added to your design. To learn how to place a notes table in a design, see *Placing a Notes Table* (page 113).

### Adding Notes to a Note Table

To add notes to a note table, follow these steps:

1. Choose **File » Design Info**. The *Design Info* dialog appears.
2. Click the **Notes** tab.
3. Click **Add**.
4. Type a note in the next available Note Text box.
5. Click **Close**.

After you add or delete revision notes to a table, you must update the table. For instructions, see *Updating Tables* (page 118).

### Importing Text Files as Notes

To import a notes file, follow these steps:

1. Choose **File » Design Info**. The *Design Info* dialog appears.
2. Click the **Notes** tab.
3. Click **Import**. The *Open* dialog appears.
4. Select the file to import and click **Open**. You return to the *Design Info* dialog.
5. Click **Close**.

### Exporting Notes to a Text File

To export a notes file, follow these steps:

1. Choose **File » Design Info**. The *Design Info* dialog appears.
2. Click **Export**. The *Save As* dialog appears.
3. Navigate to the directory in which you want to save the text file.
4. Type a name for the text file in the File name box.
5. Select **Text Files (\*.txt)** from the Save as type list.
6. Click **Save**.

## Placing a Notes Table

To place a table in a design, follow these steps:

1. Choose **DocTool » Place Table**. The *Place Table* dialog appears.
2. Select **Note Table** from the Table Type list.
3. Type a name for the table in the Table Name box. The name of the table type appears in this box by default.
4. Select a text style for the table from the Text Style list.
5. Choose the following options.
  - In the Note Numbering frame, choose **Top to Bottom** or **Bottom to Top**.
  - Enter a value in the Width of Note Numbering box.
6. Click **OK** to close the *Place Table* dialog.
7. Click the workspace at the location at which you want to place the table. A table cannot be placed outside the workspace.
8. (Optional) To place another table, click the workspace again. Then, repeat steps 2 - 6.
9. To quit placing tables, choose **Edit » Select** or another menu command.

## Power and Ground Tables

---

A power table summarizes the power and ground nets in a schematic. This table can be placed on a schematic sheet for reference. The following figure shows you a power table:

Power Table							
Ref Des	Device(Type)	Package	GND	VCC	+5V	DGND	+12V
C1	POLCAP	CAP300RP					PLUS
C5	POLCAP	CAP300RP					PLUS
C12	POLCAP	CAP300RP			PLUS		
C13	POLCAP	CAP300RP			PLUS		
C17	POLCAP	CAP300RP					MINUS
C18	POLCAP	CAP300RP					MINUS
U1	7400	DIP14	7	14			
U5	74LS373	DIP20	10	20			
U6	7408	DIP14	7	14			

The table includes the reference designator of all components that have power and ground pins. You can choose to include all pins, or restrict it to include hidden pins only. You can also restrict by reference designator prefix.

The power table summarizes the reference designator, component type, component pattern, and the power and ground nets for the component's power pins.

## Placing a Power Table

To place a power and ground table in your design, follow these steps:

1. Choose **DocTool » Place Table**. The *Place Table* dialog appears.
2. Select **Power Table** from the Table Type list.
3. Type a name for the table in the Table Name box. The name of the table type appears in this box by default.
4. Select a text style for the table from the Text Style list.
5. Choose the following options.
  - In the Pins to Include frame, choose **Hidden Pins Only** or **All Pins**.
  - In the Components to Include frame, choose **Only RefDes Prefix** or **All Components**.
6. Click **OK** to close the *Place Table* dialog.
7. Click the workspace at the location at which you want to place the table. A table cannot be placed outside the workspace.
8. To quit placing tables, choose **Edit » Select** or another menu command.

## Revision Note Tables

---

A revision block details the differences between versions of a design. An example revision block is shown as follows:

## Revision Notes

A – Initial Release

B – Added C38 & C40 to provide additional by-pass capacitance for U15 & U18,  
per ECO # 1132

In P-CAD Schematic, a revision block is placed as a Revision Notes Table. To learn how to place a table in the design, see *Placing a Revision Note Table* (page 116).

## Adding Revision Notes to a Table

To add revision notes to a table, follow these steps:

1. Choose **File » Design Info**. The *Design Info* dialog appears.
2. Click the **Revisions** tab.
3. Click **Add**.
4. Type a revision note in the next available Note Text box.
5. Click **Close**.

After you add or delete revision notes to a table, you must update the table. For instructions, see *Updating Tables* (page 118).

## Importing Text Files as Revision Notes

To import a text file as a revision note, follow these steps:

1. Choose **File » Design Info**. The *Design Info* dialog appears.
2. Click the **Revisions** tab.
3. Click **Import**. The *Open* dialog appears.
4. Select the file to import and click **Open**. You return to the *Design Info* dialog.
5. Click **Close**.

## Exporting Revision Notes to a Text File

To export a notes file, follow these steps:

1. Choose **File » Design Info**. The *Design Info* dialog appears.
2. Click the **Revisions** tab.
3. Click **Export**. The *Open* dialog appears.
4. Select the file to import and click **Open**. You return to the *Design Info* dialog.
5. Click **Close**.

## Placing a Revision Note Table

To place a table in a design, follow these steps:

1. Choose **DocTool » Place Table**. The *Place Table* dialog appears.
2. Select **Revision Note Table** from the Table Type list.
3. Type a name for the table in the Table Name box. The name of the table type appears in this box by default.
4. Select a text style for the table from the Text Style list.
5. Choose the following options.
  - In the Note Numbering frame, choose **Top to Bottom** or **Bottom to Top**.
  - Enter a value in the Width of Note Numbering box.
6. Click **OK** to close the *Place Table* dialog.
7. Click the workspace at the location at which you want to place the table. A table cannot be placed outside the workspace.
8. To quit placing tables, choose **Edit » Select** or another menu command.

## Spare Gate Tables

A spare gate table summarizes the last used, unused, and spare gates in a design. This table can be placed on a schematic sheet to determine which components have all parts included in the design. The following figure shows you a sample of a spare gate table:

Spare Gate Table		
Last Used	Not Used	Spare Gates
CB		
P12		P1:10, P1:20, P2:23, P2:24, P2:25, P2:26, P3:1, P3:3, P3:5, P3:7, P3:9, P3:11, P3:13, P3:15, P3:26, P7:18, P7:19, P7:20, P8:1, P8:4, P8:6, P8:7, P8:8, P8:9, P9:1, P9:4, P9:6, P9:7, P9:8, P9:9, P10:5, P10:6, P10:7, P10:8, P10:9, P10:10, P10:11, P10:12,
PWR28		
R13		
S2		
TB2		
U32		U5:B, U9:D, U11:D, U11 E, U11:F, U12:C, U12:D, U15:C, U15:D

The last used and not used columns contain the last placed and the remaining unused reference designators for each reference designator prefix included in the design. The spare gate column includes the reference designator and section number of any unused gate, when at least one section of the multigate component is included in the design.

## Placing a Spare Gate Table

To place a table in a design, follow these steps:

1. Choose **DocTool » Place Table**. The *Place Table* dialog appears.
2. Select **Spare Gate Table** from the Table Type list.
3. Type a name for the table in the Table Name box. The name of the table type appears in this box by default.
4. Select a text style for the table from the Text Style list.
5. Click **OK** to close the *Place Table* dialog.
6. Click the workspace at the location at which you want to place the table. A table cannot be placed outside the workspace.
7. To quit placing tables, choose **Edit » Select** or another menu command.

## Working with Tables

---

P-CAD Schematic gives you the ability to create and insert a variety of tables in your schematic design. Once you've placed a table in your design, you can modify and update the information using the procedures described in this section.

### Modifying Tables

To modify a table, follow these steps:

1. Select the table that you want to modify.
2. Choose one of the methods to open the *Table Properties* dialog.
  - Choose **Edit » Properties**.
  - Right-click and choose **Properties** from the shortcut menu.
3. (Optional) To change the table name, type a new name in the Table Name box.
4. (Optional) To change the text style, select a style from the Text Style list.
5. (Optional) To change the line width, type a new value in the Line Width box.
6. If you are updating a Power Table, you can modify the following information.
  - In the Pins to Include frame, choose **All Pins** or **Hidden Pins**.
  - In the Components to Include frame, choose **All Components** or **Only RefDes Prefix**.

7. Click **OK** to close the *Table Properties* dialog.

## Updating Tables

If you modify any of the data that is included in a table, you must update the table. To do this, follow these steps:

1. Choose one of these methods to update tables:
  - Select one or more tables and choose **DocTool » Update**.
  - Select one table. Then, right-click and choose **Update** from the shortcut menu.
  - Choose **DocTool » Update All** to update all of the tables and diagrams in your design.
2. A warning message appears to notify you that the update action cannot be undone. Click **Yes** to acknowledge this message.

## Reporting on Schematic Designs

---

The report options in P-CAD Schematic give you the ability to choose which fields appear in a report, the sort order, as well other formatting options such as lines per page, headers, footers, etc. These options are saved with the design file when you quit P-CAD Schematic.

### Reports Available in P-CAD Schematic

P-CAD Schematic provides you with a collection of standard reports to help you track and manage the schematic design process. The following reports are available:

- Attributes Report
- Bill of Materials Report
- Global Nets Report
- Last Used RefDes Report
- Library Contents Report
- Parts Location Report
- Parts Usage Report.

You can customize a standard report or you can create a custom report, based on one of the existing reports. For instructions, see *Customizing a Standard Report (page 119)* or *Creating a Custom Report (page 120)*.

### Attributes Report

The Attributes Report lists all of the design-level attributes that are assigned to the parts and nets in your schematic. The Attributes Report contains two sections: Component Attributes and Net Attributes.

For instructions, see *Generating Reports* (page 119).

## Bill of Materials Report

The Bill of Materials Report lists the parts in your schematic diagram, as well as any attributes you have selected for inclusion in the report.

To add attributes from a non-P-CAD source, click the **Import Files** button in the *Customize Report* dialog. For details on importing files, see the *P-CAD Library Executive User's Guide*.

The Count field is system-generated at the time the report is produced. Selection criteria cannot be applied to the Count field.

## Global Nets Report

The Global Nets Report lists the sheets on which a net appears, for each global net in your schematic design. A global net is any net that includes a port, a power symbol, or a hidden power pin. It can consist of one or more ported subnets, subnets connected by power symbols, one or more hidden power pins with the same net name, or a combination of all the above.

The sheet connectors must be cross-referenced between sheets in order to include zone information in a Global Nets report. For details, see *Annotating Sheet Connectors* (page 108).

## Last Used RefDes Report

The Last Used RefDes Report lists the last used reference designator for each component type in your schematic design.

## Library Contents Report

The Library Contents Report lists all of the components in any open libraries. To learn how to open a library, see *Library Setup* (page 325).

## Parts Location Report

The Parts Location Report lists the reference designator, sheet number, and sheet location of each part in your schematic design.

## Parts Usage Report

The Parts Usage Report lists all unused parts of multipart components in your schematic design. It can help you minimize the total number of components used in a design.

# Generating Reports

---

## Customizing a Standard Report

To customize one of the standard reports provided with P-CAD Schematic, follow these steps:

1. Choose **File » Reports**. The *File Reports* dialog appears.
2. Select a report in the Reports to Generate list.

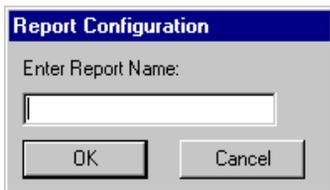
3. Click **Customize**. The *Customize Report* dialog appears. Notice that the title bar displays the report type.
4. Set your custom report options as appropriate.
5. Click **OK** to close the *Customize Report* dialog.

After you customize a report, you can generate it. For instructions, see *Generating a Report* (page 123).

## Creating a Custom Report

When you create a custom report, you base it on a standard report and give it a unique name. To create a custom report, follow these steps:

1. Choose **File » Reports**. The *File Reports* dialog appears.
2. Select the report on which you want to base your custom report from the Reports to Generate list.
3. Click **Customize**. The *Customize Report* dialog appears. Notice that the title bar displays the type of report you are customizing.
4. Set your report options as appropriate. For instructions, see one of the following topics:
  - *Setting Format Options* (page 121).
  - *Setting Data Selection Options* (page 121).
  - *Setting Sort Options* (page 122).
5. Click **Add**. The *Report Configuration* dialog appears:



6. Type a name and file name extension for the custom report in the Enter Report Name box. For example, type: Digdemo.bom
7. Click **OK** to return to the *File Reports* dialog.

Your new report is listed by name in the Reports to Generate list. You can now generate the report output. For instructions, see *Generating a Report* (page 123).

If you select and generate both the custom report and the one on which it was based, both results are output in a single file or report. To get individual outputs, generate each report separately.

## Setting Format Options

If you generate a report using the *P-CAD Report* or *Separated List* style format, you have various formatting options. To set these formatting options, follow these steps:

1. Click the **Format** tab.
2. If you are setting up the format for a *Separated List*, skip to step 6. Otherwise, continue with steps 3 - 7.
3. Type a header and footer in the Header and Footer boxes.
4. In the Page Format frame, select the following check boxes as appropriate:
  - **Use Header:** Includes the information you specified in the Header box.
  - **Use Footer:** Includes the footer information entered in the Footer box.
  - **Design Info:** Includes the information you entered in the **File Design Info** command and dialog.
  - **Date/Page:** Includes the current date and the page number.
  - **Pagination:** Allows you to create your own pagination (lines per page).
5. In the Lines per Page box, type the number of lines you want printed on each page.
6. Type a file name extension in the File Extension box. Each report has a file name extension that appears in this box by default.
7. In the *Separated List* frame, choose one of the following option buttons:
  - **Include Report Preface:** Select this check box to include a report preface.
  - **Include Column Headers:** Select this check box to include column headers at the top of your report.

To set the width of a single column, click the **Selection** tab. Next, choose a field from the spreadsheet. Then, click the **Format** tab and type a width for that column in the Column Width box. Columns are set to 20 characters in width by default.

## Setting Data Selection Options

To set data selection options for your report, follow these steps:

1. Click the **Selection** tab.
2. Select a section from the Report Section list. This list is only available for the *Attributes Report*.
3. In the Show column, select the check boxes that correspond to the fields you want to include in the report.
4. To change the order of the fields in the list, select a field and click **Move Up** or **Move Down**.

5. If you can add attribute fields to a report, the **Add Row** button is available. Click this button to open the *Select Attribute* dialog.
6. Select an attribute from the list and click **OK**. A row of data is added to the report spreadsheet in the **Selection** tab.
7. To set report criteria for a row of data, enter information in the appropriate cells of the Criteria (And) and Or columns. To learn how to enter information in these fields, see *Selecting Report Criteria* (page 122).
8. To add an Or column to the report spreadsheet, click **Add Column**. Then set additional report criteria.

## Setting Sort Options

1. Click the **Sort** tab.
2. Select a section from the Report Section list. This list is only available for the Attributes Report.
3. In the Available Fields list, select the fields you want to include in the report.
4. Add the field to the report using one of the following methods:
  - Select a field from the Selected Fields list and click **Insert**. P-CAD Schematic inserts the field above the field you select.
  - Click **Append**. The fields you want to include appear at the bottom of the Selected Fields list.
5. In the Sorting Order frame, choose **Ascending** or **Descending** to define the sort order for the list.

## Selecting Report Criteria

The following table shows the operators available for use in the Criteria (And) and the Or columns:

Use this operator	To perform this function
=	Exactly equal to  If used with a wildcard operator, * or ?, this operator becomes literal. It searches for a set of characters with, for example, a question mark at the end.
<	Less than
>	Greater than
<=	Less than or equal to
>=	Greater than or equal to
<>	Not equal to

IsLike	If used with a wildcard operator, IsLike means is similar to. For example, IsLike 5* could be 50, 510, 5, etc.  If not used with a wildcard operator, IsLike is equivalent to =.
IsNotLike	If used with a wildcard operator, IsNotLike means is not similar to. For example, IsNotLike 5* could be 14 or 20 or 42, but not 50, 510, or 5.  If not used with a wildcard operator, IsNotLike is equivalent to <>.
Exist	The attribute exists.
NotExist	The attribute does not exist.
AnyValue	The attribute exists and it has some value.
NoValue	The attribute exists, but it is assigned no value.

## Generating a Report

After you've set up your custom report options, you can generate a report. To do this, follow these steps:

1. Choose **File » Reports**. The *File Reports* dialog appears.
2. In the Reports to Generate list, select the check boxes that correspond to the type of reports you want to generate.
3. Click **Filename**. The *Reports File Save As* dialog appears.
4. Navigate to the directory in which you want to save the report file.
5. Type a name for the report file in the File name box.
6. Select **Report Files (\*.\*)** from the Save As type list.
7. Click **Save**. You return to the *File Reports* dialog.
8. In the Report Destination frame, choose one of the following buttons:
  - **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.
9. In the Style Format frame, choose one of the following buttons:
  - **Separated List**. Puts all data in character-separated format. This format can be imported into other spreadsheet and database programs.
  - **P-CAD Report**. Produces a report format with columns and spaces, etc.
  - If you choose **Separated List**, select a separator from the List Separator list.

The list separator character displayed in the box is used for both imported and exported files. The default character is your computer's regional setting.

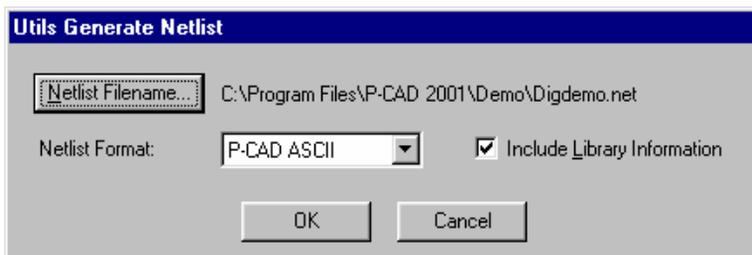
10. Set any customized options for an individual report type. For instructions, see *Customizing a Standard Report* (page 119) or *Creating a Custom Report* (page 120).
11. Click **Generate Reports**.

## Generating a Netlist

With P-CAD Schematic, you can generate netlist that lists the components and nets in your design. Typically, a netlist is used by a printed circuit board editor, such as P-CAD PCB, to form the basis of your board design.

To generate a netlist, follow these steps:

1. Open a schematic design file.
2. Choose **Utils » Generate Netlist**. The following *Utils Generate Netlist* dialog appears.



3. Click **Netlist Filename**. The following *Netlist File* dialog appears.



4. Navigate to the directory in which you want to save the netlist file.
5. Type a name for the netlist file in the File name box.

6. Select **Netlist Files** (\*.net) from the Save As type list.
7. Click **Save**. You return to the *Utils Generate Netlist* dialog.
8. In the Netlist Format list, select one of the following formats for your netlist:
  - P-CAD ASCII
  - Tango
  - FutureNet Netlist
  - FutureNet Pinlist
  - Master Designer
  - EDIF 2.0.0
  - Pspice
  - Xspice.

The P-CAD ASCII format supports attribute passing. If you select this option, you may also select the **Include Library Information** check box.

9. Click **OK** to generate the netlist. The netlist is saved with the filename and in the directory you specified.



## DDE Hotlinks

DDE Hotlinks provides a means of linking component and net highlight information between P-CAD Schematics and P-CAD PCB applications. This chapter includes the following topics:

- Setting up the DDE Hotlinks Feature
- Using DDE Hotlinks

### Setting up the DDE Hotlinks Feature

---

#### About DDE Hotlinks

The DDE Hotlinks feature gives you the ability to create a connection between two related design files created in separate P-CAD programs. With this feature, you can explore the relationships between designs when you run the following programs:

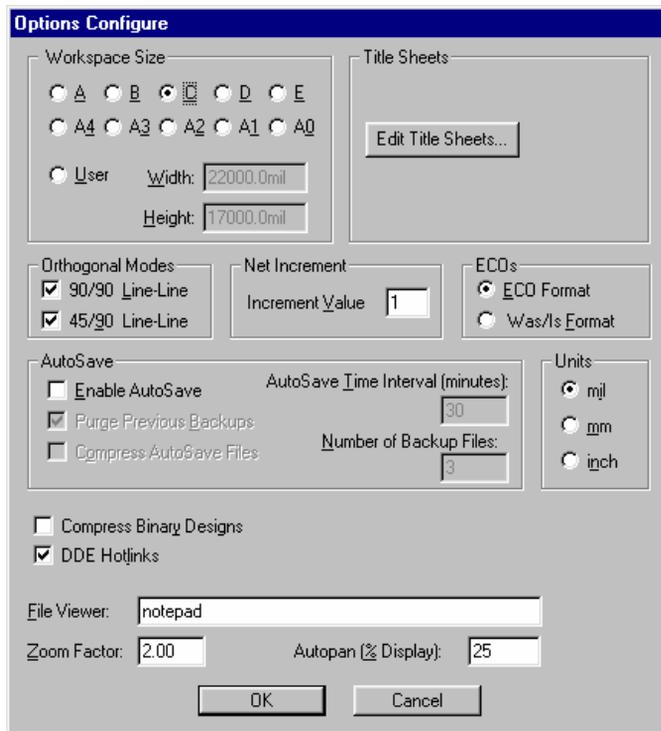
- P-CAD Schematic and P-CAD PCB/Relay
- P-CAD Schematic and P-CAD PCB Viewer

With DDE Hotlinks, you can update information in one design and the feature automatically updates the same information in a related design. For example, you could apply the current highlight color to a part in a Schematic design and the DDE Hotlinks feature updates the related part in P-CAD PCB. Or, when you highlight a net in a PCB design, the DDE Hotlinks feature highlights the corresponding net in a related Schematic design.

#### Enabling DDE Hotlinks

To the DDE Hotlinks feature in P-CAD Schematic, follow these steps:

1. Choose **Options » Configure** to open the *Options Configure* dialog.



2. Select the **DDE Hotlinks** check box.
3. Click **OK** to close the *Options Configure* dialog.

## Selecting the Current Highlight Color

To select the current highlight color, follow these steps:

1. Choose **Options » Display** to open the *Options Display* dialog.
2. Click the **Colors** tab.
3. Click **Highlight** to open the color palette.
4. Click a color in the palette.
5. Close the palette.
6. Click **OK** to close the *Options Display* dialog.

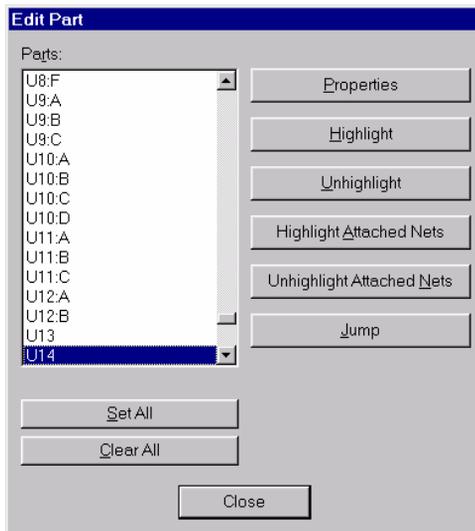
## Using DDE Hotlinks

---

### Highlighting Parts

To highlight parts so you can explore the relationships between parts in a Schematic and PCB/Relay design, follow these steps:

1. Start P-CAD Schematic and open a .sch design file.
2. Start P-CAD PCB and open a .pcb design file.
3. Choose **Edit » Parts** to open the *Edit Part* dialog.



4. Select a part in the Parts list box.
5. Click **Highlight**.
6. Click **Close** to exit the *Edit Parts* dialog box. The part remains selected in P-CAD Schematic and highlighted in P-CAD PCB/Relay.
7. Click anywhere in the P-CAD Schematic workspace to cancel the selection of the. The part appears in the current highlight color.

### Unhighlighting Parts

1. Choose **Edit » Parts** to open the *Edit Part* dialog.
2. Select a part in the Parts list box.
3. Click **Unhighlight**.

4. Click **Close** to exit the *Edit Parts* dialog box and to remove the highlight color from the part in both programs.
5. Select a part in the P-CAD Schematic workspace.

When you rest the mouse pointer over the parts in your design, a Data Tip appears.

6. Right-click the part and choose **Highlight** from the shortcut menu. The part remains selected in P-CAD Schematic and the corresponding part becomes highlighted in P-CAD PCB/Relay.
7. Click anywhere in the P-CAD Schematic workspace to cancel the selection of the part. The part appears in the current highlight color.
8. Right-click the part again and choose **Unhighlight** from the shortcut menu.

## Highlighting and Unhighlighting Attached Nets

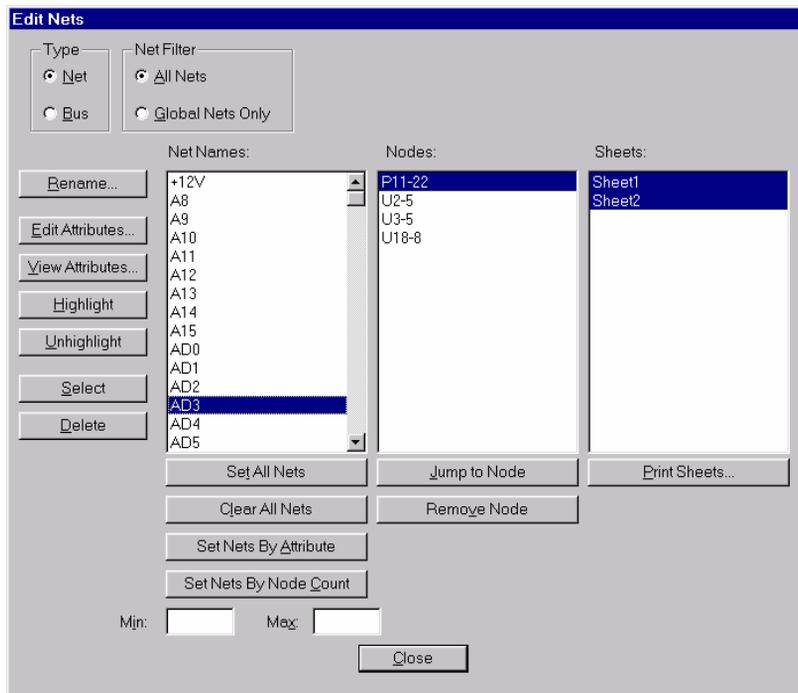
To explore the relationship between the nets in a Schematic and PCB/Relay design, follow these steps:

1. Select a part in P-CAD Schematic.
2. Right-click and choose **Highlight Attached Nets** from the shortcut menu. In both programs, all attached nets become highlighted, included bus connections, junctions, ports, wires, and pins.
3. Select the again. Then, right-click and choose **Unhighlight Attached Nets** from the shortcut menu. This removes the highlight color from all nets attached to the selected part.
4. In P-CAD Schematic, choose **Edit » Parts** to open the *Edit Parts* dialog box.
5. Select part **U23** in the Parts list.
6. Click **Highlight Attached Nets**. In both programs, all attached nets become highlighted, included bus connections, junctions, ports, wires, and pins.
7. Click **Close** to exit the *Edit Parts* dialog box. Notice that the nets appear in the current highlight color.
8. Choose **Edit » Unhighlight All** to remove the highlight color from all of the objects.

## Highlighting Nets

To highlight nets in a design, follow these steps:

1. Choose **Edit » Nets** to open the *Edit Nets* dialog.



2. Select **AD3** in the Net Names list box.
3. Click **Highlight**.
4. Click **Close** to exit the *Edit Nets* dialog box.

## Selecting All Highlighted Objects in a Design

When you want to perform editing functions on a group of highlighted objects, follow these steps:

1. Highlight one or more objects in your design.
2. Choose **Edit » Select Highlighted** to select all of the highlighted objects in a design

If a highlighted object is not selected when you choose **Edit » Select Highlighted**, open the *Options Block Selection* dialog and change your block selection criteria.



## Verifying a Schematic Design

P-CAD provides powerful Electrical Rules Checking (ERC) capabilities. The information contained in this chapter includes the following topics:

- Configuring the ERC Feature
- Using ERC Error Annotation

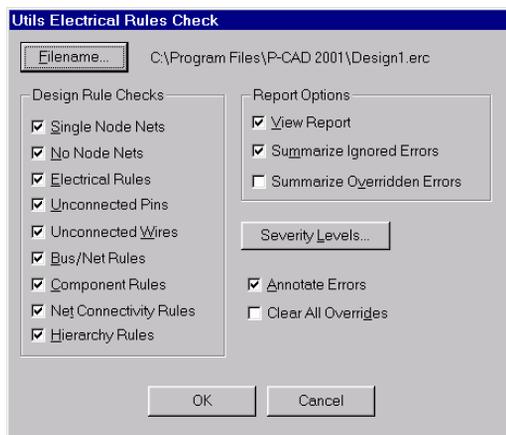
### Configuring the ERC Feature

#### Setting up Checks and Severity Levels

An electrical rules check verifies that your design does not violate the rules set up in the *Utils Electrical Rules Check* dialog.

To set up these rules, follow these steps:

1. Choose **Utils » ERC** to open the *Utils Electrical Rules Check* dialog.

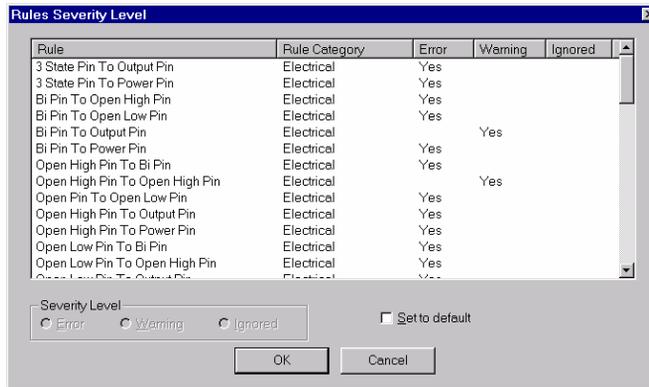


By default, a file name for the ERC report automatically appears next to the **Filename** button.

2. Click **Filename** to open the *Electrical Rules Check Report* dialog box.
3. Change the name of the file in the File name box.
4. Select the check boxes of your choice in the Design Rule Checks frame, to apply specific rules to the design for validation.
5. Select the **View » Report** check box in the Report Options frame to automatically display the report when the validation process is complete.

Before you click **OK** to start the validation process, you will set up the ERC report options and choose **Severity Levels** as described in the next lesson.

6. Select all of the check boxes in the Design Rules Checks frame. The following describes each option:
  - **Single Node Nets:** Reports all nets with only one node.
  - **No Node Nets:** Reports all nets with no nodes.
  - **Electrical Rules:** Reports incompatible pin types connected together. For example, two output pins connected together or an output pin connected to a power pin.
  - **Unconnected Pins:** Reports all pins that are not connected to any other pins (single node nets). This includes pins that are not connected to anything at all.
  - **Unconnected Wires:** Reports all wires that are unconnected (floating). An unconnected wire is one that is not attached to a pin.
  - **Bus/Net Rules:** Reports on nets only referenced once in a bus (i.e., a wire goes into a bus, but doesn't come out).
  - **Component Rules:** Reports on all components that are on top of other components.
  - **Net Connectivity Rules:** Reports on power net errors. This option is discussed in more detail later in this chapter.
  - **Hierarchy Rules:** Reports on module and link errors. Each module must have a link attribute with pins of matching electrical type and quantity. These module and link pins must be connected. All link pins must be placed within the design and have an attribute referencing a valid link component.
7. Select the **Annotate Errors** check box to annotate all errors.
8. Click **Severity Level** to open the *Rules Severity Level* dialog.



9. Select one or more rules in the list box. Then, change the severity level to **Ignored**. To do this, choose the **Ignored** button in the Severity Level frame.
10. Click **OK** to return to the *Utils Electrical Rules Check* dialog.
11. To summarize the errors you set to a severity level of **Ignored**, select the **Summarize Ignored Errors** check box.
12. Click **OK** to validate the design.

The system performs its Electrical Rules Checking according to the options you have specified.

- ERC error indicators are created for you to display, select, and view if the severity level has been set to **Error**.
- Rules with a severity level of **Warning** are written to the report but not annotated in the design.
- The rules whose severity level was set to **Ignored** are not annotated in the design, but will be summarized in the report, since you selected the **Summarized Ignored Checks** check box.

Every time you choose **Utils ERC**, all existing ERC error indicators are removed.

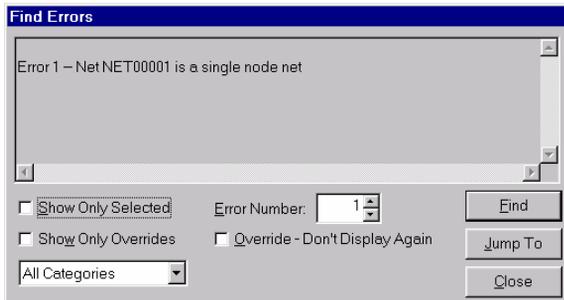
## Using ERC Error Annotation

The lessons in this section explain how to use the ERC error annotation options.

### Finding ERC Errors

To find ERC errors, follow these steps:

1. Choose **Utils » Find Errors** to open the *Find Errors* dialog box.



- From the Categories drop-down list, select **Net Connectivity**. Information about the first error in that category appears in the description area. The last error you viewed remains in the description area until another error is requested.

If your design has a number of errors, such as unconnected pins, and you do not want to see those errors when scrolling through the list of errors in the *Find Errors* dialog, check the **Override – Don't display this error again** option. Conversely, you can choose to display only the errors that have overrides applied to them by enabling the **Show Only Overrides** option.

- View another error by using one of the following methods:
  - If you know the number of the error, enter it in the Error Number box and click **Find**. Information about that error appears in the description area.
  - You can scroll through the errors using the up and down arrows next to the Error Number box until you find the error you want to view.
  - Click **Jump To** and the error finder positions the cursor in the center of the error indicator in your design.

## Overriding Error Displays

The display of overridden, annotated errors can be globally controlled. To do this, follow these steps:

- Choose **Options » Display** to open the *Options Display* dialog.
- Click the **Miscellaneous** tab.
- Select the **Display Overridden Errors** check box.

When enabled, any error indicator whose display has been overridden will appear in the design as an inverted triangle.

To hide overrides from view, clear the **Display Overridden Errors** check box in the **Miscellaneous** tab of the *Options Display* dialog. When the check box is enabled, an overridden error indicator appears as an inverted triangle in your design.

## Block Selecting Error Indicators

If you want to include ERC error indicators in a block selection,

1. Choose **Options » Block Selection** to open the *Options Block Selection* dialog.
2. Click **Clear All** to cancel the selection of check boxes in the Items frame.
3. Select the **ERC Errors** check box in the Items frame.
4. Click **OK** to close the *Options Block Selection* dialog box.

## Overriding ERC Errors

Individual ERC error indicators can be removed from the design without being resolved when you override their display. You may want to hide errors such as unconnected pins when the errors will not be resolved and you do not want to see them in the design.

To apply an override, follow these steps:

1. Select an error.
2. Right-click and choose **Override** from the shortcut menu.

To remove an override from a selected error, right-click and choose **Unoverride** from the shortcut menu.

## Fixing and Deleting ERC Errors

The recommended method for using ERC error annotation to fix design errors is as follows:

1. Choose **Utils » Find Errors** to view the errors.
2. Display an error you want to correct and go directly to it in the design by clicking the **Jump To** button. Continue resolving the errors in the design in this same manner until all errors have been fixed.
3. Select an error indicator and press the **DELETE** key. You can delete the ERC error indicators from your design as they are resolved.
4. Another way to delete ERC error indicators is to choose **Utils » ERC**; all existing error indicators are deleted, and the program flags only the remaining errors. Or you can leave the ERC errors on the sheet. The next time you choose the **Utils » ERC** command, all existing ERC error indicators will be removed automatically.



## Printing a Schematic

This chapter includes the following topics:

- Setting up Printers and Plotters
- Setting up Print Jobs.

### Setting up Printers and Plotters

---

This section explains how to set up a printer or plotter in P-CAD Schematic. Before set up a print or plot job, you must install a printer or plotter on your system. For instructions, see your Windows™ documentation.

#### Print Features

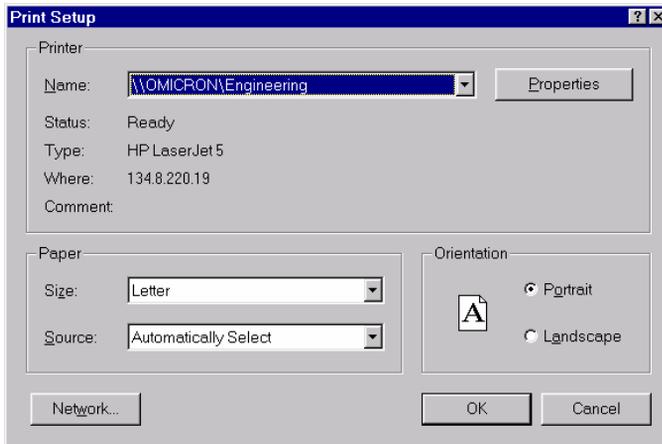
Printing in P-CAD Schematic contains many useful features:

- Flexible print options, where each sheet in a print job can be defined with unique printing options such as preset and custom scaling, print region, X and Y offset, and more.
- Color options, where you can customize color output (if you are using a color printer).
- Preview, where you can verify the layout of the output before sending to the printer.
- Batch printing of multiple sheets.

#### Selecting a Printer or Plotter

To select the printer or plotter that you want to use from P-CAD Schematic, follow these steps:

1. Choose **File » Print Setup** to open the *Print Setup* dialog.



2. Select a print device from the Name list. This list shows the printers and plotters that have been installed on your system.
3. Select a paper size and paper source from the corresponding list boxes in the Paper frame.
4. Choose **Portrait** or **Landscape** in the Orientation frame.
5. Click the **Properties** button to open a dialog box in which you can specify other print parameters. Then, click **OK** to close the *Properties* dialog.

Because print parameters are device-specific, the options in the *Properties* dialog depend upon the print device you selected.

6. Click **OK** to close the *Print Setup* dialog box.

You have selected a print device. You can now define your page setup options.

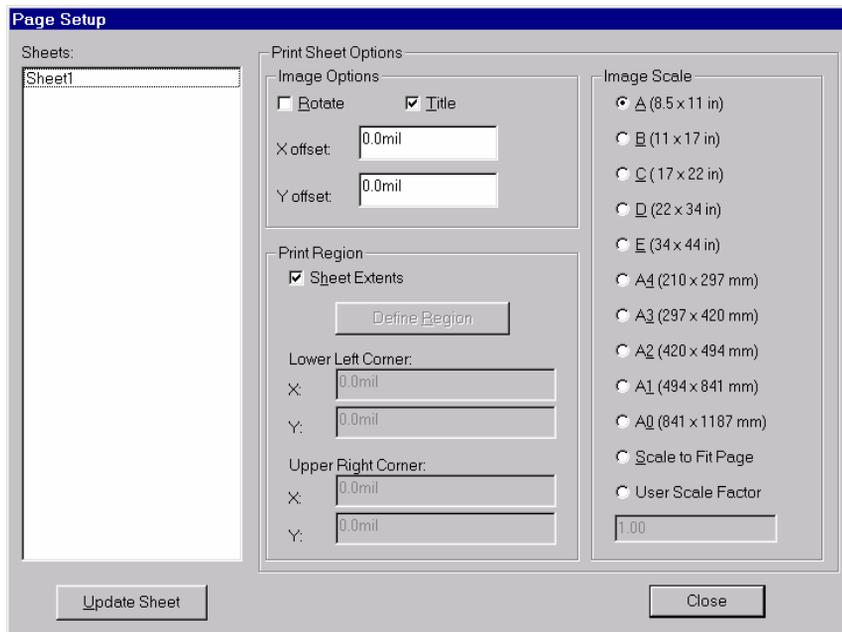
## Setting up Print Jobs

---

### Defining Image Options, Image Scale, and a Print Region

For each sheet, you can scale your image to fit standard or custom size papers, define a print region, set horizontal and vertical offsets, display a title, and rotate the image.

1. Choose **File » Print** to open the *File Print* dialog.
2. Click **Page Setup** to open the *Page Setup* dialog.



You can set different print options for each sheet in a design. However, this procedure shows you how to define your page setup options for a single sheet.

3. Select **Sheet 1** in the Sheets list.
4. Choose one of the buttons in the Image Scale frame to select an image scale to the selected sheet.

Although you will not choose the **Scale to Fit Page** button in this tutorial, choosing this option would scale the print region to fit on a single page. Page size is determined by your print driver.

5. Select the **Title** check box in the Image frame to display a title around your design. A title prints only if you have set one up with the *Options Configure* dialog.
6. In this tutorial, you will define a print region in your design. To do this, clear the **Sheet Extents** check box to enable the other fields in this frame. Then, click **Define Region**. The workspace appears.

If you prefer to print the entire design, select the **Sheet Extents** check box in the Print Region frame and skip to step 7. Select this option only if you want to print the entire design.

7. Hold down the left mouse button and drag the pointer across the workspace to draw a bounding outline around the desired print region.

8. Right-click or press **ESC**. An information message appears to confirm the coordinates of the selected print region. Click **Yes** to accept these coordinates.
9. A message appears, warning you to apply these changes to the print job/sheet. Click **OK** to acknowledge this message.

The *Page Setup* dialog reappears. In the Print Region frame, the X and Y coordinates display for the lower left and upper right corners of the print region.

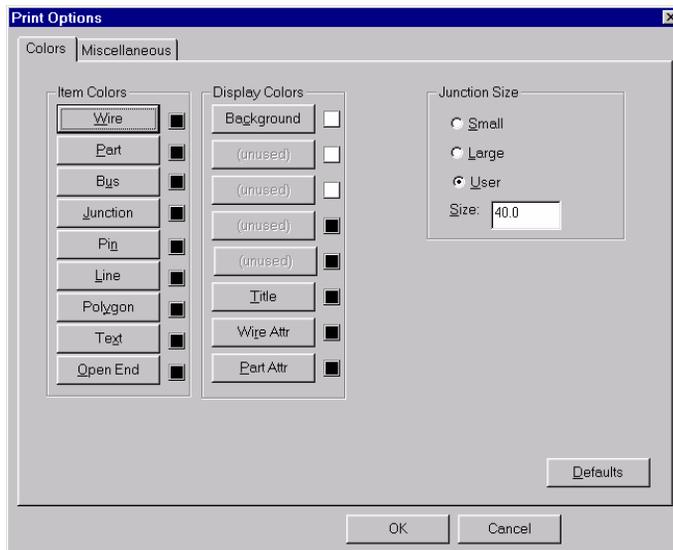
10. Click **Update Sheet** to apply these changes to the print job/sheet.
11. Click **Close** to close the *Page Setup* dialog and return to the *File Print* dialog.

You have set up your page options. Next, you will set up your print options. For instructions, see *Setting your Print Options* (page 142).

## Setting your Print Options

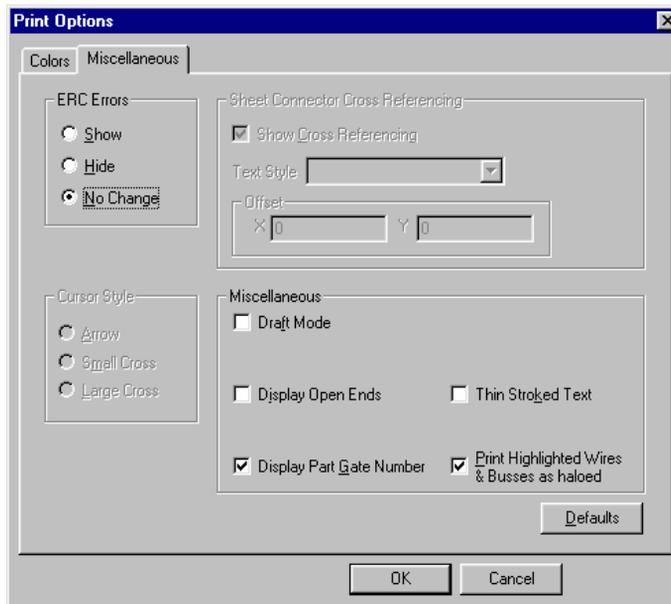
With the *Print Options* dialog, you can set up various print options. This dialog contains two tabs: *Colors* and *Miscellaneous*. To use this dialog, do the following:

1. Choose **File » Print** to open the *File Print* dialog.
2. Click **Print Options** to open the *Print Options* dialog.
3. Click the **Colors** tab.



4. Choose your print colors using one of these methods:
  - If you have a color printer, click a command button in the *Item Colors* or *Display Colors* frame. When the color palette appears, click a color swatch and then close the palette.

- If you have a black and white printer, click the **Defaults** button. This turns all colors to monochrome. We recommend that you use the defaults setting to avoid undesirable output when color settings are converted to grayscales.
5. In the Junction Size frame, choose the **Small**, **Large**, or **User** radio button.  
Notice that you can enter a value in the Size text box only when you choose the **User** radio button. You can enter a value in inches, centimeters, etc. However, the value must be less than or equal to 100 mils.
  6. Click the **Miscellaneous** tab. In this tab, you set the size or ERC error indicators and reference points. In addition, you set other miscellaneous print options as shown in the following figure:



To use the default settings, click the **Defaults** button.

7. In the ERC Errors frame, choose one of these buttons:
  - **Show**. Enables the display of ERC errors in your printed design.
  - **Hide**. Disables the display of ERC errors in your printed design.
  - **No Change**. Keeps the current display setting.
8. Select these check boxes in the Miscellaneous frame:
  - **Draft Mode**: Enables a thin outline of the design components in the printed output.
  - **Thin Stroked Text**: Prints text in its stroke mode.

- **Display Open Ends:** Prints an indicator showing any open ends.
  - **Display Part Gate Number:** Enables the printing of Part Gate Numbers.
  - **Print Highlighted Wires & Busses as haloed:** Distinguishes any wires or busses that are currently highlighted as haloed lines.
9. Click **OK** to return to the *File Print* dialog.

You have set up your print options. Next, you will learn how to preview your print job.

## Previewing a Print Job

Before you print a document, you can view it in a print preview window. To open this window, click the **Print Preview** button in the *File Print* dialog. The print preview window contains these options:

- **Print:** Sends the current print jobs to the printer.
- **Next Page:** Displays the next page of the print output in the Print Preview window.
- **Prev Page:** Displays the previous page of the print output in the Print Preview window.
- **Two Page/One Page:** Allows you to view two pages at a time or only one page.
- **Zoom In:** Zooms in on the center area of the current page. Use the scroll bars to move to the desired viewing region.
- **Zoom Out:** Zooms out on the current page.
- **Close:** Closes the Print Preview window.

## Generating Printouts

After you have defined your page setup and print options, return to the *File Print* dialog to print the sheets. The following shows you how:

1. In the *File Print* dialog, click **Set All** to enable all the sheets in the design.
2. Click **Generate Printouts** to print the highlighted sheets.

Now that you've generated a printout, you will learn how to use the override settings on print a job.

## Override Settings

You have two options for overriding your print settings. You can print your current schematic window, or you can scale the print area to fit on a single page. The following lessons guide you through each option.

## Printing the Current Window

To print the current window, follow these steps from your design workspace:

1. Use the **View** commands to select the desired window in Schematic. For example, zoom in on an area in your current design.

2. Choose **File » Print**. The *File Print* dialog appears.
3. Highlight the sheet to print in the Sheets list box.
4. In the Override Settings frame, select the **Current Window** check box.

Selecting the **Current Window** check box overrides the **Print Region** settings for the current sheet's print job.

5. Click **Generate Printouts**. P-CAD Schematic prints out the print job with the region defined by the current window. The printout may be on one page, or on many pages, depending on the size of the window and the printer settings.

## Scaling to Fit Page

When printing, you can override the page settings for all sheets so that the print output is scaled to fit on a single page. To print scaled print jobs, follow these steps:

1. Choose **File » Print** to open the *File Print* dialog.
2. Select the **Scale to Fit Page** check box in the Override Settings frame.
3. Select the desired print job(s) in the Sheets list. To define a new sheet print job, click the **Page Setup** button and follow the instructions in *Defining Image Options, Image Scale, and a Print Region* (page 140).
4. Click **Generate Printouts**. P-CAD Schematic prints out the contents of the selected sheets. The print output is scaled to fit exactly on a page. The page size is set by the print driver. See your print driver's options for the page sizes.

The **File » Print** Scale to Fit Page overrides the scale settings for all print jobs. To scale an individual sheet's print job to fit a single page, use the **Scale to Fit Page** check box in the *Page Setup* dialog.



# Design Technology Parameters

The Design Technology Parameters (DTP) feature gives you the ability to capture design information, including net class definitions and design rules. With this feature, you can create and save design information in a DTP file, which can then be used in any P-CAD design. This chapter includes the following topics:

- Introducing the DTP Feature
- Opening and Creating DTP Files
- Working with DTP Files

## Introducing the DTP Feature

---

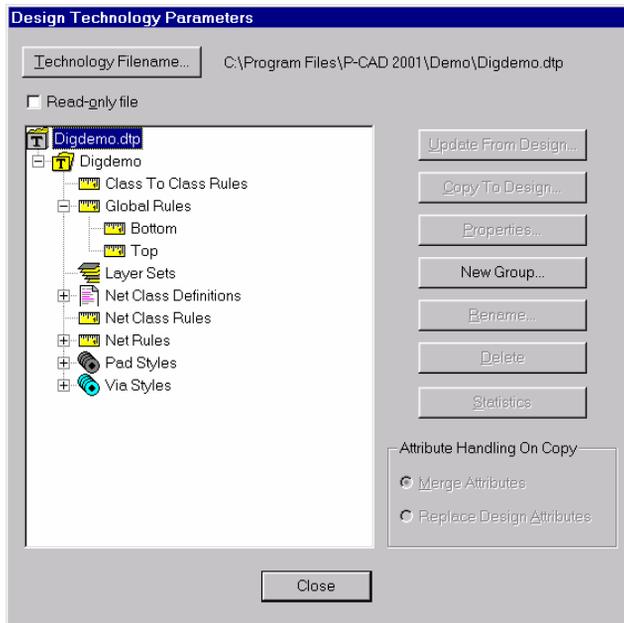
### What is DTP?

The Design Technology Parameters (DTP) feature gives you the ability to capture design information, including net class definitions and design rules. With this feature, you can create and save design information in a DTP file, which can then be used in any P-CAD design.

### About DTP Files

A design technology parameters file (\*.dtp) gives you the ability to create and save information that is associated with various aspects of a P-CAD design, such as Class to Class rules, net class definitions, and more.

The data in a DTP file is organized in a hierarchical structure with the following levels: Files, Groups, Sections, and Items. As shown in the following figure, the `Digdemo.dtp` file contains a group named `Digdemo`. Within this group are several Sections, such as `Class To Class Rules`, `Global Rules`, and so on. Each section contains a collection of items. For example, the `Global Rules` section contains two items: `Bottom` and `Top`.



When you create a design, you can also create a DTP file and organize the information as appropriate. You can then build, browse, and modify the hierarchy of your DTP file. To do this, use the options in the *Design Technology Parameters* dialog shown in the previous figure.

## Opening and Creating DTP Files

The procedures in this section familiarize you with the DTP file creation process.

### Opening a DTP File

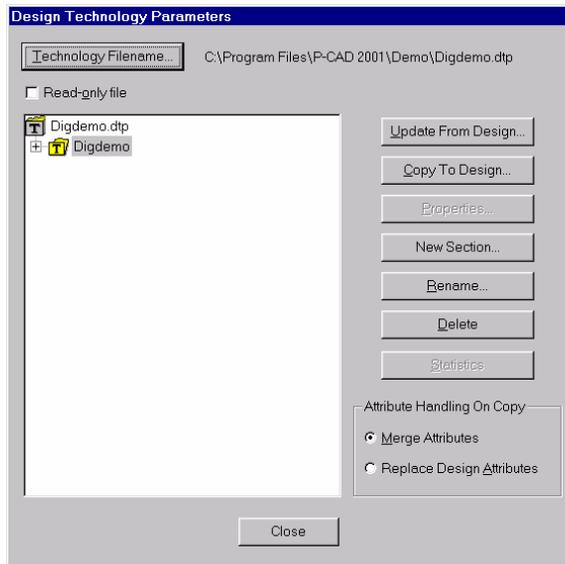
To create a new DTP file or to open an existing file, follow these steps:

1. Start P-CAD Schematic and open a design file. For example, open `Digdemo.pcb`, which is located in the `Demo` folder of your P-CAD installation directory.
2. Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog. The system path and DTP file name appears next to the Technology Filename button.

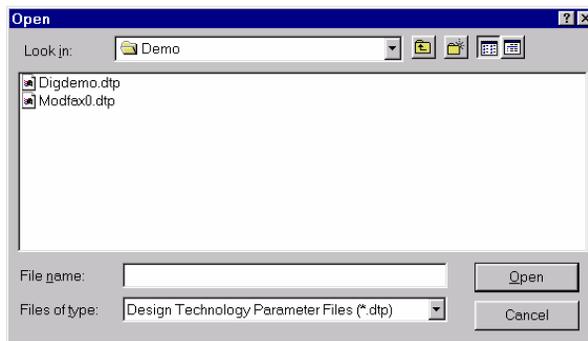
### Creating a DTP File

To create and save a DTP file, follow these steps:

1. Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog.



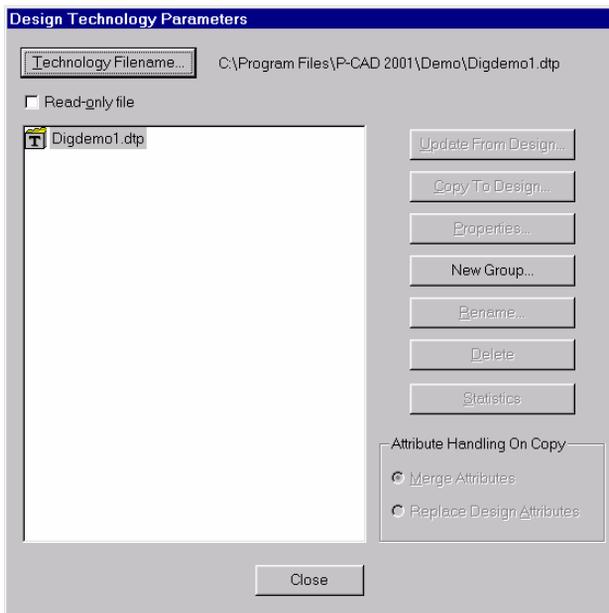
2. Click **Technology Filename** to display the *Open* dialog.



3. Type a name for your DTP file in the File name box. For example, type: Digdemo1
4. Select **Design Technology Parameter Files (\*.dtp)** in the Files of type list.
5. Click **Open** to display the following *Open* dialog:



- Click **Yes**. The new DTP file name appears next to the Technology Filename button in the *Design Technology Parameters* dialog, as shown in the following illustration.



When you create a DTP file, only the Filename appears in the list box. To add groups, sections, and items, see one of the following sections: *Adding a Group* (page 150), or *Adding a Section* (page 151).

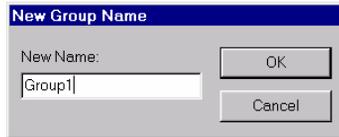
- Click **Close** to exit the *Design Technology Parameters* dialog.

## Adding a Group

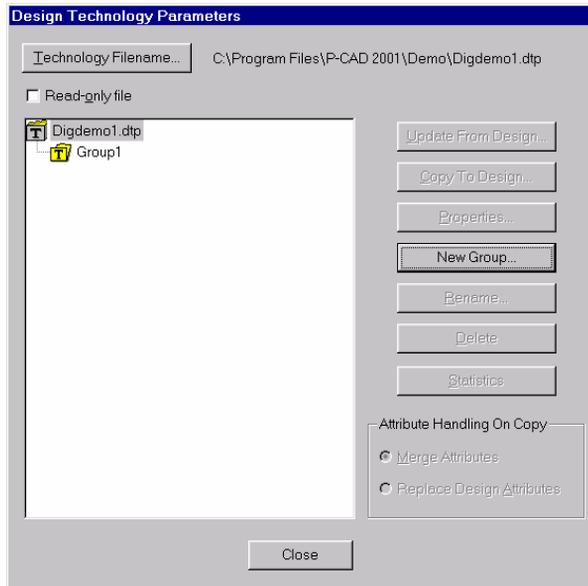
To add a group to a DTP file, follow these steps:

- Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog.

2. Open the DTP file that you want to modify by clicking the **Technology Filename** button. For example, open `Digdemo1.dtp`.
3. Click **New Group** to open the *New Group Name* dialog.
4. Type a name in the New Name box. For example, type: `Group1`



5. Click **OK** to close the *New Group Name* dialog. The new group name appears in the DTP tree, as shown in the following figure:



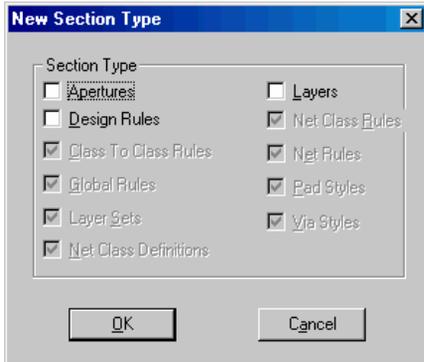
6. Click **Close** to exit the *Design Technology Parameters* dialog.

## Adding a Section

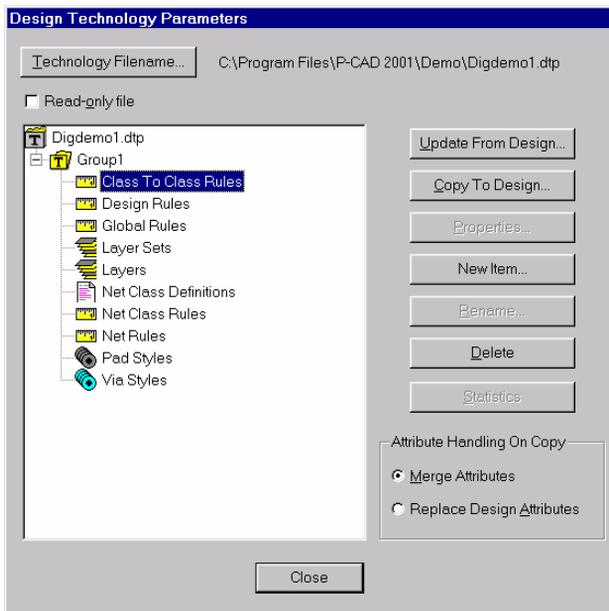
To add a section to group in a DTP file, follow these steps:

1. Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog.
2. Open the DTP File of your choice by clicking the **Technology Filename** button. For example, open `Digdemo1.dtp`.

3. Select a group in the DTP tree. For example, select **Group1**. The label on the New Group button changes to New Section.
4. Click **New Section** to open the *New Section Type* dialog.



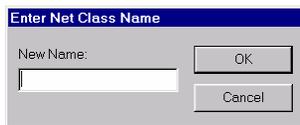
5. Select the check boxes that correspond to the sections you want to create.
6. Click **OK** to close the *New Section Type* dialog. The selected sections appear in the DTP tree, as shown in the following figure:



## Adding an Item

To add an item to a section in a DTP file, follow these steps:

1. Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog.
2. Open the DTP File of your choice by clicking the **Technology Filename** button. For example, open `Digdemo1.dtp`.
3. Select a section in the DTP tree in the *Design Technology Parameters* dialog. For example, select **Net Class Definitions**. The label on the New Section button changes to New Item.
4. Click **New Item** to open the *Enter Net Class Name* dialog.



5. Type a name in the New Name box. For example, type: `ITEM1`
6. Click **OK** to close the *Enter Net Class Name* dialog. The new item appears in the appropriate section of the DTP tree.

## Working with DTP Files

---

Once you created a DTP file, you can browse and modify the contents of that file. In addition, you can update a file with information from your active design, as well as copy DTP data to your design. This section contains procedures that show you how to perform these actions and more.

### Browsing the Contents of a DTP File

To browse the contents of a DTP file, follow these steps:

1. Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog.
2. Click **Technology Filename** to display the *Open* dialog.
3. Navigate to the directory in which the DTP file is located. For example, navigate to the `Demo` folder in the P-CAD installation directory.
4. Select the DTP file that you want to open from the list. For example, select `Digdemo.dtp`.
5. Click **Open**. The DTP filename appears next to the Technology Filename button.

If a `.dtp` file is read-only, the Read-only check box is selected and the term read-only appears next to the filename. You cannot modify or delete a read-only DTP file.

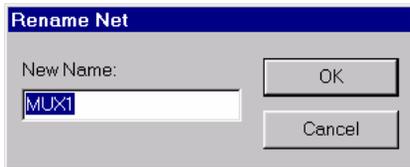
6. Expand a group or section by clicking the + sign. When a group or section is expanded, the - sign appears.
7. Collapse a group or section by clicking the – sign. When a group or section is collapsed, a + sign appears.

Use the options in the *Design Technology Parameters* dialog to set up the design technology parameters, copy parameter data to or from a design, view or modify data properties, and browse or modify the file's hierarchy. To learn more about these options, see the appropriate section in this chapter.

## Renaming a Group or Item

To rename a group or item, follow these steps:

1. Select a group or item in the DTP tree.
2. Click **Rename** to open the *Rename Net* dialog.

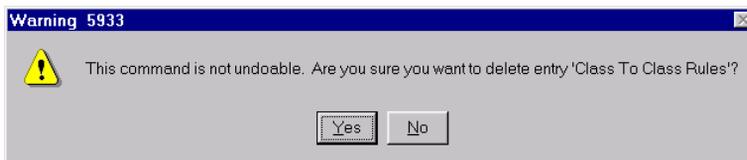


3. Type a name in the New Name box.
4. Click **OK** to close the *Rename Net* dialog. The new name appears in the DTP tree.

## Deleting DTP File Information

To delete information from a DTP file, follow these steps:

1. Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog.
2. Select a group, section or item in the DTP tree.
3. Click **Delete** to open a *Warning* dialog.

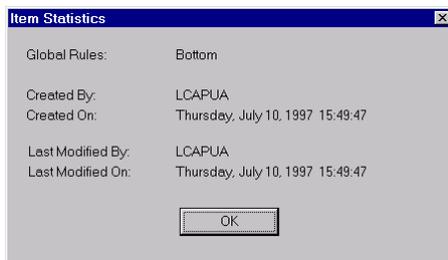


4. Click **Yes** to delete your selection. If you are deleting a group or section, all objects within that group are deleted. This action cannot be undone.

## Viewing Item Statistics

To the statistics of an item in a DTP file, follow these steps:

1. Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog.
2. Select an item in the DTP tree to make the Statistics button active. Click **Statistics** to open an *Item Statistics* dialog.

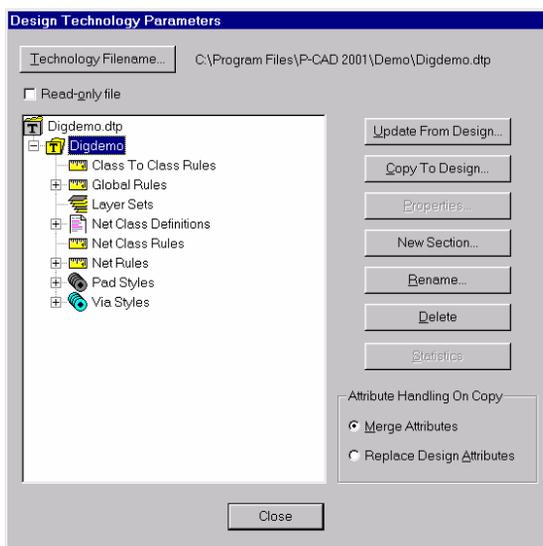


3. Click **OK** to close the *Item Statistics* dialog. This dialog contains information about modifications made to the item.

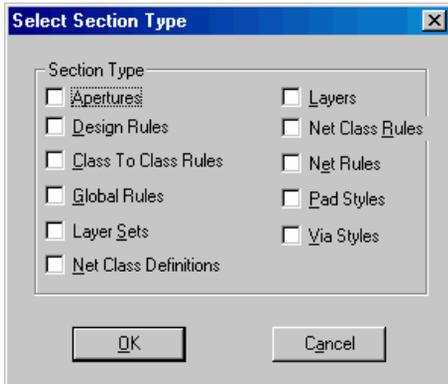
## Updating a DTP Group with Design Information

To update a group in a DTP file with information from the active design, follow these steps from the *Design Technology Parameters* dialog:

1. Select a group in the DTP tree. For example, select *Digdemo*, as shown in the following figure.



2. Click **Update From Design** to open the *Select Section Type* dialog.
3. Select the check boxes that correspond to the sections you want to update. For example, select all of the check boxes, as shown in the following figure.



4. Click **OK** to close the *Select Section Type* dialog. If the update is successful, the following message appears.



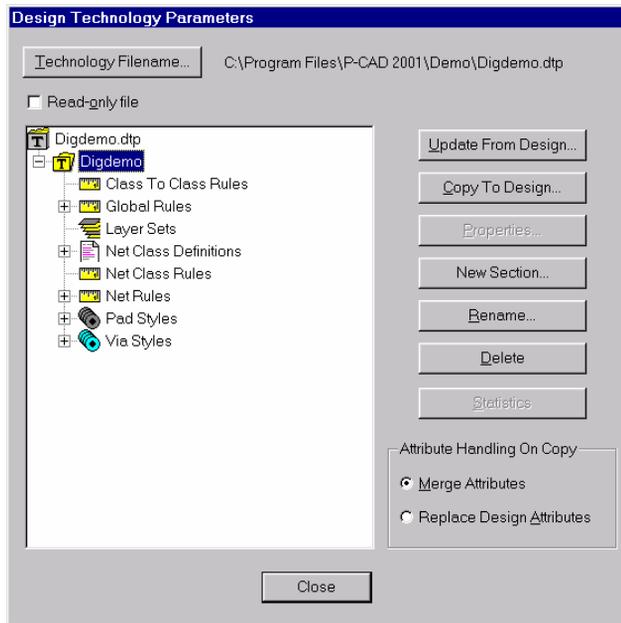
5. Click **OK** to close the Information message box.

You can also update individual sections in a DTP file by selecting a section from the DTP tree and clicking **Update from Design**.

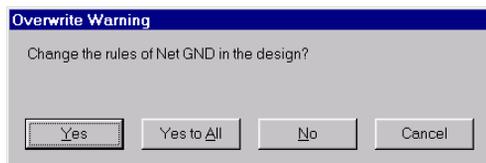
## Copying DTP Data to a Design

To copy DTP data to a P-CAD design, follow these steps:

1. Choose **File » Design Technology Parameters** to open the *Design Technology Parameters* dialog.
2. Select a group or section in the DTP tree. For additional information about your options, see *Setting up Attribute Handling for Copy Actions* (page 158).



3. Click **Copy To Design**. If the information that you want to copy will change any existing information, the *Overwrite Warning* dialog appears.



4. Click one of the following buttons to acknowledge the message:
  - Click **Yes** to overwrite the information specified in the *Message* dialog.
  - Click **Yes to All** to overwrite the information listed in the *Message* dialog, as well as any other potential warnings associated with the section.
  - Click **No** to keep the information as is.
  - Click **Cancel** to cancel the copy action.

If the selected item does not exist in the design, it is added automatically. Net rule and net class definition items are not created if they do not exist in a design.

## Setting up Attribute Handling for Copy Actions

You can copy sections to a design: Class To Class Rules, Net Class Definitions, Net Class Rules, and Net Rules. When you select one of these sections in your DTP tree, the options in the Attribute Handling on Copy frame become active as shown in the following figure:



The following describes each option in the Attribute Handling On Copy frame:

- **Replace Design Attributes:** Choose this button to replace the rules associated with an item in a design with the rules associated with an item in the DTP file.
- **Merge Attributes:** Choose this button to merge the rules associated with the item in the DTP file with rules associated with the item in the design. When you choose this button, P-CAD Schematic favors the rules of the DTP file over the rules of the design.

For information about item-specific options, see *Item-Specific Information About Copy Actions* (page 158).

## Item-Specific Information About Copy Actions

The following information explains how P-CAD Schematic supports the copy action for various objects in a DTP tree.

**Class to Class Rules.** If a design has two net classes, you can merge or replace the class to class rules. If a class to class rule doesn't already exist, a rule is created. To merge or replace class to class rules, choose the appropriate button in the Attribute Handling on Copy frame before you click the **Copy to Design** button.

- **Replace Design Attributes:** Choose this option to place a design's class-class rules with the rules from the Class to Class Rule Item.
- **Merge Attributes:** Merges the rules of the *Class to Class Rule Item* with the design's class-to-class rules, favoring the Class to Class Rule Item rules over the design.

If both net classes do not exist in the design, a class to class rule is not created.

**Layer Rules.** Layer rule items cannot be updated from P-CAD Schematic.

**Layer Sets.** Layer set items cannot be updated from P-CAD Schematic.

**Net Class Definitions.** If a net class of the same name exists in the design, its net members are replaced with the net members defined by this Item. The net class rules are not changed. If a net class of the same name does not exist in the design, it is not created.

**Net Class Rules.** If the net class already exists in the design, you have a choice of two options: to merge or to replace the net class rules. To choose between the merge or replace options, choose

the corresponding option in the *Design Technology Parameters* dialog before clicking **Copy to Design**.

- **Replace Design Attributes:** Replaces a design's net class rules with the rules from the Net Class Rule Item.
- **Merge Attributes:** Merges the rules of the Net Class Rule Item with the design's net class rules, favoring the Net Class Rule Item rules over the design.

The net class, with the Net Class Rule Item rules, is created if it does not already exist in the design.

**Net Rules.** If the net already exists in the design, you have a choice of two options: to merge or to replace the net rules. To choose between the merge or replace options, choose the corresponding option in the *Design Technology Parameters* dialog before clicking **Copy to Design**.

- **Replace Design Attributes:** Replaces a design's net rules with the rules from the Net Rule Item.
- **Merge Attributes:** Merges the rules of the Net Rule Item with the design's net rules, favoring the Net Rule Item rules over the design.

If the net does not already exist in the design, the net, with its Net Rule Item rules, is not created.

**Pad Styles.** Pad style items cannot be updated from Schematic.

**Via Styles.** Via style items cannot be updated from Schematic.

## Modifying Item Properties

To view or modify the properties of an item in a DTP tree, follow these steps:

1. Choose **File Design Technology Parameters** to open the *Design Technology Parameters* dialog.
2. Select an item in the DTP tree.
3. Click **Properties** to open an item-specific dialog.

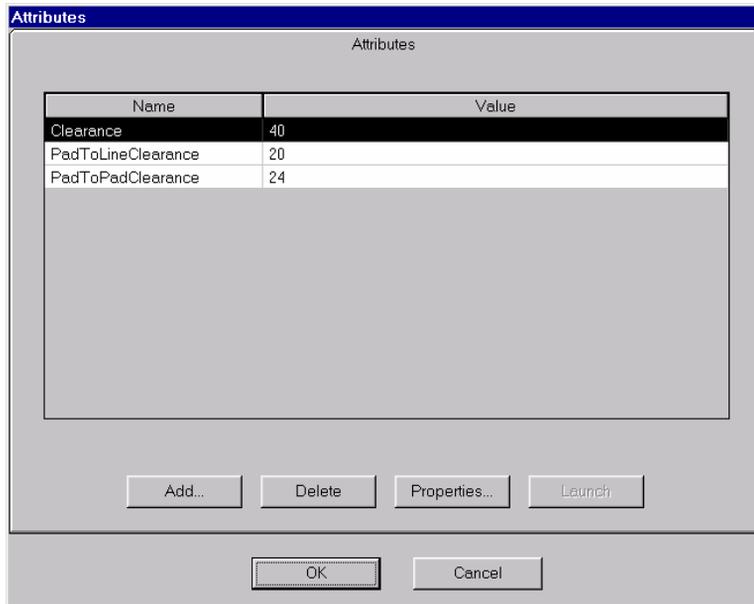
Properties of certain items (pads, vias, global rules, layer sets) cannot be accessed in P-CAD Schematic. In such cases, the Properties button is shaded and not available.

When you modify properties of items in your parameters file using *Properties*, the changes are made immediately to the file. If you want to create a new design technology file based on an existing file, first make a copy of your original design technology file, saving the copy under the new name. Then, modify the design technology file copy.

If the design technology parameters file is read-only, P-CAD Schematic discards any changes you've made.

### Class to Class, Net Class, and Net Rules Properties

To open the *Attributes* dialog, select a class to class rule, net class rule, or net rule item in the File Design Technology Parameters tree. Then, click **Properties**.



The two-column spreadsheet in the *Attributes* dialog shows the collection of net attributes. Within the collection, each attribute's name and value appear in the column.

- **Adding an Attribute:** To add an attribute, click **Add** to open the *Place Attribute* dialog. Add a pre-defined attribute by choosing first the Category and then the desired attribute. To add a new attribute, select {user-defined} in the *Name* list and type the name for the attribute. Type the *Value* for the attribute and click **OK**, and the attribute is added to the table. Complete instructions on using the *Place Attribute* dialog are found in *Place Commands* (page 259).
- **Editing Attribute Properties:** To edit an attribute's properties, select an attribute from the table and click the **Properties** button (or double-click the attribute) to open the *Place Attribute* dialog.
- **Deleting an Attribute:** Highlight an attribute in the table and click **Delete**.
- **To launch a reference link:** When the special attribute Reference, whose value is a reference link, is added to the item, you can select the Reference attribute and click **Launch** to start a program or to launch Internet Explorer to reference additional information about the item.

### Global Layer Rules Properties

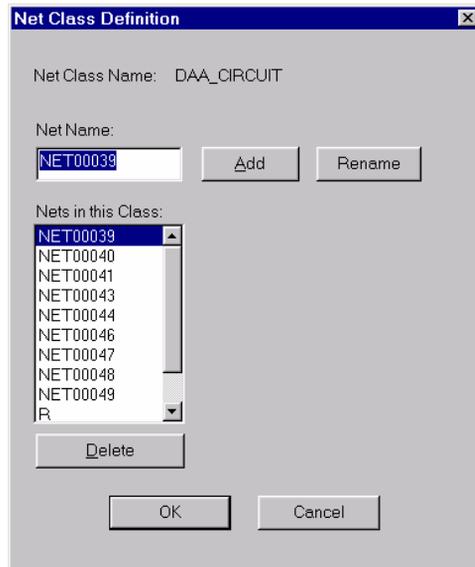
P-CAD Schematic does not support Layer rule properties.

### Layer Set Properties

P-CAD Schematic does not support Layer set properties.

### Net Class Definition Properties

To open the *Net Class Definition* dialog, select a net class definition item in the File Design Technology Parameters tree. Then, click **Properties**.



Use this to create net classes and add nets to the class. For details, see *Options Net Classes* (page 309).

To add a net class definition:

1. Type a layer name in the Net Name box.
2. Click **Add**.
3. Repeat the process until you have added all nets to the class.

### Pad Style Properties

Pad style properties are not available from Schematic.

### Via Style Properties

Via style properties are not available from Schematic.



## Simulating a Circuit Design

With the Mixed-Signal Circuit Simulator you can perform an array of mixed-signal simulations on your design. Simulations are run directly from your schematic (including multi-sheet designs), providing an easy way to investigate the performance of a circuit throughout the design cycle.

The analog/mixed-signal simulation engine uses an enhanced version of Berkeley SPICE3f5/XSPICE, allowing you to accurately simulate any combination of analog and digital devices without manually inserting D/A or A/D converters. This “mixed-signal” or “mixed-mode” simulation is possible because of the inclusion of accurate, event-driven behavioral models for digital devices, including TTL and CMOS.

Due to the complexity of digital devices it is generally not practical to simulate them using standard, non-event-driven, SPICE instructions. For this reason, the simulator includes a special descriptive language that allows digital devices to be simulated using an extended version of the event-driven XSPICE. The digital devices included in the simulation-ready schematic libraries are modeled using the Digital SimCode language.

The simulator allows unlimited circuit-level analog simulation and unlimited gate-level digital simulation. Circuit size is only limited by the amount of RAM you have in your system.

The Mixed-Signal Circuit Simulator runs in a separate application, called the Design Explorer. When you select Run from the new Simulate menu in the P-CAD 2002 Schematic Editor the schematic is analyzed and an XSPICE netlist is generated. This netlist is automatically opened in the Design Explorer, a simulation performed, and the results displayed as waveforms in the Design Explorer's oscilloscope-like waveform display window.

The types of analyses supported include: AC small signal, Transient, Noise and DC transfer, Monte Carlo analysis, parameter and temperature sweeping, and Fourier analysis. Analyses can be configured in either the P-CAD 2002 Schematic Editor, or in the Design Explorer. Refer to the on-line help for more information on using the circuit simulator.

## Using the P-CAD Circuit Simulator

---

### Configuring the Schematic

Before you can successfully simulate your circuit you need to ensure that your schematic contains all the necessary information. In general, the following rules must be adhered to before you will be able to run any of the available simulations:

- All components and parts in the schematic must properly reference a simulation device model.
- You must place and wire up suitable signal sources to provide drive to the circuit during simulations.
- The circuit must include a Ground net.
- If necessary, you must set the initial simulation conditions of the circuit using the .IC and .NS devices.

### Selecting simulation-ready schematic components

To perform simulation analyses, all parts placed on your schematic must contain special simulation-specific information which tells the simulator how these components are to be treated. In general this means that schematic parts must include a reference to an appropriate SPICE device model. This information is defined as attributes of the component symbol.

The easiest way to create a schematic for simulation is to use simulation-ready parts from the new P-CAD 2002 component libraries. There is a complete Library Index included with the new P-CAD 2002 component libraries (Library Index.XLS). Refer to the *New features.WRI* file for details on how to use this index to locate simulation-ready components. In most cases you simply place a simulation-ready component, set the value (such as the resistance) and simulate. Each component includes all the information required for SPICE simulation, including designator prefix information and pin mapping for multiple-gate components.

SPICE supports many extended features that allow you to more accurately model component behavior, for example specifying the operating temperature of a transistor. This extra information is entered into the pre-defined SimFields in the **Attributes** tab of the components' Properties dialog after the component has been placed on the schematic sheet.

### Setting up the Simulation Analyses

To select which simulation analyses are enabled, select **Simulate » Setup** from the P-CAD 2002 Schematic Editor menus. The *Analyses* dialog will pop up, the required analyses are enabled in the **General** tab of the dialog.

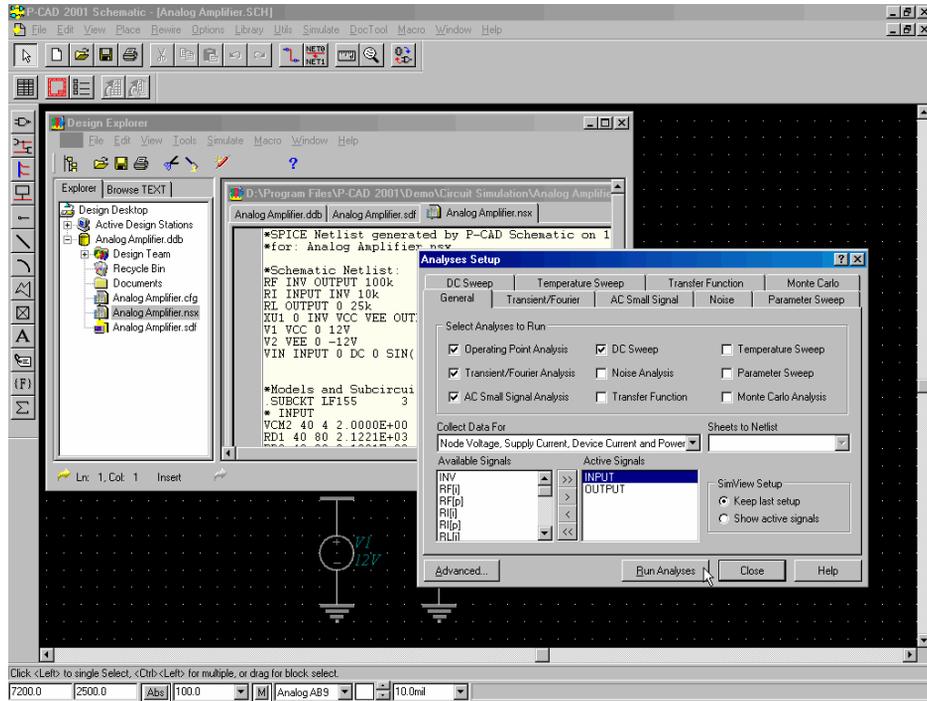
When you run a simulation, data for all enabled analyses will be calculated and displayed in the simulation waveform viewer window. Click the **Help** button in this dialog for information about each type of analysis, and for information on the other options in the *Analyses Setup* dialog.

## Running a Simulation

Once the required analyses are enabled and the parameters of the enabled analyses are configured in the appropriate tab in the dialog, you are ready to run the simulation.

To run a simulation:

1. Choose **Simulate » Setup** from the main menu in P-CAD Schematic to setup the analysis. The Design Explorer window opens and the *Analyses Setup* dialog displays.



Alternatively, choose **Simulate » Run** and the simulation is run straight away. You can also run the analysis from the Design Explorer menus when the simulation netlist is the active document.

2. Select the criteria for the simulation in the appropriate tab of the *Analyses Setup* dialog and click **Run Analysis**. The Circuit Simulator runs the analysis. Refer to the online help for more information about the setup dialog.

An Xspice netlist is generated from the design and placed in the same directory as the schematic. This is then imported into a design database (.ddb file) of the same name as the schematic and the database opened in the Design Explorer 99 SE. The netlist that was generated in P-CAD Schematic is then loaded and passed to the simulation engine.

The first time a simulation is run on a design, a default configuration file (.cfg) will be created. When you make any changes in the *Analyses Setup* dialog, they are stored in this .cfg file and subsequently applied to future simulations of that particular design. The setup information in the configuration file is added to the netlist when the simulation is run. No setup information is contained in the netlist generated by P-CAD.

You can also generate the Xspice netlist from your P-CAD Schematic design, using the **Utils » Generate Netlist** command. You are then free to import the netlist into Design Explorer 99 SE and run the simulation at a later stage, from the Mixed-Signal Circuit Simulator.

Information can also be directly added into the netlist, by manually editing the .nsx file using the Design Explorer's Text Editor. Upon running the simulation, a dialog will appear, giving you the option to use the netlist file as is, or to ignore the setup information that has been added and use the configuration file instead.

The attributes that must be specified in order to make a part simulatable, are all part of the Simulation category in the *Place Attribute* dialog / *Attribute Properties* dialog. These attributes consist of: SimType, SimModel, SimFile, SimPins, SimNetlist, SimDefaults and SimFields 1-16.

The .ddb file is created if it does not already exist (in the same directory as the .sch and .nsx files). If it does exist, the netlist file generated will replace the previous version of the same name.

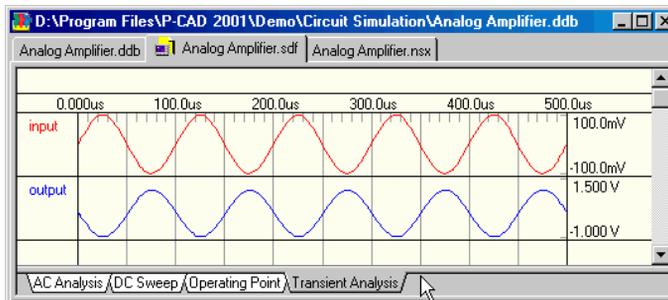
The XSpice netlist exported directly from Schematic does not contain any setup information. This information is added to the netlist from the configuration file, to create a modified netlist which is stored external to the .ddb, when the simulation is run. Upon completion of the simulation, the modified netlist file is deleted.

If a simulation is already running (started from the Mixed-Signal Circuit Simulator) and you start a simulation with the same netlist from P-CAD, then the netlist will be imported to the .ddb, but simulation will not proceed because a simulation is already running. This is the same for a simulation running on a different netlist.

## Working with Simulation Waveforms

When you run a simulation, a simulation results document is created and displayed in the waveform analysis viewer.

When the waveforms appear the results of each analysis performed during the simulation run are displayed on separate tabs (these tabs are at the bottom of the window), click on a tab to examine the results of that analysis, as shown below.



The waveform viewer window operates much like an oscilloscope, simply adjust the scale options in the *Browse SimData* panel to show exactly the part of the waveform that you would like to examine. The Waveform analysis pane also includes measurement cursors for taking measurements directly from the waveforms.

Click on the appropriate tab at the bottom of the window to display the results for that type of analysis. Operating Point results are displayed as a list of voltage, current and power calculations for nodes or devices.

## Troubleshooting simulation problems

When a circuit will not simulate you must identify if the problem is in the circuit, or the process of simulation. To troubleshoot simulation problems, read the topics listed in the Simulation troubleshooting topics listed in the on-line help and work through the suggested points, trying one at a time.

Sometimes during a simulation a message will be displayed reporting errors or warnings. These messages are saved in a text document (*DesignName.ERR*).

### Warning Messages

Warning messages are not fatal to the simulation. They generally provide information about changes that SPICE had to make to the circuit in order to complete the simulation. These include invalid or missing parameters, and so on. Digital SimCode warnings may include information such as timing violations (*tsetup*, *thold*, *trec*, *tw*, etc.) or significant drops in power supply voltage on digital components. Note that valid simulation results are normally generated even if warnings are reported.

### Error Messages

Error messages provide information about problems that SPICE could not resolve and were fatal to the simulation. Error messages indicate that simulation results could not be generated, so they must be corrected before you will be able to analyze the circuit.



## File Commands

### Using the File Commands

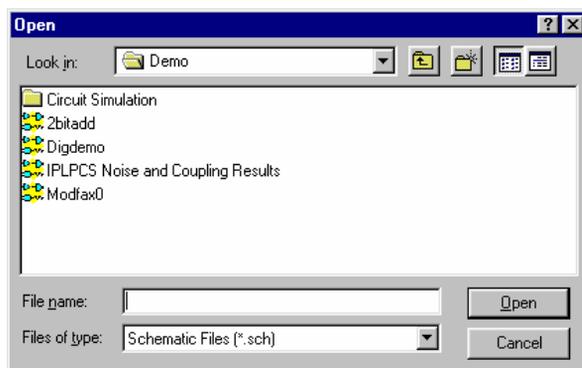
File commands allow you to open, save, and print designs and output reports in P-CAD Schematic. Most of the File commands cannot be undone: once an action is taken, it cannot be reversed using the Undo command

### File New

Choose **File » New** to open a new, untitled design window. The **File » New** command clears the styles, sheet definitions, and sheet sizes for the new design. All design parameters are returned to their default settings.

### File Open

Choose **File » Open** to open a file the P-CAD Schematic window. When you choose this command, the *Open* dialog appears. In this dialog, you can choose the directory and filename of the file you want to open.



Notice that the default file name extension in the Files of Type list is \*.sch. This file name extension applies to binary P-CAD Schematic designs as well as ASCII files. Other file name extensions include:

- .s01 to open Tango Series II ASCII files.
- .cfg to open PCAD CFG files.

To learn more about the options in the *Open* dialog box, see your Windows™ documentation. To learn how to open files in P-CAD Schematic, see *Opening Design Files* (page 21).

## File Close

---

Choose **File » Close** to close the currently active design window. If the design has been changed but not yet saved, you are asked whether or not you want to save your changes before closing. If you have opened more than one design window, the next window will become the active window. If you close the last open design, a new, untitled design window appears.

## File Save

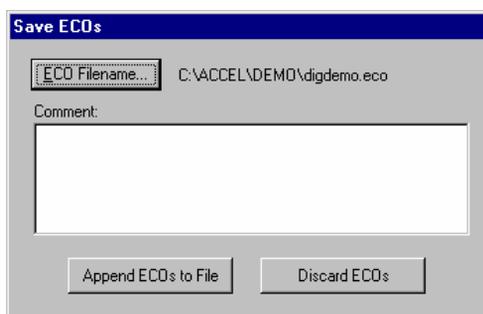
---

Saves the changes to the active design and creates a backup file (.bak). When you choose **File » Save**, the file remains open so you can continue working on it, and a backup file is created. The file name and location are unchanged by this command.

### Saving a File with ECOs

To save a file that has ECOs attached, follow these steps:

1. Choose **File » Save**. Pending Engineering Change Orders (ECOs) must be appended to an ECO file or discarded when a design file is saved. If there are pending ECOs, the following dialog appears:



2. The ECO filename appears at the top of the dialog. It is the last used ECO file. To change it, click **ECO Filename**.
3. Type a file name in the File name box, or select a file from the list area. Click **Save** to return to the *Save ECOs* dialog.

4. In the **Comments** box, type any comments that can help document the ECOs.
5. To append ECOs to the ECO file, click **Append ECOs to File**.
6. To discard ECOs, click **Discard ECOs**. Once discarded they cannot be recovered.

For more information on ECOs, see the following sections:

- *Utils Record ECOs (page 338)*
- *Utils Import ECOs (page 339)*
- *Utils Export ECOs (page 340)*

## Saving an ASCII File

If a P-CAD ASCII file type was selected in the File Type list box in the *File Save* dialog, a P-CAD ASCII file is generated instead of a P-CAD binary file.

## File Save As

---

Choose **File » Save As** to save a copy of the active design. When you save a copy of the design, you can specify a unique file name and location to save the file. This command also creates a backup file (.bak). You can either name a new file or save an existing file to a new name.

When you choose this command, the *File Save As* dialog appears. In this dialog, choose the folder and type the new filename. The Save in list displays the current folder and any files in that folder. The File name box lets you type or select a file name with the extension specified in the Save as Type list box. The default file name extension is .sch. This extension applies to P-CAD Schematic designs as well as ASCII files.

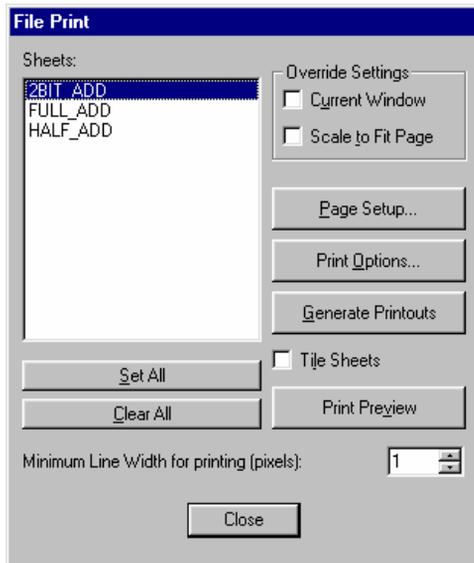
## Saving a Copy of a File

To save a copy of a file, to a different name or location, follow these steps:

1. Choose **File » Save As** to open the *Save As* dialog.
2. Type a name for the file in the File name box.
3. If the current folder is not appropriate, then navigate to the appropriate directory in which you will save the file.
4. Make sure the correct file name extension appears in the Save As Type list box.
5. Click **Save** to save the file as you have specified.
6. If there are pending ECOs, you are prompted to save them. For detailed instructions, see *File Save (page 170)*.

## File Print

Choose **File » Print** to print one or more sheets in your current schematic design. When you choose this command, the *File Print* dialog appears.



From the *File Print* dialog, you can create print settings for all of the sheets in a design, or you can create custom print settings for each sheet. When you close this dialog, your settings become part of the design file.

The *File Print* dialog contains the following options:

- **Sheets** (list). Shows all of the sheets in your active design. Choose one or more sheets from this list.
- **Set All** (button). Click this button to select all sheets in the *Sheets* list box.
- **Clear All** (button). Click this button to cancel the selection of all sheets in the *Sheets* list box.
- **Page Setup** (button). Click this button to open the *Page Setup* dialog. For more information, see *Page Setup Dialog* (page 173).
- **Print Options** (button). Click this button to open the *Print Options* dialog. For more information, see *Print Options Dialog* (page 174).
- **Generate Printouts** (button). Click this button to send your current print job to the printer.
- **Tile Sheets** (checkbox). Select this check box and click **Print Preview** to display all sheets of the schematic on your screen. The sheets are resized and arranged side-by-side so that all are

visible and none overlap. When you print with this option selected, the tiled sheets are output to a single printed page.

- **Print Preview** (button). Click this button to view your design in a print preview window.
- **Minimum Line Width** (spin box). Select or type the minimum desired printed line width.

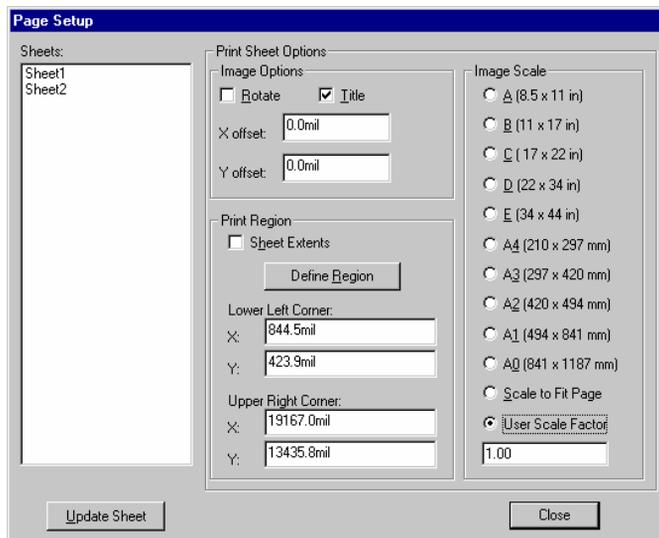
## Override Settings Frame

**Current Window.** Select this check box to print only what appears in your active window. For example, if you are zoomed in on your design, select this check box to print only the area in view.

**Scaling to Fit Page.** Select this check box to override the page scale settings for all sheets. For example, if you select this box the print output for each sheet is adjusted to fit on one page.

## Page Setup Dialog

To open the *Page Setup* dialog, click **Page Setup** in the *File Print* dialog.



The *Page Setup* dialog contains the following options:

- **Sheets** (list box). Select one or more sheets from the list to define page setup options. After you select your print options, click **Update Sheet**
- **Update Sheet** (button). Change any print options and then click this button to update the sheet's print options.
- **Close** (button). Click this button to return to the *File Print* dialog.

## Image Options Frame

**Rotate** (check box). Select this check box to rotate the printed image clockwise by 90 degrees.

**Title** (check box). Select this check box to print the design title with your design. To assign a title to a design, use the *Options Configure* dialog. Clear this check box to omit the title block from the printout.

**X Offset and Y Offset** (boxes). Type a value to offset your printout horizontally or vertically by that value. The default unit is the value set in *Options Configure*. You can override the default units by entering the units with the value (e.g., mil, mm, in).

## Print Region Frame

**Sheet Extents** (check box). Select this check box to print the entire design. Notice that the *Lower Left Corner* and *Upper Right Corner* boxes are not active when this check box is selected. Clear this check box to define a print region.

**Define Region** (button). Choose this button to define a print region. The *Page Setup* dialog is hidden and the design window is displayed. Click and hold the left mouse button to define a selection window. Click the right mouse button to return to the *Page Setup* dialog.

**Lower Left Corner/Upper Right Corner** (value box). Manual alternative to the *Define Region* button. Type the lower left and upper right limits of the desired print region.

## Image Scale Frame

**User Scale Factor** (button). Choose this button to increase or decrease the size of the print job by a specific factor. The value you enter affects the X axis and Y axis equally. For example, 0.100 reduces the output by a factor of 10, while 10.00 increases the output by a factor of 10.

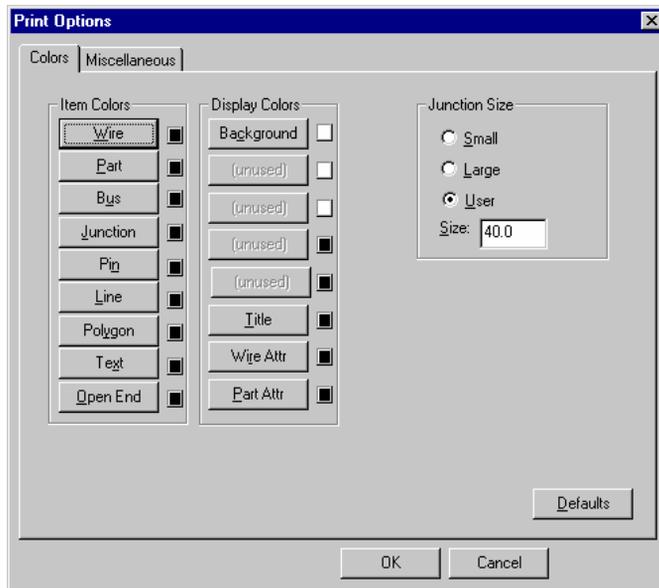
**Scale to Fit Page** (button). Choose this button to scale the print job to fit on a single page. When selected the *User Scale Factor*, *X offset*, and *Y offset* boxes are not active.

## Print Options Dialog

To open the *Print Options* dialog, click **Print Options** in the *File Print* dialog. This opens the *Print Options* dialog. Like the *Options Display* dialog, this dialog contains both a *Colors* tab and a *Miscellaneous* tab.

### Colors Tab

The following figure shows you the options in the *Colors* tab. To learn about these options, see *Options Display* (page 297).



The **Colors** tab contains the following options:

**Default** (button). Click this button to restore the default color scheme for your printed output.

### Item Colors Frame

To select a color for an item, click the corresponding button in this frame. When you do, a color palette appears. Choose a color from the palette by clicking the appropriate button. The palette will automatically close.

### Display Colors Frame

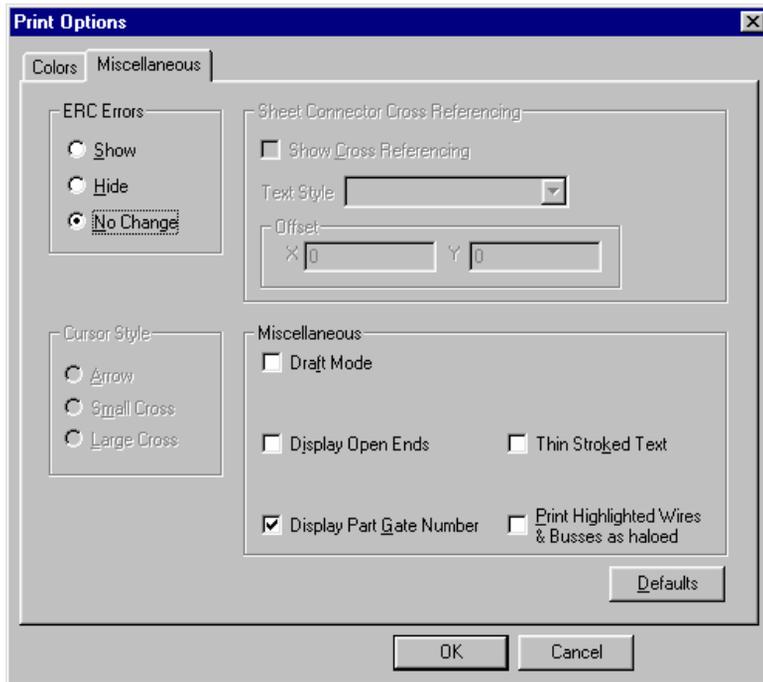
To select a color for a display item, click the corresponding button in this frame. When you do, a color palette appears. Choose a color from the palette by clicking the appropriate button. The palette will automatically close.

### Junction Size Frame

- **Small** (option button). Choose this option to use the small junction size in the printed output.
- **Large** (option button). Choose this option to use the large junction size in the printed output.
- **User** (option button). Choose this option to define a custom junction size for the printed output. Then, enter a value in the *User* box.
- **User** (box). When you choose the **User** button, enter a value to define a custom junction size for the printed output. Your entry can be in inches, centimeters, etc., as long as the value is between 0 - 10 mm.

## Miscellaneous Tab

The following figure shows you the options in the *Miscellaneous* tab. To learn about these options, see *Options Display* (page 297).



The *Miscellaneous* tab contains the following options:

### ERC Errors Frame

- **Show** (button). Choose this button to show ERC errors in the workspace.
- **Hide** (button). Choose this button to hide the display of ERC errors in the workspace.
- **No Change** (button). Choose this button to keep the current display setting.
- **Size** (text box). Type a value in this box to set the size of ERC errors. Your entry can be in inches, centimeters, etc., as long as the value is between 0 - 10 mm. If you do not choose a size, the default is 100 mil.

### Miscellaneous Frame

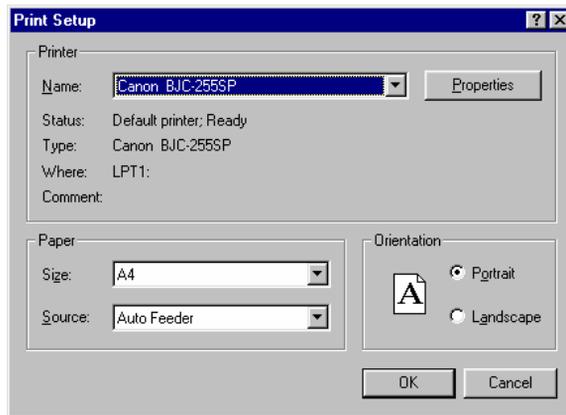
- **Draft Mode** (check box). Select this check box to show these two items in your design: **(1)** a thin, single-pixel outline for pads, vias, and text. **(2)** a segmented and outlined representation of arcs, lines, and any line segment objects such as, polygons, cutouts, etc. Draft mode improves redraw speed and helps you view segment overlaps.

- **Thin Stroked Text** (check box). Select this check box to show text in thin line mode. Clear this check box to display text in regular mode.
- **Display Open Ends** (check box). Select this check box to show open ends on unconnected pins and wires. Open ends appear as open squares, and no longer appear when a positive connection is made.
- **Display Part Gate Number** (check box). Select this check box to show a part's reference designator gate number. You can also use the *Print Options* dialog to show or hide the gate number in your hardcopy. The options are independent of each other, so you can show the gate number on screen for editing, but hide it from your hardcopy at the same time.
- **Print Highlighted Wires and Busses as Haloed** (check box). Select this check box to distinguish any wires or busses that are currently highlighted in your Schematic as haloed lines.

## File Print Setup

---

Choose **File » Print Setup** to view a list of installed printers. This command also lets you to set the current printer. You can also change device-specific parameters. When you choose **File » Print Setup**, a dialog similar to the one in the following figure appears.



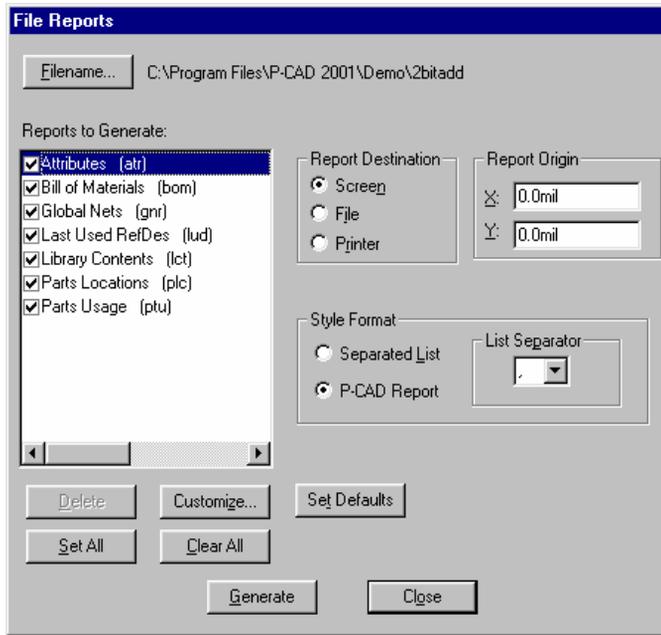
To learn about the options in this dialog, see your Windows™ and printer documentation.

## File Reports

---

Choose **File » Reports** to generate various reports. P-CAD Schematic provides you with a collection of standard reports to help you track and manage the schematic design process. For information on a specific report, see *Reporting on Schematic Designs* (page 118).

When you choose **File » Reports**, the following *File Reports* dialog appears:



The *File Reports* dialog contains the following options:

- **Filename** (button). Click this button to select a design file. By default, the name of the active design file appears here. Any report settings defined in this dialog apply only to the file that appears in the *Filename* area. This is also the file to which reports are saved and stored.
- **Delete** (button). Click this button to remove a custom report from the design. You cannot delete any predefined reports.
- **Customize** (button). Click this button to open the *Customize Report* dialog. For details on the fields in this dialog, see *Customize Report Dialog* (page 179). To learn how to customize reports, see *Customizing a Standard Report* (page 119).
- **Set Defaults** (button). Click this button to restore the default settings for P-CAD Schematic.
- **Set All** (button). Click this button to select all of the reports in the *Reports to Generate* list.
- **Clear All** (button). Click this button to cancel the selection of all reports in the *Reports to Generate* list.
- **Generate** (button). Click this button to generate the selected reports.

### Reports to Generate Frame

This list contains a collection of standard reports to help you track and manage the schematic design process. To generate a report, select the appropriate check box. For details on each report, see *Reporting on Schematic Designs* (page 118).

### Report Destination Frame

- **Screen** (button). Choose this button to send the output to a file and open the file using Notepad.
- **File** (button). Choose this button to send the output to a file.
- **Printer** (button). Choose this button to sends the output directly to the printer without creating files.

### Report Origin Frame

**X and Y coordinates** (boxes). By default, the report origin is 0,0, which represents the lower left corner of the design. To offset the report origin, enter an X and Y coordinates in the appropriate boxes. Coordinates printed in the report output indicate their position relative to the designated coordinates. Report origins are printed in the first line of the output report in the standard P-CAD Report format.

### Style Format Frame

The Style Format you select determines which customization options are available.

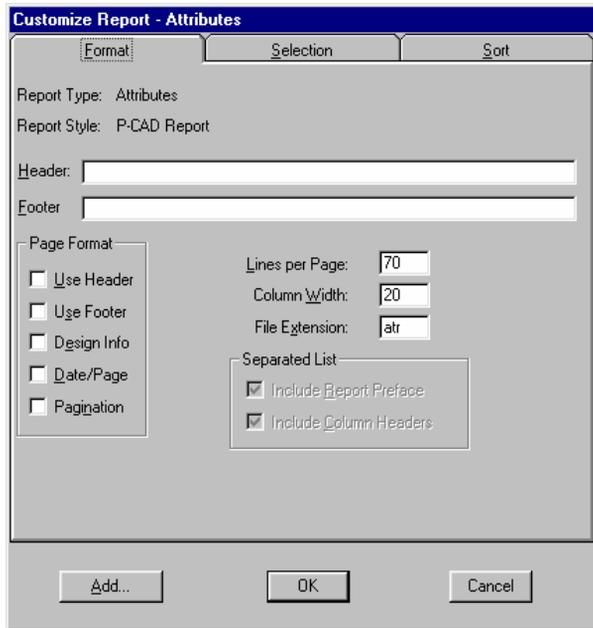
- **Separated List** (button). Choose this button to place all data in character-separated format. This format can be imported into other spreadsheet and database programs.
- **List Separator** (list). If you choose **Separated List**, select the character you want to use as the separator. The separator that appears in the box is used for imported and exported files. P-CAD Schematic uses the character defined in your Windows™ Regional Settings by default.
- **P-CAD Report** (button). Choose this button to produce a report format with columns and spaces, etc. When you choose this button, you can gain access to most of the options in the *Format* tab of the *Customize Report* dialog.

### Customize Report Dialog

To open the *Customize Reports* dialog, select a report from the Reports to Generate list and click **Customize**. When the dialog appears, the **Format** tab is selected by default. In addition, the type of report you selected appears in the title bar.

### Format Tab

Use the options in this tab to set up the page format of your report. Depending on the style format of your report (i.e., P-CAD Report or Separated List) different options are available in this tab. The following figure shows you the **Format** tab.



- **Header** (box). Type a report header. This box is only available when you choose the P-CAD Report style format.
- **Footer** (box). Type a report footer. This box is only available when you choose the P-CAD Report style format.

### Page Format Frame

If you choose the P-CAD Report style format, you can select one of the following check boxes to enable the corresponding page format option:

- **Use Header.** Includes the information you specified in the Header box.
- **Use Footer.** Includes the footer information entered in the Footer box.
- **Design Info.** Includes the information you entered in the **File Design Info** command and dialog.
- **Date/Page.** Includes the current date and the page number.
- **Pagination.** Gives you the ability to create your own pagination (lines per page).
- **Lines per Page:** Type the number of lines you want printed on each page. This box is only available when you choose the P-CAD Report style format
- **Column Width:** Enter a value in this box to set the width of a single column. To do this, click the **Selection** tab. Next, choose a field from the spreadsheet. Then, click the **Format** tab and

type a width for that column in this box. Column width is set to 20 characters by default. This box is only available when you choose the P-CAD Report style format

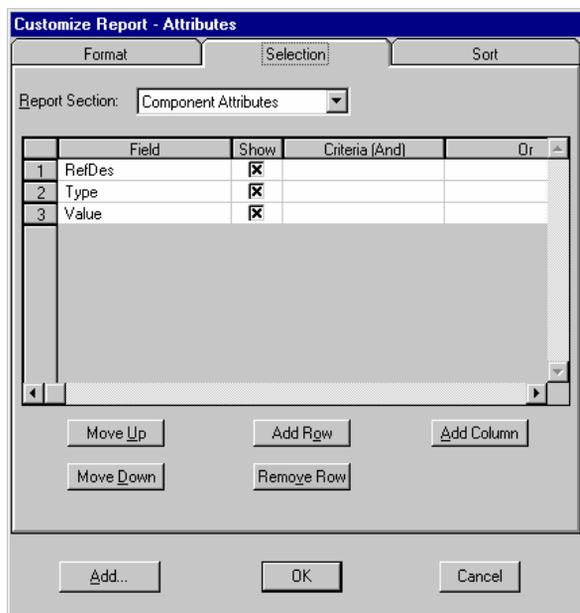
- **File Extension:** This box displays the default file name extension for the selected report. You can also type a new extension if desired. This box is available with the P-CAD Report and Separated List style formats.

### Separated List Frame

- **Include Report Preface** (check box). Select this check box to include a report preface. This box is only available when you choose the Separated List style format.
- **Include Column Headers** (check box). Select this check box to include column headers at the top of your report. This box is only available when you choose the Separated List style format.

### Selection Tab

The following figure shows you the **Selection** tab.

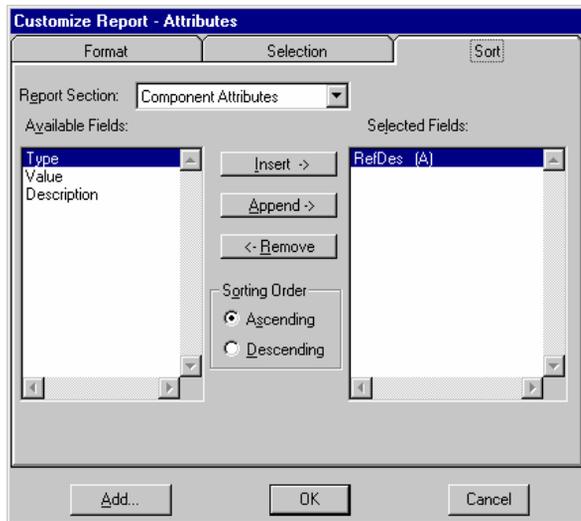


- **Report Section** (list). Select a report section from this list. This option is only available with the Attributes report.
- **Field** (column). Lists the fields specific to the selected report.
- **Show** (column) Displays the selected fields in the report. Unchecked fields do not appear in the report output.

- **Criteria (And)** (column). Contains the selection criteria used to filter the report data. To learn how to enter information, see *Selecting Report Criteria (page 122)*.
- **Or** (column). Type additional selection criteria here if necessary. To learn how to enter information, see *Selecting Report Criteria (page 122)*.
- **Add Column** (button). Click this button to add an *Or* column to the report spreadsheet.
- **Add Row** (button). Click this button to add a row of data to the report spreadsheet. This button is only available when you can add attribute fields to the report. When you click this button, the *Select Attribute* dialog appears.
- **Move Up** (button). Click this button to move the selected row of data up one position.
- **Move Down** (button). Click this button to move the selected row of data down one position.
- **Add Row** (button). Click this button to add a row of data to the spreadsheet.
- **Remove Row** (button). Click this button to remove a row of data from the spreadsheet. You cannot remove a row of predefined data from the report spreadsheet.

## Sort Tab

The following figure shows you the **Sort** tab.



- **Available Fields** (list). The list of fields in the selected report.
- **Selected Fields** (list). The fields used to sort the data, the order in which the sort is applied and the field's sorting order, (A) for Ascending or (D) for Descending, are displayed in the *Selected Fields* list.

- **Insert** (button). Click this button to move a field from the Available Fields list and insert it above the selected field in the Selected Fields list.
- **Append** (button). Click this button to move a field from the Available Fields list to the bottom of the Selected Fields list.
- **Remove** (button). Click this button to move a field from the Selected Fields list to the Available Fields list.

### Sorting Order Frame

- **Ascending** (button). Choose this button to sort report data in ascending order.
- **Descending** (button). Choose this button to sort report data in descending order.

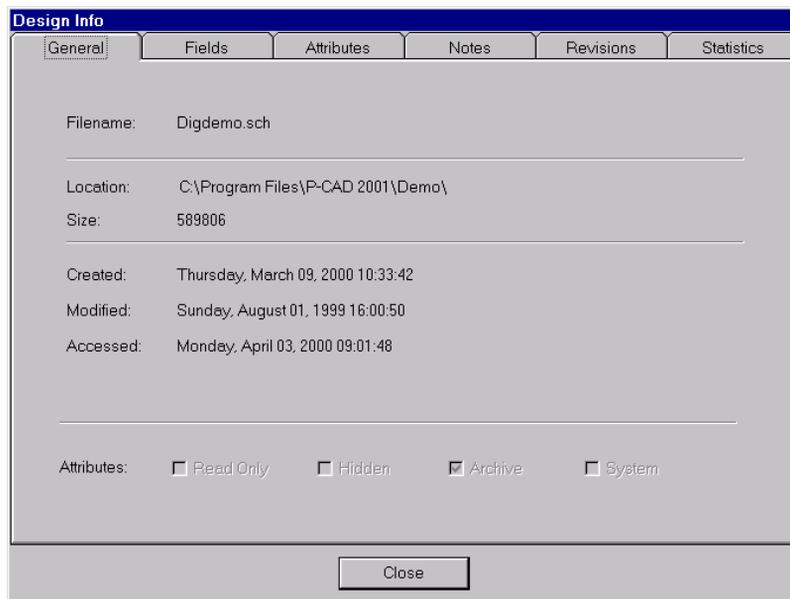
## File Design Info

---

Choose **File » Design Info** to enter design information, review design statistics, and query/modify design attributes and fields. When you choose this command, the *Design Info* dialog appears.

### General Tab

Click the **General** tab to view basic file information about the active design. The options in this tab are read-only.

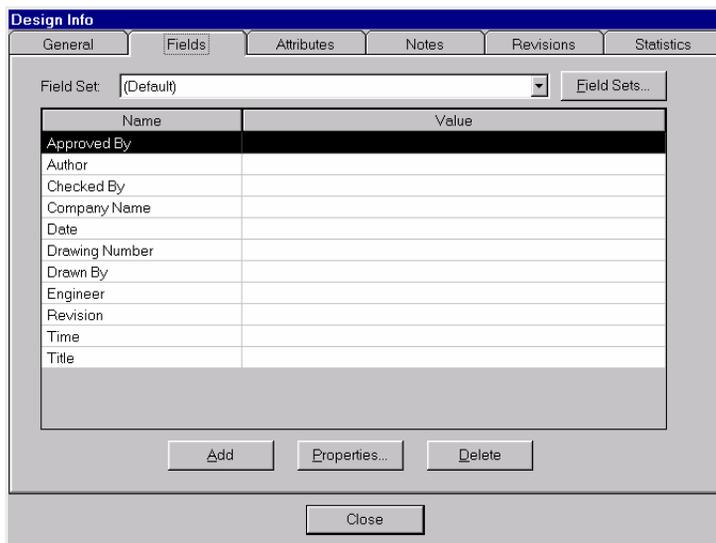


- **Filename.** Displays the name and file name extension of the schematic design. To place this field in a design, choose **Place Field** and select **Filename**.

- **Location.** Shows where the design is located on your computer or network.
- **Size.** Displays the size of the design file.
- **Created.** Shows the date the file was created.
- **Modified.** Shows the date and time the file was last modified. To place this field in a design, choose **Place Field** and select **Modified Date**.
- **Accessed.** Shows the date and time the file was last viewed
- **Attributes Frame**
- To modify the attributes shown in this frame, you must change the design file properties in Microsoft® Windows™. For more information, see your Windows™ documentation.
- **Read-Only** (check box). A check mark in this box indicates that the design has been saved as a read- only file.
- **Hidden** (check box). A check mark in this box indicates that the design is hidden, which means that you cannot see or use it unless you know the filename.
- **Archive** (check box). A check mark in this box indicates that the file should be backed up.
- **System** (check box). A check mark in this box shows that the design has been marked as a system file.

## Fields Tab

Click the **Fields** tab view a list of all the fields that can be placed in a design. To learn how to add fields to this list or to place a field in a design, see *Fields and Field Sets* (page 103).



- **Field Set** (list box). Select a field set from this list. For information on field sets, see *Fields and Field Sets* (page 103).
- **Field Sets** (button). Click this button to add or modify a field set. For information, see *Adding a Field Set* (page 106).
- **Name/Value** (spreadsheet). The Name column lists all fields that can be placed in the design. The Value column lists any values assigned to the field.
- **Add** (button). Click this button to open the *Field Properties* dialog. You use this dialog to add a name and value to the fields list.
- **Properties** (button). Click this button to open the *Field Properties* dialog. You use this dialog to view or modify properties associated with a field.
- **Delete** (button). Select a user-defined field from the list and click this button to delete it from the list.

## Field Properties Dialog

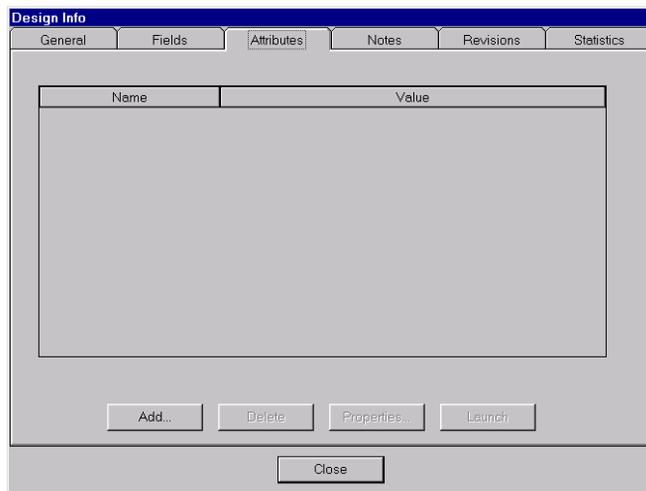
To open the *Field Properties* dialog, click the **Add** or **Properties** button in the *Design Info* dialog. This dialog contains the following options:

- **Name** (text box). Enter a name for the selected field. This field is only available for user-defined fields.
- **Value** (text box). Enter a value for the selected field.

For details about using Fields and Field Sets, see *Fields and Field Sets*, (page 103).

## Attributes Tab

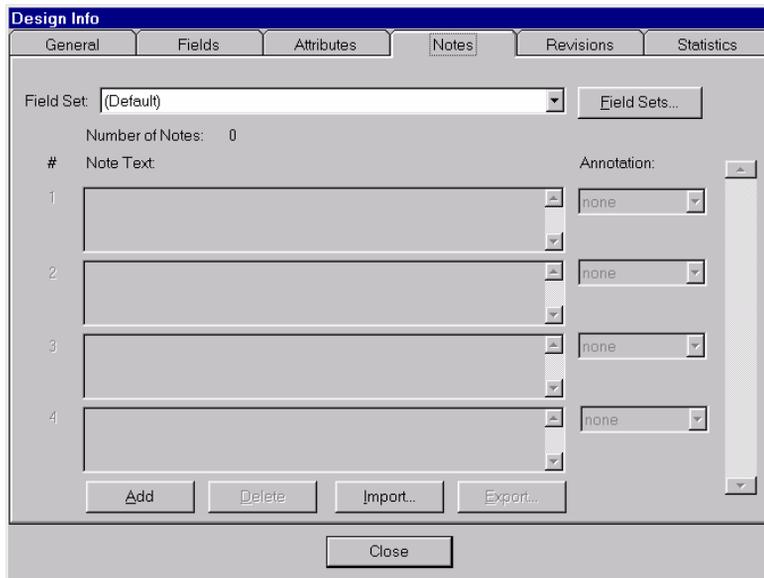
Click the **Attributes** tab to view and modify design-level attributes:



- **Name/Value** (spreadsheet). Displays a list of attributes that have been added to the design, as opposed to a component or net.
- **Add** (button). Click this button to open the *Place Attribute* dialog. For details about this function and a complete listing of attributes, see *Place Attribute* (page 279).
- **Delete** (button). Select an attribute from the Name/Value spreadsheet and click this button to remove the attribute from the design.
- **Properties** (button). Select an attribute from the Name/Value spreadsheet and click this button to open the *Attribute Properties* dialog.
- **Launch** (button). Select a Reference link from the attributes list and click this button to open the program associated with the link, or to launch Internet Explorer and go to the website associated with the link.

## Notes Tab

Click the **Notes** tab to add design or drawing notes that can later be inserted in a design as a Notes field or Notes table:



The **Notes** tab contains the following options:

- **Field Set** (list box). Select a field set from this list. For information on field sets, see *Fields and Field Sets* (page 103).
- **Field Sets** (button). Click this button to add or modify a field set. For information, see *Adding a Field Set* (page 105).

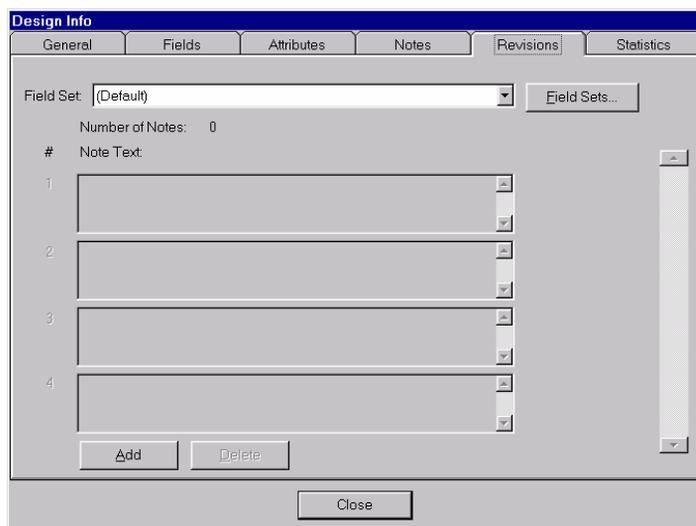
- **Number of Notes** (data field). Shows the total number of notes that have been added
- **Note Text** (text box). Click **Add** and then type new notes or modify the notes that appear in this box. Or, modify the notes that appear here.
- **Annotation:** Select one of the following annotation markers from this list: box, circle, triangle, or none. The following figure shows you what each annotation looks like:



- **Add** (button). Click this button to enter notes or to add a note to the end of the list.
- **Delete** (button). Place the text insertion point in one of the Note boxes. Then, click this button to remove that note.
- **Import** (button). Click this button to import a note from an ASCII text file. To learn more about importing notes, see *Importing Text Files as Notes* (page 112).
- **Export** (button). Click this button to export a note to an ASCII Text file. To learn more about exporting notes, see *Exporting Notes to a Text File* (page 112).

## Revisions Tab

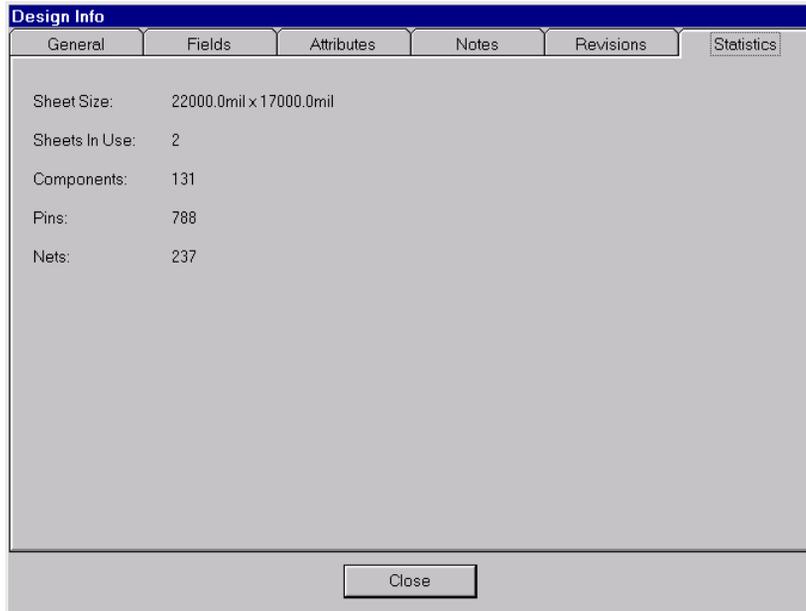
Click the **Revisions** tab to add, import, and export revision notes. Revision notes are used specifically to document changes between drafts of a design.



For details about the dialog box and its options, see *Notes Tab* (page 186).

## Statistics Tab

Click the **Statistics** tab to design statistics for the current design. The information in this tab is read-only.



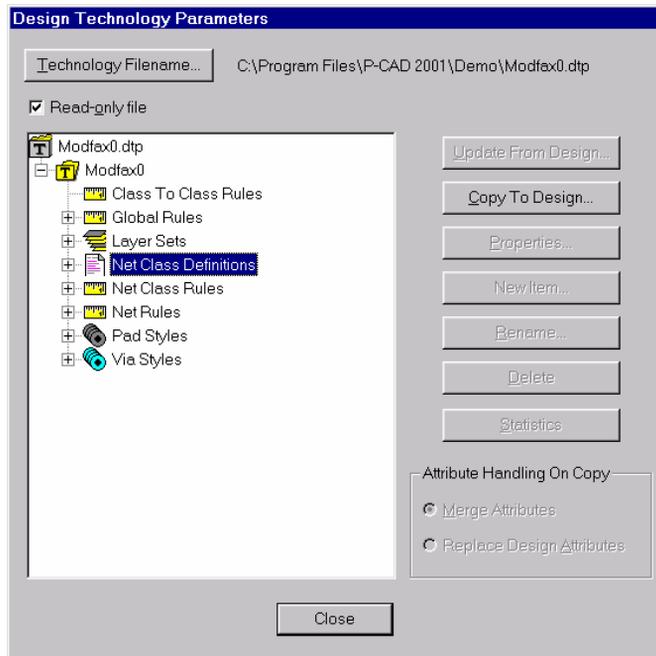
## File Design Technology Parameters

---

Choose the **File » Design Technology Parameters** command to view design data such as net class definitions and design rules. When you choose this command, the *File Design Technology Parameters* dialog appears. The information in this dialog is stored in a design technology parameters file (\*.dtp), which contains data that can be updated or copied to any P-CAD design.

### Design Technology Parameters Dialog

When you choose the **File » Design Technology Parameters** command, the following *File Design Technology Parameters* dialog appears.



- **Technology Filename** (button). Click this button to display the *Open* dialog, from which you can choose the folder and filename of the file you want to open or create.
- **System Path** (field). The field displays the system path to the active DTP file.
- **Read-only file** (check box). If a *.dtp* file is read-only, this check box is selected and the term “read-only” appears next to the filename. To make a *.dtp* file read-only, select this check box. If a *.dtp* file is read-only, you cannot update it with data from a design, modify an item’s properties, rename an item, or delete an item.
- **DTP tree** (directory list box). The contents of the open parameters file appear as a tree structure, or hierarchy, in the *Design Technology Parameters* dialog. You can browse the tree to view its contents. Also, you can highlight groups, sections, or items in the tree to select them for copying to a design, editing properties, and completing other functions described in this section. The tree structure allows you to view your entire file 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.
- **Update From Design** (button). The **Update From Design** button allows you to add information from your current design to the design technology parameters file.
- **Copy to Design** (button). The **Copy to Design** function allows you to modify your current design using data contained within your design technology parameters file.

- **Properties** (button). This button allows you to view and modify an item's properties. When you highlight an item in the design technology parameters file tree and click **Properties**, an item-specific dialog appears
- **New Group** (button). Select the DTP filename in the tree and click this button to add a new group to your DTP file.
- **New Section** (button). Select a group from the DTP tree and click this button to create a new section in your DTP file.
- **New Item** (button). When you highlight the filename in the tree, click this button to create a new item in your DTP file.
- **Rename** (button). Select a group or item from the DTP tree and click this button to rename the selected object. You cannot rename sections with this button.
- **Delete** (button). Click the **Delete** button to delete objects from the tree.
- **Statistics** (button). Click the **Statistics** button to view statistics about the DTP file. When you click this button, the *Statistics* dialog appears. This dialog contains information about modifications made to the item.

### Attribute Handling on Copy Frame

- **Replace Design Attributes** (button). Replaces a design's rules with the rules from the design technology parameter file item.
- **Merge Attributes** (button). Merges the rules of the design technology parameter file item with the design's rules, favoring the rules of the design technology parameter file item over the design.

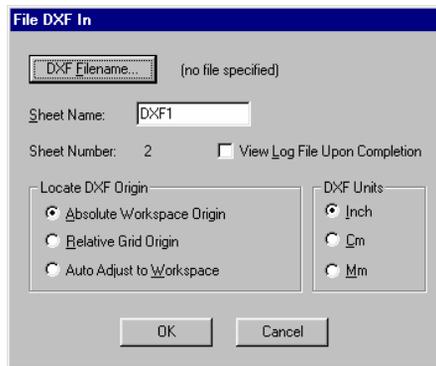
## File DXF In

---

DXF (Drawing Interchange Format) files generated using AutoCAD® Version 9.0 through 14 or other conforming CAD programs can be opened in P-CAD Schematic. Using this command, you can create dimensions, board outlines, manufacturing instructions, artwork, logos, etc., and then import the resulting DXF file into P-CAD Schematic.

### Opening a DXF File

To open a DXF file in P-CAD Schematic, choose the **File » DXF In** command. The *File DXF In* dialog appears as shown in the following figure:



### DXF Filename

Click the **DXF Filename** button to open the *Open* dialog where you can navigate to the desired file.

### Sheet Name/Number

The name and number of the sheet where the DXF items are placed when imported. If the sheet name contains any AutoCAD® reserved keywords, a syntax error occurs which causes the file load to abort. Rename the layer in AutoCAD® to work around this limitation.

### View Log File Upon Completion

Select the **View Log File Upon Completion** check box to display the log file when the import is complete.

### DXF Units

The DXF Units section of the *File DXF In* dialog provides the ability to choose which units will be used to apply proper scaling when loading the DXF file. Select the appropriate option to set the unit for inches (inch), centimeters (cm) or millimeters (mm).

### Locate DXF Origin

The P-CAD Schematic workspace uses positive X and Y coordinates. Since DXF items can be placed at negative X and Y coordinates, you may have to direct P-CAD Schematic to adjust the position of your imported DXF geometry. In the *File DXF In* dialog, select a method of translation to positive coordinates in the **Locate DXF Origin** radio box.

- **Absolute Workspace Origin:** If your DXF geometry is positioned only at positive coordinates, select **Absolute Workspace Origin**. No coordinate translation is made.
- **Relative Grid Origin:** Loads the DXF geometry so its origin is placed at the Relative Grid Origin of your workspace. The Relative Grid Origin is specified in the *Options Grids* command.
- **Auto Adjust to Workspace:** Automatically adjusts the placement of your DXF geometry into positive coordinates.

When you import data into P-CAD Schematic from a DXF file, and you enable the **Auto Adjust to Workspace** option, P-CAD Schematic automatically translates the position of the lower left extent of the DXF file data into the positive P-CAD Schematic workspace. The DXF data's lower left extent is the coordinate defined by the EXTMIN variable in the header section of the DXF file. If the EXTMIN variable is absent from the DXF file, then P-CAD Schematic automatically calculates a suitable solution based on the extents of the geometry in the DXF file.

It is suggested that you set the current grid to a uniform value consistent with the grid of your DXF design. This grid spacing optimizes the on-grid placement of items. Set your current grid in the *Options Grids* dialog before importing your DXF file.

If DXF geometry items still fall outside the P-CAD workspace, a warning is written to the log file and the items are not imported. Your DXF geometry can fall outside the workspace if the translated items are still positioned at negative coordinates or if the geometry is larger than the workspace size. If the geometry is larger than the workspace size, use the *Options Configure* command to increase your workspace size and import the DXF file again.

### Importing a DXF File

To import a DXF file with the correctly positioned geometry:

1. Click the **DXF Filename** button to open the *Open* dialog.
2. Type a file name in the File name box, or navigate to the directory where the file is located.
3. Select the appropriate **Locate DXF Origin** option button on the *File DXF In* dialog.
4. From the *File DXF In* dialog, type a sheet name or use the one selected for you. The sheet name is used to create a sheet in P-CAD Schematic onto which all the DXF items are placed. After the DXF items have been translated and placed onto the new sheet, these items are equivalent to other P-CAD primitives. The new lines, arcs, text, and polygons can be modified as usual.
5. Click **OK**.

A progress indicator shows the status of the DXF translation. Some items supported in the DXF language cannot be translated for use by P-CAD Schematic (see list below). Errors and warning messages are placed in a report file, which you may view at the end of the translation.

### Items Supported for Translation

The following items and features are supported for translation:

#### Header Variables

The AutoCAD® state variables are grouped together at the top of the file in the HEADER section. Supported variables are listed and described below, with default values in parentheses:

\$ACADVER	The AutoCAD® drawing database version number (must be 9.0 or higher.)
\$ANGBASE	The angle zero direction. The DXF coordinate system is rotated by this angle. (0)

\$ANGDIR	The angular orientation: 1 = clockwise;
0 =	counter clockwise. (0)
\$MIRRTEXT	Mirror text if non zero. (0)
\$TEXTSIZE	Default text height.

## Tables

**DXF In** supports LTYPE and STYLE tables.

## Blocks

The only entries in the BLOCKS section that are supported are the dimension blocks. All dimensions that are translated come from this section.

## Entities

The majority of a DXF file is made up of entities. These include lines, arcs, text, block insertions and others. Only two-dimensional entities are supported; z-axis values are ignored. All block insertion entities, except dimensions, are ignored. Information embedded in the entities for color are also ignored.

LINE	DXF LINE entities have infinitesimal width. They are translated into P-CAD lines of 10 mil.
ARC	DXF ARCs are translated into P-CAD arcs of 10 mil.
CIRCLE	DXF CIRCLES are translated into P-CAD arcs with a sweep angle of 360 degrees and a 10 mil width.
POLYLINE	DXF POLYLINES are a sequence of possibly tapered, straight and curved lines that are connected end-to-end. These may be open or closed. P-CAD does not support tapering and only supports normal 2-D (unflagged) DXF vertices. POLYLINES are translated into P-CAD arcs and lines with a thickness equal to the initial POLYLINE thickness.
LWPOLYLINE	DXF LWPOLYLINES are translated into P-CAD arcs and lines with a thickness equal to the initial LWPOLYLINE segment thickness.
MTEXT	Same as TEXT except it handles multiple lines and supports Bold and Italic styles.
VERTEX	Vertices define DXF POLYLINES. When translated, they become the defining vertices of the translated P-CAD item. Only normal 2-D DXF vertices are supported (no spline-fit, curve-fit, 3-D mesh, or other special flags).

SOLID	DXF solids are filled three or four sided polygons. They are translated into P- CAD polygons. Four sided solids that form a “Complex” polygon will be ignored.
TRACE	DXF traces are lines with thickness that can be filled or unfilled. They are treated the same as DXF solids and are translated into polygons.
TEXT	DXF text is translated into P-CAD text. Oblique angle text is not supported. The font used for translation when a direct match cannot be made is the P- CAD default font. Due to the difference in fonts, translated text strings may be of different total width than the DXF version. The bar over barred text may not align exactly with the text.

The following is a specific (but not comprehensive) list of items that are not supported:

- All BLOCKS, except the dimension blocks.
- SHAPE, ATTDEF, and ATTRIB entities.
- 3DLINE and 3DFACE entities.
- Curve- or spline-fit vertices or meshes for POLYLINE and VERTEX entities. Tapering POLYLINES are also not supported.
- Oblique angle text and font are not supported for TEXT entities. Font is supported for True Type text.
- Three-dimensional entities and coordinates; thickness for all entities will be ignored, and only the first two values of a coordinate-triplet will be used.
- Dashed and dotted lines are converted to the P-CAD thin width.
- Color values for individual entities. Color values for entities will depend on the P-CAD primitive and layer to which the entity is translated.

## DXF In Notes

The following is a list of notes that are important or useful when using the *DXF In* command:

- The EXPLODE command in AutoCAD® can be used to transform blocks into individual entities.
- An important item to be translated from a DXF file is the dimension. AutoCAD® creates a new dimension block every time a dimension is moved, edited, or altered in any way. The user should use the PURGE command to eliminate any unreferenced copies of the dimension blocks created by AutoCAD®. The user should then output the design to DXF format immediately after the PURGE command, before editing or modifying dimensions.
- Text fonts and styles are more accurately translated from AutoCAD® to P-CAD when the style has been defined prior to entering text.

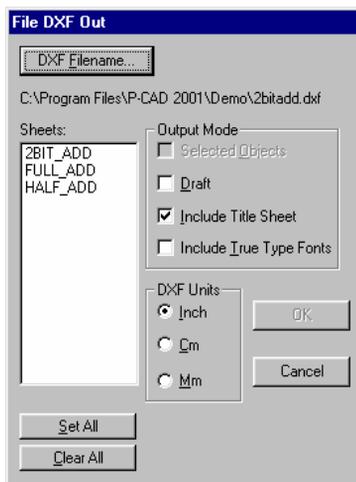
## File DXF Out

Choose **File » DXF Out** to create DXF (Drawing Interchange Format) files of your Schematic designs. These files can then be transferred to AutoCAD® and other mechanical CAD packages. The output is compatible with AutoCAD® Version 9.0 or later.

### Creating a DXF File

To create a DXF file, follow these steps:

1. Open a schematic design.
2. Choose **File » DXF Out**. The following *File DXF Out* dialog appears.



3. Click **DXF Filename**. The following *File DXF Out* dialog appears.



4. Type a name in the *File name* box and select the folder in which you want to save the file. Make sure *DXF Files (\*.dxf)* is selected in the Save As Type list.
5. Click **Save**. You return to the *File DXF Out* dialog.
6. In the Sheets list, select the sheets you want to output. Each sheet is output to a separate DXF layer.

You can click **Select All** to select all of the sheets in the design, or click **Clear All** to cancel the selection of all sheets in the design.

7. In the Output Mode frame, select the **Draft** check box to output in draft mode.

If you selected specific objects in the design before choosing the **File DXF Out** command, you can output only those objects by selecting the **Selected Objects** check box.

DXF polylines are normally used for all lines, wires, arcs, pins, and buses. Solids are normally used for polygons. Polylines are filled lines with thickness. When you select the **Draft** check box DXF outputs arcs, lines, and circles instead of polylines, and polygon outlines instead of DXF solids. Draft mode produces smaller files that process faster, but the drawings are not technically accurate since the lines have no width and areas are not filled.

8. To include title sheets in the DXF file, select the **Include Title Sheet** check box. The title sheets must be displayed in the design before they can be output to DXF.

To display title sheets in a design, select the **Display Title Sheets** check box in the *Options Configure* dialog.

9. Select the **Include True Type Fonts** check box to output true type text. Clearing this check box gives you the ability to export a DXF file that is compatible with Revision 9 DXF. True Type Fonts are compatible only with Revision 14 DXF.
10. In the DXF Units frame, choose **Inch**, **Cm** or **Mm**.
11. Click **OK** to generate the DXF output file. All selections are saved in the design file. While the file is generating, the Status line indicates progress by displaying the current layer being output. While the file is being generated, you right-click or press **ESC** to cancel the operation.

DXF Out normally tries to produce as faithful a picture as possible: lines and arcs have thickness. Draft mode does everything with DXF LINES, ARCS, and CIRCLES, which have no thickness: lines and arcs become thin and polygons become outlines.

## DXF Output Considerations

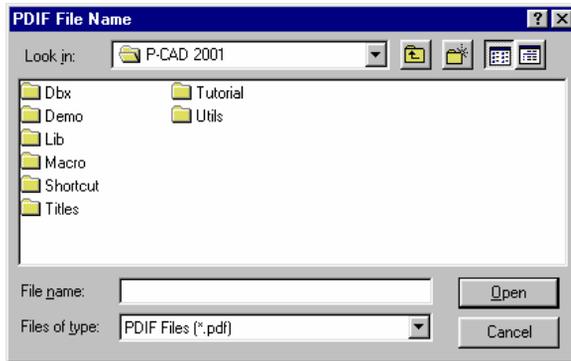
- **Line Styles:** The solid, dashed, and dotted line types are supported for output. The line styles are mapped as follows: P-CAD Dashed to AutoCAD Dashed, P-CAD Dotted to AutoCAD Dot and P-CAD Solid, Thick or User to AutoCAD Continuous. These specifications may not match those of your CAD package for lines of the same style.

- **Sheets:** P-CAD sheets become DXF layers with the same name. DXF substitutes the underscore for unsupported characters such as spaces to maintain compatibility with AutoCAD naming conventions.
- **Blocks:** DXF blocks are used to combine individual entities into a common unit, to be treated as a whole by the CAD package, analogous to a part or component. Blocks are used, where possible, to make CAD processing easier, and to reduce the size of the DXF file. For example, a part block has the name of the reference designator; exploding a part block produces text (for attributes) and pin blocks. These in turn can be exploded.
- **Polygons:** P-CAD polygons are also represented as blocks containing a collection of three- or four-sided solids. In this way they can be processed as a unit.
- **Lines:** When not in Draft mode, lines consist of a straight polyline and two round endcaps. Note that due to limitations in how blocks are scaled in DXF, lines are not blocks; the endcaps and polylines are separate from one another. In Draft mode, lines become DXF LINES with no endcaps.
- **Arcs:** When not in Draft mode, arcs consist of a curved polyline and two round endcaps. Note that due to limitations in how blocks are scaled in DXF, arcs are not blocks; the endcaps and polylines are separate from one another. In Draft mode, arcs become DXF ARCs with no endcaps.
- **Text/Mtext:** True Type font text styles are created and included in the DXF file. P-CAD stroke fonts text strings are converted into DXF text strings of the same height, rotation, mirroring, and justification; the AutoCAD® STANDARD font is used. Note that due to the difference in fonts, translated text strings may be of different total width than in P-CAD. The bar over barred text may not align exactly with the text. Mtext is output if True Type is selected.
- **Pins:** A Schematic pin is composed of DXF POLY LINE, LINE, and ARC entities; POLY LINES are not used in Draft mode.
- **Wires:** These become DXF 2D POLYLINES.
- **Busess:** When not in Draft mode, these become POLYLINES; in Draft mode, they are DXF LINES.
- **IEEE Symbols:** These become individual LINE entities, much as they are now for plotting.
- **Title Sheets:** Title sheets are output to DXF in the block format. These blocks consist of the lines, text and other objects that form the title sheet and title block. Title blocks are named TITLE\_SHEET\_xx, where xx is the sheet number on which the title sheet resides.

## File PDIF In

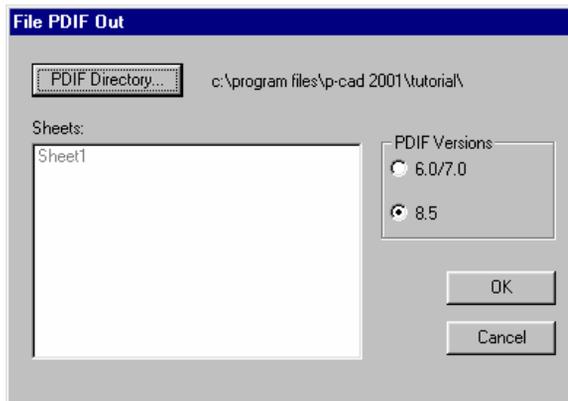
---

Choose **File » PDIF In** to import Pcad.pdf files. When you choose this command, the *PDIF File Name* dialog appears:



## File PDF Out

Choose **File » PDIF Out** to export schematic files to the PDF file format. When you choose **File » PDIF Out**, the *File PDF Out* dialog appears:



The *File PDF Out* dialog contains the following options:

- **PDF Directory** (button). Click this button to select the directory in which you want to save the file.
- **Sheets** (list box) Lists the sheets by name.

### PDF Versions Frame

- **6.0/7.0** (button). Choose this button to select the file format for version 6 and 7.
- **8.0** (button). Choose this button to select the file format for version 8.

## File Exit

---

Choose the **File » Exit** command to quit P-CAD Schematic.

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

The program writes information to the `Sch.ini` file when you choose **File » Exit**. This information, which applies to subsequent P-CAD Schematic sessions, consists of parameters and settings such as workspace size, units and values set in **Options » Configure** and report file settings from **File » Reports** and **Utils » ERC**.



## Edit Commands

### Using the Edit Commands

---

Use the Edit menu commands to modify objects in your design. The following list summarizes some of the actions you can perform with various Edit menu commands.

- You can choose **Edit » Undo** to undo recent actions and, if you decide you didn't want to undo an action, you can choose **Edit » Redo**.
- Some actions cannot be undone, but can be unwound by pressing the **BACKSPACE** key. For details, see *Edit Undo* (page 201).
- When you choose **Edit » Paste**, you have a number of paste options. See *Edit Paste* (page 204) for more information.
- Before you can choose some **Edit** commands, you must select an object. See *Edit Select* (page 247) and *Options Block Selection* (page 287) for more information.

### Edit Undo

---

Choose **Edit » Undo** to undo your last action. For example, if you place a part in a design, you can choose **Edit » Undo** to remove the part you placed. As a shortcut for choosing this command, you can click the **Undo** button on the *Command* toolbar, press **CTRL+Z**, or press the **U** key.

By default, you can undo up to 10 actions with the **Edit » Undo** command. To change the undo limit, open the `Sch.ini` file. Then, modify the `UndoLimit` value.

You can set the `UndoLimit` to any number. However, each stored, undoable action requires memory. Setting a large `UndoLimit` may use up your computer's available memory, causing slow performance and often unpredictable results.

If an action cannot be undone, the **Edit » Undo** command is shaded and not available. The list of undoable actions is deleted when you save the design. Any of the commands in the following list also cause the undo list to be cleared:

- Place Commands
- Edit Delete
- Edit Copy
- Edit Cut
- Moving Objects
- Rotating Objects
- Flipping Objects
- File Save
- File Save As
- Delete TextStyle
- Utils Renumber
- File DBX In
- File Design Technology Parameters (**Close** or **ESC**)
- File Design Info (**Close** or **ESC**)
- Utils Force Update
- Utils Record ECOs
- Utils Import ECOs
- Delete Sheet
- Utils ERC
- Utils Resolve Hierarchy
- Utils Module Wizard

## Edit Redo

---

Choose **Edit » Redo** to repeat an action that has been undone.

Each modification made to a design results in a copy of the design being placed in the undo list, as explained in *Edit Undo* (page 201). If you have stepped backwards in the list using **Edit » Undo**, and find that you want to move forward to a later version of the modifications, choose **Edit » Redo**.

The redo button on the toolbar, the **CTRL+Y** and **SHIFT+U** keys are equivalent to the **Edit » Redo** command. The `UndoLimit` setting in the `Sch.ini` file also applies to the **Edit » Redo** command. To change the undo limit, open the `Sch.ini` file. Then, modify the "UndoLimit" value.

## Edit Cut

---

Choose **Edit » Cut** to remove selected objects from your design and save them to the Clipboard. From there you can paste them into another design, into another location within the current design, or into another program.

You need to enable the **Select** tool (**Edit » Select**) and select at least one object to choose this command; otherwise the menu command is shaded and the **CTRL+X** shortcut key is unavailable.

You can copy selected items to a disk file instead of the Clipboard. For details, see *Edit Copy to File* (page 203). You can also cut multiple objects by using multiple select and block select operations. For information, see *Edit Select* (page 247).

## Edit Copy

---

Choose **Edit » Copy** to copy selected objects to the Clipboard, from where you can paste them to another design, to another location within the same design, or to another location.

You need to enable the **Select** tool (**Edit » Select**) and select at least one object to choose this command; otherwise the menu command is shaded and the **CTRL+C** shortcut key is unavailable.

Instead of Copy and Paste, you can use **CTRL + left mouse button** (a drag-and-drop operation) to copy the selection within the same design quickly. The **CTRL + left mouse** action does not affect the clipboard.

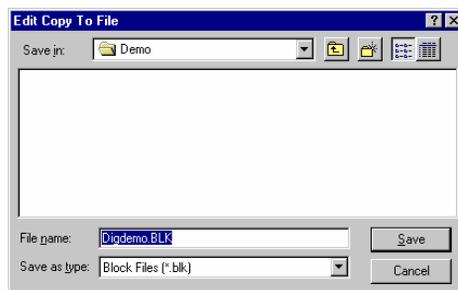
You can copy selected items to a disk file instead of the Windows™ clipboard. For details, see *Edit Copy to File* (page 203).

You can copy multiple objects by using multiple select and block select operations. For information, see *Edit Select* (page 247).

## Edit Copy to File

---

Choose **Edit » Copy to File** to copy selected objects to a block file or meta file. You can later paste objects from these files into a schematic design. To copy objects to a file, you must select at least one item and then choose **Edit » Copy to File**. When you choose this command, the following dialog appears:



You use the options in the *Edit Copy to File* dialog to name the file and select one of the following file formats:

- Block Files (\*.blk). A file format that stores the information that you want to transfer between P-CAD Schematic designs.
- Meta Files (\*.wmf). An image file format that can be transported between computers.

To save storage space, select the **Compress Binary Designs** check box in the *Options Configure* dialog before you save the file. This is beneficial when working with large design files.

For detailed instructions, see *Copying Objects to a File* (page 91).

## Edit Paste

---

Choose **Edit » Paste** to paste objects into your design file from either the clipboard or a block file. You must be in **Select** mode to choose this command.

When you choose **Edit » Paste**, a sub-menu appears, from which you can choose a paste method. The paste choices range from the simple options **Paste From Clipboard** and **Paste From File** in which objects are pasted without net information, to the more intelligent **Paste Circuit** and **Paste Circuit From File** commands which allow you to control changes to component and net names and retain net information.

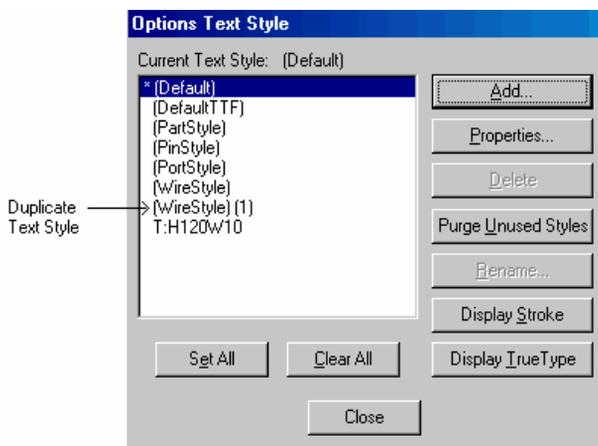
### Paste Behavior

After items have been copied and you press and hold down the left mouse button in the workspace, the ghosted outline of the copied item(s) appears until you release the mouse button to commit them to the desired location. Before releasing the left mouse button, you can drag the items to a more precise location in the workspace.

There must be sufficient space to accommodate the objects being pasted. Error messages inform you if the target space is not large enough or if you are attempting to paste too close to or outside of the edge of the workspace.

If, while pasting a part, parts of the same name but from a different library already exist in the destination design, the part may not be pasted/placed depending on possible conflicts in pin assignments. This conflict could also occur when parts from a Tango Series II schematic are mixed with parts of the same name in a P-CAD library. In effect, the first instance of the part name establishes the standard.

When you paste text of a style that has the same name but a different definition than in the current design, the incoming style name has a bracketed number appended to it to indicate the style conflict.



The new, bracketed style name is added to the list of available styles in the current design. For style information, see *Edit Properties* (page 211) and *Options Text Style* (page 314).

If you are pasting components or nets that have the same names as objects that already exist in the design, you have the option to specify how their names should be changed if you choose the **Edit » Paste Circuit** and **Edit » Paste Circuit From File** commands. With these commands you can also force a specified net to be pasted into the design while retaining the same name.

## Paste From Clipboard

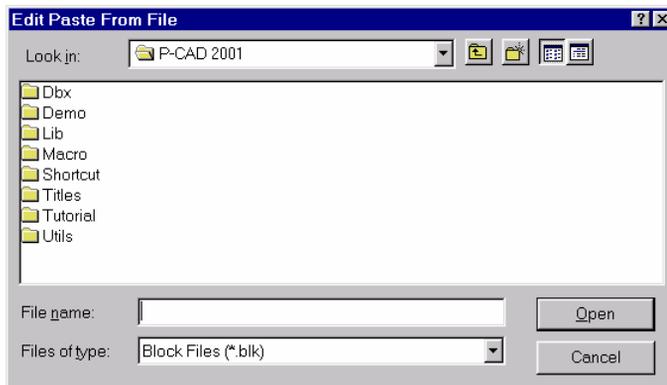
Choose **Edit » Paste From Clipboard** to paste Clipboard objects into your design. This command is available only when data has been copied to the Clipboard and when you are in **Select** mode.

When you choose **Edit » Paste From Clipboard**, Schematic places the copied information in the new location in the design, renames the design objects to the next sequentially available name and retains the component attributes and their values.

## Paste From File

Choose the **Edit » Paste From File** command to paste items from a block file into the current file. To create a block file, select one or more objects and choose the **Edit » Copy to File** command.

To paste objects from a block file into the current file, choose **Edit » Paste From File**. The following dialog appears:

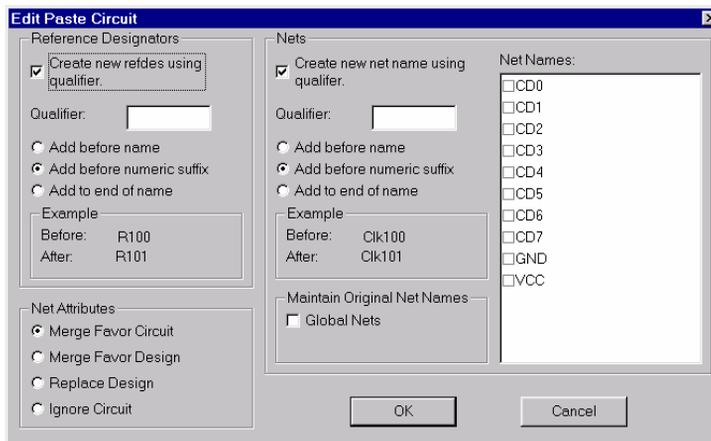


The Look In list box displays the current folder; a list of files in that folder appearing directly underneath. The File name box lets you select or enter a design file, with the extension specified in the Files of type list.

Select the block file containing the item(s) you wish to paste. Once the file is selected, this command works like the **Edit » Paste From Clipboard** command. For details, see *Paste From Clipboard* (page 205).

## Paste Circuit

Choose **Edit » Paste Circuit** to control the naming of components and nets being pasted into a design and stipulate net attribute bias. When you choose **Edit » Paste Circuit**, the following dialog appears:



## Using Paste Circuit

As long as you have not disabled the paste circuit function, you can paste the data from the clipboard as many times as necessary. Each time you paste the same data with a left mouse click,

the component and net names are incremented starting from the last net or component name resident in the design. If you have added a qualifier for the component or net names, the qualifier is incremented with each paste until the component or net name is unique. You can, however, choose to retain the current net names.

When you have finished pasting and you have exited the paste function by right-clicking, you can begin pasting the data again with another **Paste Circuit** command. This time, when you paste this same data, the component and net names begin incrementing from their original names.

### Reference Designators

In the **Reference Designators** frame you can choose how the components are named when pasted. You can either allow Schematic to incrementally change the component names, or you can control the way components are renamed by adding a qualifier in a specific position in the new name.

To rename the components using a qualifier:

1. Select the **Create new refdes using qualifier** check box.
2. Type the characters (no more than four) that you want to add to the name in the Qualifier box.
3. Choose the desired position within the RefDes name for the qualifier. Each of following choices places the qualifier in a different location:
  - Add before name.
  - Add before numeric suffix.
  - Add to end of name.

The Example frame shows a sample, so you can view the change before it is made.

If you do not want to designate the way the RefDes names are modified on paste, clear the **Create new refdes using qualifier** check box.

Each time a component is pasted, the RefDes is sequentially incremented alphanumerically to the next available RefDes for that component. If you have added a qualifier, the qualifier is incremented first in the same manner. Then, if the component name still conflicts with an existing component name, it will be incremented until it is unique.

### Net Attributes

When pasting data containing net information, either from the same design or another design, you can choose how the net attributes are handled choosing one of the following radio buttons in the *Net Attributes* frame:

- **Merge Favor Circuit:** With this option, incoming net attributes take precedence over existing net attributes. When the net attributes in the incoming circuit information match those already in the design, but the values are different, the design attributes are modified to match those of the incoming circuit. All other incoming net attributes are merged with those in the design.
- **Merge Favor Design:** Existing net attributes are retained when you merge attributes favoring the design. When the design and incoming circuit have matching net attributes with different

values, the design attribute values are retained. Other incoming net attributes are merged with those in the design.

- **Replace Design:** This option removes all existing net attributes and replaces them with those of the incoming circuit.
- **Ignore Circuit:** Net attributes attached to the incoming circuit are ignored and existing net attributes left unchanged.

## Nets

In the *Nets* frame you can choose which nets are renamed and how they are named when pasted. This process is similar to the way the components are renamed in the *Reference Designators* frame.

To rename the nets, follow these steps:

1. Select the **Create new net name using qualifier** check box.
2. Type the characters (no more than four) that you want to add to the name in the *Qualifier* box.
3. Choose the desired position for the qualifier within the net name. Each of the following choices places the qualifier in a different location:
  - Add before name.
  - Add before numeric suffix.
  - Add to end of name.

The Example frame shows a sample, so you can view the change before it's made.

If you do not want to designate the way the net names are modified on paste, disable the **Create new net name using qualifier** box.

Each time a net is pasted, the net name is sequentially incremented alphanumerically to the next available net name. If you have added a qualifier, the qualifier is incremented first in the same manner. Then, if the net name still conflicts with an existing net name, it will be incremented until it is unique.

4. The last step is to choose which nets should be renamed using the qualifier and which should retain their current names. You can designate individual nets to retain the current name by clicking the specific nets in the Net Names list box. You can also choose to retain all global net names by enabling the **Global Nets** option in the Maintain Original Net Names section.

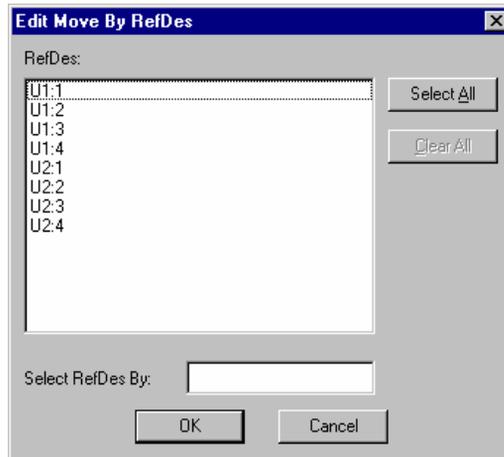
Nets connected to ports are considered global if the port is also copied. If the port is not copied, the nets are considered part of a signal net. This distinguishes the global nets so that you can choose to keep the same net name for the global nets when they are pasted.

## Paste Circuit From File

Choose the **Edit » Paste Circuit From File** command to open the *Edit Paste Circuit From File* dialog and select the file to be pasted. Once the file is selected, the paste operation functions in the same manner as the **Edit » Paste Circuit** command. For details, see *Paste Circuit* (page 206).

## Edit Move By RefDes

Choose the **Edit » Move By RefDes** command to move parts by reference designator. This opens the following dialog.



Use this dialog to select the reference designators of the parts you'd like to move. The following describes this dialog in more detail:

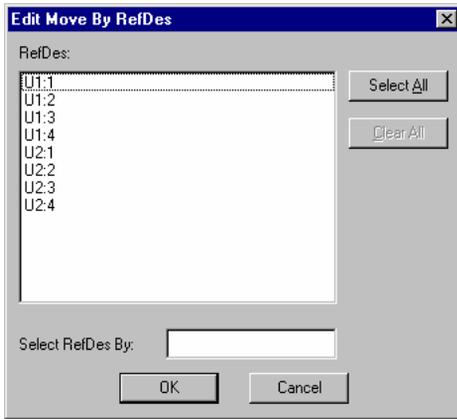
- **RefDes.** This list shows all of the parts on your current sheet. You can select one or more parts from this list. To select multiple parts, hold down the **CTRL** or **SHIFT** key and click the parts to choose.
- If you select parts in your design before you open this dialog, those parts are selected in the RefDes list box when the dialog appears.  
  
When you double-click a part in this list, the dialog closes. At this time, you can place the part into a new position by clicking a location in your workspace.
- **Select RefDes By.** Type search criteria in this box to search the RefDes list for a match. You can enter wildcard characters in this field.
- **Select All.** Click this button to select all of the items in the RefDes list.
- **Clear All.** Click this button to deselect all of the items in the RefDes list box.

After selecting parts from the dialog, click **OK** to return to your workspace. The status line shows the Next RefDes to move. To skip to the next RefDes, press **N**. To move to the previous RefDes, press **B**.

### Moving Parts by RefDes

To move parts by reference designator, follow these steps.

1. Choose **Edit » Move by RefDes** to open the *Edit Move by RefDes* dialog.



If you select parts in your design before you open this dialog, those parts are selected in the *RefDes* list box when the dialog appears.

2. Use one of these methods to select parts from the *RefDes* list:
  - **To select one part:** Double-click a *RefDes* in the list. Or, select a *RefDes* from the list and then click **OK** or press **ENTER**.
  - **To select all parts:** Click **Select All**. Then, click **OK**.
  - **To select a range of parts:** Hold down the **SHIFT** key and click the first and last parts in the range to select. Then, click **OK** or press **ENTER**.
  - **To select various parts:** Hold down the **CTRL** key and click each part to select. Then, click **OK** or press **ENTER**.
  - **To search for parts in the *RefDes* list:** type search criteria in the *Select RefDes By* box and then press **ENTER**.

When the *Edit Move by RefDes* dialog closes, the status line shows the name of the Next *RefDes* to move. You are in placement mode.

To cancel your selection, press **ESC** or right-click. This reopens the *Edit Move by RefDes* dialog, so you can choose another part to move.

3. Choose one of these methods to place the parts:
  - For quick placement, click the location at which you want to place the part.
  - For precise placement, do the following: (1) press and hold down the left mouse button or press the **SPACEBAR**. (2) Move the ghosted part to the location you want. If appropriate,

press **F** to flip or **R** to rotate the part. (3) Place the part by releasing the mouse button or by pressing the **SPACEBAR**.

If you selected multiple parts, the status line shows the Next RefDes to move. To skip to the next *RefDes*, press **N**. To move to the previous RefDes, press **B**.

4. Once you've moved all of the selected parts, the *Edit Move By RefDes* dialog appears. You have these options:
  - To move another part, select it from the RefDes list.
  - To close the *Edit Move by RefDes* dialog, click **Cancel** or press **ESC**.

## Searching the RefDes List

To search for parts in the *RefDes* list, follow these steps:

1. Enter search criteria in the Search by RefDes box. You can type wildcard characters as follows:
  - Type an asterisk (\*) at the end of your search criteria. For example, type **RN\*** to search for any reference designators that begin with RN (e.g., RN1, RN2, RN3).
  - Type a question mark (?) to match a single character in that position. For example, type **C??** to search for reference designators, such as C12, C22, C32.
2. Click **OK** or press **ENTER** to initiate a search of the RefDes list. Then, do the following:
  - If parts matching your wildcard search are selected in the Edit Move by RefDes list, verify the accuracy of the match, and click **OK**. You can now move these parts.
  - If the *Edit Move by RefDes* dialog closes and you return to the workspace. You can now move these parts on the current sheet.
  - If a message notifies you that no match was found, click **OK** to close the message box. Then, try another search.

## Edit Properties

---

Choose **Edit » Properties** to open the *Properties* dialog for the objects you select. With this dialog, you can query and modify the selected object's properties. Before you can choose **Edit » Properties**, you must enable the **Select** tool and then select the objects.

The *Properties* dialog that appears is specific to the object you select. If multiple objects are selected, they must all be of the same type (e.g., arcs), otherwise the command is shaded and no dialog will appear. If the objects are of the same type, the changes you make apply to all selected objects.

## Right-Click to Select Properties

As a shortcut for choosing the **Edit » Properties** command, you can select an object, right-click to open a shortcut menu, and choose **Properties**. The commands available from this shortcut menu vary depending on the object you select.

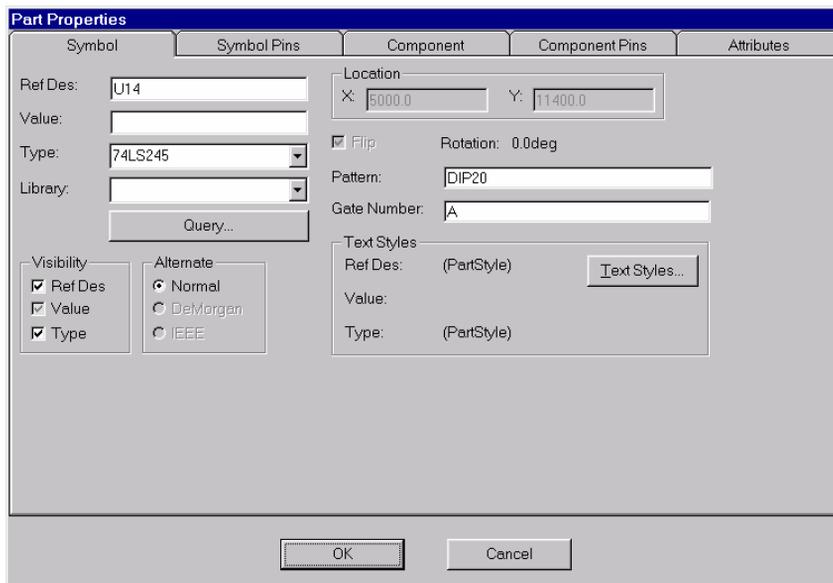
For more information, see *Shortcut Menu Commands* (page 13).

## Double Click to Select Properties

As a shortcut for choosing the **Edit » Properties** command, double-click an object to open the *Property* dialog. This shortcut is available only if the **Double Click Displays Properties** check box is selected in the **Mouse** tab of the *Options Preferences* dialog. For more information, see *Options Preferences* (page 301).

## Part Properties

When you select one or more parts and choose the **Edit » Properties** command, the *Part Properties* dialog appears, as shown in the following figure:



Use the *Properties* dialog to examine properties for the selected part and to modify certain symbol properties. From this dialog, you can selectively replace the component type of one or more gates at the same time. This feature is useful because it lets you replace several gates in your design automatically, preserve reference designators and, where possible, preserve gate connectivity.

Additional tabs provide access to additional information:

- Symbol Pins
- Component
- Component Pins
- Attributes

## Symbol Tab

The following information appears on the symbol tab:

- The Ref Des box shows the reference designator name. To change the reference designator, type a new value in the Ref Des field. If you selected more than one part, this value cannot be changed.
- The Value box shows the part's value. To change the value, type a new value in the Value field.
- The Type list box shows the component type. You can change this value by selecting a different component Type from the drop-down list that appears when you click the down arrow button.

A component Type swap follows the rules used by the **Utils » Force Update** command if you choose the **Merge Attributes** (Favor Library) option. The rules are:

- If an attribute exists in the library component and not in the design component, the library attributes and their values are moved into the design component.
- If the same attribute exists in both the design component and the library component with different values, the value of the library component attribute will replace the value of the design component attribute.
- An existing design component attribute, which has no matching library component attribute, is retained, unchanged, in the design component.

When the Value attribute is present in both the design component and the library component, and has a different value in both places, you must choose which value to keep in the design when prompted.

- The Library list box shows the part libraries available, and displays the currently selected library.
- The **Query** button provides a direct link to the Library Executive Query function where a selected part from the *Query* output dialog can replace the current part and its properties by clicking the **Replace** button.
- The Visibility frame contains check boxes indicating whether the selected part(s) have visible, invisible, or undetermined RefDes, Value, and Type attributes.
- If a box is checked, the attribute is visible. If the box isn't checked, the attribute is invisible. If the box is shaded, then the attribute either does not exist (e.g., there is no Value attribute for the selected part), or there is a conflict between multiple parts selected (e.g., the attribute on one part is visible, but is invisible on another).
- The Alternate frame lets you select the **Normal**, **IEEE** or **DeMorgan** representations of a part if they exist. The symbols displayed for these representations are assigned in Library Executive.
- The Location frame shows the **X** and **Y** coordinates of the part's reference point.
- The **Flip** check box indicates whether or not the symbol has been flipped.
- The Rotation field shows the rotation amount if the symbol has been rotated.

- The Pattern box displays the pattern name. To change the pattern, type a new pattern name in the box. Changing the pattern in this dialog doesn't change the pattern attached to the component in the library.
- A netlist loaded into PCB after these pattern changes will load a component and its pattern for the modified component, depending on the contents of the open libraries. P-CAD PCB will load the first component in an open library having a type matching the modified component. If a component with a matching type cannot be located, a component with a type name matching the modified component pattern name is loaded.
- The Gate Number box shows the section number of this part within the component. To reassign this gate, type a new gate number in the box. You can use a numeric or alphabetic part designation, within the range shown by the Number of Symbols field (Component tab).
- The Text Style frame shows the symbol's text font (PartStyle).

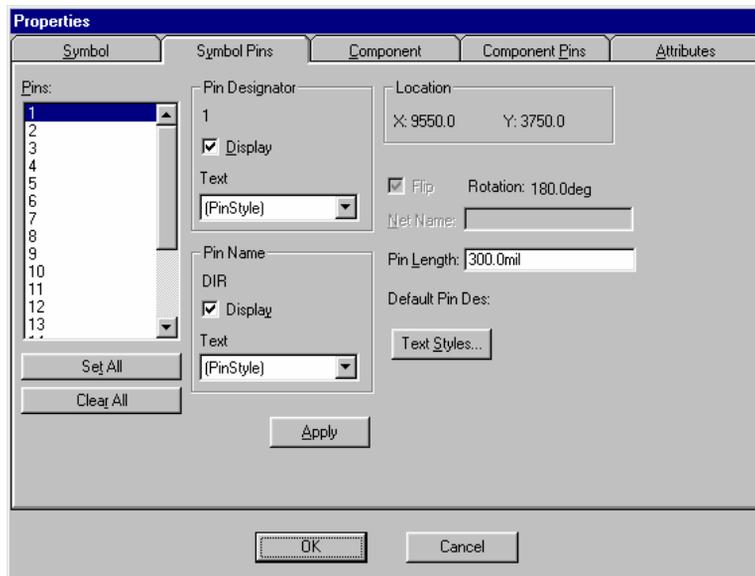
### Text Styles Button

The part's text style (PartStyle) appears in the Text Style field. Click **Text Styles** to open the *Options Text Style* dialog. From this dialog you can edit the part's text style.

Refer to *Options Text Style* (page 314) for more information.

### Symbol Pins Tab

The **Symbol Pins** tab shows the following dialog:



This tab allows you to change several pin properties of one or more pins within the part.

## Pins

The Pins list box lists the pin designators in the selected part as well as power pins in the component. The information fields to the right display the information about the pin that is highlighted.

### Pin Designator/Pin Name

The Pin Name and Pin Designator boxes each contain a display check box and a combo box. The **Display** check box indicates whether the pin designator or pin name on the selected pin(s) are visible, invisible, or indeterminate. In the Text list boxes, you can set the text style for the pin name and pin designator.

### Net Name/Pin Length

You can change the pin length of visible pins and the net name of hidden, global power pins by typing new values in the appropriate Pin Length and Net Name boxes. If a pin is connected to a wire or net, you cannot change the pin length, and the Pin Length box is shaded.

### Set All/Clear All

The **Set All** and **Clear All** buttons can be used to highlight and unhighlight all pins in the list box.

### Query Fields

The following fields allow you to see information about the selected pin:

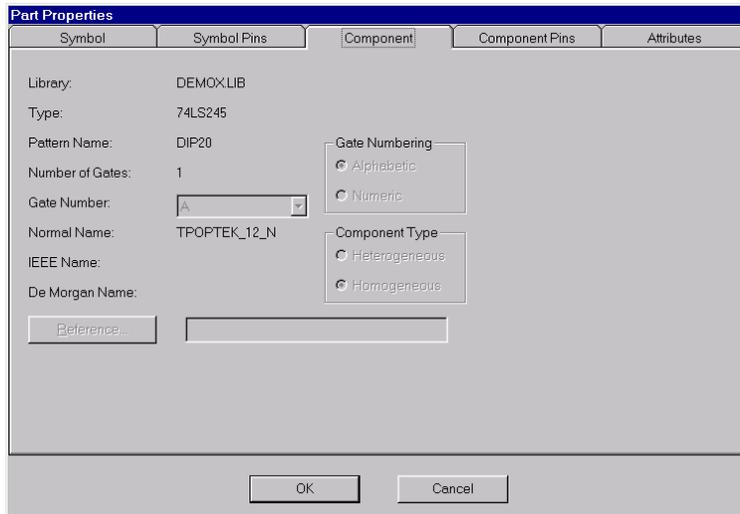
- **Location:** The X and Y coordinates of the selected pin.
- **Flip:** The Flip box indicates whether or not the pin has been flipped.
- **Rotation:** The Rotation field shows the rotation amount if the pin has been rotated.
- **Default Pin Des:** When a default pin designator has been assigned to the pin, the value is displayed here. If multiple pins are selected, and all have the same default pin designator, the value is displayed. When multiple selected pins have different default pin designators, this area remains blank.

### Text Styles

Click **Text Styles** to open the *Options Text Style* dialog. From this dialog you can edit all text style's properties. If you modify a text style in this dialog, all text in your design with that style changes. For more information, see *Options Text Style* (page 314).

## Component Tab

When you select the **Component** tab, the *Port Properties* dialog appears as follows:



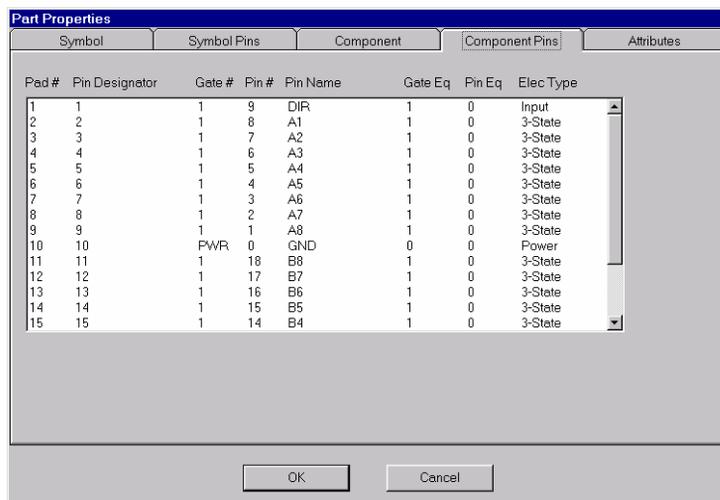
This tab shows information for the component(s) you selected on a gate-by-gate basis. This information is display only; it can't be modified from this dialog. To show information for a different gate, select the gate from the Gate Number drop-down list box.

The **Reference** button, when activated by the presence of the Reference attribute, quickly launches the reference link entered as the Reference attribute's value. If the link is a web address the Internet Explorer is launched and the web site opened. If the link is a document, the associated program is started and the document displayed.

See the *Library Executive User's Guide* for information about this dialog's fields.

## Component Pins Tab

The **Component Pins** tab appears as follows:



Use this dialog to look at pin information for the component pins within the symbol or component. The following information appears:

- **Pad #:** The number of the corresponding pad on the attached pattern. Pad numbers must be unique, and they must exist in the attached pattern.
- **Pin Designator:** The pin designator of each pin in the component.
- **Gate #:** The part number defines the part that the pin is associated with. In multipart components, the parts are uniquely numbered from 1 through n.
- **Pin #:** The number of the corresponding pin on the attached symbol. Pin numbers must be unique and must exist in the attached symbol.
- **Pin Name:** The pin name associated with that pin designator.
- **Gate Eq:** The gate equivalence column defines which gates are equivalent. All gates with the same Gate Eq number are defined to be equivalent. This information is used by P-CAD Schematic when automatically incrementing reference designators (e.g., **Place » Part and Utils » Renumber** commands) and by P-CAD PCB to determine which gates can be swapped during manual or automatic gate swapping. You cannot set the gate equivalency of pins in the same gate to be different. When you change a part number or gate equivalence for a gate, the spreadsheet updates the gate equivalence field of the other pins of that gate to match.
- **Pin Eq:** Indicates which pins within a gate are logically equivalent. The pin equivalence values must be non-zero and identical for a swap to occur between two pins. Non-swappable pins are indicated with a zero value.
- **Elec Type:** The type of pin.

## Attributes Tab

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



You can view, add, modify, or delete a collection of component attributes and access a web site. The dialog contains a two-column table showing the collection of component attributes. Within the collection, each attribute's name and value appear in the column.

## Wire Properties

When you select a wire and choose the **Edit » Properties** command, the *Wire Properties* dialog appears as shown in the following figure:



The following information appears on the *Wire* tab of the *Wire Properties* dialog:

- The Net Name field contains the name of the net to which the wire is associated.
- Select the **Display** option to display or hide the wire name.
- The Width frame contains the width of the selected wire.
- The End Points frame shows the X, Y coordinates for the wire's start and end points.
- The Text Style field shows the text style used for the wire name.

## Wire Width

In the Width grouping, you can modify the width of the selected wire. The wire width can be Thin (10 mils), Thick (15 mils), or User defined. User defined wire widths can range from 0.1 to 100 mils.

For example, to set the wire width to 11 mils:

1. Choose the **User** radio button in the Width frame. The box becomes active.
2. In the box, type: 11mil and click **OK**.

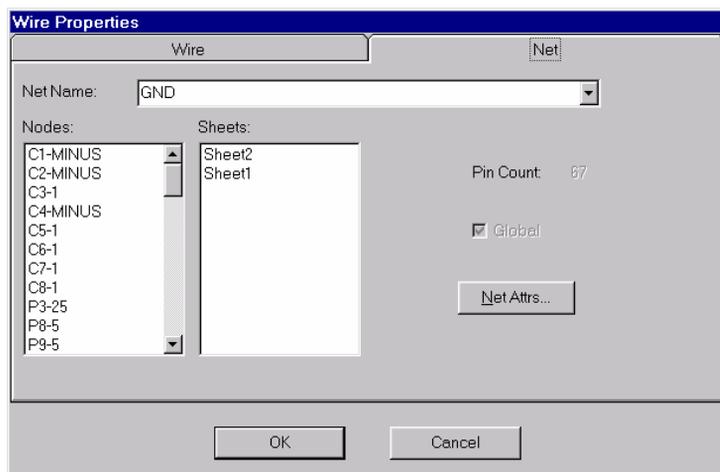
To maintain the appearance of P-CAD Schematic designs before V13.0, all wires less than 11mils wide are displayed as 1 pixel. To set the default wire width, see *Options Current Wire* (page 312).

## Text Styles Button

Click **Text Styles** to open the *Options Text Style* dialog. From this dialog you can add, delete, rename, or edit text styles including the wire style. For more information, see *Options Text Style* (page 314).

## Net Tab

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

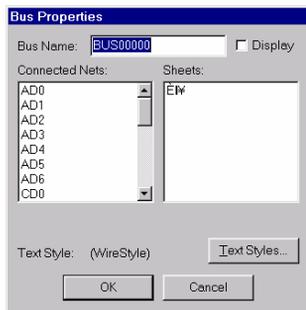


On the **Net** tab of the *Wire Properties* dialog, you can set the following fields:

- **Net Name:** The Net Names list box contains the name of the net associated with this wire. If you attempt to rename the net with the name of an existing net, and one of the nets is not global, an error message appears. You must confirm that you want to autoplacement ports to make both nets global by clicking the **Yes** button. The nets are merged and renamed.
- **Nodes:** The Nodes list box contains the names of all nodes in the net.
- **Sheets:** The Sheets list box contains the names of all sheets on which the net exists.
- **Global:** The Global check box indicates whether or not the net is a global net.
- **Pin Count:** The Pin Count field displays the pin count for the selected net.
- **Net Attrs:** When you click the **Net Attrs** button, the *Attributes* dialog appears.

## Bus Properties

When you select a bus and choose the **Edit » Properties** command, the *Bus Properties* dialog appears:

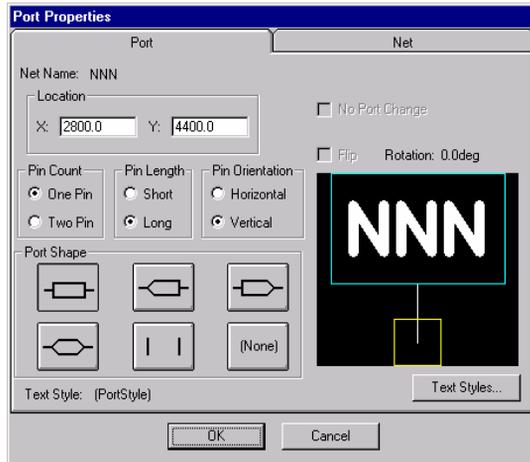


The following information appears on the *Bus Properties* dialog:

- **Bus Name:** The Bus Name list box contains the name of the bus.
- **Display:** Select this check box to display the bus name. Clear the check box to hide the bus name.
- **Connected Nets:** This list box contains the names of all nets connected to the bus.
- **Sheets:** This list box contains the names of all sheets on which the bus is placed.
- **Text Style:** This field shows the text style used for the wire name.
- **Text Styles:** Click the **Text Styles** button to open the *Options Text Style* dialog. From this dialog you can add, delete, rename, or edit the text style of the bus name. Refer to *Options Text Style* (page 314) for more information.

## Port Properties

When you select a port and choose the **Edit » Properties** command, the *Port Properties* dialog appears with the **Port** tab selected:



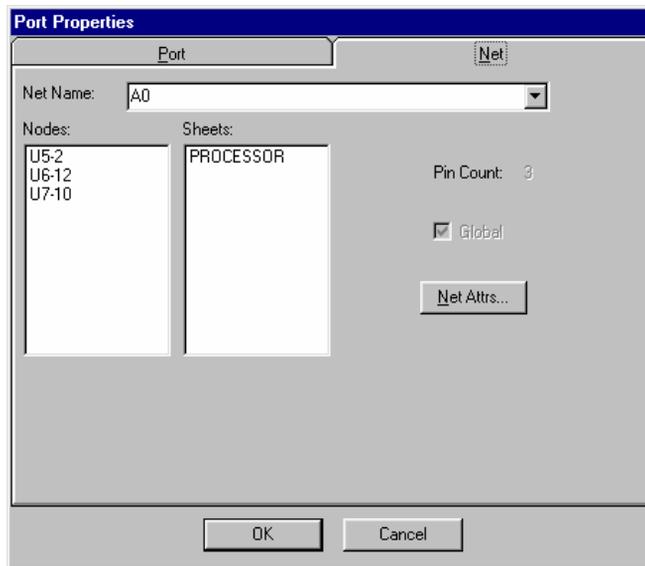
Ports consist of the net name, an optional box surrounding it, and one or two pins to connect it to wires.

The following information appears on the dialog:

- **Net Name:** Shows the name of the net to which the selected port is attached.
- **Pin Count:** Choose the **One Pin** or **Two Pin** radio button.
- **Pin Length:** Choose the **Short** (100 mil) or **Long** (500 mil) radio button.
- **Pin Orientation:** Choose either **Horizontal** or **Vertical**.
- **Port Shape:** Select one of the **Port Shape** buttons. If you don't want a port outline, choose the **None** button.

## Net Tab

When you click the **Net** tab, the *Port Properties* dialog appears:



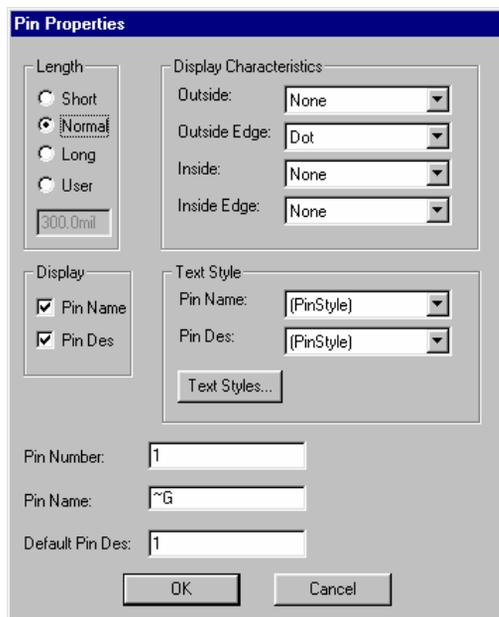
The following information appears on the dialog:

- **Net Name:** The Net Names list box contains the name of the net associated with the port. You can change the net by typing a new net name in the *Net Name* box or by choosing a new net from the drop-down list box. If you attempt to rename the net with the name of an existing net, and one of the nets is not global, an error message appears. You must confirm that you want to autoplacement ports to make both nets global by clicking the **Yes** button. The nets are merged and renamed.
- **Nodes:** The Nodes list box contains the names of all nodes in the net.
- **Sheets:** The Sheets list box contains the names of all sheets on which the net is placed.
- **Global:** The Global field indicates whether or not the net is a global net.
- **Pin Count:** The Pin Count field displays the pin count for the selected net.
- **Net Attrs:** When you click the **Net Attrs** button, the *Attributes* dialog appears.

## Pin Properties

If the pin is part of a symbol, you can access its properties using the subselect feature described in *Selecting Objects* (page 78) or through the *Parts Properties* dialog discussed above.

When you select a pin and choose the **Edit » Properties** command, the *Pin Properties* dialog appears:



The following information appears on the dialog:

- **Length:** Select **Short** (100 mil), **Normal** (300 mil), **Long** (500 mil), or set your own length by clicking the **User** option and entering a length in the box. You cannot change the length of a subselected pin if it is connected to a wire.
- **Pin Number:** You can renumber the pin, by typing over the value that appears in the Pin Number box. You cannot renumber a symbol pin.
- **Display:** Select the **Pin Name** and **Pin Des** check boxes to display the pin name and pin designator. Clear the check box to hide the name or designator.
- **Text Style:** In the Pin Name and Pin Des list boxes, you can set the text style for the pin name and pin designator. Click **Text Styles** to open the *Options Text Style* dialog. From this dialog you can edit all text style's properties. If you modify a text style in this dialog, all text in your design with that style changes.
- **Pin Name:** For a symbol pin, the pin name, like a pin number, cannot be edited. For a free pin, the pin name, or default pin name, is a placeholder for the real pin name. The Pin Name box may be left blank. Use this default label to change the orientation or position of the pin name.
- **Default Pin Des:** Use the Default Pin Des box to assign or change a pin designator for the selected pin. If you have selected multiple pins, which have different default pin designators, the *Default Pin Des* box displays "Hetero\_Selection". Changing the value in this box changes the default pin designator for all the selected pins.

- **Display Characteristics:** Display characteristics include all attribute symbols that may be attached to a pin for design clarification. They are for graphical appearance only. Refer to *Place Pin* (page 269) for examples.

After reviewing or updating the information, click **OK** to close the *Pin Properties* dialog.

## Pin Name and Pin Designator Properties

When you subselect a pin name or pin designator and choose **Edit » Properties**, the *Text Properties* dialog appears:



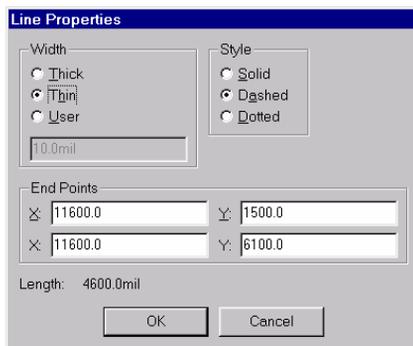
You can view the text properties including style and rotation. Also, you can modify the location or justification of the subselected pin name or pin designator.

A sub-selected pin name or pin designator can be rotated, flipped, and moved.

To modify the text or text style, see *Pin Properties* (page 222). For more information about text properties, see *Options Text Style* (page 314).

## Line Properties

When you select a line and choose the **Edit » Properties** command, the *Line Properties* dialog appears as follows:



The following information appears on the dialog:

- The Width frame lets you choose a radio button to set Thick, Thin, or User (defined) lines. If you select **User**, enter a line width in the corresponding box.
- The Style frame lets you choose a radio button to set a Solid, Dashed, or Dotted line style for thin lines.
- The End Points frame lets you change the lines start and end points by typing new **X** and **Y** coordinates over the existing values.
- The Length box shows the line's length.

If you select multiple lines to modify that are of differing styles and/or widths, the values in the dialog appear blank. You can select a value and make all the selected lines the same style and/or width.

## Arc Properties

When you select an arc and choose the **Edit » Properties** command, the *Arc Properties* dialog appears:

Property	Value
Radius	1774.4
Width	10.0mil
Start Angle	222.4deg
End Angle	248.2deg
Center Point X	2709.6
Center Point Y	10797.2
Start Point X	1399.3
Start Point Y	9600.7
Start Point Tangent Slope Angle	227.6deg
End Point X	2050.6
End Point Y	9149.7
End Point Tangent Slope Angle	201.8deg

The *Properties* dialog for arcs shows you the start and end points for the selected arc.

## Query Fields

For the free arc Center, Start, and End points, see one of the following:

- **X and Y coordinates:** The X and Y location of the Start and End point.
- **Tangent Slope Angle:** The tangent slope angle of the arc.

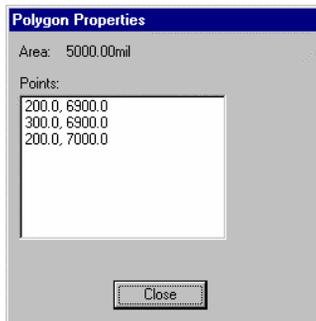
## Changing Arc Properties

To change the properties of an arc, follow these steps:

- **Start Angle:** Click the scroll buttons (up and down arrows) to scroll through arc start angles.
- **X and Y coordinates of the Center Point:** Type over the existing X and Y values in the Center Point box.
- **End Angle:** Click the **scroll buttons** (up and down arrows) to scroll through arc end angles.
- **Radius:** Type a new radius over the existing value.
- **Width:** Type a new line width over the existing value.

## Polygon Properties

When you select a polygon and choose **Edit » Properties**, the *Free Polygon Properties* dialog appears as follows:

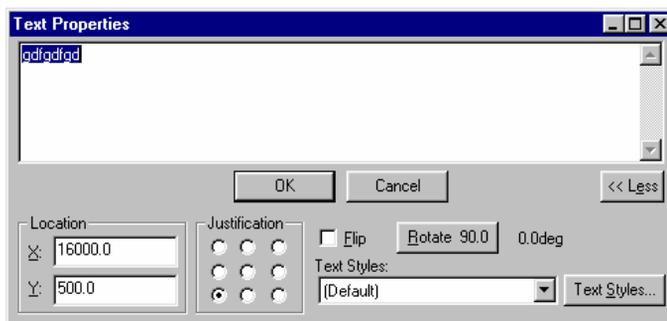


The *Free Polygon Properties* dialog for free polygons shows you the following information:

- **Area:** The area of the polygon.
- **Points:** The X and Y location of each point in the free polygon.

## Text Properties

When you select one or more text items and choose **Edit » Properties**, the *Text Properties* dialog appears:



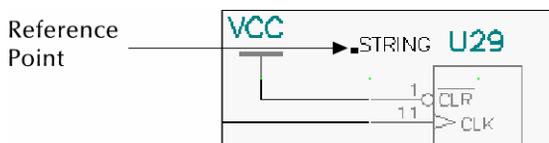
The *Text Properties* dialog remains displayed in your workspace until you close it or choose another tool. Changes made to the text in the box are instantly displayed in the selected text in the design. You can adjust the zoom factor in the design to more easily view the text while the dialog remains on the screen.

From this dialog, you can change the text content, justification, style and location. The text can also be rotated and flipped, and any non-default text styles can be modified.

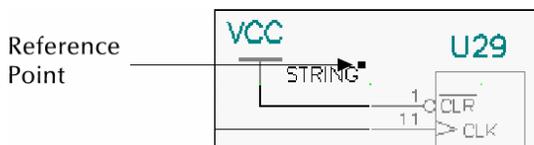
## Changing Text Properties

To change the properties of text, follow these instructions:

- **Text:** Type over the text displayed in the Text box and click **OK**. For multi-line text, **ENTER** creates a line break. You can enter a maximum of 2,000 characters. While the Text box has focus, you can use **CTRL+V** to paste text from the Clipboard.
- **Location:** The X and Y coordinates of the selected text appear in the Location box. You can move the text by typing new coordinates.
- **Justification:** Under Justification are nine buttons, which allow you to change text justification. The reference point of the text string is, by default, at the location of the first character in the string and set to the lower left corner. When you change the Justification button, the text moves around the reference point. For instance, if the text "STRING" is displayed in its default settings the reference point is as shown in the following illustrations:



If you change the Justification to the upper right corner button, the text "STRING" rotates around the reference point and appears in the design as shown in the following illustration:

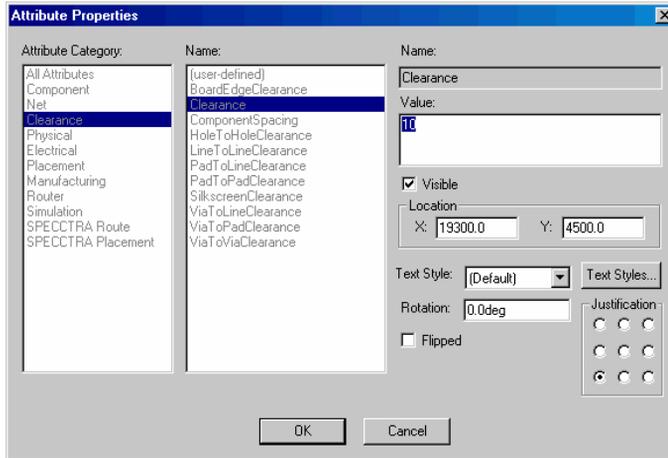


- **Flip:** Enable the **Flip** option by clicking in the box. The text appears flipped.
- **Rotate:** Click the **Rotate 90.0** button to rotate the text 90 degrees. The degree of rotation is displayed next to the **Rotate 90.0** button.
- **Text Style:** Click on the text style you want from the Text Style list box.
- **Text Styles:** Click the **Text Styles** button to display the *Options Text Style* dialog. From this dialog you can add, delete, rename, or edit non-default text styles.

Refer to the *Options Text Style* (page 314) for information.

## Attribute Properties

When you select an attribute and choose the **Edit » Properties** command, the *Attribute Properties* dialog appears:



The following information appears in the dialog:

- **Attribute Category:** This list box displays a list of all attribute categories, All, Component, Net, Clearance, Physical, Electrical, Placement, Manufacturing, Router, and SPECCTRA. Selecting a category brings up a list of predefined attributes for that category.
- **Name:** This list box displays all predefined attributes for the specified category. The first entry in the list is User-defined. The currently-selected attribute also appears in the Name box, unless User-defined is selected. In that case, the Name box is blank so that you can enter a user-defined attribute name.
- **Name:** For user-defined attributes, enter a name for the attribute in this box.

If the dialog is accessed for an attribute that already has a name, then the Category list box, Name list box, and Name box are filled in, but shaded. If the attribute doesn't have a name, these controls are enabled.

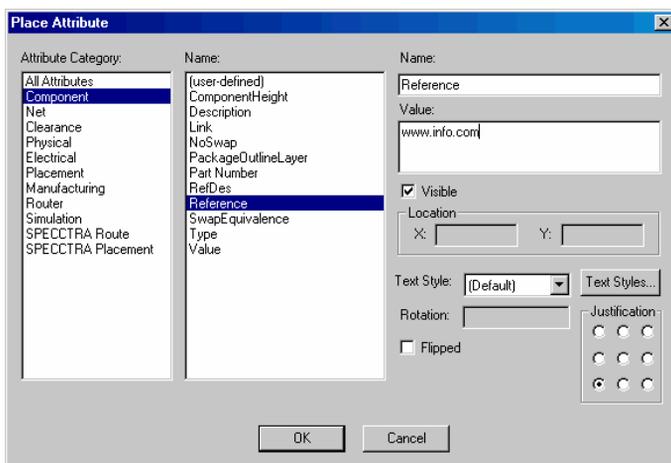
- **Value:** Use this box to enter a value for the attribute.
- **Visible:** This check box indicates whether or not the attribute is visible.
- **Location:** This frame shows the X and Y coordinates of the attribute's reference point.
- **Text Style:** This frame lets you select the attribute text style. Text styles appear in the Text Style drop-down list box. To change the selected text style, click on the text style you want from the list box. To modify the text style, click the **Text Style** button.
- **Rotation:** Shows the rotation amount if the pattern has been rotated.

- **Flipped:** This box indicates whether or not the pattern has been flipped.
- **Justification:** In the Justification frame nine buttons allow you to change text justification by setting the reference point of the text string. For example, if you enable the middle button, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

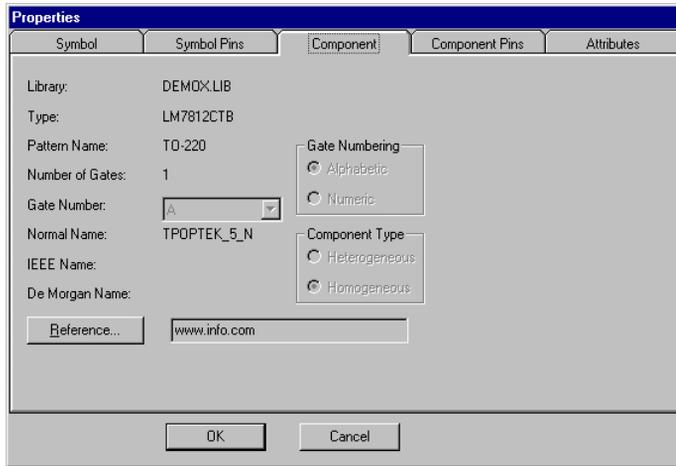
## Component References

A special attribute, which provides direct access to information located in a document or web site, can be added to the part using the *Attribute Properties* dialog. The attribute is named Reference and is associated with the Component Attribute Category. The Reference attribute's Value should be a document or web site address where the additional information about this part is located.

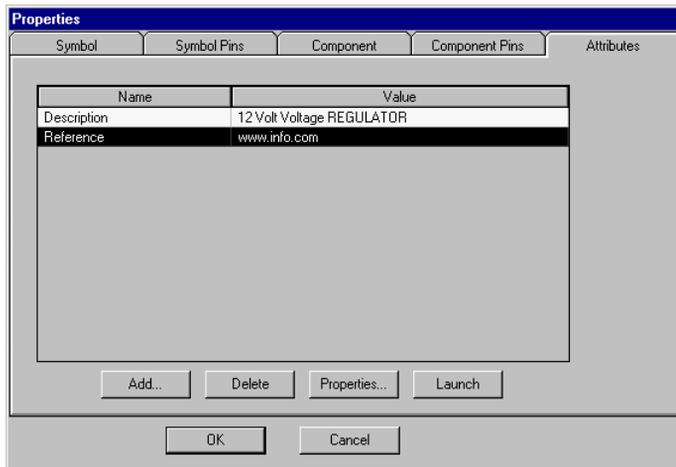
When you add the Reference attribute and Reference location, the *Place Attribute* dialog appears.



Once the Reference attribute has been added, the **Reference** button on the **Component** tab of the *Properties* dialog is activated and displays the reference link (in this case a web address), as shown:



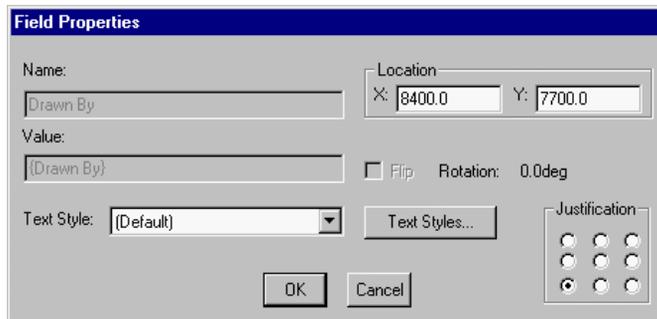
Also, when the Reference attribute and reference link have been added and selected, the **Launch** button on the **Attributes** tab becomes available, as shown in the following figure:



Click the **Launch** button on the **Attributes** tab or the **Reference** button on the **Component** tab to start Internet Explorer and go directly to the designated web address. If the reference link is a document, the associated program is started and the document appears.

## Field Properties

When you select a field and choose the **Edit » Properties** command, the *Field Properties* dialog appears as shown on the following page.



The *Field Properties* dialog lets you view and/or change information about the selected field's properties. You can modify the following fields:

- **Text Style** (list). Select a text style for the field from this list.
- **Text Styles** (button). Click this button to open the *Options Text Style* dialog. You use this dialog to add or modify a text style.
- **Rotation** (data field). Displays the degree of rotation for the field.
- **Justification** (frame). Choose a button in this frame to set the text justification for a field.
- **X and Y** (boxes). Enter an X and Y coordinate to set the location of the field in the design.

## Query Fields

The following information about the selected field can be viewed:

- **Value:** The Value assigned to the field.
- **Name:** The name of the existing field is displayed here.
- **Flip:** The Flip box indicates whether or not the field has been flipped.
- **Rotation:** The Rotation field shows the rotation amount if the field has been rotated.

## Editable Fields

The following information about the selected field can be modified:

- **Text Style:** Choose a Text Style from the list of styles that appear when you click the down arrow button.
- **Text Styles:** If the text style you need does not appear on the Text Style drop-down list, click the **Text Styles** button to display the *Options Text Style* dialog where you can define a new text style. Complete information on the *Options Text Style* dialog can be found in *Options Text Style* (page 314).

- **Location:** The current X and Y coordinates of the field's location. You may enter new coordinates and the field will be placed in the new location when you exit the dialog by clicking the **OK** button.
- **Justification:** Choose the reference point of the field by clicking the desired button. For example, if you enable the middle button, the field reference point (the lower-left corner) moves to the center of the bounding rectangle of the field.

## ERC Errors Properties

When you select ERC error indicators and choose the **Edit » Properties** command, the *Find Errors* dialog appears.

Each indicator represents an error or violation detected in the electrical rules checking pass (**Utils ERC**). You can select multiple error indicators with block select and cycle through them by error number.

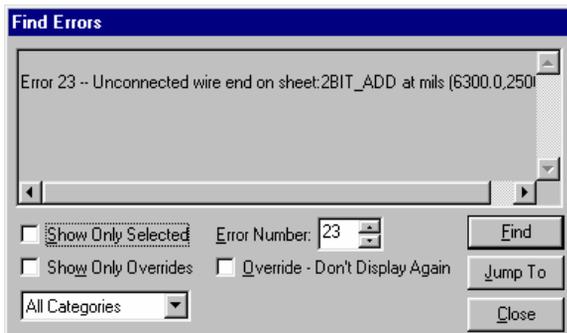
To generate ERC error indicators, use the **Utils ERC** command to enable the **Annotate Errors** option.

To block select ERC error indicators, use the **Options » Block Selection** command to include the indicators in a block selection, then perform the block select.

## Viewing ERC Errors

To view ERC errors, follow these instructions:

1. Select ERC error(s) either individually or through a block select.
2. Right-click the error and choose **Properties** from the shortcut menu, or choose **Edit » Properties**. The *Find Errors* dialog appears as follows:



The *Find Errors* dialog options are:

- **Error Text** (scroll box): Displays the error text.
- **Show Only Selected** (checkbox): Click this box to view selected errors.
- **Show Only Overrides** (checkbox): Click this box to view errors that have been marked as overridden.

- **Category** (list box): Select one or all categories to ERC errors to view.
- **Error Number** (spin box): Click the **Error Number** spin box to scroll forward and backward through the errors associated with the selected ERC error indicators.
- **Overrides – Don't Display Again** (check box): Click this box to override an error.
- **Find** (button): To find an error within the design.
- **Jump To** (button): To jump to a particular error.

## Replacing Component Types

P-CAD Schematic lets you selectively replace the component type of one or more gates at the same time. To replace component types, you need to delete an existing gate, then add a new one. With this feature, you can replace several gates in your design automatically, preserve reference designators and, where possible, preserve gate connectivity.

If you're replacing a group of gates, each gate in the selection must be of the same type. Also, if you select one gate in a multigate component, P-CAD Schematic automatically selects all the gates in that component, even if some of them are not on the current sheet.

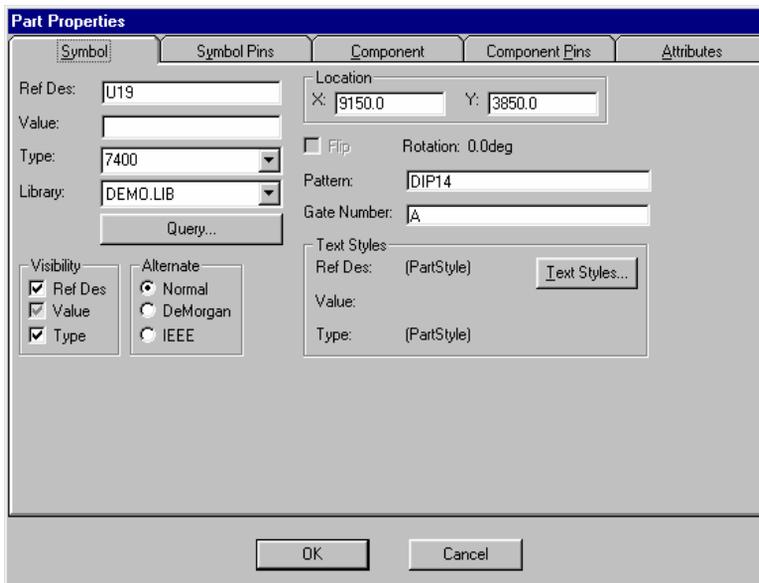
Although P-CAD Schematic does its best to preserve gate connectivity, there are some cases where gates lose their connectivity. This occurs if:

- The old and new component types have different pin designators or different numbers of gates. In this case, the program replaces gates by gate ordering. That is, the first gate of the old component type is replaced by the first gate of the new component type, and so on until all gates are replaced. If the old component type has more gates than the new component type, the program removes the extra gates from the design. However, if the new component type has more gates than the old component type, the extra gates are left unused.
- The new symbol is different from the old symbol.

If P-CAD Schematic cannot maintain gate connectivity, a warning message appears.

To replace component types, follow these steps:

1. Select the component(s) you wish to replace.
2. Choose **Edit » Properties**. The *Part Properties* dialog appears, as shown in the following figure:



Notice the dialog has two list boxes, **Type** and **Library**.

- **Type:** Lists the component types available, and displays the component type of the currently selected component.
  - **Library:** Lists the part libraries available, and displays the currently selected library.
3. Click the Type list box. A drop-down list box shows the available component types in the current library.
  4. Select the desired component type. If the desired component type is not in the current library, select another library from the Library list box.
  5. Click **OK**. P-CAD Schematic replaces the selected components with the new component type.

When type swapping, the program does not apply other Parts Properties changes.

## Edit Delete

Choose the **Edit » Delete** command to delete all selected objects. As a shortcut for choosing this command, press the **DEL** key or select an object, right-click and choose **Delete** from the shortcut menu.

This command does not cut the data to the clipboard as does **Edit » Cut**. To reverse the **Delete** action choose the **Edit » Undo** command.

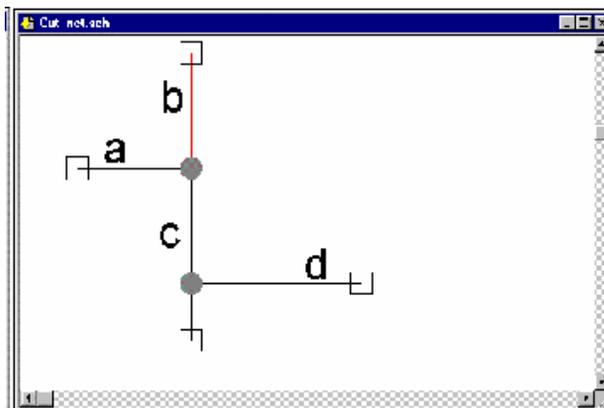
## Deleting Objects

1. Choose **Edit » Select** or click the **Select** button on the toolbar. Click the object you want to delete.
2. Choose **Edit » Delete**. If you inadvertently delete an object, choose **Edit » Undo** to reverse the delete action.

This operation can also be performed with multiple objects by using multiple select and block select operations. Refer to *Edit Select* (page 247).

## Deleting Objects from Nets

When you use the **Edit » Delete** command to remove objects from nets, the results depend on whether the net is a global or local net.



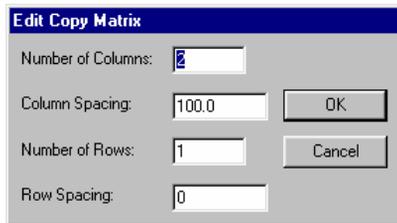
If you delete a wire that isolates a pin from the rest of the net, you end up with a disconnected node that is no longer part of any net. For example, in the above figure, if you delete wire **cd**, the node **d** becomes isolated from the net.

- If the deleted wire has a system-assigned net name (for example, NET00001) then one of the nets is given a new system-assigned net name, while the other retains the original net name.
- If the deleted wire was connected to a global net, the subnet still attached to the port or power symbol retains the original net name, while the other net is renamed to a new system-assigned net name.
- If both subnets are connected to a port or power symbol, then both subnets retain the original name.
- If the wire was connected to a jumper pin and nothing else is connected to that jumper pin. Then all the jumpered pins are removed from the net.

## Edit Copy Matrix

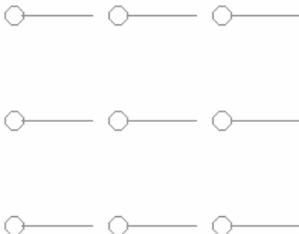
Choose **Edit » Copy Matrix** to duplicate all the selected objects according to the parameters you specify. The objects must be selected before Copy Matrix can function.

When you choose **Edit » Copy Matrix** the following dialog appears:



In the *Edit Copy Matrix* dialog, the Number of Columns and Number of Rows boxes determine the number of X (horizontal) and Y (vertical) duplications, respectively, of a selected object.

The Column Spacing and Row Spacing boxes allow you to enter a value, in current units, to determine the spacing between the duplicated objects. With Column Spacing, a positive value duplicates to the right, a negative value to the left. With Row Spacing, a positive value duplicates up, a negative value down. For example, if you specify 350 mil for Column Spacing and 350 mil for Row Spacing, and specify 3 rows and 3 columns, the result is a matrix with 9 objects 350 mils apart, as shown in the following illustration.



The values represented default to **mm** (millimeters), **mil**, or **in** (inches) depending on what you have set in **Options » Configure** (your current units). You can specify a measurement value (overriding **Options » Configure**) by typing in *mil*, *mm*, or *in* after the numeric value.

### Duplicating an Object

To duplicate an object, follow these steps:

1. Select the object(s) you want to duplicate by clicking the objects to highlight them.
2. Choose **Edit » Copy Matrix** to open the *Edit Copy Matrix* dialog.

3. In the Number of Columns box, specify how many duplications you want to perform horizontally. In the Column Spacing box, enter a value to determine the spacing between duplications and in which direction (positive=right, negative=left) to duplicate.
4. In the Number of Rows box, specify how many duplications you want to perform vertically. In the Row Spacing box, enter a value to determine the spacing between duplications and in which direction (positive=up, negative=down) to duplicate.

You will receive an error message if what you specify for your duplication is too large to fit in the Workspace.

5. Click **OK**. If your duplication is unsatisfactory, select **Undo** to reverse the action.

## Copying Nets

When you copy a contiguous net, P-CAD Schematic always creates a new system-assigned net name. CTRL/Drag and Copy Matrix provide automatic net name incrementation based on increment values you set using the *Options Configure* dialog.

## Edit Explode Part

---

Choose the **Edit » Explode Part** command to convert a part back to its basic primitives and creates a collection of editable graphic objects. When you explode a symbol, the collection of objects is no longer a part and the default pin designators are updated with the current pin designators.

This feature is useful for modifying an existing part or creating a new part from an existing one. After you explode the symbol, you can then perform changes to the objects, such as adding more pins, changing line size or thickness, renumbering pin designators, etc. To aid in creating a new component, the pins default pin designators are updated with the values of the current pin designators after exploding.

Make sure that the part you are editing has not been flipped because if some objects in the symbol are flipped and others aren't, the behavior might be unexpected when you flip the symbol.

## Edit Align Parts

---

Parts can be aligned around a selection reference point either horizontally or vertically, and as an option, equally spacing the parts. Or, if a number of parts are off-grid, these parts can be aligned back on-grid.

There are two important things to remember about the **Edit » Align Parts** command:

- The alignment of parts cannot be undone.
- The alignment of parts has full macro support.

## Edit Select All

---

Choose **Edit » Select All** to select all items on the current sheet. For more information on this command, see *Selecting All Objects* (page 80).

## Edit Deselect All

---

Choose **Edit » Deselect All** to cancel the selection of all selected items.

## Edit Highlight

---

Select one or more items in your design and choose **Edit » Highlight** to apply the current highlight color to those items. The objects retain the highlight color until you use the **Edit » Unhighlight** command to remove the highlight color from an item.

You can also access this command by selecting an item or items, right-clicking the items and choosing **Highlight** from the shortcut menu.

This command interacts with the DDE Hotlinks feature.

## Edit Unhighlight

---

Choose **Edit » Unhighlight** to remove the highlight color from the items you select. As a shortcut for choosing this command, you can also select an item, right-click and choose **Unhighlight** from the shortcut menu.

This command interacts with the DDE Hotlinks feature.

## Edit Unhighlight All

---

Choose **Edit » Unhighlight All** to remove the highlight color from all items on all sheets in the design. When you use this commands, the objects typical color is restored. This command applies to all highlighted objects, regardless of whether they are selected or not.

This command interacts with the DDE Hotlinks feature.

## Edit Select Highlight

---

Choose the **Edit » Select Highlight** command to select all highlighted objects in your design. This is useful when you want to edit a group of highlighted objects.

Your block selection criteria determines which highlighted items are selected when you use this command. To set block selection criteria, see *Options Block Selection* (page 287).

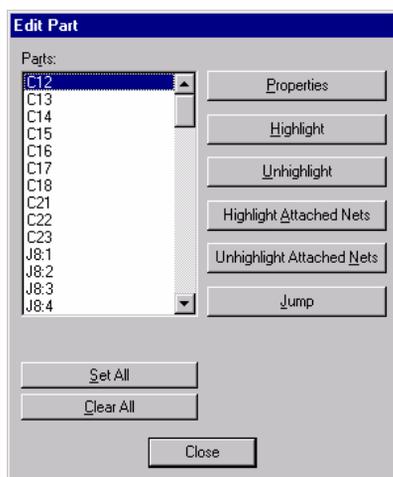
In addition, this command won't select highlighted sub-items within an object. For example, if you highlight two pins within a component and choose **Edit » Select Highlight**, neither the pin nor the component will be selected.

If any items are selected, but not highlighted, when you choose **Edit » Select Highlight**, those items remain selected. For example, highlight three components on a sheet and then select another component, but do not highlight it. Now, choose **Edit » Select Highlight**. P-CAD Schematic selects the three highlighted components, and the other component remains selected.

## Edit Parts

---

Choose **Edit » Parts** to edit parts within your design and to jump to a particular part. This command also allows you to highlight parts and nets attached to a particular part.



### Parts List Box

The Parts list box contains the names of all parts in the active design. You can select individual or multiple parts in the list box. Once selected, you can highlight and unhighlight parts and attached nets and jump to a part.

When you close the dialog, parts selected in the list box are automatically selected in the design.

### Set All/Clear All

If you want to select all parts in the Parts list box, click the **Set All** button. If you don't want any parts selected, click the **Clear All** button.

### Properties

The **Properties** button accesses the *Part Properties* dialog for the selected part or parts. See *Part Properties* (page 212) for details.

## Highlight/Unhighlight

The **Highlight** button highlights the part or parts selected from the Parts list box in the current highlight color set in the **Colors** tab of the *Options Display* dialog. You can use this feature to highlight objects using different colors. When you change the highlight color of one object, it does not affect the highlight color of the other highlighted objects.

When you choose this command, the selected parts are drawn in the highlight color and remain displayed in that color until the highlighting is removed. The selected color overrides the highlight color, so you won't see the highlights until the parts are deselected.

If P-CAD Schematic and P-CAD PCB are both running, and if the **DDE Hotlinks** check box in the *Options Configure* dialog is selected in both programs, then part highlight information is communicated between the two programs. Highlighting a part in P-CAD Schematic highlights the corresponding component in the P-CAD PCB.

If you change the highlight color of an object in P-CAD PCB, the corresponding object in P-CAD Schematic is automatically updated with the same highlight color.

The **Unhighlight** button removes the highlighting from the selected part or parts.

## Highlight/Unhighlight an Attached Net

You can highlight nets, which are attached to the part or parts selected in the Parts list box:

1. To highlight attached nets, select a part from the Parts list box. (You can use the **Set All** button if you want to select all parts in the list for highlighting.)
2. Click the **Highlight Attached Nets** button. The attached nets, including bus connections, junctions, ports, wires and pins, are highlighted with the highlight color set in the *Colors* tab of the *Options Display* dialog.
3. To remove a highlight, select the part from the Parts list box.
4. Click the **Unhighlight Attached Nets** button.

## Jump

This button allows you to jump to the selected part:

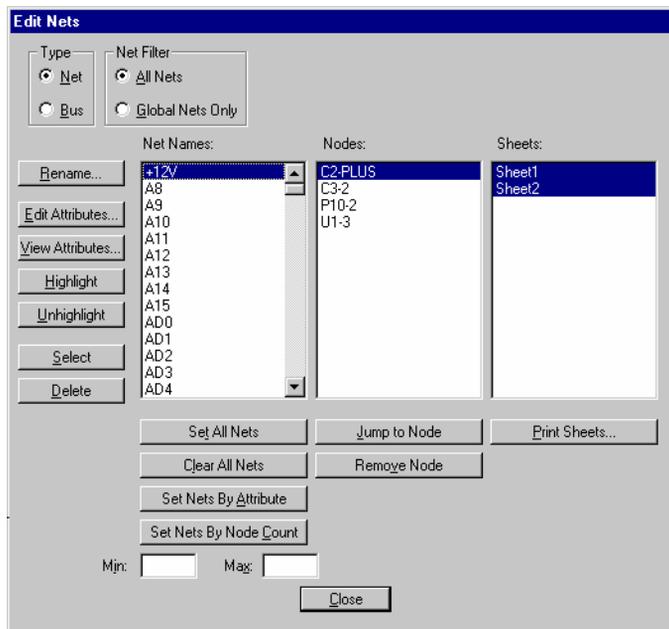
1. Select one part from the list.
2. Click **Jump** to jump to the specified part. The specified part appears in the center of your workspace.

## Edit Nets

---

Choose **Edit » Nets** to select nets, edit net attributes and to rename and delete nets and buses. A net is an electrical connection (e.g., two pins connected by a wire). Buses are a graphical representation of a bundle of one or more wires used to show multiple parallel wires on the schematic.

When you choose **Edit » Nets**, the following dialog appears:



- **Rename** (button). Click this button to open the *Net Name* dialog, which gives you the ability to rename the selected net.
- **Edit Attributes** (button). Select a net in the *Net Name* list and click this button to open the *Attributes* dialog, which gives you the ability to modify net attributes.
- **View Attributes** (button). Select a net in the *Net Name* list and click this button to open the *Attributes* dialog, which gives you the ability to view net attributes.
- **Highlight** (button). Select a net and click this button to highlight the net.
- **Unhighlight** (button). Select a net a click this button to remove the highlight color from the selected net.
- **Select** (button). Select a net and click this button to select the net in the schematic design.
- **Delete** (button). Select a net a click this button to delete the selected net from the schematic design.

### Type Frame

- **Net** (button). Choose this button to edit nets.
- **Buses** (button). Choose this button to edit buses.

### Net Filter Frame

- **All Nets** (button). Choose this button to view all nets.
- **Global Nets Only** (button). Choose this button to view only global nets.

### Net Names/Bus Names

- **Net Names** (list box). This list contains the names of all nets in the active design. Nets from global power pins also appear in the list box. You can select individual or multiple nets in the list box. Once selected, you can edit net attributes, rename a single net, delete nets, and select nets.
- **Bus Names** (list box). This list contains the names of all buses from the active design. You can select individual or multiple buses in the list box. Once selected, you can delete or rename buses.
- **Set All Nets** (button). Click this button to select all nets listed in the Net Names or Bus Names list box.
- **Clear All Nets** (button). Click this button to cancel the selection of all next listed in the Net Names or Bus Names list box.
- **Set Nets by Attribute** (button).
- **Set Nets by Node Count** (button). Nets with a specific number or range of nodes can be selected by entering the Min and Max values and clicking the **Set Nets By Node Count** button.

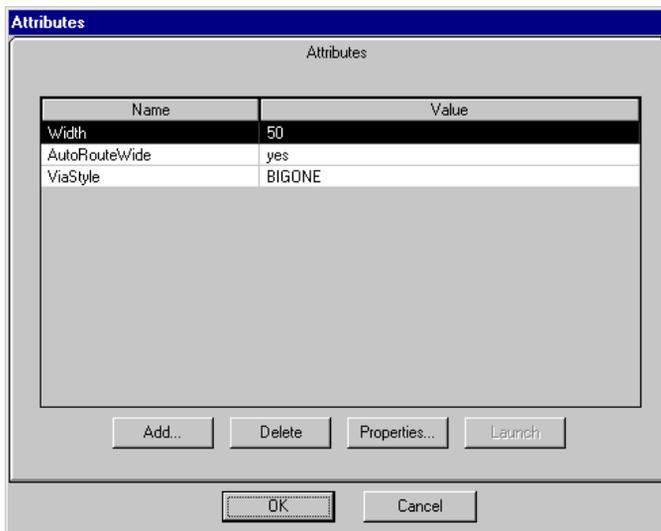
### Nodes

- **Nodes** (list box) contains the list of nodes in the selected Net or Bus.
- **Jump to Node** (button). After selecting a Net Name and then one of the Nodes in the net, click the **Jump to Node** button to be placed in that node's location.
- **Remove Node** (button). You can select a node from the list of nodes on a net and remove that node from the net by clicking the **Remove Node** button.

### Sheets

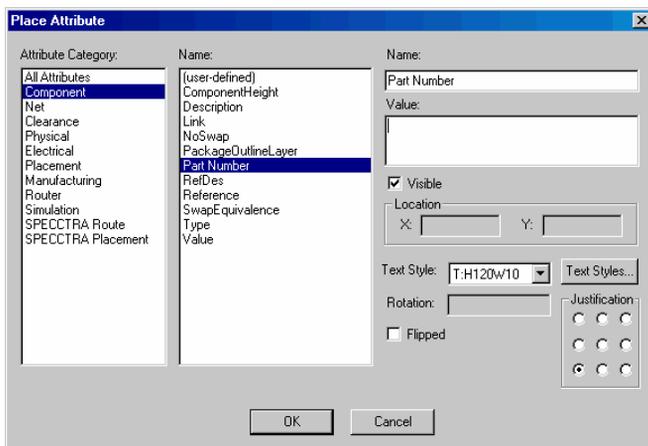
- **Sheets** (list box). When a net from the Net Names box is selected, the *Sheets* box contains a list of the sheets on which the select net appears.
- **Print Sheets** (button). Click this button to open the *File Print* dialog, with the sheets selected in the Sheets box of the *Edit Nets* dialog selected in the Sheet box of the *File Print* dialog. For details on the *File Print* dialog, see *File Print* (page 172).

## Attributes Dialog



The dialog contains a two-column spreadsheet showing the collection of net attributes.

The *Place Attribute* dialog appears as follows:



The following information appears in the dialog:

- Attribute Category** (list box): Displays a list of all attribute categories: All, Component, Net, Clearance, Physical, Electrical, Placement, Manufacturing, Router, and SPECCTRA. Selecting a category brings up a list of predefined attributes for that category.

- **Name** (list box): Displays all predefined attributes for the specified category. The first entry in the list is User-defined. The currently-selected attribute also appears in the Name box, unless User-defined is selected. In that case, the Name box is blank so that you can enter a user-defined attribute name.
- **Name** (text box): For user-defined attributes, enter a name for the attribute.

If the dialog is accessed for an attribute that already has a name, then the *Category* list box, *Name* list box, and *Name* box are filled in, but shaded. If the attribute doesn't have a name, these controls are enabled.

- **Value:** Use this box to enter a value for the attribute.
- **Visible:** This check box indicates whether or not the attribute is visible.
- **Location:** This frame shows the X and Y coordinates of the component's reference point.
- **Text Style:** This frame lets you select the attribute text style. Text styles appear in the Text Style drop-down list box. To change the selected Text Style, click on the text style you want from the list box. To modify the text style, click the **Text Style** button.
- **Rotation:** Shows the rotation amount if the pattern has been rotated.
- **Flipped:** This box indicates whether or not the pattern has been flipped.
- **Justification:** Under **Justification** are nine buttons, which allow you to change text justification by setting the reference point of the text string. For example, if you enable the middle button, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

## Rename

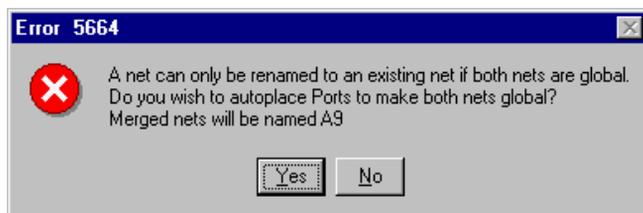
The **Rename** button is enabled only when you have highlighted a single net or bus.

### For Nets

When you highlight a single net name in the Net Names list box, the **Rename** button becomes active. Click **Rename** and the *Net Name* dialog appears:



Type a new name and click **OK** to rename the net and return to the *Edit Nets* dialog. If the name is already assigned to another net, and one of the nets is not global the following error message appears:



You must confirm that you want the autoplacement Ports to make both nets global by clicking the **Yes** button. Ports are placed on both nets and the net is renamed.

### For Buses

When you highlight a single bus name in the Bus Names list box, the **Rename** button becomes active. Click **Rename** and the *Bus Name* dialog appears:

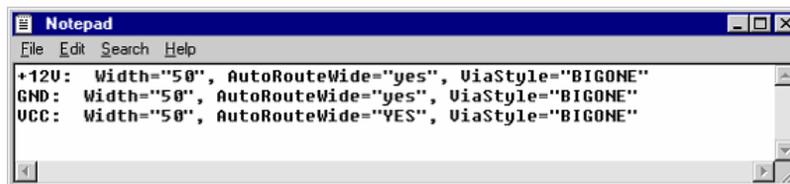


Type a new name and click **OK** to rename the bus and return to the *Edit Nets* dialog. If the name is already assigned to another bus, you are asked if you want to merge the buses.

### View Attributes

When you click the **View Attributes** button, a report showing the attributes and their values is generated and displayed in the Notepad.

An example of the report is shown in the following figure:



### Highlight/Unhighlight Nets

When the **Net** radio button is selected, the **Highlight** button highlights the net or nets selected from the Net Names list box in the current highlight color set in the **Colors** tab of the *Options Display* dialog. Highlighted items include bus connections, junctions, ports, wires and pins. If hotlinks are enabled, the corresponding nets are highlighted in PCB.

This options works in the same way as the Edit Highlight command discussed above.

The **Unhighlight** button removes the highlighting from the selected net(s) and included items. If hotlinks are enabled, the corresponding nets are no longer highlighted in P-CAD PCB.

## Highlight/Unhighlight Buses

When the **Bus** radio button is selected, the **Highlight** button highlights the bus or buses selected from the *Bus Names* list box in the current highlight color set in the **Colors** tab of the *Options Display* dialog.

The **Unhighlight** button removes the highlighting from the selected bus or buses.

## Delete

The **Delete** button deletes nets or buses highlighted in the list box.

## Select

The **Select** button selects all items in the nets highlighted in the list box. Select is sensitive to the settings in Options Block Selection.

You can select a net or multiple nets in the design by highlighting the net names in the list box in the *Edit Nets* dialog, then clicking the **Select** button (the button is available only if a net is chosen). The nets in the design are displayed in the selection color. You can also select nets through the **Select** shortcut menu.

## Edit Measure

---

Choose the **Edit » Measure** command to measure the X distance, Y distance, and total distance between two points. As a shortcut for choosing this command, click the **Edit Measure** toolbar button. You can measure vertical, horizontal, and diagonal distances and the results will be displayed on the status line. The measurements appear either in mils, millimeters or inches depending on the current settings in *Options Configure*.

The Measure tool remains enabled until you enable another tool.

## Using Edit Measure

To measure between two points on your workspace:

1. Choose **Edit » Measure** or click 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. The results will be displayed on the status line for X distance, Y distance, and T (for total) distance (in mm or mils). You can't see the measurements if the status line is disabled (View » Status Line).

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

## Edit Select

---

Choose the **Edit » Select** command to enable the **Select** tool. When you are in Select mode, you can select objects in a design and perform various actions on those objects. As a shortcut for choosing this command, you can click the toolbar button or press the **S** key.

### Status Line Information

The status line information area identifies selected items, either specifically (part reference designator, net name, or bus name) or generally (number of items selected).



### Select Commands

When you are in **Select** mode, the following commands are available:

- Edit Cut
- Edit Copy
- Edit Copy to File
- Edit Paste
- Edit Paste From File
- Edit Delete
- Edit Copy Matrix
- Modify
- Edit Explode Part
- Edit Highlight
- Edit Unhighlight

In addition, **Select Net** is available from the **Select** shortcut menu or through **Edit » Nets**.

Select actions are possible only if an object is selected. For example, you cannot move a part unless it is selected; you cannot modify a line unless it is selected.

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

- For keyboard equivalents to standard mouse functions in P-CAD Schematic, see *Keyboard Reference* (page 377).
- For the Edit menu command descriptions, refer to the respective command in this chapter.

## Selecting Objects

- **To single select**, click an object. All other selected objects are deselected. Selected objects are drawn in the selection color, and are contained within a boundary box. Primitive objects (arcs, lines, etc.) have handles (small squares used for sizing an object).
- **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, and the selection box 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.

These are the default keys, but you can use the *Options Preferences* dialog to change these select keys.

- **To subselect**, hold down the **SHIFT** key and **left mouse click** the object. This command lets you select a single item on an object (a pin on a part). Once selected, you can view and, in some cases, modify properties for the item selected.
- **To select all objects**, choose the **Edit » Select All** command. All objects on the current sheet in the current design are selected.
- **To deselect all objects**, click on an empty area of the Workspace to deselect all items 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 or delete objects to or from the block selection individually by doing a *multiple select*. To cancel while dragging the selection box, right-click.
- If you have the **Outside Block** radio button selected in the *Options Block Selection* dialog, then the selection occurs outside of the selection block. If you have the **Inside Block** radio button selected, then the selection occurs inside of the selection block. If you have the **Touching Block** option enabled, a block selection includes everything inside and touching the selection box.
- 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 to set the selection options; refer to the command documentation for complete information.

## When Objects Overlap

When objects overlap, it may seem difficult to select an underlying object. Continue clicking without moving the cursor and Select cycles through all objects underneath the cursor. The **SPACEBAR** is easier to use than the mouse in this situation (pressing the **SPACEBAR** twice equals click and release for the left mouse button).

## Moving and Copying Objects (Drag-and- Drop)

To move an object, first select it, then click on the object and drag the cursor (with the object attached) to the new location. Release to place the object.

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

You can copy objects in the same manner; after you select the objects, hold the **CTRL** key down and drag a copy of the objects to where you want to place them. When copying a part, the RefDes (reference designator) changes in the copy. When copying non-global nets, the copy's net name is incremented or decremented depending on the value you set for the **Options » Configure Net Increment** option. To cancel a move or copy in progress, right-click while holding down the left mouse button.

## Resizing Objects



You can resize a selected object by clicking on one of its handles, and dragging to stretch the object. The resize function varies for the different objects. For example, to resize an arc you click on one of the endpoint resize handles and drag the endpoint to increase the sweep angle.

To resize a polygon, you can grab one of its vertex handles and move it to change the polygon. When you move a handle that is on an edge between two vertices, a new vertex is created. You can delete a vertex by moving it to an adjacent vertex and releasing.

Wires, buses, lines, arcs, and polygons can be resized with the Select mode resize function.

## Rotating and Flipping

Select an object. This function works on multiple- or block-selected objects as well.

Press **R** to rotate 90 degrees counterclockwise. To flip the object in the X direction (about the Y axis), press **F** while the object is selected. For items with associated connections, press the left mouse button while rotating or flipping to prevent reconnection attempts each time you rotate or flip. Connections are committed when the left mouse button is released.

## Edit Properties

With the **Select** tool enabled, choose **Edit » Properties** or use the shortcut menu after an object is selected. This displays a *Properties* dialog for the selected object. For details, see *Edit Properties* (page 211).

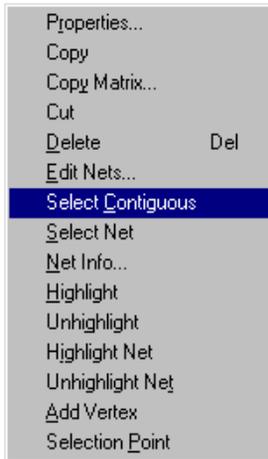
## Select Contiguous

The **Select Contiguous** command allows the selection of all contiguous net objects of a selected net.

For example, to use the **Select Contiguous** command:

1. Select a wire or port.

2. Right-click the wire or port to open a shortcut menu.



3. From the shortcut menu, choose the **Select Contiguous** command.

All contiguous net objects of the net are selected unless you set the Block Selection mask to filter out wires or ports.

This feature will only work for one net at a time.

## View Commands

### Using the View Commands

---

The View commands allow you to temporarily alter the display characteristics of your workspace to better pinpoint locations and objects in your design. You can also alternately enable or disable the information displays and the toolbar.

For more permanent adjustments to your display characteristics, such as color display and style changes to objects, refer to the **Options** menu.

### View Redraw

---

Choose **View » Redraw** to clear and then repaint the active window without changing the current view.

Use **View » Redraw** when you have leftover wires and shapes from moving or deleting objects, to erase leftover graphics.

To interrupt a redraw in progress, right-click or press **ESC**.

### View Extent

---

Choose **View » Extent** to display the extent of all objects on the current sheet in the active window.

P-CAD Schematic computes and draws the workspace so that all objects on the current sheet are visible. Any items on the title sheet are not considered in the extent.

### View Last

---

Choose **View » Last** to redraw the previous view for the active window, if you have altered the view in any way in this window.

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

---

Choose **View » All** to redraw the entire workspace in the active window.

The workspace size displayed is determined by the **Workspace Size** option set in *Options Configure*. If you want to make the workspace smaller to fit the design you are working on, then size it accordingly using *Options Configure*. The scroll bars don't appear at this zoom level.

**View » All** is the default view when you start up P-CAD Schematic with an empty workspace.

## View Center

---

Choose the **View » Center** command to redraw the screen using the cursor as the relative center point.

When you choose **View » Center**, the cursor takes the shape of a magnifying glass. Move the cursor to a point on your design and click the workspace. The point where you click is centered on the screen. To cancel the zoom after the magnifying glass cursor appears, you can right-click or press **ESC**.

As a shortcut for choosing this menu command, move the cursor to a point in your design and press the **C** key. This is quick way to pan across your workspace. You don't need to click the mouse button, just move the cursor to the point you want centered and press **C**.

## View Zoom Out

---

Choose the **View » Zoom Out** command to zoom out by the magnification factor set in *Options Configure*. This is a temporary mode that does not affect the 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 reinvoke the command for every zoom action. To cancel the zoom after the zoom cursor appears, right-click or press **ESC**.

### Minus Key (-)

An easier way to zoom out is to press 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 the workspace.

## View Zoom Window

---

Choose the **View » Zoom Window** command or click the toolbar button to zoom in on an area of the workspace. When you choose this command, the zoom window tool becomes active. As a shortcut for choosing this command, press the **Z** key.

You use the zoom window tool to draw a bounding outline around a selected region of your workspace. A bounding outline is dotted rectangle that appears when you drag the mouse cursor across your workspace, to select a range of items or a design region. The bounding outline is also called a selection block or zoom window.

The zoom window mode is a temporary mode that does not affect whatever other mode you may be operating in.

## Zooming In with a Zoom Window

To zoom in on a region of the design with a zoom window, follow these steps:

1. Press the **Z** key, click the **Zoom** button on the toolbar, or choose the **View » Zoom Window** command. The cursor takes the shape of a white magnifying glass shape.
2. Press and hold down the left mouse button. Then, drag the mouse cursor across your workspace to draw a bounding outline around the region you want to view.
3. Release the mouse button when you have completed the bounding outline. Whatever you surround with the outline will be enlarged on the active window.

The mode you were in is still active and will resume after you do the zoom window action (the zoom window action is a temporary mode).

You must drag the cursor to create a zoom window. If you click and release in the active window 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, right-click or press **ESC**.

## View Jump Location

---

Choose the **View » Jump** Location command to move the cursor to a specified location (X, Y coordinates).

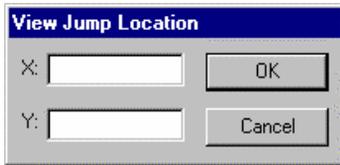
This command pans the active window to the specified location, attempting to center the location. Your current zoom setting is not changed by the jump location panning (except **View » Last** is updated). If the specified location is already visible on the active window, no panning is necessary.

The units used for the location value (mil, mm, or in) are determined by the setting in *Options Configure*. Choose **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 295)* for more information.

## Jump to a Location

1. Choose the **View » Jump** Location command. The following dialog appears:

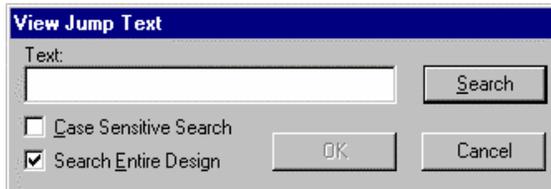


2. In the dialog, specify the **X** and **Y** coordinates (in the boxes) of where you want your cursor to be in the workspace.
3. Click **OK**.

## View Jump Text

---

Choose the **View » Jump Text** command to search through all text strings to locate a specific combination of letters. The character string that you specify can be part of a word or a complete word.



**Case Sensitive Search.** Select this check box to search for text matches based on case. If you disable this option, text case will be ignored in the search.

**Search Entire Design.** Click this check box to search all sheets for the specified text string. It will search only the current sheet when you clear the check box.

## Jump to Text

1. Choose **View » Jump Text**. The *View Jump Text* dialog appears.
2. Type the text you want to search for in the Text box. Notice that the *OK* button is grayed out. Click **Search**.
3. If the matching text is not already on the active window, the program pans the active window at the current zoom level to locate the text and center its reference point. You can set the text reference point by setting the justification with the **Options Text Style** command.
4. After P-CAD finds the first instance of the specified text string (highlighting it in the highlight color), the **Search** button changes to **Next**, allowing you to find subsequent instances of the same text string. You can continue to cycle through all occurrences of the search pattern until you click **Cancel** or **OK**.

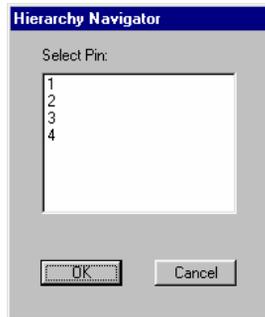
Once a string has been found, the **OK** button is no longer grayed; if you click it, the program moves the cursor to the highlighted text and the highlighting disappears.

Highlighted text may be obscured by the dialog; move the dialog to view the text if necessary.

## View Descend

---

In a hierarchical schematic design, you can easily move down the hierarchy by choosing the **View » Descend** command. When you select a module, the **View Descend** command becomes available. Choose **View » Descend** and the *Hierarchy Navigator* dialog appears:



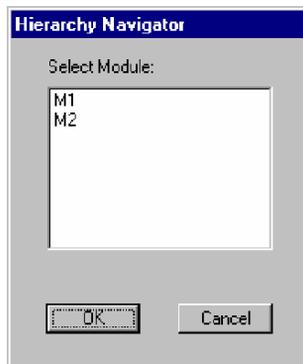
Select a link pin number from the Select Pin list and click **OK**. The selected link becomes the worksheet's focus.

A shortcut command is available for View Descend.

## View Ascend

---

In a hierarchical schematic design, you can easily move up the hierarchy by choosing the **View » Ascend** command. When you select a link pin, the **View Ascend** command is enabled. Choose **View » Ascend** and the *Hierarchy Navigator* dialog appears:



Highlight a module reference designator and click **OK**. The selected module becomes the worksheet's focus.

A shortcut command is available for View Ascend.

## View Command Toolbar

---

Choose the **View » Command Toolbar** command to 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 used editing commands (such as the **Select** tool).



Disabling the command increases the space within the active window.

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

## View Placement Toolbar

---

Choose the **View » Placement Toolbar** command to show or hide the *Placement* toolbar. The toolbar gives you quick access to the most frequently used Place commands.



Disabling the command increases the space within the applicable window.

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

## View Custom Toolbar

---

Choose **View » Custom Toolbar** to either show or hide the custom toolbar. A check mark next to this menu command indicates that the *Custom* toolbar is visible.

If no tools have been added to the custom toolbar, the display of the toolbar is turned off. As soon as the first custom tool has been added using the **Utils » Customize** command, the *Custom* toolbar display is turned on. When custom tools exist in the *Custom* toolbar, you can turn off the display of the toolbar by choosing **View » Custom Toolbar**.

## View Prompt Line

---

Choose **View » Prompt Line** to either show or hide the prompt line. The prompt line gives instructions on what to do in certain modes.

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

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

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

## View Status Line

---

Choose **View » Status Line** to either show or hide the P-CAD Schematic status line. The status line provides status information and allows you to change sheets and execute temporary macros.



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 `Sch.ini` file when you exit the program, and restored when you restart it.

## View Snap to Grid

---

Choose the **View » Snap to Grid** command to turn on and off a snappy cursor. When a check mark precedes the command, a snappy cursor moves from grid point to grid point. When the command is not checked, a free floating cursor moves freely between grid points.

A snappy cursor can create a predictable point of reference when you move and rotate objects or measure distances. A free floating cursor can enhance your ability to select items in the workspace, such as when you want to select a line that does not run true along grid points.

The **View Snap to Grid** command does not support the DataTips feature. When **View Snap to Grid** is a checked command, the DataTips feature is temporarily disabled.

Your current **View Snap to Grid** setting (on or off) is saved in the `Sch.ini` file when you quit the program.



## Place Commands

### Using the Place Commands

---

Use the commands in the **Place** menu to place various items and objects in a design. After you choose a command, click the workspace to place the selected item. With some commands, a dialog box appears so you can select specific properties for the item you are about to place.

Before an item is placed in a design, it appears in a highlighted color to distinguish it from placed items. Once placed, they appear in their normal color. To modify an object after it has been placed in a design, select the object and choose **Edit » Properties**.

### Place Part

---

Choose **Place » Part** to place a part in your active design. With this command, you can select the type of part to place, as long as the part and its corresponding symbol has been created and assigned to an open library.

Parts of the same type in different libraries can be placed only if they have the same pin mapping (i.e., pin designator to pin name to pad number). In other words, if you place the “same” part from different libraries, the first instance of the part type establishes the standard pin mapping for that type of part. Any parts of that type placed subsequently have to conform to the pin order of the first or they cannot be placed.

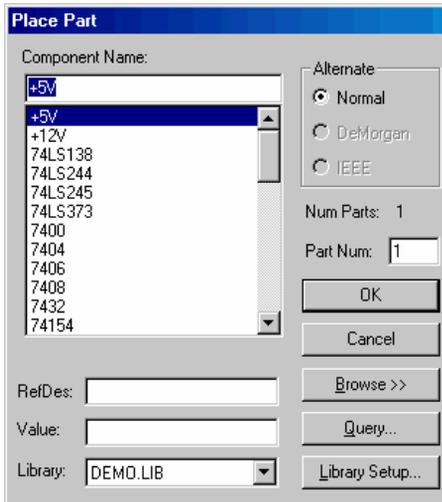
When you place parts that include text with styles that have the same names but different definitions than those in the current design, the new style names are bracketed to indicate the style conflict. The bracketed style names are added to the list of available styles in the current design. For additional information, see *Edit Properties* (page 211) and *Options Text Style* (page 314) and the *P-CAD Library Executive User's Guide*.

### Placing a Part

To place a part, follow these steps:

1. Choose **Place » Part** or click the **Place Part** button in the *Placement* toolbar.

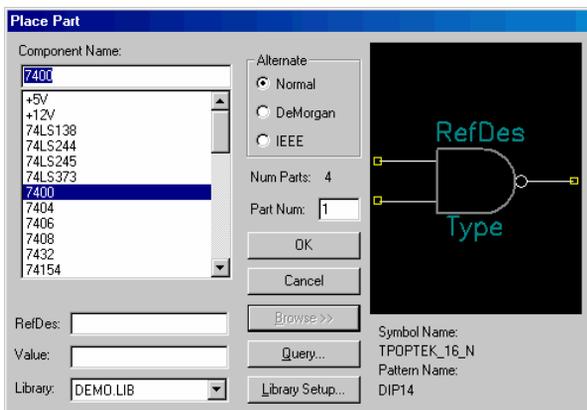
- Click the workspace. The following *Place Part* dialog appears.



- Select a part from the Component Name list. The list shows all of the components in the library currently selected in the Library list. To view the contents of another library, select a library from this list.

To add a library to this list, click **Library Setup**. For more information, see *Library Setup* (page 325).

- Click **Browse** to expand the dialog and view a graphical representation of the part, as shown in the following figure:



5. In the Part Num box, select which gate in the highlighted component will be placed first. To do this, type the part number in the box. Notice that the Num Parts field shows the number of parts associated with the selected component.
6. In the Alternate frame, choose one of the following radio buttons to select a graphical representation: **Normal**, **IEEE**, or **DeMorgan**.

In addition to the typical commercial representation of components, P-CAD Schematic includes IEEE and DeMorgan representations of many parts. The IEEE alternate representations are based on the component specifications from *IEEE/ANSI Standard 91-1984*.

7. Specify a reference designator in the RefDes box.

If you leave this field blank, the program uses the default RefDes prefix assigned to the part at part creation. If you do not assign a number, the program automatically assigns the next available number.

8. If the component being placed has a value, it is displayed in the Value box. You may change the value in the Value box, when necessary (e.g., electrical values for resistors and capacitors; typically blank for logic parts). The new value overrides the value taken from the library component.

When a Value attribute exists in the component, its value is displayed in the *Value* box based on the following rules:

- If the symbol has a Value attribute and there is no component Value attribute, the symbol's Value attribute value is used.
- If the component has a Value attribute, the component's Value attribute value will take precedence and be used even if the symbol has a Value attribute value.

9. Click **OK** when the desired information is displayed in the Value box. The *Place Part* dialog closes. You are now in part placement mode.
10. Choose one of the following methods to place the part:
  - Move the cursor to the location in the design where you want to place the part. Click to place it.
  - Press and hold the left mouse button down to display a ghosted outline of the part. Drag the ghosted outline to the desired location and release the left mouse button to place the part.
  - Hold down the **ALT** key and click the left mouse button. Release the **ALT** key and move the part with the mouse. You do not have to hold down the left mouse button. Then, click to place the part. To cancel placement of the part, right-click.

To rotate or flip a part, press **R** or **F** while holding down the left mouse button. For details, see *Rotating Objects During Placement* (page 86).

11. Continue to place the same part by clicking the desired locations in the workspace. Notice that each part is given a unique reference designator.

To change the displayed part number to the next available value, press **P**; to change to the previous available value, press **SHIFT+P**. To select the next available reference designator, press **D**. To select the previous available reference designator, press **SHIFT+D**.

12. If you want to place a different part, right-click or press **ESC** to stop placing the selected part. As you are still in part placement mode, you can click the workspace to open the *Place Part* dialog and select another part to place (as in step 2).

To gain access to the embedded Query utility, click the **Query** button. For details on this feature, see the *Library Executive User's Guide*.

## Power Components

Power Components are used to rename nets. For example, connecting a GND power component to a net will force that net to have a net name of GND. Although they have reference designators, power components do not appear in any netlist you generate. Create a power component by selecting the **Power** radio button in the *Save Component As* dialog; the pin name is the name of the net that will be created. See *Library Symbol Save As* (page 326) for more information.

If you place a power component that has a pin name such as GND or VCC when the design contains a non-global net with the same name, a warning message appears. The message informs you that a port has been added to the net and shows the sheet number where the port was placed.

## Splitting a Net

This section explains how the **Place Part** command handles the resulting net splitting when a two-pin part is placed, copied, or moved on top of a single wire segment. The portion of the wire between the two pins is deleted so that the wires connect to the pin endpoints.

### Net Naming for Delete and Insert Part

Splitting a net, by inserting a part over a wire segment (or deleting a wire segment) results in retaining the original net name on one side, and renaming the other to a new system-assigned net name.

If a port or power symbol is attached to a net created by the splitting process, that portion of the net retains the original name. If both portions are attached to a port or power symbol, then both keep the original net name.

## Jumper Pins

Components with jumper pins are handled in a special way. Any time this command adds a pin to a net, P-CAD Schematic checks to see if the pin is a jumper pin. If it is, the component behaves as if all of the pins marked as being jumpered together are connected. Jumper pins are ported to force them to belong to the appropriate net.

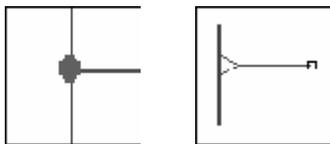
## Place Wire

Choose **Place » Wire** to place a wire or series of wire segments on the current sheet.

When you place a wire, the wire is the width specified in the *Options Current Wire* dialog. To change the current wire width, choose **Options » Current Wire**.

Wires can start or end on a pin, on a bus, in an open space or on another wire. If you place a wire so that it ends on a wire or pin, the wire is automatically connected. Unconnected wires display open ends if the **Display Open Ends** check box is selected in the **Miscellaneous** tab of the *Options Display* dialog. To indicate that the wire is unconnected, open ends take the shape of small boxes at the end of a wire. Starting or ending a wire on an existing wire automatically creates a junction. Starting or ending a wire on a bus creates a bus entry.

The illustration on the left shows three wire segments. Where they intersect is a junction. The illustration on the right shows a bus connection where the wire is connected to the bus, and an open end where the wire is unconnected.



1. Choose **Place » Wire** or click the **Place Wire** toolbar button.

While you draw a wire or wire segments, the cursor is displayed as a crosshair shape to indicate that wire placement is in progress.

2. With the cursor in the active window, click and hold the button at the starting point, then drag the wire to its second point and release to place the wire. You can continue with connected segments in the same manner.

You can press **ALT+left click** instead so you don't have to hold the button down while dragging wire segments.

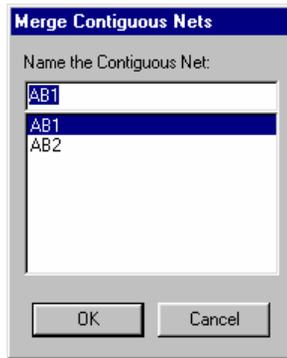
You can unwind any wire segments by pressing the **BACKSPACE** key or backtracking the mouse over the wire segments. If you have finished the wire segments, press **ESC** or the right mouse button, choose **Edit » Undo**, or click the toolbar button to undo the whole series of finished segments.

3. To finish the wire or wire segments, right-click or press **ESC**. Then you can begin another wire beginning at a new location.
4. If you are creating a new net, the Net Name is automatically assigned. Each successive net is incremented by the increment value you set up in the *Options Configure* dialog.

You remain in wire placement mode until you choose another command.

## Adding Wires to Nets

When you connect a new wire to an existing net, it automatically places a junction. If the existing wire belongs to a different net, the following dialog appears:



With the *Merge Contiguous Nets* dialog, you choose the net name after the nets are merged. The Name the Contiguous Net combo box shows the net names of the existing nets. Select a net name from the list or type a new name and click **OK**.

## Jumper Pins

Any time a wire adds a pin to a net, P-CAD Schematic checks to see if the pin is a jumper pin. If it is, the component behaves as if all of the pins that have been as being jumpered together are connected. Jumper pins are ported to force them to belong to the appropriate net.

## Orthogonal Modes

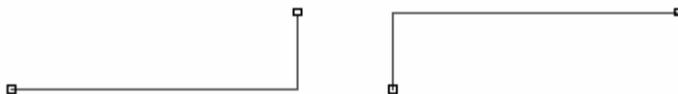
There are orthogonal modes that you can use while placing wires. While you are in wire placement mode and dragging a wire segment, you can press the **O** key while holding down the left mouse button to switch between the enabled orthogonal modes.

Orthogonal modes use lines that are horizontal, vertical, and at 45-degree angles. Press the **O** key to switch between the enabled modes. You can enable or disable the orthogonal modes in the *Options Configure* dialog. The non-orthogonal mode, with straight line placement at any angle, is always enabled.

The 90/90 and 45/90 orthogonal modes are provided as mode pairs. Press the **F** key to switch between the current mode pair.

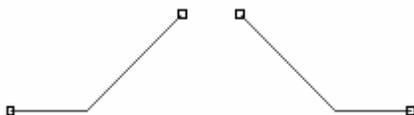
### 90/90 Line-Line

Wires are placed horizontally and vertically in 90/90 Line-Line mode. In the first mode, the first wire segment is always longer than the second. In the second mode, the first segment is shorter. You can switch between the two by pressing the **F** key.



### 45/90 Line-Line

With 45/90 Line-Line mode, you can press the **F** key to switch between the two modes. The first mode makes the first segment displayed at a 45-degree angle and the second segment is either horizontal or vertical. The second mode makes the first segment either horizontal or vertical and the second segment is displayed at a 45-degree angle.



## Place Bus

---

Choose the **Place Bus** command to place a bus (or series of bus segments) on the current sheet. Buses are used to show multiple parallel wires on the schematic, such as address and data lines. Buses graphically represent a bundle of one or more wires.

To place a bus, follow these steps:

1. Choose **Place » Bus** or click the **Place Bus** toolbar button.

While you draw a bus or segments, the cursor is displayed as a crosshair shape to indicate that bus placement is in progress. When you finish the bus segments, the cursor returns to its normal shape.

2. With the cursor in the active window, click and hold the left mouse button at the starting point, then drag the bus to its second point and release to place the bus. You can continue with connected segments in the same manner.

You can press **ALT+click** instead so you don't have to hold the button down while dragging bus segments.

You can unwind any previous bus segments by pressing the **BACKSPACE** key or backtracking the mouse over the segments. If you have finished the bus by pressing **ESC** or right-clicking, choose **Edit » Undo** or click the toolbar button to undo the whole series of finished segments.

- To finish the bus or bus segments, right-click or press **ESC**. Then you can begin another bus beginning at a new location.

You remain in bus placement mode until you choose another command.

## Status Line Measurements



The status line information display area (on the right side of the status line) displays bus measurements for delta X and delta Y while you are dragging a bus segment. When the bus segment is finished, the total length measurement of the segment(s) is displayed.

When the bus is finally placed, all measurements on the status line disappear.

## Orthogonal Modes

The orthogonal modes that you can use while placing buses work in the same way as they do for wires. While you are in bus placement mode and dragging a bus segment, you can hold down the left mouse button and press the **O** key to switch between the enabled orthogonal modes.

Orthogonal modes use lines that are horizontal, vertical, and at 45-degree angles. Press the **O** key to switch between the enabled modes. You can enable or disable the orthogonal modes in the *Options Configure* dialog. The non-orthogonal mode, with straight line placement at any angle, is always enabled.

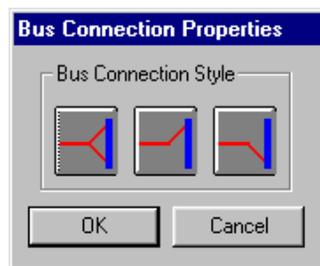
The 90/90 and 45/90 orthogonal modes are provided as mode pairs. Press the **F** key to switch between the current mode pair. For illustrations, see *Place Wire* (page 263).

## Bus Connections

Bus connections are formed automatically when you connect a wire to a bus. They let you know that the connection has been made.

The bus connection you place defaults to the type of connection you using the *Bus Connection Mode* field in the **Colors** tab of the *Options Display* dialog.

To change a connection, select it, then right-click and choose **Properties** from the shortcut menu. Press the **B** key to switch between bus connection styles while placing wires.



Select one of the three styles for the required connection.

## Place Port

Choose the **Place » Port** command to place ports in your design. Ports indicate connectivity between non-contiguous portions, or subnets, of the same net. In other words, ports provide a visible means to connect contiguous portions of a net in different locations of the design without having to wire them together directly. Ported nets may be on the same or different sheets.

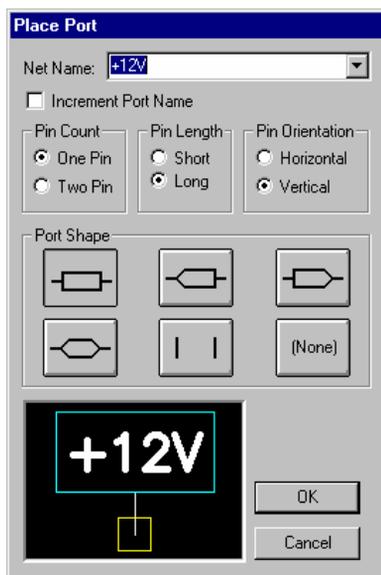
### Display Characteristics

Ports consist of the following items: the net name, an optional box surrounding it, and one or two pins to connect it to wires. Sixteen different styles of ports are provided.

### Placing Ports

To place ports, follow these steps:

1. Choose the **Place » Port** command or click the **Place Port** toolbar button.
2. Click the workspace to open the *Place Port* dialog. If you click over a wire, the Net Name box is preset with the net name. Otherwise, it is preset with the last name you used.



3. To increment net names when placing multiple ports, select the **Increment Port Name** check box. Net names are incremented only when the name ends in a number.
4. Type the net name in the Net Name box.
5. In the Pin Count frame, choose the **One Pin** or **Two Pin** radio button.
6. In the Pin Length frame, choose the **Short** (5 mils) or **Long** (50 mils) radio button.

7. In the Pin Orientation frame, choose the **Horizontal** or **Vertical** radio button.
8. Click **OK** to close the *Place Port* dialog. You are now in port placement mode.
9. In your workspace, move the cursor to the location where you want to place the port. Click the workspace to place it.  
  
If you elected to increment ports, subsequent mouse clicks place ports with incremented net names, otherwise each click places a port with the same name.
10. To rotate the port, press **R** while holding down the left mouse button. To change the port display type, press **P** while holding down the mouse button.  
  
If you are placing a port anywhere on a non-global net, an error message appears. The message informs you that a net can only be renamed to an existing net name if both nets are global, and asks if you want to autoplace ports to make both nets global. If you click the **Yes** button, the ports are placed, and nets are merged and renamed.
11. Right-click to end the place port session.

## Hotkeys

While placing a port, you can change the orientation of the port using a hot key. Press **R** to rotate the port or **F** to flip it. You can also switch between the following port styles by pressing the **P** key:

- End-wire
- Inline
- Off-wire-side
- Off-wire-top
- Off-wire-bottom

## Placing an Inline Port

If you place an Inline port over a wire, the occluded wire is automatically deleted. This is especially useful for labeling bus entries. If a port is not placed at a wire end or inline, a junction will automatically be created.

## Jumper Pins

Any time placing a port adds a node to a net, P-CAD Schematic checks to see if the node is a jumper pin. If it is, the component behaves as if all of the pins marked as being jumpered together are connected. In Schematic, pins are ported to force them to belong to the appropriate net.

## Moving Ports

Ports may be moved like components. If a port is moved any attached wires will be rubber banded to the new location.

## Renaming Ports

The net to which a port is attached may be renamed as follows:

- Selecting the port and clicking the right mouse button to modify.
- Using the *Edit Nets* dialog.
- Using the *Net Rename* tool.
- Selecting and modifying the net name of a wire to which the port is attached.

## Modifying a Port's Text Style

The font and text size of all the ports can be changed by entering the *Options Text Style* dialog, and modifying the [Wirestyle].

## Deleting Ports

Ports may be selected and deleted, like any other object, however, when an inline port is deleted, a wire is automatically drawn to maintain connectivity. If you delete a port where opposing pins are connected to wires of different widths, the resulting contiguous wire will be of uniform width.

## Backward Compatibility

The following rules apply when loading designs and block files from P-CAD versions earlier than 2.60:

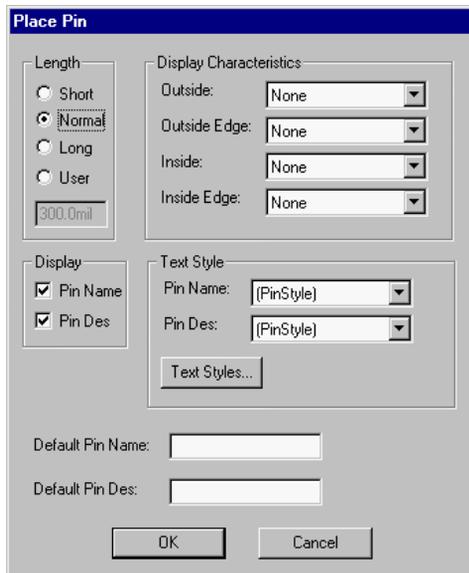
- If a net is comprised of a single contiguous net, then the net is a local net.
- If a net is comprised of two or more subnets, then the net is a global net and will have a port attached to each non-contiguous subnet.
- If a net has a wire attached to a power symbol or is part of a hidden power net, the net is considered global.
- If a wire has a power symbol attached to it, it does not require a port to be attached; bus entries do.
- If a block is pasted into the design, the rules for porting apply as well. Nets of existing net names will retain their net names if they were ported; if they were not ported they are given unique system-assigned net names.

## Place Pin

---

Choose the **Place Pin** command to place a free pin in the workspace. To place a pin, follow these steps:

1. Choose **Place » Pin** or click the **Place Pin** toolbar button.
2. Click the active window to open the *Place Pin* dialog shown in the following figure:



3. Select the pin length from the options in the Length box. In the Display frame, select the **Pin Name** or **Pin Des** check box to choose the visibility of the pin name and pin designator.
4. In the Text Style frame, set the text style for the pin name and pin designator. Click the **Text Styles** button to open the *Options Text Style* dialog. From there you can add or modify existing text styles for your design.
5. Type the default pin name, if desired, in the Default Pin Name box. The default pin name is a placeholder for the real pin name. Use this default label to change the orientation or position of the pin name. The default pin name, like a pin number, cannot be edited once attached to a symbol.
6. Enter the default pin designator, if desired, in the Default Pin Designator box. The default pin designator can be edited once attached to a symbol.
7. In the Display Characteristics frame, choose the desired characteristics by selecting or typing information in the four combo boxes. Click **OK**.
8. You are now in the place mode of Place Pin. Move the cursor to the location in the active window where you want to place the pin. Click the left mouse button to place it. Or you can hold the left mouse button down to make a ghost, then drag and drop (release) to place it more accurately. (An alternate method for drag-and-drop is **ALT+click**, then release the **ALT** key. You can then move the pin freely with the mouse without having to keep the button depressed.) To cancel ghosting of a pin, right-click.

You can rotate a pin while placing it by pressing **R** while holding down the left mouse button. A pin can also be flipped during placement using the **F** key while holding down the left mouse button.

You can continue to place similar pins by clicking in additional locations. Pin attributes previously specified (in the *Place Pin* dialog) are also displayed.

- To place a different pin, right-click or press **ESC** to exit the place mode for that particular pin. As you are still in pin placement mode, you can click the active window to open the dialog and select another pin to place.

Once placed, pins can be renumbered. For more information, see *Utils Renumber* (page 329).

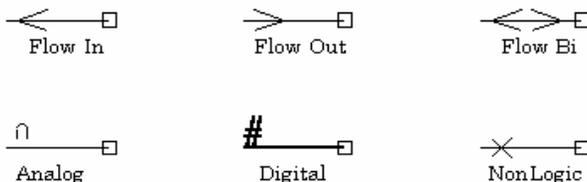
A part's sub-selected pin can be connected to a specified net using the **Add to Net** command in the shortcut menu.

Pin names and pin designators can be sub-selected for moving, flipping and rotating.

### Display Characteristics

Display characteristics include all attribute symbols that may be attached to a pin for design clarification. They are for graphical appearance only. You may specify display characteristics for each item shown in the following illustrations:

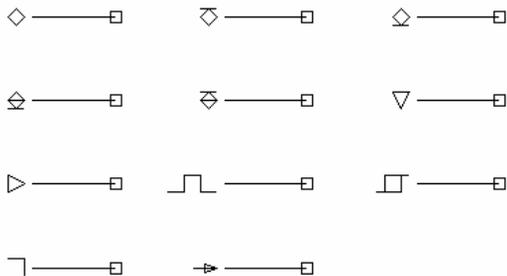
#### Outside



#### Outside Edge



#### Inside



**Inside Edge****Place Line**

Choose the **Place » Line** command to place a series of line segments of the current line width on the current sheet. In P-CAD Schematic, lines are graphical representations only; they carry no net information.

To change the current line width, choose the **Options » Current Line** command.

**Placing a Line**

To place a line on the workspace, follow these steps:

1. Choose the **Place » Line** command or click the **Place Line** toolbar button.
2. With the cursor in the workspace, click and hold the left mouse button at the starting point, then drag the line to its second point and release to place the line. You can continue with connected segments in the same manner.

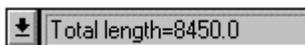
You can press **ALT+click** instead so you don't have to hold the mouse button down while dragging line segments.

While drawing a line or segments, the cursor is displayed as a crosshair shape. When you finish the line or segments, the cursor returns to its normal shape.

You can unwind any line segments by pressing the **BACKSPACE** key or backtracking the mouse over the segments. If you have finished the line segments by pressing **ESC** or right-clicking, choose **Edit » Undo** or click the toolbar button to undo the whole series of finished segments.

3. To finish the line or line segments, right-click or press **ESC**. Then you can begin another line beginning at a new location.

You remain in line placement mode until you select another tool or command.

**Status Line Measurements**

The status line information display area (on the right side of the status line) displays line measurements for delta X and delta Y while you are dragging a line segment. When the line segment is finished, the total length measurement of the segment(s) appears.

When the line is finally placed, all measurements on the status line disappear.

## Orthogonal Modes

The orthogonal modes used while placing lines work in the same way as for wires. While you are in place line mode and dragging a line segment, you can press the **O** key while holding down the left mouse button to switch between the enabled orthogonal modes.

Orthogonal modes use lines that are horizontal, vertical, and at 45-degree angles. Press the **O** key to switch between the enabled modes. You can enable or disable the orthogonal modes in the *Options Configure* dialog. The non-orthogonal mode, with straight line placement at any angle, is always enabled.

The 90/90 and 45/90 orthogonal modes are provided as mode pairs. Press the **F** key to switch between the current mode pair. For illustrations, see *Place Wire* (page 263).

### 90/90 Line-Line

Both lines are either horizontal or vertical (displayed perpendicular to each other). For long, the first segment is always longer than the second. For short, the first segment is shorter. You can switch between the two by pressing the **F** key.

### 45/90 Line-Line

You can switch between the two modes by pressing the **F** key. The first mode makes the first segment displayed at a 45-degree angle and the second segment is either horizontal or vertical. The second mode makes the first segment either horizontal or vertical and the second segment is displayed at a 45-degree angle.

## Place Arc

---

Choose the **Place » Arc** command to place an arc or circle of the current line width on the current sheet. With this command, you can create arcs of varying length and radius and circles of varying radius. In P-CAD Schematic, arcs are graphical representations only; they carry no net information.

To change the width of an existing arc, choose the **Edit » Properties** command. To change the arc width for new arcs, choose the **Options » Current Line** command.

Arcs are partial circles. Arcs and circles are constructed counter-clockwise; the click (down) and release (up) define the start and end point of the arc, therefore a stationary click/release defines a circle. In this case, the second click and drag moves the center point away from the defined point on the circumference.

## Placing an Arc

To place an arc on the workspace, follow these steps:

1. Choose **Place » Arc** or click the **Place Arc** toolbar button.
2. Move the cursor into the active window to where you want the starting point of the arc.

While you draw an arc, the cursor is displayed as a crosshair shape. When you finish the arc, the cursor returns to its normal shape.

3. Click and drag to where you want the end point of the arc to be. Release and the start and end points are established as a 180-degree arc.

The arc sweeps counterclockwise as you place it. (e.g., left-to-right the arc sweeps up; right-to-left the arc sweeps down, etc.) If the start and end points are the same (i.e., you click and release without dragging), you create a circle.

4. After the start and end points are established, click and drag the cursor to define/alter the center point, thereby increasing or decreasing the sweep angle and radius of the arc.
5. You can flip the arc (swapping the end points) by pressing **F** while the arc is still unfinished. When you release the mouse button, the arc is permanently placed.
6. To cancel ghosting of an arc, right-click.

## Changing Arcs

You use select mode to move, resize, rotate, flip or perform other types of changes to the arc (after it is placed):

- To rotate or flip an arc, select it, and press **R** to rotate or **F** to flip while the button is depressed.
- To move the arc, click within the selection box (not on the handles) and drag the arc.
- To resize, click and move the center handle to change the radius or move the start or end point to change the sweep angle.

## Changing an Arc Center Point

While placing an arc, you can alter the center point before the final placement. After you click, drag, and release, the arc start and stop points are defined. A second click defines the center point and the radius. If you click and drag, you can alter the center point of the arc before you release.

- To toggle between the arc's beginning and end points, press the **F** key. For more information, see *Place Arc* (page 273).

## Place Polygon

---

Choose the **Place Polygon** command to place a solid filled polygon on the current sheet. In P-CAD Schematic, polygons are graphical representations only; they carry no net information.

1. Choose **Place » Polygon** or click the **Place Polygon** toolbar button.

While you draw a polygon, the cursor appears as a crosshair. When you finish the polygon, the cursor returns to its normal shape.

2. Put the cursor at the starting point of where you want your polygon. Click and drag to the second point and release.
3. Click and drag to a third point and you have a triangle. Any subsequent polygon points are connected by a line to the first point you selected, e.g., the fourth point of a polygon is connected to the first point automatically.

4. When you have established all points for the polygon, right-click or press **ESC** to finish and fill the polygon.

Complex polygons are not allowed. A complex polygon is a self-intersecting or self-crossing polygon. Polygon sides can touch each other but not cross each other.

## Draft/Outline Display Mode

Polygons can be drawn and printed as outlines. They are drawn in outline form when the **Draft Mode** check box is selected in the **Miscellaneous** tab of the *Options Display* dialog. To switch between draft mode on and off, press the **Q** key.

## Rotate/Flip

To rotate a polygon, select it and press **R** to rotate the polygon by 90 degrees. To flip a polygon, select it and press the **F** key.

## Altering the Shape

You can alter the shape (move, add, or delete vertices) by selecting the polygon, then clicking and dragging one of the handles. When you move one of the handles that lies between two vertices, a new vertex is created. To delete a vertex, grab and move it to an adjacent vertex and release.

## Place Ref Point

---

Choose the **Place » Ref Point** command to place a reference point on the active sheet. When you place a reference point, it appears in the 1x grid color set in the **Colors** tab of the *Options Display* dialog. You can also set reference point size in the **Miscellaneous** tab of *Options Display*. For more information, see *Options Display* (page 297).

When you place a reference point on a symbol, the symbol moves with the cursor at the reference point. Symbols are flipped and rotated about the reference point. If you are creating a symbol, you should place a reference point on the symbol object before saving it in a symbol library using the **Library » Symbol Save As** command explained in *Library Symbol Save As* (page 326).

## Placing a Reference Point

1. Select **Place » Ref Point** or click the **Place Ref Point** toolbar button.
2. Move the cursor to the location on an object or with a collection of objects. Click to place the reference point; or click, drag and release to place the
3. You can move a reference point by selecting it and dragging it to a new location.

To cancel placement of a reference point, right-click.

## Place Text

---

Choose the **Place Text** command to place text on your design using the following basic steps:

1. Choose the **Place » Text** command or click the **Text Placement** toolbar button.
2. Click the workspace at the location where the text is to be placed.
3. Type the text you want to place in the design in the Text area of the *Place Text* dialog.
4. Click the **Place** button in the dialog or click the **Place Text** tool again to commit the text in the design.

## Place Text Dialog

Text entered in the *Place Text* dialog is instantly displayed at the location in the design where you clicked the mouse button to begin the place text function. You can change the location of the text by dragging it to a new location.

The *Place Text* dialog can be displayed in the expanded version, an intermediate size or minimized to a one-line display. The display size and location at the time the dialog is closed become the default settings that are used the next time you enter the *Place Text* mode.

To open the expanded version of the *Place Text* dialog, click the **More>>** button located directly beneath the scroll bar, on the right side of the Text edit box. To reduce the dialog to the intermediate size, click the **<<Less** button, located where the **More>>** button used to be. To reinstate the minimized dialog to its previous size, click the **Maximize** button next to the **Minimize** button at the top right corner of the dialog. To reduce the dialog to its minimized size, click the **Minimize** button at the top right corner of the dialog.

The minimized version of the dialog appears as shown below:



The intermediate version of the dialog appears as follows:



The expanded version of the dialog appears as follows:



The *Place Text* dialog contains the following options:

- **Text:** The Text edit box is where you type the text you want to place in the design. You can also paste text from the clipboard into the edit box, using **CTRL+V**. If you want to enter multiple lines (e.g., a list), you can start new lines in the box by using a carriage return (the **ENTER** key). A single text object can have up to 2,000 characters.
- **Location:** If the text has not been placed yet, the Location section displays the X and Y coordinates of the location of the cursor when the mouse button was released. While placing the text the coordinates cannot be modified. When you select committed text and choose the **Edit » Properties** command, the location coordinates can be modified to move the text to a different area in the design.
- **Justification:** In the Justification box you can specify the text reference point. The reference point defines the cursor location on text placement, and also the point on which text is rotated or flipped. The default reference point for text is the lower-left corner of the text block. There are nine different reference point options.
- **Flip:** The *Place Text* dialog provides the option to flip the text by checking the **Flip** box.
- **Rotate:** To rotate the text by 90 degrees, click the **Rotate** button. Each time you click the **Rotate** button the text rotates another 90 degrees. The degree of rotation is displayed next to the **Rotate** button.
- **Text Styles:** You can view the default text styles and add, modify and delete a non-default text style by clicking the **Text Styles** button to open the *Options Text Style* dialog. Changes to the text style appear instantly in the design at the time they are made.

The text style for the selected text or the text being placed can be changed by choosing the new style from the drop down list in the Text Style edit box. Fonts and font sizes cannot be mixed within the same text item. Barred text can be inserted in the text item by placing a tilde (~) on either side of the text you want to bar. Type two consecutive tildes to cause a single tilde to be displayed. Changes that cause the text to be placed outside the workspace are not allowed.

See *Options Text Style* (page 314) for detailed information on adding, modifying, viewing and deleting text styles.

- **Place/Cancel:** Commit the text to the current location in the workspace by clicking the **Place** button or the **Place Text** button on the *Placement* toolbar.

To cancel the text placement dialog and remove the temporary text from the design, click the **Cancel** button or press the **ESC** key. The **Text** tool is still active, but the dialog disappears from view.

Whenever you are typing, flipping or rotating text, and zooming or panning the workspace, the dialog remains on the screen. At the time the text is committed to the design by clicking the **Place** button on the dialog, choosing the **Place » Text** command or clicking the **Place Text** button on the *Placement* toolbar, the dialog disappears from the screen. The **Place Text** tool remains active so that you can click the workspace to indicate the location for the next text item, display the dialog again and place more text.

## Place Text Features

The **Place Text** tool provides additional features that enhance the basic placement functionality and make placing the text easier and more accurate. This section describes how to rotate, flip, zoom and pan text during or after placement.

### Rotating Text

You can rotate text during placement. Click and hold the left mouse button on the text, then press **R** or click the **Rotate** button. This rotates the text by 90 degrees. The text moves about its reference point, but always reads from left to right.

To rotate text after it has been placed, select it and press **R** to rotate while the left mouse button is depressed or choose the **Edit » Properties** command to open the *Text Properties* dialog where you can click the **Rotate** button.

Attributes, text, and fields share the same rotation memory. In other words, if you place and rotate text at 90 degrees, then immediately place a field, the field's text will be placed at a 90-degree angle.

Click again in the box in the *Place Text* dialog to continue typing new text.

### Flipping Text

You can flip text as you are placing. If you change focus from the dialog to the workspace, click and hold the left mouse button on the box, and then press the **F** key. While in the dialog you can flip the text by checking the **Flip** box.

The **F** key and **Flip** check box functions flip the text 180 degrees to the right or left of the original location. The text moves about its reference point, but always reads from left to right.

To flip text after it has been placed, select it and then press **F** to flip, or choose the **Edit » Properties** command to open the *Text Properties* dialog and select the **Flip** check box.

Click again in the box in the *Place Text* dialog to continue typing new text.

### Zooming and Panning While Placing Text

The *Place Text* dialog remains on the screen while you zoom or pan the workspace.

Click the workspace to assure that the focus is in the workspace before choosing any of the zoom commands. To zoom in or out of the workspace while placing text press the **Plus (+)** and **Minus (-)**

keys, the **View » Zoom In** and **View » Zoom Out** commands or click the **Zoom Window** button on the *Command* toolbar.

The workspace can be panned by moving the scroll bars up, down, right and left, or by pressing one of the arrow keys.

Click again in the box in the *Place Text* dialog to continue typing new text.

## Place Attribute

Choose the **Place » Attribute** command to place an attribute according to the **Name** and **Value** options you select in the dialog. This command allows you to place an attribute within a collection of objects comprising a symbol or part.

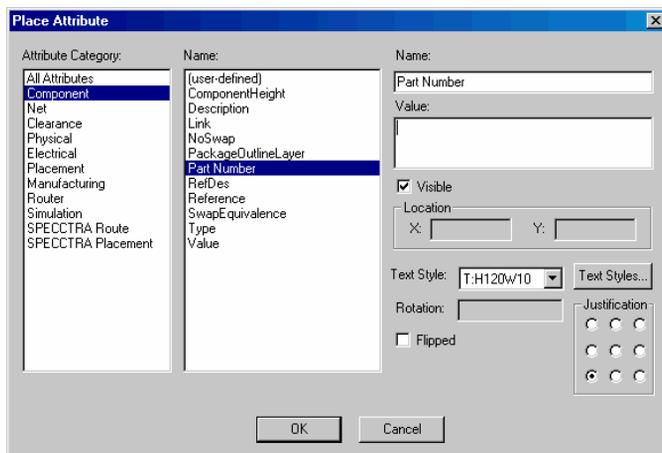
The predefined Names are: Filename, Name, Part Number, RefDes, Type and Value.

The Value box specifies the attribute definition, e.g., the actual filename rather than just the placeholder {Filename}.

### Placing an Attribute

To place an attribute, follow these steps

1. Choose **Place » Attribute** or click the **Place Attribute** toolbar button.
2. Click the workspace to open the dialog:



3. Choose an attribute category from the Attribute Category list.
4. All pre-defined attributes for the category appear in the Name list. To place a predefined attribute, select an attribute from the Name list.
5. To place a user-defined attribute, select user-defined from the Name list. Type an attribute name in the Name edit box.

6. Type a value for the attribute in the Value edit box.
7. Set attribute properties. See *Attribute Properties* (page 228) for more information.
8. Click **OK**.
9. Move the cursor into the workspace and click, drag and place the attribute. Before you release the mouse button to place it, you can move, rotate, or flip the placement box (see the following section).

## Rotate/Flip

To rotate or flip an attribute after it has been placed, select it and press **R** to rotate or **F** to flip while it is selected.

The **R** key rotation is 90 degrees.

You can rotate or flip an attribute as you are placing it. For rotation, the angle that is the result of the rotation applies to the next attribute you place. For example, you are placing an attribute, and you rotate it 90 degrees before you finish it. Then you place another attribute; it is placed at the same 90-degree angle without any rotation action. You can place multiple attributes at the same angle by rotating the first attribute as you place it.

Attributes, text, and fields share the same rotation memory. In other words, if you place and rotate text at 90 degrees, then immediately place an attribute, that attribute will be placed at a 90-degree angle. This rotation memory derives only from rotation action during placement, not from select and rotate actions that take place after object placement.

If you don't specify a value for an attribute, the attribute key name will be displayed in brackets, e.g., {Type}. See *File Design Info* (page 183) for information on design-level attributes.

## Place Field

---

Choose the **Place » Field** command to place a field containing design information such as date, time, author, etc.

The information that appears when you place certain fields is determined by what you specify in the **Field** tab of the *File Design Info* dialog. You can place a field from a selection of predefined field types including: Approved By, Author, Checked By, Company Name, Current Date, Current Time, Date, Drawing Number, Drawn By, Engineer, Filename, Modified Date, Note, Number of Sheets, Revision, Revision Note, Sheet Name, Sheet Number, Time, and Title.

You can also define and place additional fields using **File » Design Info**. For instructions on adding a user-defined field, see *File Commands* (page 169).

Current date and current time are taken from the computer's clock. If you just select date then you must specify the date in the *File Design Info* dialog. The same is true for time (as opposed to current time).

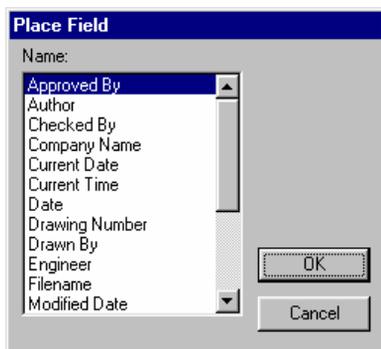
The value of a field placed in the design, with the exception of Current Date/Time, Filename and Modified Date, must be specified in the *Field Properties* dialog, otherwise you place a generic field, e.g., {Author} rather than "W. Shakespeare". To open the *Field Properties* dialog, choose the **File »**

**Design Info** command and click the **Properties** button in the **Fields** tab. Or, select a field in your design, right-click, then choose **Properties** from the shortcut menu.

Fields are handy for use within title blocks.

## Placing a Field

1. Choose **Place » Field** and click the workspace. A dialog appears with a list box containing field types.



2. From the Name list box, select what kind of field you want to place. Click **OK**.
3. Move the cursor to where you want to place the field; click to place it. If you click again, the *Place Field* dialog will be displayed so that you can choose another type of field for placement.
4. You can rotate or flip a field as you are placing it. See the following section for more details.

To cancel before a field is placed, right-click.

## Rotate/Flip

To rotate a field, select it and press **R** to rotate the field by 90 degrees. To flip a field, select it and press **F**.

You can rotate or flip a field as you are placing it. For rotation, the angle that is the result of the rotation will apply to the next field you place. For example, you are placing a field and you rotate it 90 degrees before you finish it. When you place another field; it is placed at the same 90-degree angle without any rotation action. If you decide to rotate the second field, it is incremented 90 degrees more, resulting in a 180-degree angle. Therefore, you can place multiple fields at the same angle but only have to perform the rotation action on the first field.

Attributes, text, and fields share the same rotation memory. In other words, if you place and rotate text at 90 degrees, then immediately place a field, that attribute is placed at a 90-degree angle. This rotation memory derives only from rotation action during placement, not from select and rotate actions that take place after object placement.

## Place IEEE Symbol

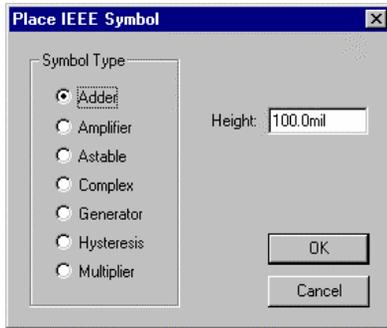
---

Choose the **Place » IEEE Symbol** command to place an IEEE symbol in the design. Once placed, an IEEE symbol cannot be modified. It must be deleted and replaced with a new symbol. You can place the following symbols:

	Adder
	Amplifier
	Astable
	Complex
	Generator
	Hysteresis
	Multiplier

### Placing a IEEE Symbol

1. Choose **Place » IEEE Symbol** or click the corresponding toolbar button.
2. Click the active window. A dialog appears listing the IEEE symbol types.



3. From the Symbol Type group, select the kind of IEEE symbol you want to place. You may also change the height of the symbol.
4. Click **OK** to close the *Place IEEE Symbol* dialog.
5. Move the cursor in the active window to where you want to place the IEEE symbol; click to place it. If you click again, the *Place IEEE Symbol* dialog appears so that you can choose another type of IEEE symbol for placement.

To cancel before an IEEE symbol is placed, right-click.



## Rewire Commands

### Using the Rewire Command

---

There is a single Rewire command in P-CAD Schematic, **Rewire » Manual**.

### Rewire Manual

---

Choose the **Rewire » Manual** command to reroute wire segments quickly and easily. This lets you make room for other objects, or to make aesthetic enhancements to a design. The **Rewire » Manual** command maintains the width of the wire.

### Using the Rewire Manual Command

To enter rewire mode:

1. Choose the **Rewire » Manual** command or click the **Rewire Manual** toolbar button.
2. Point to the wire segment you want to change.
3. Click the segment at the location where you would like to begin rerouting, then move the cursor to a new location and click to complete the new segment. Each click in a new location adds a new wire segment.

Press the **BACKSPACE** key to undo the last rewired segment. After the current rewire session is ended, the entire session is undoable.



## Options Commands

### Using the Options Commands

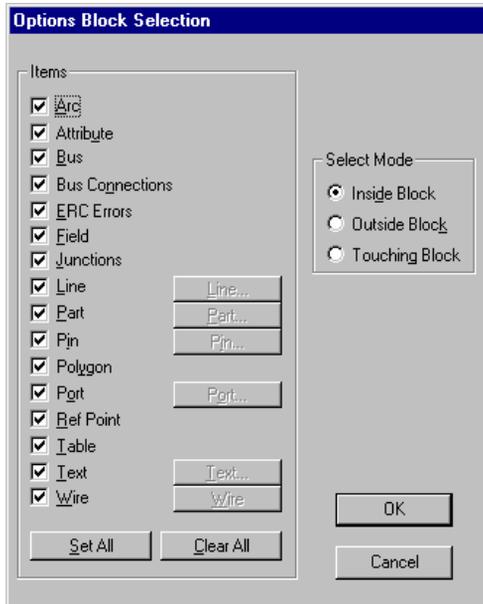
---

Use the commands in the **Options** menu to change various settings that affect a number of P-CAD Schematic actions such as block selection criteria (selection mask), default units (mm, mils, or in), grid settings, object display colors, and object styles.

### Options Block Selection

---

Choose **Options » Block Selection** to open the *Options Block Selection* dialog. You use the options in this dialog, to set block selection criteria that affects the selection of specific objects when you are in Select mode, as shown in the following figure:



## Select Mode Frame

The Select Mode frame contains the following radio buttons:

- **Inside Block.** This option is selected by default. Choose this button to select items within the selection block, according to the selection criteria you establish in this dialog.
- **Outside Block.** Choose this button to select items outside the selection block. All of the selection criteria that you specify in the *Options Block Selection* dialog functions outside the block rather than inside.
- **Touching Block.** Choose this button to select all items inside or touching the selection block. This is a more inclusive selection option than **Inside Block**.

## Items Frame

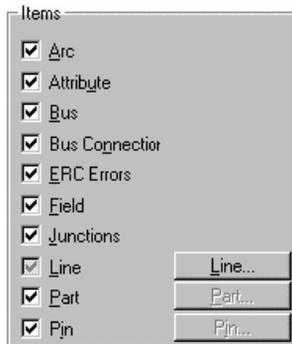
In the Items frame, you specify all of the objects listed in the dialog individually determining whether or not they are included in a block select. If you want to include only one or two items in your selection, click **Clear All** to clear all check boxes, then select individual check boxes to enable specific items. If you want to exclude only one or two items in your selection, click **Set All** to select all check boxes (if they are not already enabled), then individually clear the check boxes you want to exclude.

## Selection Mask Dialogs

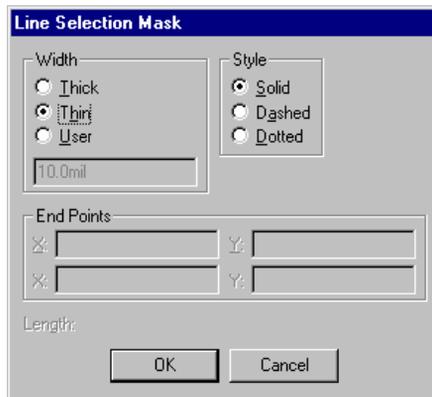
Some check boxes in the Items frame have corresponding command buttons (e.g., Line, Part, Pin). These check boxes have three states:

- included (selected).
- excluded (cleared).
- masked (shaded).

This feature allows you to narrow your selection further by setting specific properties as selection criteria for any item with a corresponding button. To access these property “masks”, click the check box until a shaded check mark appears. For example, the **Line** button becomes available when a shaded check mark appears in the **Line** check box, as shown in the following figure.



Then, click a button to open the corresponding dialog. For example, click the **Line** button to open the following *Line Selection Mask* dialog:



With the *Selection Mask* dialog, you can specify parameters for the particular object that you want as part of the selection mask or screening process. In the above example, if you specify a thin, solid line, your block selections will only select thin, solid lines.

## Selection Mask Parameters

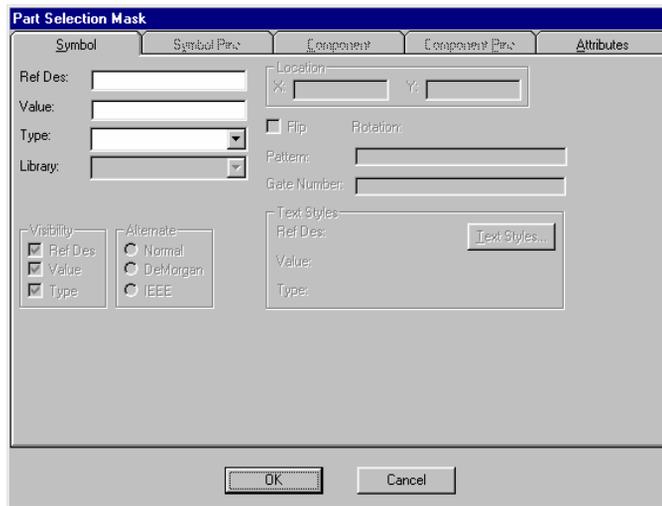
This is a basic summary of the parameters you can work with in the *Selection Mask* dialogs. Refer to *Edit Properties* (page 211) for detailed information about a specific object's properties.

### Line

For **Line**, you can specify line width and style. For details, see the illustration on the previous page.

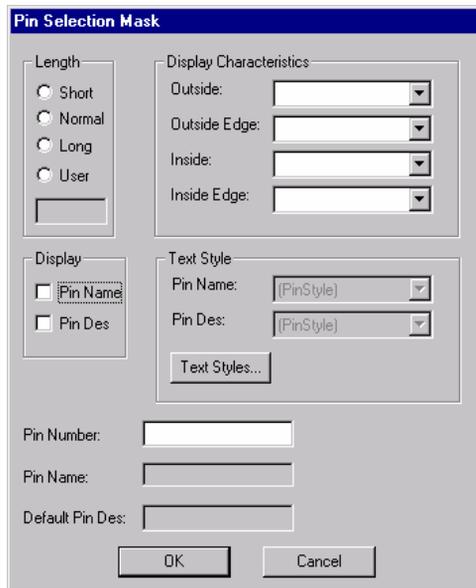
### Part

For **Part**, you can specify reference designator, value, type, and attributes, as shown in the following dialog:



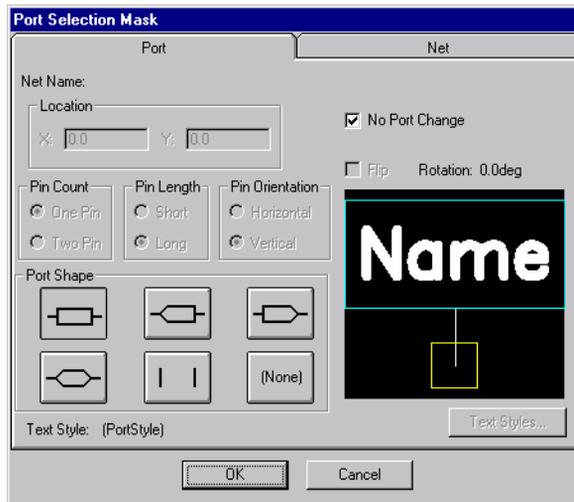
### Pin

For **Pin**, you can specify Length, Display Characteristics, Display, and Pin Number for a block select, as shown in the following dialog:



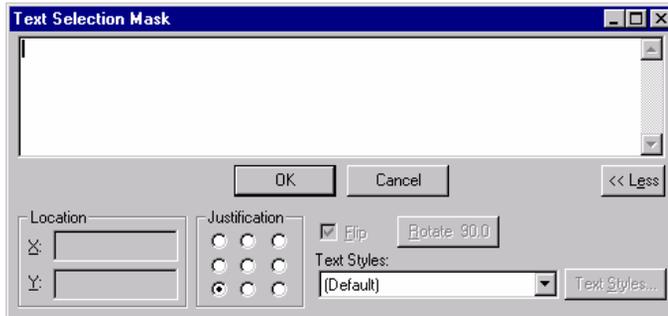
## Port

When the **No Port Change** check box is selected, you can select the **Nets** tab to specify Net Name. When the **No Port Change** check box is cleared, you can specify Pin Count, Pin Length, Pin Orientation, Port Shape, and Net Name, as shown in the following dialog:



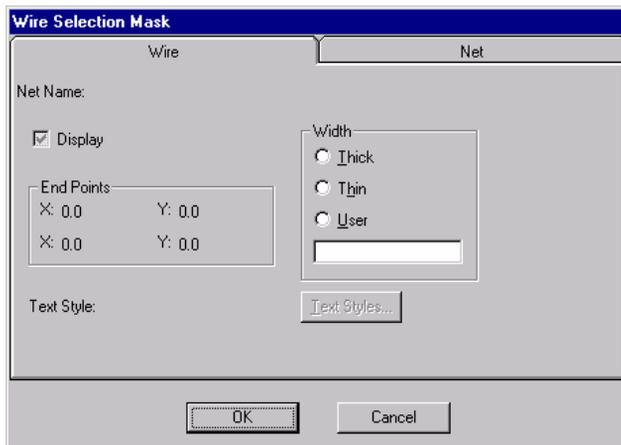
## Text

For **Text**, you can specify a specific text string, Text Styles, and Justification, as shown in the following dialog:



## Wire

For **Wire**, you can specify a Width, as shown in the following dialog:



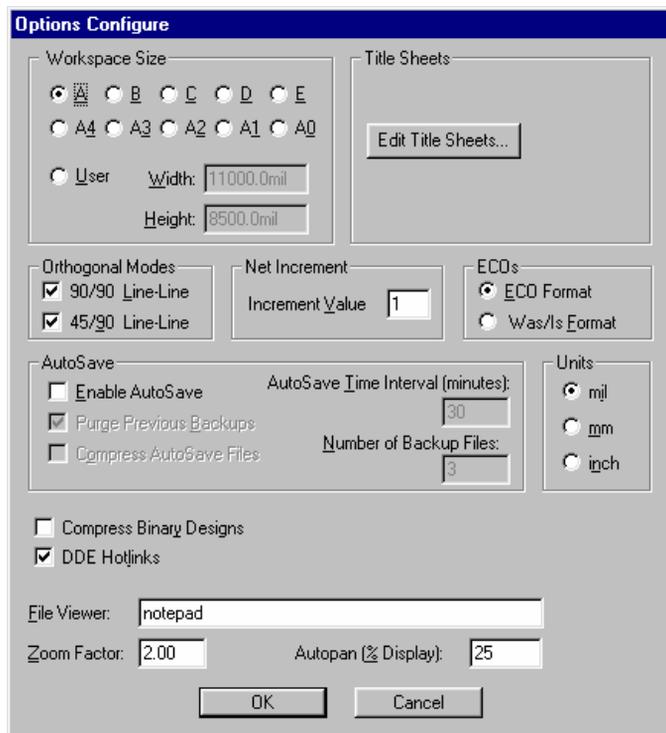
## Related Topics

For more information on using the *Selection Mask* dialogs, see *Edit Properties* (page 211).

For information about block selection, refer to *Edit Select* (page 247).

## Options Configure

Choose the **Options » Configure** command to open the *Options Configure* dialog. Use this dialog to set many of the Schematic parameters. Some of these parameters affect all open designs, and some affect only the active design.



## Workspace Size

Use the controls in this frame to set the size of the current design to one of five, standard imperial sizes (A-E) or five standard metric sizes (A4-A0). Choose the **User** radio button to create a custom workspace size by providing values for Width and Height. The minimum size for both Width and Height is 1 inch; the maximum is 60 inches.

## Title Sheets

The **Edit Title Sheets** button allows you to specify global and individual title sheet attributes. See *Title Sheets*, (page 98) for information on how to define and modify your title sheets.

## Units

Allows you to alter your display units between mils, millimeters, and inches. Dimensions are not altered, only the display of measurements are updated. A **mil** equals 0.001 inch or .0254 mm. A **mm** equals 0.001 meter. When the selected unit is mm, you can control the number of digits displayed to the right of the decimal point by modifying the MillimeterPrecision setting in the .ini file. The default displayed precision is three decimal digits, but can be changed to an integer value of 2 through 5.

This setting will affect 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.

## Orthogonal Modes

Orthogonal modes are for use while placing wires, buses and lines, using segments that are horizontal, vertical, and at 45-degree angles.

Press the **O** key to switch between the enabled modes. You can enable or disable the orthogonal modes in the *Options Configure* dialog. The non-orthogonal mode, with straight-line placement at any angle, is always enabled.

The 90/90 and 45/90 orthogonal modes are provided as mode pairs. Press the **F** key to switch between the mode pair of the current mode.

- 90/90 Line-Line creates true 90-degree angles, long and short.
- 45/90 Line-Line creates 45/90 and 90/45 angles.

Refer to the *Place Wire* (page 263), *Place Bus* (page 265), and *Place Line* (page 272) for more details about the orthogonal modes.

## Net Increment

The Net Increment frame contains an Increment Value edit box. The Increment Value edit box lets you specify the step value for incrementing net names: a negative value causes the name to decrement instead of increment, and a zero value disables the copy increment feature. **CTRL/Drag Copy** and **Edit » Copy Matrix** are the commands that use this increment value.

## ECOs

Engineering Change Orders (ECOs) are recorded when you choose the **Utils » Record ECOs** command. From the *Options Configure* dialog, you can select a format for the ECO files. This setting applies to the active design.

The ECO file is generated in an ECO format or a Was/Is format, which is compatible with Tango Series II. Click either the **ECO Format** or the **Was/Is Format** radio button to select a format:

- ECO format records full ECOs.
- The Was/Is format records only Was/Is ECOs. It keeps track of RefDes changes only and generates a file that is compatible with Tango Series II.

## AutoSave

Use the controls in the AutoSave frame to enable the AutoSave feature, which regularly saves your files at a user-defined interval in a compressed or non-compressed format. AutoSave won't be performed during autorouting or if a tool is busy.

The following options are available:

- **Enable AutoSave:** Select this check box to turn on the AutoSave feature.

- **AutoSave Time Interval:** Enter the time between saves. AutoSave uses a rolling backup to save files, incrementing each subsequent autosave file.
- **Purge Previous Backups:** When enabled, the option causes all backups saved from the previous design session to be deleted when you begin a new design session.
- **Number of Backup Files:** Allow you to specify the number of design files to be archived before file names are reused. This must be a number between 1 and 99.
- **Compress AutoSave Files:** Compresses the file so that it uses less space when saved.

## File Viewer

Enter a value in this text box to define the viewer to be used for viewing reports, log files, error reports, etc. Enter the application name (e.g., Notepad). If the application is in a directory that is not included in your `Autoexec.bat` path statement, include the complete pathname here.

## Zoom Factor

Enter a value in this text box to adjust the amount of zoom that occurs when you choose the **View » Zoom In** or **View » Zoom Out** command. A factor of 2.00 doubles (or halves) the size of objects in the Workspace, etc. Zoom factors must be greater than 1.00.

## 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.

## Compress Binary Designs

Select the **Compress Binary Designs** check box to enable the automatic compression of binary files whenever those files are saved.

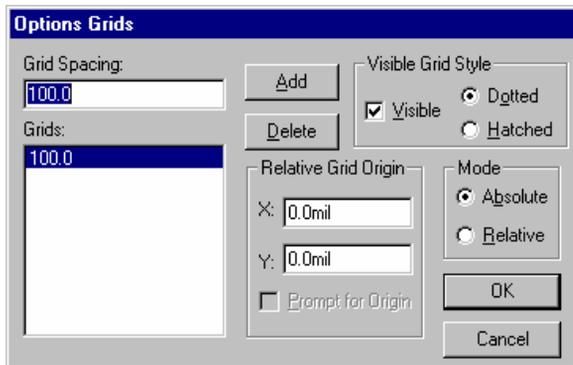
## DDE Hotlinks

If selected, this check box enables exchange of hotlink data with P-CAD PCB. Hotlink data consists of highlighting and unhighlighting commands for parts and nets. The state of the DDE Hotlinks option is saved in the `Sch.ini` file.

## Options Grids

---

Choose the **Options » Grids** command to define various properties for your grids. Grid values are stored with each, individual design file. When you choose the **Options » Grids** command, the following dialog appears:



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

## Grid Spacing

Use the controls in the Relative Grid Origin frame to 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 box, then click **Add** to add it to available choices in the Grids list box. To delete a grid-spacing value, highlight it in the Grids list box and click **Delete**.

## Grid Toggle Buttons

The Status Line **Grid Toggle** button and the Grid combo box, allow you to easily switch between grid settings and add new grid settings. The toggle switches between absolute grid (**Abs**, with white background) and relative grid (**Rel**, with colored background). Your absolute and relative grid values can be changed from the combo box. The **A** key toggles between absolute and relative grids.

If you use the **Grid Toggle** button to switch to Relative, you are prompted for the origin point if you have the **Prompt for Origin** option enabled in *Options Grids*. The cursor takes the shape of a crosshair cursor while the system is awaiting input; when you click in the active window, that becomes the relative origin point (X=0, Y=0) and the cursor returns to normal.

## Visible Grid Style

The **Visible Grid Style** check box allows you to either show or hide grids. When you select the check box, two grid style options are available. **Dotted** pinpoints grid points, while **Hatched** draws lines along the grids to show grid intersections (like graph paper).

## Relative Grid Origin

Use the controls in the Relative Grid Origin frame to specify your X and Y relative grid origins by entering the coordinates. You must have the **Prompt for Origin** option disabled for it to work.

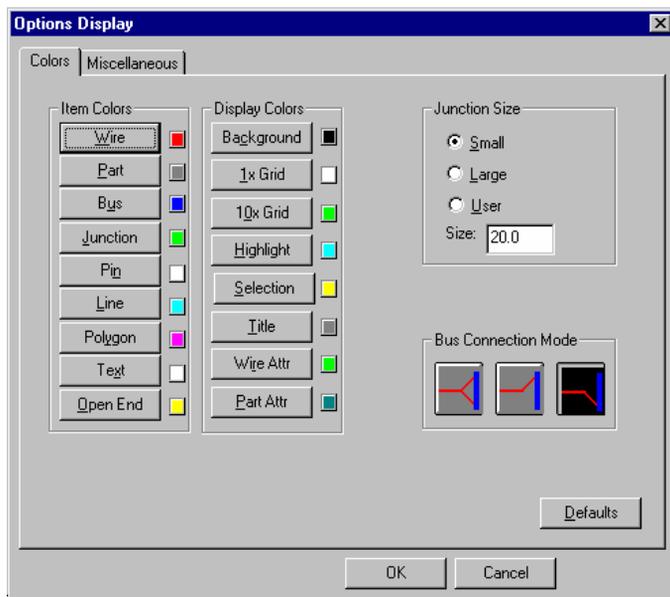
If you switch to Relative grid in the dialog and the **Prompt for Origin** check box is selected, you are prompted for the new origin point after you click **OK**. You are prompted every time you switch between Absolute to Relative grids by clicking the **Grid Toggle** button or by pressing the **G** key.

## Mode

In the Mode frame, choose the **Absolute** radio button when you want the grid origin point to be the lower-left corner of the Workspace. You can choose the **Relative** radio button to allow any point as an origin point.

## Options Display

Choose the **Options » Display** command to open the *Options Display* dialog. There are two tabs in this dialog: **Color** and **Miscellaneous**. Use these tabs to define your workspace preferences.



## Colors

This tab contains four frames and a **Defaults** button. Click the **Defaults** button to return your display setup to the default scheme.

Any color settings established here are saved in your `Sch.ini` file when you close the dialog. These settings affect your designs in subsequent sessions, until you change them.

The following sections describe how to use the controls in these frames:

If you have a color printer, clicking the **Defaults** button also restores default color settings for printing.

- Item Colors
- Display Colors

- Junction Size
- Bus Connection Mode

### Item Colors

To select a color for an item, click the corresponding button in this frame. When you do, a color palette appears. Choose a color from the palette by clicking the appropriate button. The palette closes automatically.

You can also set a custom color. To close the color palette without choosing a color, press **ESC** or click the **Close** button.

### Setting A Custom Color

To set a custom color, click the **Custom** button in the palette to open the *Color* dialog. Then, choose a color from this dialog using one of these methods:

#### Method 1: Select a Defined Custom Color

1. Click one of the colors in the Basic Colors frame.
2. Click **OK** to return to the color palette.

#### Method 2: Define a Custom Color

1. Click the color matrix to define the Red/Green/Blue settings, or type the desired settings in the Red/ Green/Blue boxes.
2. Click the **Add to Custom Colors** button.
3. When the selected color appears in the Custom Colors section, click **OK** to return to the color palette.

Some custom colors are displayed only when supported by your video display settings. If your video display supports 256 Colors, custom colors are approximated to the nearest solid color, while retaining the Red/Green/Blue settings. If your video display supports more than 256 colors, custom colors display accurately.

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 color palette, however, colors not selected for an item/display color are forgotten once the P-CAD application is exited.

### Display Colors

In the Display Colors frame, there are several command buttons. Click one of these buttons to determine various Schematic display colors. The settings configured with these buttons appear in your workspace, regardless of layer or item colors.

When setting your display colors, make sure that the colors you pick do not conflict with any layer or item colors. For example, if the background color is set to be the same as the line color, lines will not be visible in your design.

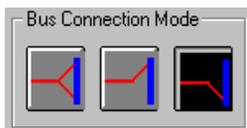
### Junction Size

To set a size for junctions, choose one of the following radio buttons in the Junction Size frame:

- **Small.** When you choose this option, the junction size appears in the Size box.
- **Large.** When you choose this option, the junction size appears in the Size box.
- **User.** When you choose this option, you can type a value in the Size box. Your entry can be in inches, centimeters, etc., as long as the value is between 0 - 10 mm.

### Bus Connection Mode

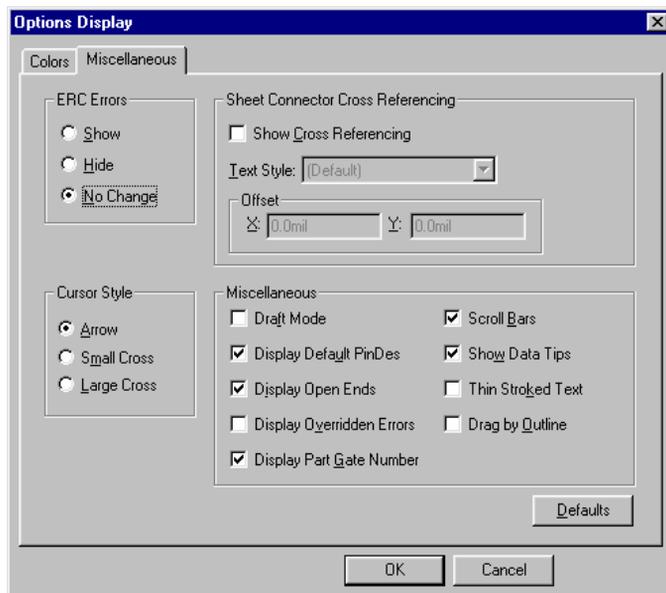
Click one of these buttons to determine the bus connection mode:



In a design session, bus connections form automatically when you connect a wire to a bus. For more information, see *Place Bus* (page 265).

## Miscellaneous

To set up other display options for your workspace, click the **Miscellaneous** tab in the *Options Display* dialog. As shown in the following figure, this tab contains a number of options:



## ERC Errors

In the ERC Errors frame, you have these options:

- **Show:** Choose this button to show ERC errors in the workspace.
- **Hide:** Choose this button to hide the display of ERC errors in the workspace.
- **No Change:** Choose this button to keep the current display setting.

## Sheet Connector Cross Referencing

See *Annotating Sheet Connectors*, (page 108) for details on how to define sheet connector cross-referencing.

## Cursor Style

To change the style of your cursor, choose from an arrow, small cross or large cross.

The large cross style stretches horizontally and vertically to the edges of the Schematic window.

The Large Cross cursor style does not support the DataTips feature. Enabling the Large Cross cursor clears the Show DataTips check box and makes the feature unavailable.

## Miscellaneous

The Miscellaneous frame contains a number of check boxes. To enable an option, select its check box. Clear a check box to disable an option.

- **Draft Mode:** Select this check box to show these two items in your design: (1) a thin, single-pixel outline for pads, vias, and text. (2) a segmented and outlined representation of arcs, lines, and any line segment objects such as, polygons, cutouts, etc. Draft mode improves redraw speed and helps you view segment overlaps.
- **Thin Stroked Text:** Select this check box to show text in thin line mode. Clear this check box to display text in regular mode.

You can use the draft mode shortcut key to switch between draft and non-draft display modes. Based on the combination of options enabled (e.g., Draft Mode and Thin Stroked Text), the current selection is retained and used to produce a comparable, opposite display when the draft mode shortcut key is pressed. For instance, if you begin with both Draft Mode and Thin Stroked Text enabled, and you switch to non-draft mode, the display produced is normal figures and regular text.

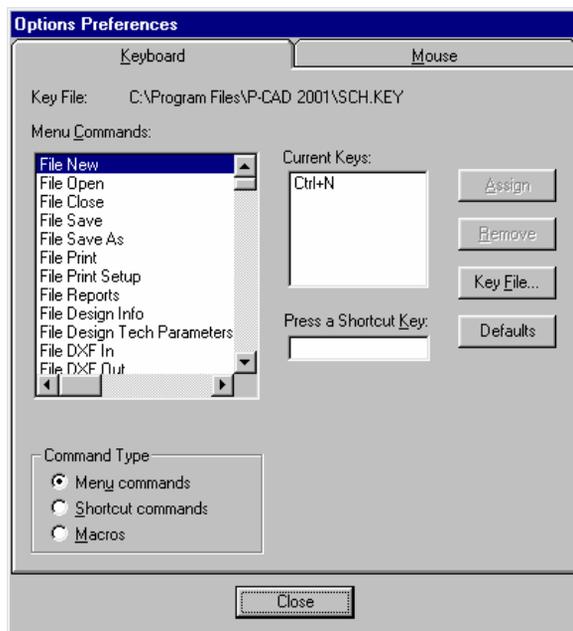
- **Display Open Ends:** Select this check box to show open ends on unconnected pins and wires. Open ends appear as open squares, and no longer appear when a positive connection is made.
- **Display Part Gate Number:** Select this check box to show a part's reference designator gate number. You can also use the *Print Options* dialog to show or hide the gate number in your hardcopy. The options are independent of each other, so you can show the gate number on screen for editing, but hide it from your hardcopy at the same time.

- **Drag by Outline:** Clear this check box to show a ghosted image of the part and a bounding outline when you move or copy objects in your design. Select this check box to show only the bounding outline. This setting affects only the move and copy operations associated with the **Edit Select** command.
- **Display Overridden Errors:** Select this check box to show overridden ERC error indicators. Clear the check box to hide overridden errors.
- **Display Default PinDes:** Select this check box to show the default pin designator for free pins. Clear this check box to display only pin numbers.
- **Scroll Bar:** Select this check box to show scroll bars in the active window. Clear this box to hide the scroll bars.
- **Show DataTips:** Select this check box to show DataTips in the active window. Clear the check box to hide the display of DataTips in the active window.

The Large Cross cursor style and the **View Snap to Grid** command do not support the DataTips feature. For details, see *Cursor Style* (page 300) and *View Snap to Grid* (page 257).

## Options Preferences

Choose the **Options » Preferences** command to define keyboard, mouse, and toolbar preferences used to set up the application. When you choose this command, the following dialog appears with the **Keyboard** tab selected:



## Keyboard Tab

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

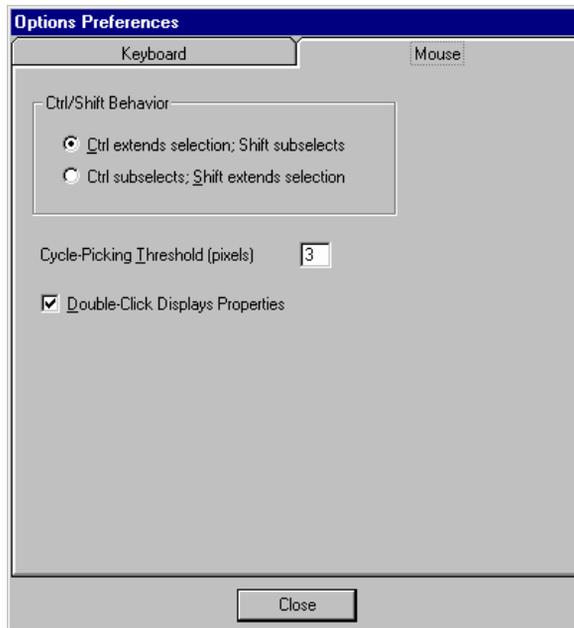
- **Command Type** (checkbox): Choose the type of command for which you want to change shortcut key assignments.
- **Menu Commands/Shortcut Commands/Macros** (list box): Select the command or macro you want to add a shortcut key assignment to or from which you want to remove a shortcut key assignment.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

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

- **Assign** (button): Assigns the key appearing in the Press a Shortcut Key box to the selected command or macro. If the shortcut is currently assigned, the current assignment disappears.
- **Remove** (button): Removes the key you select in the Current Keys box.
- **Key File** (button): Allows you to select 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.
- **Defaults** (button): Restores original default key assignments to all commands or macros.

## Mouse Tab

When you select the **Mouse** tab the *Options Preference* 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 (extended) selections and which to use for Sub 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 an object to bring up the *Properties* dialog for that object.

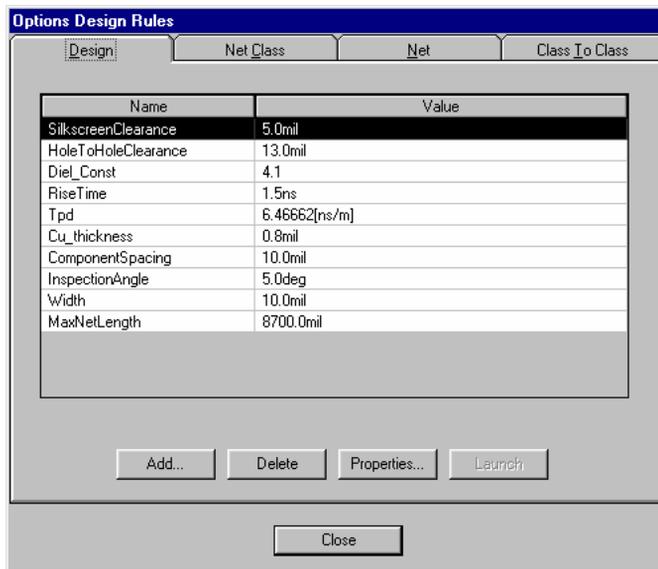
## Options Design Rules

---

When you choose the **Options » Design Rules** command, the *Options Design Rules* dialog appears.

### Design Tab

The **Design** tab shows global clearance rules that are applied to the entire design.



## Clearance Rules

When a clearance rule for a specific object is requested (e.g., DRC), the design rules category is searched in the following order of priority:

- Class to class rules (highest priority).
- Net rules.
- Net class rules.
- Global rules (lowest priority).

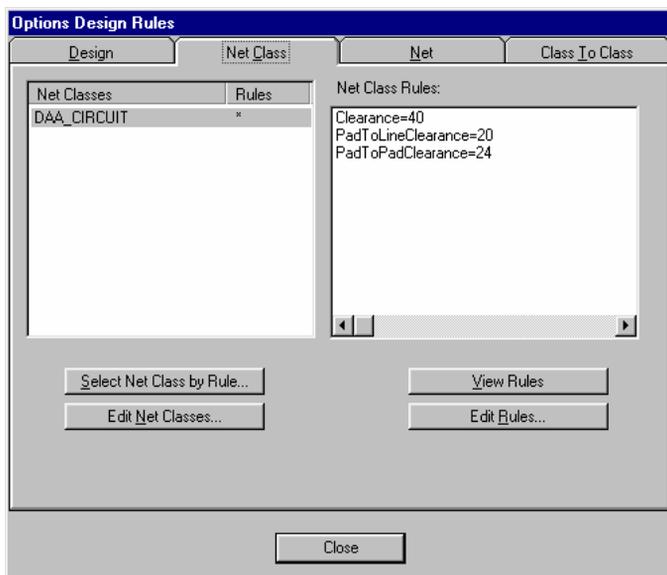
Within each category, the clearance rules are searched in the following order of priority:

- Object pair clearance rules (e.g., Pad to Line clearance).
- Clearance rules.

The order of evaluation matches the order of evaluation used by the CCT SPECTRA Router. P-CAD PRO Route uses only global clearance rules.

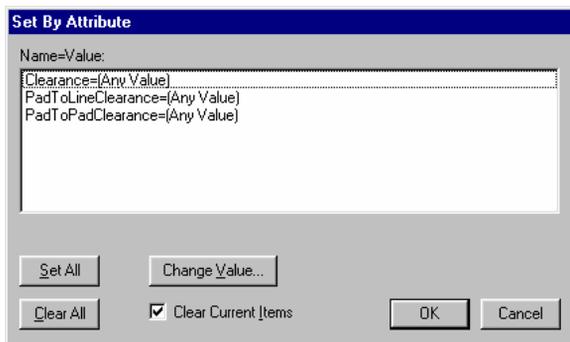
## Net Class Tab

Use the **Net Class** tab to specify clearance rules for pads, vias, lines, and object pairs (such as pad to pad or line to via). The dialog lists all net classes, indicates the presence of rules in the Rules list, and shows the rules and their values in the Net Class Rules list box, as shown in the following figure:



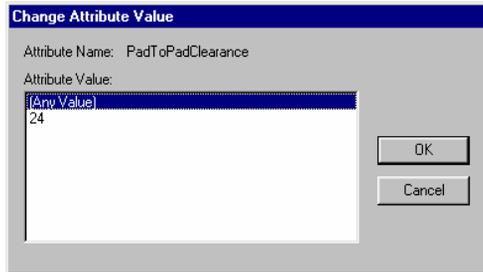
When an attribute exists for multiple Net Classes, and the value of the attribute is different for each Net Class, the Net Class Rules list shows the attribute and indicates the value as mixed when you select more than one Net Class.

Click the **Select Net Class by Rule** button to display the *Set By Attribute* dialog, shown in the following figure, where you can choose the attribute(s) you want to use to select the Net Classes. Selecting an attribute from the list box allows you to find all net classes containing that attribute, regardless of the attribute value.



If you select an attribute from the Name=Value list box and click **OK**, you are returned to the *Options Design Rules* dialog where all net classes that have the selected attribute defined are highlighted in the Net Class list box.

To find net classes with attributes having a specific value, select an attribute from the Name=Value list box and click the **Change Value** button to display the *Change Value Attribute* dialog, shown in the following figure.

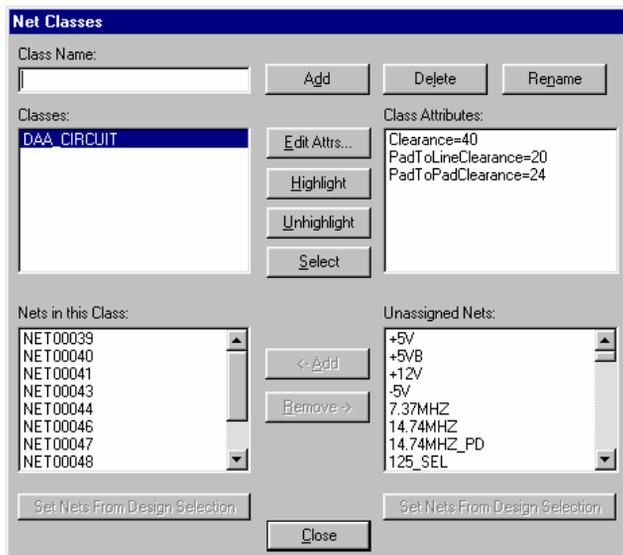


The *Change Attribute Value* dialog lists all values assigned to the selected attribute and also includes the option to select {Any Value}. Select the desired **Value** and click **OK** to return to the *Set By Attribute* dialog.

To set all of the attributes, click the **Set All** button. To clear the selected attributes, click the **Clear All** button. Choose the **Clear Current Items** check box if you want to remove previously selected items each time you modify your selection criteria.

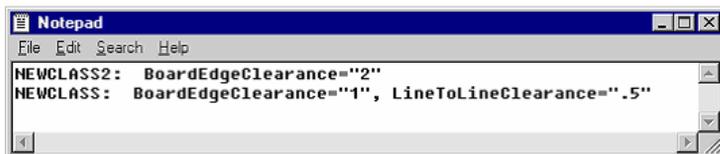
If you select multiple attribute names and values in the Name=Value list box, only those nets with all of the selected attribute names and values are highlighted when you return to the *Options Design Rules* dialog.

To make changes to the Net Classes, click the **Edit Net Classes** button to display the *Net Classes* dialog:



In the *Net Classes* dialog you can define a group of nets that share common rules. Collections of nets sharing the same rules are known as a Net Class. For more information on using the *Net Classes* dialog, see *Options Net Classes* (page 309).

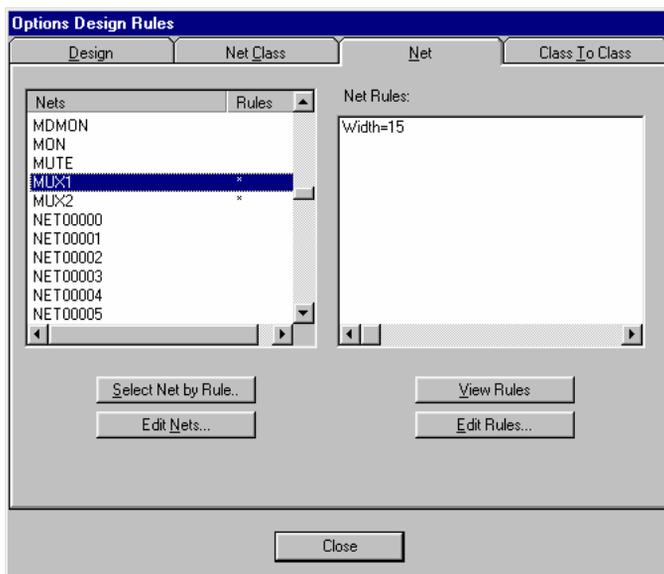
Click the **View Rules** button to see the rules in the Notepad as shown in the following figure:



To modify the rules click the **Edit Rules** button to display the *Attributes* dialog.

## Net Tab

When you click the **Net** tab, the *Options Design Rules* dialog appears as shown in the following figure:



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

Click the **Select Net by Rule** button to display the *Set By Attribute* dialog where you can choose the attribute(s) you want to use to select the Net. The *Set By Attribute* dialog is explained in *Net Class Tab* (page 304).

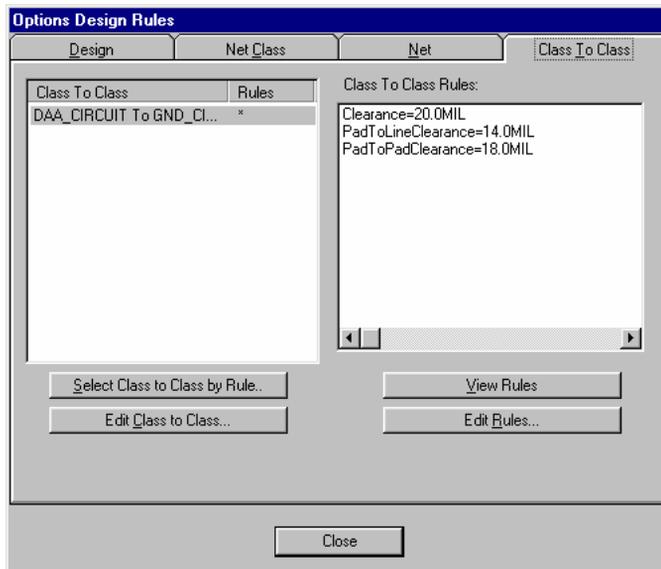
To modify the nets in the design, click the **Edit Nets** button to display the *Edit Nets* dialog. Complete information on the *Edit Nets* dialog is found in *Edit Nets* (page 240).

Click the **View Rules** button to see the rules in the Notepad format.

To add, delete or edit rules, click the **Edit Rules** button to access the *Attributes* dialog.

## Class to Class Tab

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

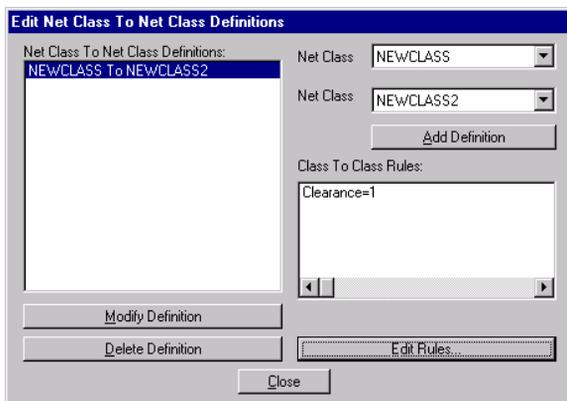


The **Class to Class** tab allows you to specify clearance rules between nets grouped into a Net Class. The dialog lists all class to class definitions and the rules associated with each.

Click the **Select Class to Class By Rule** button to display the *Set By Attribute* dialog, where you can choose the attribute(s) you want to use to select the **Class to Classes**. The *Set By Attribute* dialog is explained in *Net Class Tab* (page 304).

Click the **View Rules** button to see the rules in the Notepad format.

To create a new Class to Class definition or modify an existing Class to Class, click the **Edit Class to Class** button to display the *Edit Class to Class Definition* dialog shown in the following figure:

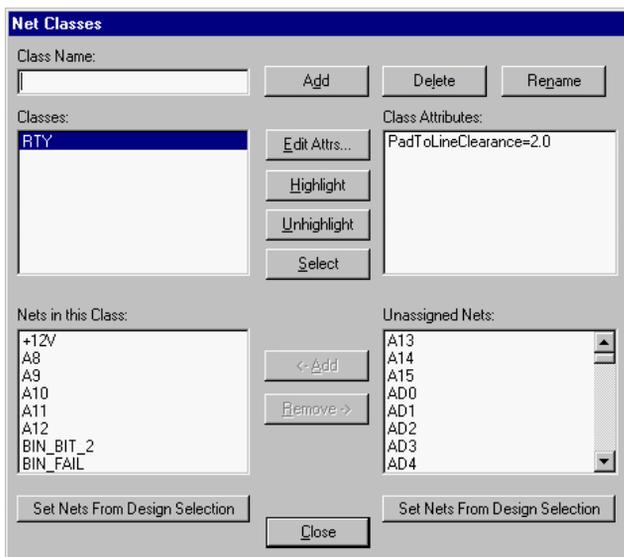


To create a Class to Class definition, select **Net Classes** from the two Net Class name list boxes, and click **Add Definition**. Once created, the Class to Class definition can be modified or deleted by clicking the **Modify Definition** or **Delete Definition** buttons. Rules associated with the Class to Class definition appear in the Class to Class Rules list box.

To add, delete or modify rules, click the **Edit Rules** button to access the *Attributes* dialog.

## Options Net Classes

Choose the **Options » Net Classes** command to define a group of nets that share common rules. Collections of nets sharing the same rules are referred to as a 'net class'. When you choose this command, the *Net Classes* dialog appears:



This class editor allows you to create named net classes using pre-defined clearance rules or pre-defined SPECCTRA autorouter clearance rules and then assign nets to that class. You can also add user-defined attributes to the net classes for your own use.

Net classes are transferred from the Schematic design to PCB via the P-CAD format netlist or through ECO's. PCB Design Rules Checking verifies clearances and the attributes listed below when they have been defined in the net class:

- MaxNetLength
- MaxVias
- MinNetLength
- ViaStyle
- Width

For net clearances the rules can be further refined by specifying clearance rules for pairs of objects, like pad to pad clearances or line to via clearances.

The net class information is written to binary and ASCII design files. P-CAD Master Designer PCB does not support Net Classes so this information is lost when exporting a P-CAD design to P-CAD format.

To create named net classes, follow these steps:

1. Enter a class name in the Classes box.
2. Click **Add**.
3. To include a net from the Unassigned Nets area to the new net class, you may use any of the following methods:
  - Select a single net and click the **Add** button.
  - Double click on a net to move it from **Unassigned** to **Nets in this Class** and vice versa.
  - Select multiple nets in a block by either 1) holding the **SHIFT** key while selecting the first and last nets in the block or, 2) select the first net in the list, hold the left mouse button and drag the cursor to the last net in the list and release. Then click **Add**.
  - Select individual nets by holding the **CTRL** key while clicking on each net, then click **Add**.
4. To remove a net from the Nets in this Class area, use the same methods detailed above, but click the **Remove** button instead of the **Add** button.

In addition to the normal selection process you may use the **Set Nets From Design Selection** buttons to quickly place all currently selected nets in the design into either the Unassigned Nets or Nets in this Class areas. When no nets are selected in the design, the inactive Set Nets From Design Selection buttons are gray. If nets in either area are selected and you click the **Set Nets From Design Selection** button, the other selected nets become unselected.

5. Use the **Edit Attributes** button to assign one or more attributes to this new net class. Refer to *Edit Nets* (page 240) for details.

## Options Sheets

Choose the **Options » Sheets** command to switch to a different sheet by defining the current sheet. You may also add, delete, reorder and change the name of any sheet in a design.



With P-CAD Document Toolbox, the *Options Sheets* dialog is enhanced to include a **Titles** tab. On this tab you can specify independent borders, zone divisions, and title blocks.

### Sheets

In the Sheets box are listed all sheets, excluding the title sheet, defined for the design. The current sheet has an asterisk next to it.

### Buttons

The **Add** button adds a sheet to the list using the Sheet Name you have specified. The specified sheet name must be unique.

The **Modify** button allows you to change a sheet name. Choose a sheet name from the list, make the desired changes in the Sheet Name box and click **Modify**.

The **Delete** button deletes the selected sheet. You can delete any empty, non-current sheet.

The **Current** button makes the selected sheet the current sheet, indicated by the asterisk. This sheet then appears in your display.

The **Move Up** button positions the selected sheet one step above its current location in the list. The first sheet on the list can only be moved down in the list. When the first sheet is selected the **Move Up** button is grayed and unavailable.

The **Move Down** button positions the selected sheet one step below its current location in the list. The last sheet on the list can only be moved up in the list. When the last sheet is selected the **Move Down** button is grayed and unavailable.

Whenever a sheet is moved up or down, the sheet number changes to indicate its new position in the list and is reflected across the application wherever the sheet number appears.

## Adding a Sheet

When you add (create) a sheet, you must give it a unique name. Schematic assigns Sheet Numbers sequentially, using the next available number.

To add a sheet, follow these steps:

1. In the *Options Sheets* dialog, type the new sheet name in the Sheet Name box.
2. Click **Add**. The new sheet name is listed in the list box with the sheet number automatically specified.

Once a sheet has been created/added, the sheet number displayed represents the order in which it appears in the list. If the sheet is moved up or down to a different position in the list the sheet number is adjusted accordingly.

## Options Current Wire

---

Choose the **Options » Current Wire** command to set the current wire width for the **Place » Wire** command. When you choose this command, the *Options Current Wire* dialog appears. In this dialog, select a wire width and click **OK**.



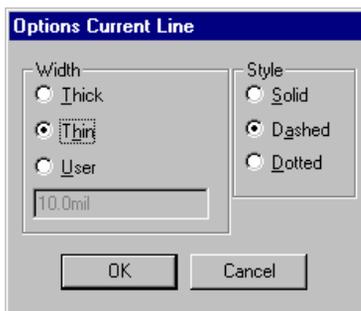
Wire Width options are:

- **Thick** wires are 15 mils.
- **Thin** wires are 10 mils.
- **User** defined wires can range from 0.1 to 100 mils. To maintain the appearance of P-CAD Schematic designs before V13.0, all wires less than 11mils wide are displayed as 1 pixel. The current wire settings are saved to the `Sch.ini` file.

To change the wire width of existing wires, choose the **Edit » Properties** command.

## Options Current Line

Choose the **Options » Current Line** command to set the current line width and style for the **Place » Line** and **Place » Arc** commands. Select a line style and width and click **OK**.



Thick and user-defined lines are always solid. Thin lines can be solid, dashed or dotted. Arcs are always solid, but can be thick or thin. The current line settings are saved to the `Sch.ini` file.

You can also change the current line width using the line width combo box on the status line.

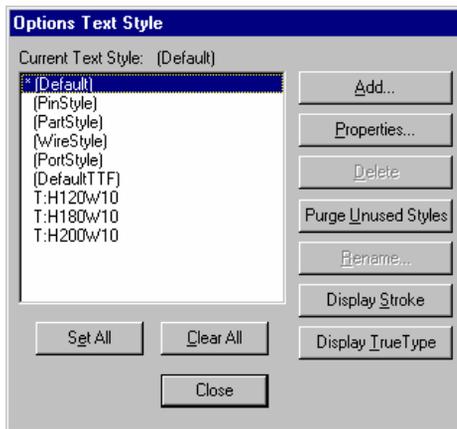
To change the line width (thickness) of existing lines and arcs, choose the **Edit » Properties** command.

## Options Text Style

Choose the **Options » Text Style** command to set the current text style for the **Place » Text** command. This command also allows you to add, delete, rename, or edit text styles by using the series of available dialogs. The text styles you create or edit here are available when you enable the **Place » Text** command, or when you want to modify already placed text with the **Edit » Properties** command.

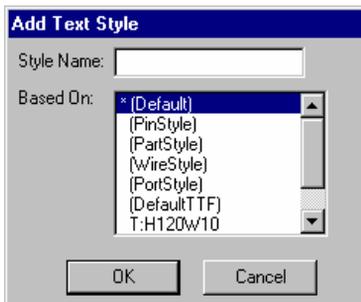
The Default and DefaultTTF styles cannot be deleted, renamed, or modified. The other default styles (PinStyle, PartStyle, WireStyle and PortStyle) can be modified, but not deleted or renamed. You can base a new text style on any of the default styles.

Choose **Options » Text Style** to open the following dialog:



### Add a Text Style

1. To add a text style, follow these steps:
2. In the *Options Text Style* dialog, click **Add**. The *Add Text Style* dialog appears:



3. Select the style of text from the Based On list box.

4. Enter the text style name you are adding (e.g., Bus style).
5. Click **OK** and the *Text Style Properties* dialog appears. For a picture of this dialog, see *Text Style Properties* (page 315).

## Delete a Text Style

To delete a text style:

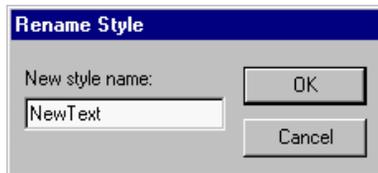
1. Highlight the non-default text style you want to delete.
2. Click **Delete**. A message appears asking you to confirm your deletion.
3. Click **Yes**, and the selected text style disappears from the list.

You cannot delete default text styles or a text style that is currently in use.

## Rename a Text Style

To rename a text style:

1. Select the non-default text style you want to rename.
2. Click **Rename**. In the Rename Style box, the existing style name appears.



3. Type over the existing style name to specify the name of the new text style.
4. Click **OK** to return to the *Options Text Style* dialog.

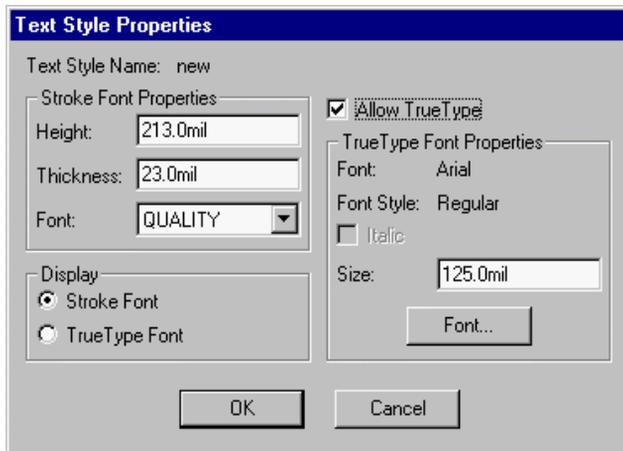
## Text Style Properties

The *Text Style Properties* dialog lets you add and modify font properties of the selected (non-default) text styles. You cannot change the properties of Default text style; you can only query it.

To query and edit text style properties:

1. Select a text style from the Current Text Style list.
2. Click **Properties** and the *Text Style Properties* dialog appears (shown below).
3. From this dialog, you can modify the Height, Thickness, and Font fields for non-default fonts.

As shown below, the *Text Style Properties* dialog appears when you click the **OK** button in the *Options Add Text Style* dialog or the **Properties** button in the *Options Text Style* dialog:



The stroke font text style is automatically assigned to text in a Schematic design. You may change the **Height**, **Thickness** and **Font** properties of the stroke font in the Stroke Font Properties area. In addition, when the **Allow TrueType** box is enabled, you may choose the **Font**, **Font Style** and **Size** for the selected style's corresponding TrueType representation using the standard *Font* dialog.

### Stroke Font Properties

You can set the following *Stroke Font Properties*.

- **Height:** The font's height.
- **Thickness:** The text thickness.
- **Font:** Choose between **QUALITY**, **BASIC**, or **LCOM** fonts. QUALITY and BASIC fonts are interchangeable. BASIC is simpler and therefore draws faster. LCOM is a serif font (a little fancier).

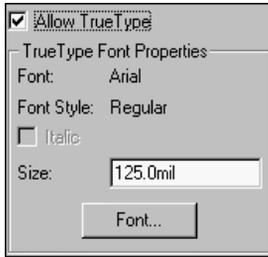
quality font

basic font

lcom font

### TrueType Font Properties

When the **Allow True Type** check box is selected, you can change the font type, size and style of the text's TrueType representation.



The TrueType Font Properties frame, shown in the previous figure, displays this information:

- **Font:** The TrueType font name.
- **Font Style:** The font style (E.g., Regular, Bold, etc.)
- **Italic:** An X indicates an italic font style.
- **Size:** The font size in mils or mms depending on what you have set in the *Options Configure* dialog.

To change TrueType font properties, click the **Font** button.

Click the **OK** button to apply your selection and return to the *Text Style Properties* dialog.

### Display Area

If you have enabled the **Allow TrueType** option in the *Text Style Properties* dialog, you may choose to display the text in either **Stroke Font** or **TrueType Font** by clicking the appropriate button.

Click **OK** to return to the *Options Text Style* dialog.

### Display Stroke and Display TrueType Buttons

Whenever a text style has had its **Allow TrueType** option enabled, you can quickly change its display mode by clicking the **Display Stroke** or **Display TrueType** button in the *Options Text Style* dialog. This changes the selection in the Display area of the *Text Style Properties* dialog without having to go into the dialog and make a new selection.



## Library Commands

### Using the Library Commands

---

With the Library commands, you can create new libraries, create item aliases, copy items from one library to another, delete library items, rename library items, open libraries for access, save symbols that you use to create a component libraries, and create an archive library.

P-CAD libraries are a combination of components, Schematic symbols and PCB patterns (not used by Schematic). The component section of a library contains component information, such as what symbol is attached to a particular part, what its type is, etc. The symbol section contains the graphical information for the symbol that is attached to a part. Without a symbol, a part has no graphical representation, and is not placeable. A symbol by itself is only a graphical structure. A component and its symbol reside in the same library; the component references a particular symbol, its structure, when that symbol is attached to it. For example, a 7400N component with its component attributes and pin assignments occupies the same library with a SYMBOL100\_N symbol, which it references.

It is typical for multiple components within a library to reference the same symbol. A library could conceivably contain 100 components while only containing five symbols (multiple components referencing the same symbol). If you changed one of the symbols, then all of the components referencing that symbol would be affected.

If you have purchased P-CAD Library Executive, the **Library** menu has two additional commands: **Query** and **Verify Design**. These commands access the embedded component search utility and the design verification utility, respectively. See your *P-CAD Library Executive User's Guide* for details.

### Library New

---

Choose **Library » New** to create a new library. The new library is empty; it has no components and no symbols.

When you choose **Library » New**, the *Library New* dialog is displayed, which is a Windows™ common dialog. In the dialog, you can specify the filename of your new library.

## Library Alias

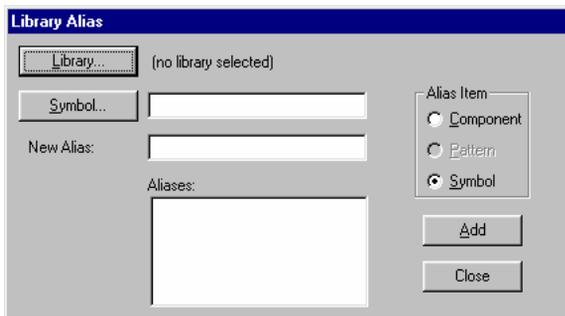
Choose the **Library » Alias** command to create an alias for a component or symbol. An alias is an alternate name for an item (component or symbol). You can create multiple equivalent names for the same item with this command.

When you create aliases for an item, it is not the same as creating copies or renaming. For copying or renaming, see the respective Library commands.

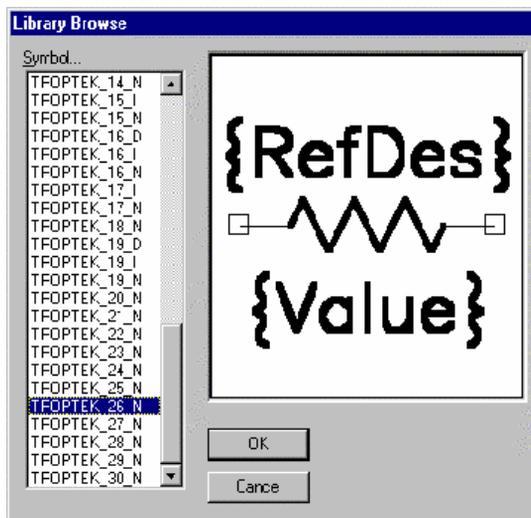
Aliases allow flexibility in using a variety of naming conventions for components or symbols, without renaming them. For example, what P-CAD calls an SN7400N, you may want to use a generic alias of 7400. Or, if you have components from a vendor using a particular naming convention, and you want to continue using that system, you can use alias names and display them on your design as such.

### Creating an Alias

1. Choose **Library » Alias** to open the following *Library Alias* dialog:



2. If the appropriate library is not current, click the **Library** button to open the *Library Select* dialog.
3. The *Library Select* dialog is similar to the *File Open* dialog.
4. Select the library you want, click **Open**, and the *Alias* dialog reappears.
5. In the Alias Item frame, choose the **Component** or **Symbol** radio button. The Pattern radio button is grayed in Schematic.
6. Click the **Component** or **Symbol** button to open the *Library Browse* dialog. Select the item you want from the list and click **OK**.



7. You have now returned to the *Library Alias* dialog. Enter the alias for the symbol in the New Alias box, and click **Add** to append it to the Aliases list. Click **Close** when finished.

## Library Copy

Choose the **Library » Copy** command to copy one or more items from one file to another (either in the same or in a different library). The items you copy can be a component or symbol.

It's important to note that a library part consists of a component section (type, reference designator, etc.), a symbol section (for Schematic graphics), and a pattern section (for PCB graphics). To place the copied component on a schematic design, you need to copy the component and its symbol when copying between different libraries (notice that the *Copy Item* frame of the dialog has a choice between **Symbol** and **Component**). When you copy a component, you are prompted whether you want to include its associated symbols; you would typically respond **Yes**. When you copy a symbol, no components are included in the copy.

Also, when copying items from one library to another, you can choose to preserve items in the destination library or overwrite them. And when copying components, you can choose to copy the corresponding symbols.

The dialog allows you to select source library and symbol name as well as destination library and destination name.

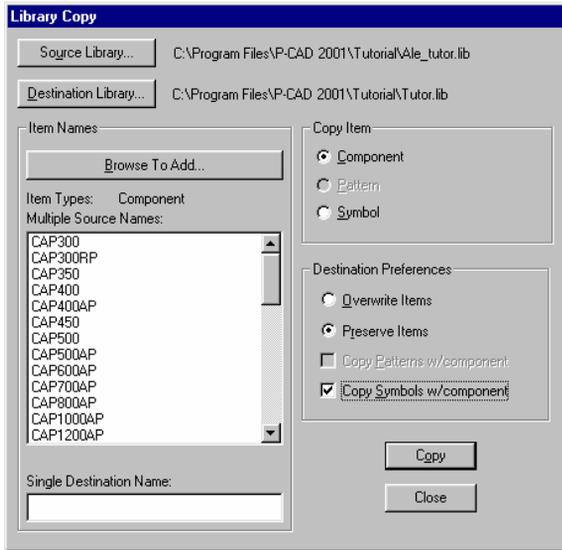
The source library and destination library that are used with **Copy** will remain current if you re-invoke the command during the same P-CAD session.

If you are copying a component or symbol but are not changing its name, you can leave the Destination Name box blank.

## Copying Symbols/Components

To copy one or more symbols or components, follow these steps:

1. Choose **Library » Copy**. The *Library Copy* dialog appears as shown in the following figure:



2. Click **Source Library**. The *Library Select* dialog appears. The *Library Select* dialog is similar to the *File Open* dialog.
3. Select the source library. Click **Open**. Notice that the source library you selected appears in the *Library Copy* dialog.
4. Click **Destination Library**. The *Library Select* dialog appears again.
5. Select the destination library. Click **Open**. The *Library Copy* dialog displays the paths and filenames of the source and destination libraries you selected.
6. In the Copy Item frame, select the type of item you want to copy. The available items are Component and Symbol. For the item type you select, the names of the available items appear in the Multiple Source Names list box.
7. Select the item(s) to copy by using the standard Windows™ selection key combinations. You can select either a single item, a group of items, or all the items in the list. The table below explains how to select items for copying:

Do this:	To select this:
Left mouse button	A single item. If you select a single item, you can enter its destination name in the Single Destination Name box.

SHIFT+Left mouse button	All items between the last selected (highlighted) item and the item you click on.
CTRL+Left mouse button	Add items to or subtract items from the selection.

8. You can also click the **Browse to Add** button to add single items to the your selection. To do this, click **Browse to Add**. Then select the desired item, then click **OK**. The program highlights the selected item in the Multiple Source Names list box.
9. In the Destination Preferences section of the dialog, indicate whether to overwrite existing items in the destination file, preserve existing items, and copy patterns and symbols with a component. The last two check boxes are only available if you're copying components.
10. Click **Copy**. P-CAD Library Executive copies the selected objects from the source library to the destination library.
11. Click **Close** to exit the *Library Copy* dialog.

## Library Delete

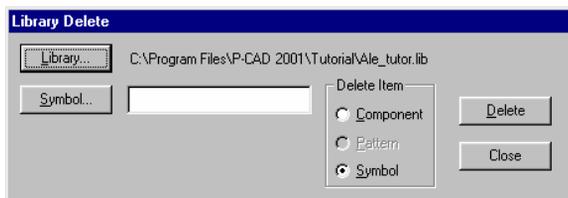
Choose the **Library » Delete** command to delete a library item or its alias.

This command deletes the item in name only, if it has aliases. The alternate names (aliases) still exist unless you delete them. If the item has only one name and no aliases and you delete it, then the item itself is deleted from the library. Use the **Library » Alias** command to check whether an item has aliases.

**IMPORTANT:** If you delete a symbol, then all of the components in the library that reference that symbol have no graphics, and therefore cannot be placed. Typically you would want to delete a symbol alias only, which is not dangerous unless a component used a symbol alias.

### Deleting from a Library

1. Choose the **Library » Delete** command to open the following dialog:



2. In the Delete Item frame, select the **Component** or **Symbol** radio button. The Pattern radio button is shaded in Schematic.
3. Click the **Library** button. The *Library Select* dialog is displayed, from which you can select the library in which you want to delete an item.

4. The library you selected in Library Select appears in the *Library Delete* dialog.
5. Click the item button (**Symbol** or **Component**) and the items within the displayed library are listed in the *Library Browse* dialog. Select one and it is listed in the *Library Delete* dialog.
6. Click the **Delete** button and the item box becomes blank. You can continue to delete items from the same library.
7. Click **Close** to exit the *Library Delete* dialog.

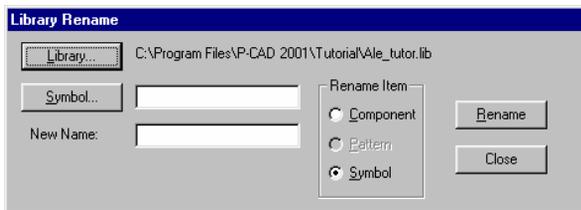
## Library Rename

Choose the **Library » Rename** command to rename a symbol or a component.

If you rename a symbol, then all of the components in the library that reference that symbol by the original name have no symbol graphics, and cannot be placed. If you want to use a different naming convention for a symbol, then create an alias for the symbol (**Library Alias** command) and use that alias name. Likewise for components: if you want to use a different naming convention, using aliases is much safer than renaming.

### Renaming a Symbol/Component

1. Choose the **Library » Rename** command to open the following dialog:



2. Select the Rename Item type (**Component** or **Symbol** radio button) to specify the item you want to rename. Pattern is shaded in Schematic.
3. Click the **Library** button to display the *Library Select* dialog, where you can choose the library to access.

The *Library Select* dialog is the standard Windows™ *File Open* dialog. The library you select in *Library Select* is displayed in the *Library Rename* dialog.

4. Click the **Symbol** or **Component** button and the items within the displayed library are listed in the *Library Browse* dialog. Select one and it is listed in the *Library Rename* dialog.
5. In the New Name text box, type the new name of your item, then click **Rename**. Both the old and new name disappear if the rename action is successful. Then you can continue renaming items in the same library.
6. Click **Close** to exit the *Library Rename* dialog.

## Library Setup

---

Choose the **Library » Setup** command to open libraries from which you can access parts.

The **Place » Part** command uses the open library list to place parts. The **Library » Symbol Save As** command also uses the open library list.

When you want to place a part, the library file where the part resides must be open. You can open up to 100 libraries at one time.

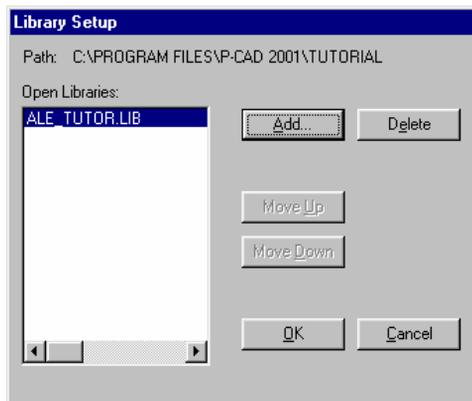
### Drag and Drop File Load

You can use the drag and drop method for opening library files (.lib) from the File Manager or other Windows™ file maintenance utilities. Drag the .lib icon(s) onto your Schematic icon or workspace.

### Setting Up a Library

To setup a library, follow these steps:

1. Choose the **Library » Setup** command to open the following dialog:



2. The dialog lists libraries that are already open in the Open Libraries list box.
3. To add another library to the list, click **Add** to open the *Library File Listing* dialog. From there you can gain access to the library directory to select a library file.

When you select a file from the *Library File Listing* dialog (and click **OK**), that filename appears in the **Open Libraries** list box in the *Library Setup* dialog.

4. To rearrange the list order, select a library name and click the **Move Up** or **Move Down** buttons.
5. To remove a library from the list, select the library name from the Open Libraries list box and click **Delete**.

- Click **OK** and the libraries that you have specified are now open and accessible for part placement or saving symbols.

In order to save a symbol or a component, you need to setup the library in advance. The list of open libraries is saved to the `Sch.ini` file and therefore saved for subsequent P-CAD sessions.

Many commands start looking for components in the first open library. To ensure your component is found in the correct library, open custom libraries first or move them to the top of the list.

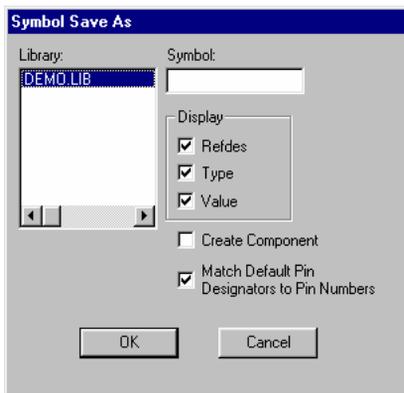
## Library Symbol Save As

Choose the **Library » Symbol Save As** command to save a symbol to a library. Optionally creates a component based on the symbol (you can later attach the symbol to a component using P-CAD Library Executive), or match the default pin designators to the pin numbers. You must have a library already open to save a symbol to it.

### Saving a Symbol

To save a symbol to a library, follow these steps:

- Use the block select function (**Edit Select** mode, draw a selecting rectangle) to include all objects you want to be included in your symbol. At a minimum, a symbol must include a RefDes attribute, a Type attribute, and a reference point. All pins must be consecutively numbered, starting with 1. No duplicate default pin designators are allowed except for blank.
- Any pins numbered **0** will be automatically renumbered for you.
- While the objects are selected in your workspace, choose **Library » Symbol Save As** to open the following dialog:



- From the Library list, choose the library that you want to save the symbol to, then specify a symbol name.

5. In the Display frame, select the appropriate check boxes to set the default display characteristics for the display of reference designators, type, and value.

To automatically create a component that corresponds to the symbol, select the **Create Component** check box.

6. To set the default pin designators to match the pin numbers, select the **Match Default Pin Designators to Pin Numbers** check box. You must confirm that you want to overwrite existing data if default pin designators had previously been assigned.
7. Click **OK**. The symbol is saved to the library.
8. If the **Create Component** check box is selected, the *Save Component As* dialog appears:



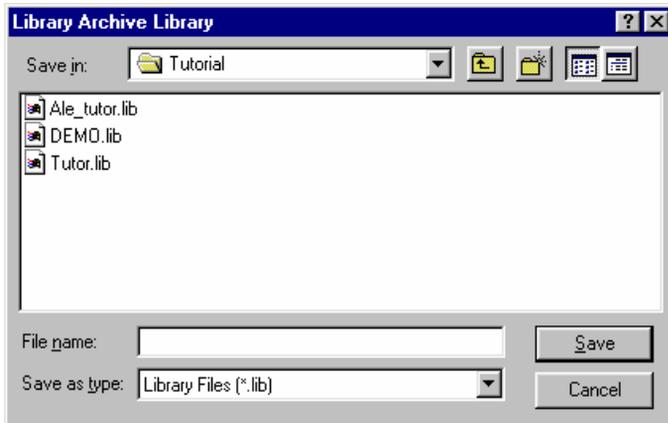
9. Type a component name in the Component Name box and select a component type. Make additional modifications using the Library Executive. See the *P-CAD Library Executive User's Guide* for more information.
10. Click **OK**.

## Library Archive Library

Choose the **Library » Archive Library** command to copy the component and symbol information from the open libraries for components used in the active design file. Saving the component information in one library file can be extremely helpful, especially for large designs, to keep the design component information in a single, readily accessible location. This eliminates the need to store the complete set of libraries used to create the design.

The definitions for each component in the current design must reside in at least one open library.

1. When you choose **Library » Archive Library**, the following file search dialog appears:



2. The Save In list box shows the current folder and any files in that folder. In the File Name box, enter or select a file with the file name extension specified in the Save As Type list box.
3. If the folder you want is not displayed in the Save In list box, move through the directory tree to select the proper folder.
4. Type a new library file name in the File Name area or select one from the list displayed in the Save In area.
5. Click **Save**.

If the library already exists, and you want to replace the data in the file, you must confirm that you want to overwrite the file.

The results of the **Archive Library** command are displayed in an output file named `ArchiveLibrary.err` located in the same directory as the destination library. This file contains errors, warnings and informational messages logged during the archive process.

## Utils Commands

### Using the Utils Commands

Use the commands in the **Utils** menu to renumber pins and reference designators, run electrical rule checking, record Engineering Change Orders, generate netlists, start other P-CAD or third-party program, and gain access to various web sites.

### Utils Renumber

Choose **Utils » Renumber** to manually assign sequential pin numbers and default pin designators to pins, and to automatically renumber reference designators on parts. You must be in Select mode to choose this command.

For reference designators, renumbering is automatic across all sheets.

### Renumbering Reference Designators

1. Choose **Utils » Renumber** to open the *Utils Renumber* dialog:

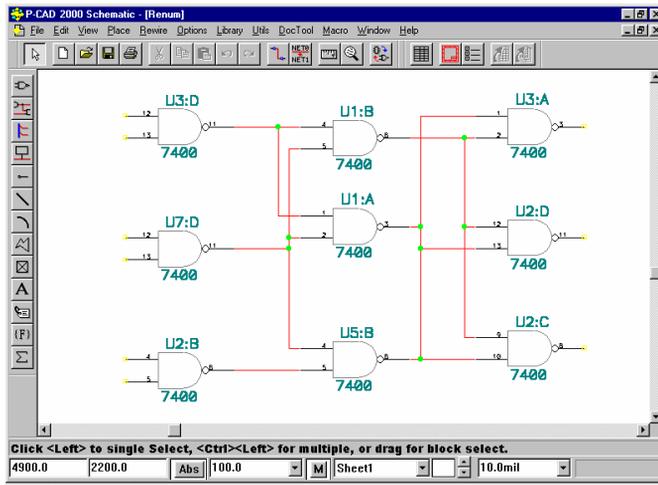


2. In the Type frame, choose **RefDes**. The controls in the Direction and RefDes frames become available and the box labels change to **Starting RefDes** and **Increment Value**.

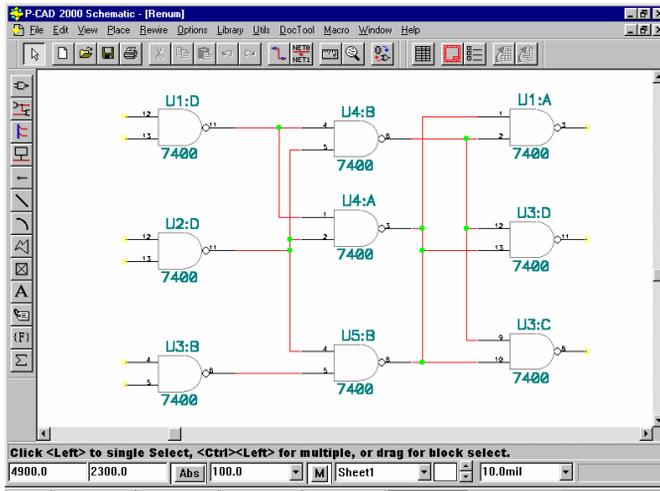
3. In the RefDes frame, choose one of these buttons:
  - Choose **Auto Group Parts** to minimize the number of components by renumbering parts so that they fill existing components.
  - Choose **Keep Parts Together** to renumber components, but leave every component you placed, regardless of the number of parts of that component you placed.
4. In the Direction frame, choose one of the buttons:
  - **Top to Bottom**: Renumbers reference designators so that lower numbers appear at the top of the design and higher numbers appear at the bottom.
  - **Left to Right**: Renumbers reference designators so that lower numbers appear to the left of the design and higher numbers appear to the right.
5. Click **OK**.

### Examples

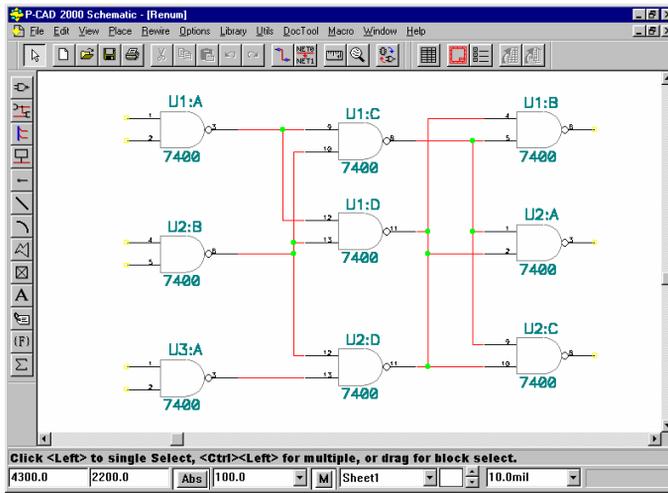
The following examples show you how the renumbering options work. The illustration below shows you the design before renumbering:



If you chose the **Left to Right** and **Keep Parts Together** buttons in the Direction and RefDes frames of the *Utils Renumber* dialog, your design should now appear as shown in the following figure:

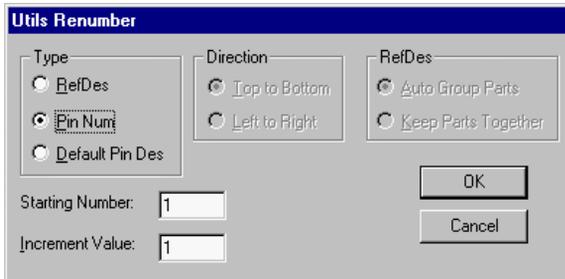


If you choose the **Top to Bottom** and **Auto Group Parts** options in the Direction and RefDes frames of the *Utils Renumber* dialog, your design should now appear as follows:



## Renumbering Pins

1. Choose **Utils » Renumber** to open the *Utils Renumber* dialog.
2. In the Type frame, choose **Pin Num**. The controls in the Direction and RefDes frame become shaded and the box labels change to Starting Number and Increment Value, as shown in the following figure:



3. Click **OK** to begin pin renumbering manually. You are in a temporary mode of assigning numbers, so every left-button click on a pin renumbers it.

For example, the first pin you click on would be number 1 (if Start Value was specified as 1), the second pin number 2 (if the Increment Value was specified as 1). As you click on a pin while in the Renumber mode, it highlights to show that a new number has been assigned.

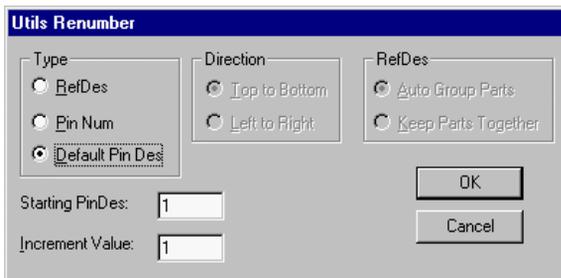
The status line shows the next pin number every time you number a pin.

You can use the unwind feature (**BACKSPACE**) to reverse the renumbering process.

4. Right-click or press **ESC** to end the renumber process.

## Renumbering Default Pin Des

1. Choose **Utils » Renumber** to open the *Utils Renumber* dialog.
2. Choose the **Default Pin Des** button. The controls in the Direction and RefDes frame become shaded and the box labels change to Starting PinDes and Increment Value, as shown here.



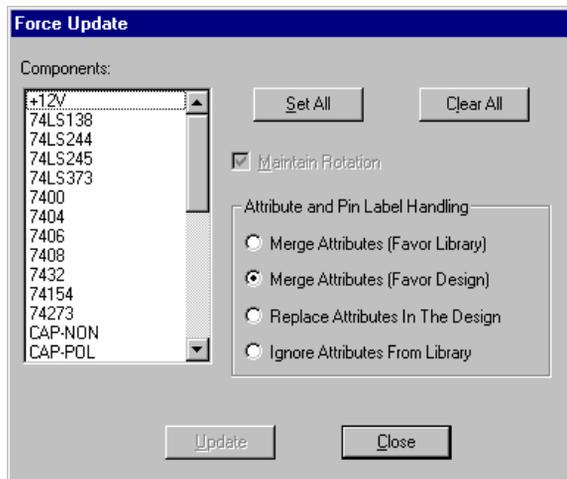
3. Click **OK**. Then click on the pins in the order in which you want the default pin designators to be numbered. The status line shows the next pin number every time you number a pin. You can use the unwind feature (**BACKSPACE**) to reverse the renumbering process.
4. Right-click or press **ESC** to end the renumber process.

## Utils Force Update

Choose **Utils » Force Update** to replace all components of a given type in your design with the first component of that type in an open library. This is especially useful when you have modified a component in the Library Executive and want to place the modified component in a Schematic design that has the original one.

For example, you change the number of pins in a component in the library. When you try to place the modified component into the design you will receive an error. Choose the **Utils » Force Update** command to update all occurrences of that component in the design to reflect the changes. When all existing components of that type have been changed in the design, you can place more of the newly modified component into the design without incurring an error.

When you choose the **Utils » Force Update** command, the *Force Update* dialog appears.



Select a component type, or click the **Set All** button to select all component types. Then, click the **Update** button. To cancel your selections, click the **Clear All** button. The command looks in all open libraries in the order they are listed in the Library Setup list for the replacement components.

Select the **Maintain Rotation** check box to maintain the rotation of any rotated components. It does not maintain rotations for components in designs loaded from Tango Series II.

In the Attribute and Pin Label Handling frame, choose the method used to update component attributes in your design with the same type of components from the library. Remember that there are rules used to determine an attribute value.

- **Merge Attributes (Favor Library):** This merge option combines library and design component attributes in the component being used in the design. It favors component attributes from the library over the component attributes of the same name in the design.

The following rules determine which attributes and values are kept in the design component:

- If an attribute exists in the library component and not in the design component, the library attribute and its value is copied into the design component.
- If the same attribute exists in both the design component and the library component with different values, the value of the library component attribute will replace the value of the design component attribute.
- An existing design component attribute, which has no matching library component attribute, is retained, unchanged, in the design component.
- **Merge Attributes (Favor Design):** Again, this merge option combines library and design component attributes in the component being used in the design. This merge option favors component attributes in the design over the component attributes of the same name in the library.

These rules decide which attributes and values are kept in the design component:

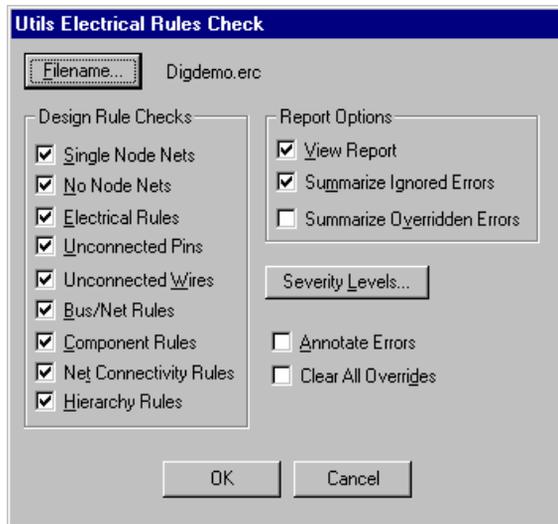
- If an attribute exists in the design component and not in the library component, the design attribute and its value is kept, unchanged, in the design component.
- If the same attribute with different values exists in both the design component and the library component, the value of the design component attribute takes precedence and is retained in the design.
- **Replace Attributes in The Design:** Choose this option to replace all of the attributes and their values in the design component with those defined in the library component. Current design attributes are removed from the component.
- **Ignore Attributes From Library:** Keeps attributes in the design and ignores those in the library. This feature gives you more control over what to update in the design if you want to preserve any specific attribute modifications.

## Utils ERC

---

Choose the **Utils » ERC** command to perform electrical rules checking on your design. Schematic then produces a report listing design errors. This process looks for design errors such as single node nets and output pins connected to output pins.

When you choose **Utils » ERC**, the following dialog appears:



- **Filename button:** Click this button to open the *Electrical Rule Check Report* dialog (a Windows™ common *File Save As* dialog). From this dialog choose an ERC report file to which you can save the report information.
- **Design Rule Checks frame:** Select a check box to include that item on the ERC report:
- **Single Node Nets:** Reports all nets with only one node.
- **No Node Nets:** Reports all nets with no nodes.
- **Electrical Rules:** Reports pins of incompatible types connected together, for example, two output pins connected together or an output pin connected to a power pin.
- **Unconnected Pins:** Reports all pins that are unconnected to other pins. This includes pins that are not connected to anything at all.
- **Unconnected Wires:** Reports all wires that are unconnected (floating).
- **Bus/Net Rules:** Reports on nets only referenced once in a bus (i.e., a wire goes into a bus, but doesn't come out).
- **Component Rules:** Reports on all components that are on top of other components.
- **Net Connectivity:** Lists conditions that are ambiguous regarding power and ground nets. This may occur when the nets have been renamed or merged.

There are three possible warnings that can occur on the report:

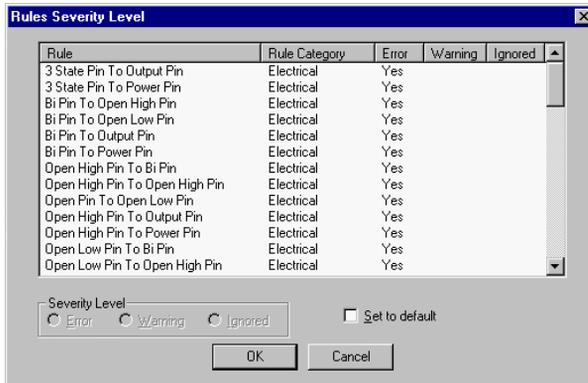
- A power symbol is connected to a net causing the net name to be ambiguous.

- A component's global hidden pin is attached to a net, which is different from its default net name. (For example, the net merge of a hidden pin with a net can cause a hidden pin to no longer be connected to its default net name.)
- Warning nets (X) and (Y) have been merged as (X), but net (Y) still exists. This warning is designed to let you know you must use the Edit Nets function to rename the nets and any connections that may have been added after the merge occurred.
- **Hierarchy Rules:** Reports on module and link errors. Each module must have a link attribute with pins of matching electrical type and quantity. These module and link pins must be connected. All link pins must be placed within the design and have an attribute referencing a valid link component.
- **Report Options frame:** Select a check box to enable one of these features:
- **View Report:** Select this check box to display the ERC report file when the ERC is complete. When this check box is cleared, the ERC report file is generated, but not displayed.
- **Summarize Ignored Errors:** When the severity level of an error has been set to Ignored, the error annotation is not displayed in the design. You can show the summary of these errors by selecting this check box.
- **Summarize Overridden Errors:** When a design has a number of errors that you do not wish to see again, such as unconnected pins, and you have overridden their display using the Edit Override command, you can include a count of these errors by selecting this check box.
- **Annotate Errors:** Select this check box to display ERC error indicators in your design. These indicators can then be selected for viewing of error information. The error information is determined by the other options that you enable in rule checking.

To view the error associated with an error indicator, select it and choose **Edit Properties**. The selection criteria for ERC error indicators (for block selecting) is determined by the Options Block Selection command.

You can also define the size of ERC error indicators in a design in the **Miscellaneous** tab of the *Options Display* dialog. For information, see *Options Display* (page 297).

- **Clear All Overrides:** When errors in the design have been overridden, and you want to clear or delete them from the design, enable the **Clear All Overrides** option.
- **Severity Level:** Click the **Severity Level** button to open the *Rules Severity Level* dialog shown in the following figure:



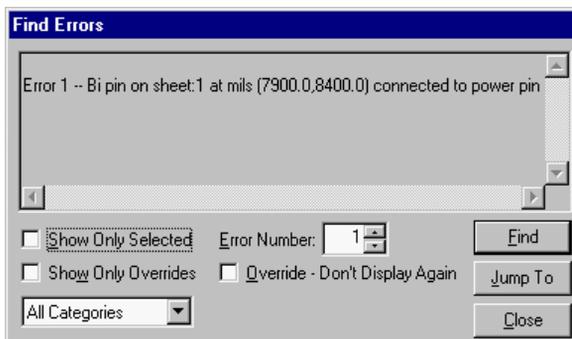
In this dialog you can apply a severity level of **Error**, **Warning** or **Ignored** to individual rules. Rules marked **Ignored** are not listed in the ERC report, but you can summarize the number of **Ignored** rules that have violations if you enable the **Summarize Ignored Checks** option in the *Utils Electrical Rules Check* dialog.

To change the severity level, select one or more rules and click the radio button next to the desired severity level. You can remove the changes you’ve made and return to the original settings by clicking the **Set to default** box. Click **OK** to return to the *Utils ERC* dialog.

When you are finished setting up the ERC options, click **OK** to begin the rule checking process. Existing error indicators are cleared when you choose **Utils » ERC**.

## Utils Find Errors

Choose the **Utils » Find Errors** command to open the *Find Errors* dialog.



If you have not run the *Utils Electrical Rules Check*, then the *Find Errors* dialog displays a message informing you that there are no ERC error points. When you have run an ERC command, you have access to all the errors through the *Find Errors* dialog.

Information contained in the *Find Errors* dialog is as follows:

- **Categories:** The drop-down list displays the rule categories. You can view the errors for one category by selecting it from the list. To select all the categories choose **All Categories**.
- **Description:** Displays the error number along with a brief explanation for the error.
- **Error Number:** In the Error Number box you can type the number of the error you want to see, and click the **Find Specific** button to go directly to it. If you want to scroll through the errors sequentially, use the up and down arrows next to the box.
- **Show Only Selected:** If you have block selected an area in the design, and you only want to view the errors in that area, check the **Show Only Selected** option.
- **Jump To:** When you find an error in the report that you want to locate in the design, click the **Jump To** button. The error finder positions the cursor in the center of the error indicator. If you have block selected an area in the design the **Jump To** button positions the cursor inside that area and retains the selections.
- **Override – Don't display this error again:** As you display each error in the design, individual errors can be removed from the error display by checking the **Override – Don't display this error again** option. The overridden errors can be summarized in the reports by enabling the **Summarize Overridden Errors** option in the *Utils Electrical Rules Check* dialog.
- **Show Only Overrides:** Provides access to scroll through only those errors to which an override has been applied.

The *Find Errors* dialog remains on the screen until you click the **Close** button. As long as there are error indicators in the design, you can retrieve the error information in the *Find ERC Errors* dialog with the **Utils » Find Errors** command.

## Utils Record ECOs

Choose the **Utils » Record ECOs** command to record Engineering Change Orders (ECOs). When you choose this command, the following dialog appears:



In the ECO Recorder frame, choose the appropriate radio button to turn the ECO recorder on or off. As a shortcut for choosing this command, click the **ECO** button on the toolbar.

If there are pending ECOs, you are prompted when a design is saved on whether to append the pending ECO to the current ECO file.

## Types of ECOs

The following types of ECOs can be recorded:

- RefDes change (Was-Is).
- Net and net class name changes.
- Additions, deletions, and modifications of components and parts.
- Component swaps (Replace).
- Additions and deletions of nets, net classes and class to classes.
- Net additions to and deletions from net classes.
- Additions and deletions of net nodes.
- Additions, deletions, and modifications of attributes (i.e., in nets, net classes, class to classes, etc.).
- Pin and gate swap changes.

The format of the ECO file is determined by the setting in the *Options Configure* dialog. Full ECO format files have an .eco extension, and Was/Is ECO files have a .was extension.

When using the ECO feature to import NetNodeAdds into Schematic, the wire stubs that are auto-created may be off-grid. If this happens, you can place another wire on-grid so that it overlaps the existing wire stub; the two wires will be merged.

Was-Is ECO events record only reference designator changes, not part number changes. If the ECO recorder is on and is recording only Was-Is events, the **Utils » Renumber** command should be used with the **Keep Parts Together** option to ensure that the generated ECOs are correct.

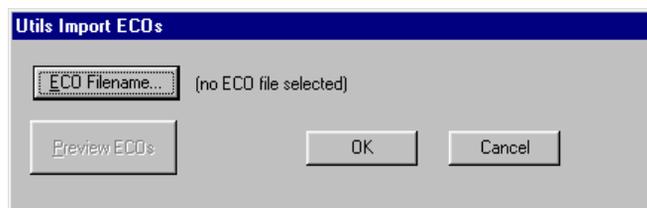
## Utils Import ECOs

---

Choose the **Utils » Import ECOs** command to import an ECO file and to apply the ECO changes to the current design file. The ECO file is created in P-CAD PCB to capture post-schematic changes made to your design.

If a component attribute is modified through ECOs, it is updated in all parts of the component.

When you choose this command the following dialog appears:



## ECO Filename

1. In the *Utils Import ECOs* dialog, click the **ECO Filename** button. The following *ECO Filename* dialog appears:

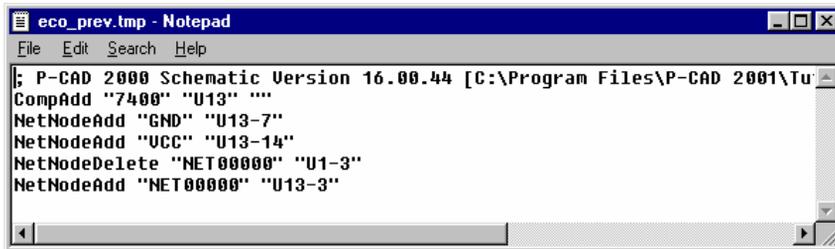


2. Navigate to the file you want to open or type the name of the file to open in the File Name text box.
3. Click **Open** to return to the *Utils Import ECOs* dialog.

.ECO files are assumed to be full ECO format; .WAS files are assumed to be *Was/Is* format.

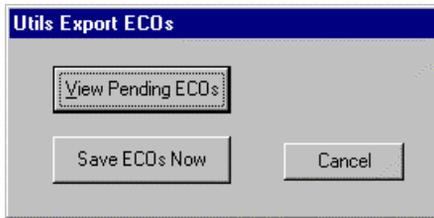
## Preview ECOs

When you select an ECO filename and then click the **Preview ECOs** button, you can view a list of ECOs before you import them. When you click **Preview ECOs**, any ECOs appear in Windows™ Notepad:



## Utils Export ECOs

Choose the **Utils » Export ECOs** command to save ECOs to the ECO file at any time, without saving the design file. If there are pending ECOs, the following dialog appears when you choose this command:



The **Utils » Export ECOs** command is grayed and unavailable when no ECOs exist.

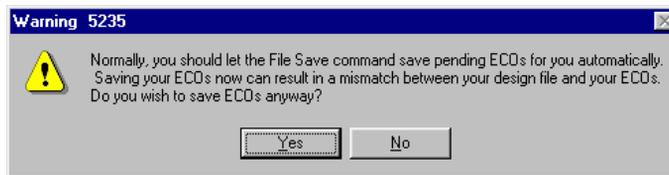
## View Pending ECOs

When you click the **View Pending ECOs** button, you can view pending (outgoing) ECOs, which are still stored in memory. The pending ECO data are written to a temporary ASCII file and displayed in Windows™ Notepad. The format displayed is either full or Was/Is, depending on the setting in the *Options Configure* dialog.

## Save ECOs Now

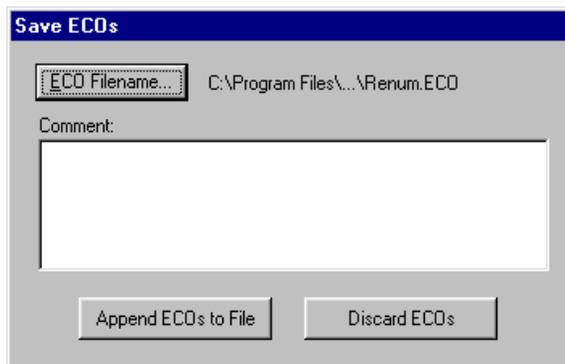
To save pending ECOs, follow these steps:

1. Click the **Save ECOs Now** button in the *Utils Export ECOs* dialog. The following warning message appears:



It is important to remember that if you save ECOs without saving the design, your file and the ECOs may not match. That is, the ECOs might not reflect the current state of the design.

2. To continue, click **Yes**. The *Save ECOs* dialog appears:



- The ECO filename appears at the top of the dialog. It is the last used ECO file. To change it, click the **ECO Filename** button and the following dialog appears:



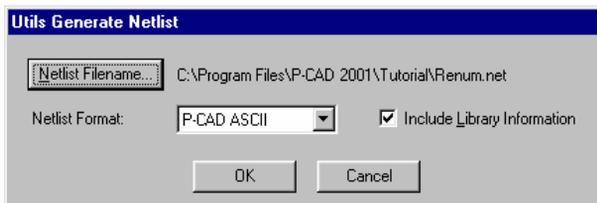
- The *ECO Filename* dialog is a standard *File Open* dialog. Type, or select from the list, the name of the file you want to open in the File name box. Click **OK** to return to the *Save ECOs* dialog.
- Full ECO files must have an .eco extension, and Was/Is files must have a .was extension.
- In the Comments box, type any comments that can help document the ECOs.
- To append ECOs to the ECO file, click the **Append ECOs to File** button.
- To discard ECOs, click the **Discard ECOs** button. Once discarded they cannot be recovered.

## Utils Generate Netlist

---

Choose the **Utils » Generate Netlist** command to create a netlist from the combination of connections you have established (or loaded) in a design file.

- With your design file opened, choose the **Utils » Generate Netlist** command. The following dialog appears:



- To choose a netlist file, click the **Netlist Filename** button. This will display the *Netlist File* dialog, which is a standard Windows™ *File Open* dialog.

3. Select the destination format in Netlist Format combo box to specify the destination format. Available formats are:
  - P-CAD ASCII
  - Tango
  - FutureNet Netlist
  - FutureNet Pinlist
  - PCAD
  - EDIF v2.0.0
  - Pspice
  - Xspice.
4. Select the **Include Library Information** check box to write an optional library section to the netlist.
5. Library information is read by P-CAD PCB but not processed; it is merely informational. The Library section may be used to create a P-CAD library, however, by removing the non-library information from the file and using the Library Executive to translate the ASCII form of the Library into a P-CAD binary library.
6. Click **OK** in the *Netlist Generate* dialog to create the netlist with the filename and netlist format you have specified.

## Utils Rename Net

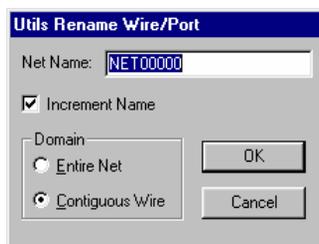
---

Choose the **Utils » Rename Net** command to input a seed net name and an increment value, and rename one or more nets by selecting one net at a time.

### Renaming a Net

To rename a net:

1. Choose the **Utils » Rename Net** command or click the **Rename Net** button in the toolbar.
2. In your design workspace, click a wire or port. The *Utils Rename Wire/Port* dialog appears:



3. Use this dialog to set your initial selection for net renaming. Depending upon the selected object and which options you set, you can rename a net, split a net, or merge a net.
4. Set the seed name for renaming the nets and indicate whether or not to increment the subsequent names.
5. To split a net, choose the **Contiguous Wire** button.
6. A net can be split if the selected wire or port is part of a subnet and there are other subnets in that net. The net is split with the selected subnet getting the new name and the other subnets keeping the original name.
7. A subnet can be split from one net and merged with another, if the net name already exists and is global. If the net is split, the selected subnet is merged with the existing net, while the other subnets retain their original name.
8. To rename an entire net, you must either select the **Entire Net** option or select a net that is comprised of a single contiguous subnet.

If the new name does not already exist within the design, the entire net is renamed. If you attempt to rename a net with the name of an existing net, and one of the nets is not global, an error message appears. You must confirm that you want to autoplacement ports to make both nets global by clicking the **Yes** button. The nets are merged and renamed.

9. Click **OK** to return to your design. Each time you make a selection, the net is renamed using the values set in the dialog.

## Jumper Pins

Jumpers behave as if all of the jumper pins are connected. In Schematic, pins are ported to force them to belong to the appropriate net.

## Utils Resolve Hierarchy

---

Once you create a module and place it in your design, you must resolve the hierarchy before performing ECOs or generating netlists.

To resolve a hierarchy, follow these steps:

1. Choose the **Utils » Resolve Hierarchy** command. A message box appears, asking you if you want to save the design before proceeding. Resolving hierarchies alters your design, so you should save it before proceeding.
2. Click **Yes** to save your design, **No** to proceed without saving your design, or **Cancel** to abort the command. Clicking **Yes** displays the Save As dialog, which prompts you for a filename for saving the design.
3. Select the desired design name, or enter it in the File Name field.
4. Click **OK**. P-CAD Schematic resolves the hierarchy. In the case of a complex hierarchy, the program copies the definition sheet for each module that references it. In this process, ports are added to module and link pins.

If a problem occurs, P-CAD Schematic displays an error message indicating that the hierarchies were not resolved and explains why the hierarchies were unresolved. In this case, you can click Yes to view the errors in Notepad.

One important condition that causes an error message to appear is called a passthrough condition. A passthrough condition occurs when two links are connected together by wires, ports, or a combination of the two.

## Utils Module Wizard

---

P-CAD Schematic supports hierarchical schematic designs. The **Utils » Module Wizard** command makes it easier for you to create and manage complex designs.

With a hierarchical design, you can reference several sub-circuits from the main, or top-level, schematic design.

You create a hierarchical design by placing a module, or black box, on your design. P-CAD Schematic represents this module as a component with a component type MODULE. A module is an abstract representation of the sub-circuit to which you are referring. Each module has pins representing the interface to the sub-circuit. These pins connect to the inputs and outputs of the sub-circuit.

Each module also requires a link, which connects the module to its definition. A definition, or defined sheet, is a sheet that contains the actual circuitry of a module. P-CAD Schematic represents a link as a component with a component type LINK. Both the module definition and its link must be on the same sheet, and a sheet can only contain one module definition.

Your design can have simple or complex hierarchies. A simple hierarchy has one module referencing one link, while a complex hierarchy has multiple modules referencing a single link.

You can place modules and links the same way you place any other part.

P-CAD Schematic stores modules and links as components in a component library, complete with type names, reference designators, and user-defined attributes.

After placing modules and links, you must resolve the hierarchy. This converts a complex hierarchy to a simple hierarchy so that you can generate a netlist. Once you resolve a hierarchy, you can import and export ECOs, generate netlists, and save your schematic to a PDF file.

You can ascend and descend the unresolved designs using the Ascend and Descend commands found on the right mouse button pop-up menu.

To create and use a hierarchy in your design, you must:

- Create a module and its link. You can either create a new module and its link or reuse an existing module.
- Place the module in the schematic design.
- Resolve the hierarchy.

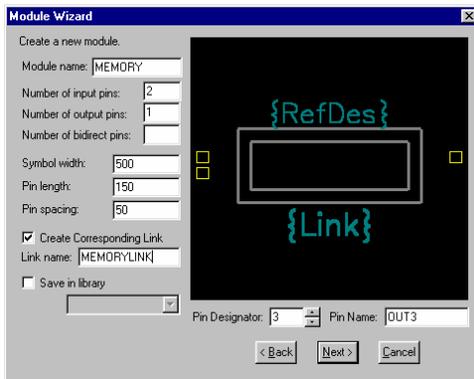
## Creating a New Module and its Link

This section explains how to create a module and its link, then place the module in a schematic design:

1. Choose **Utils » Module Wizard**. The *Module Wizard* dialog appears:



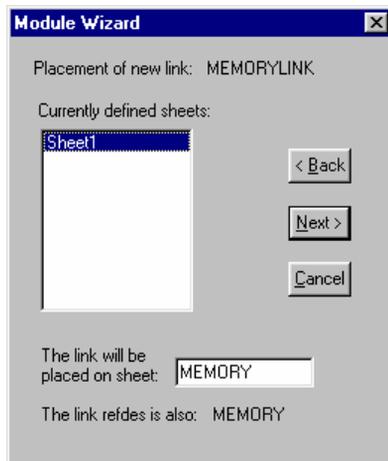
2. Choose the **Create a new module and its link** button. Then, click the **Next** button.
3. Another *Module Wizard* dialog appears, as shown below.



The *Module Wizard* dialog allows you to specify module and link parameters. The preview window displays the module symbol. As you define the module's parameters, the preview window reflects these changes in the module's appearance. For instance, if you change the number of input pins from two to three, this change appears in the preview window.

- **Module name:** The name of the module.
- **Number of input pins:** The number of input pins for the module symbol.
- **Number of output pins:** The number of output pins for the module symbol.
- **Symbol width:** The width (in mils) of the module symbol.
- **Pin length:** The length (in mils) of the input and output pins.
- **Pin spacing:** The spacing (in mils) between each input and output pin.

- **Create Corresponding Link:** Indicates whether or not to create a corresponding link for the module. If you don't select this check box, the program asks you to select a link already placed on the schematic.
  - **Link name:** The name of the link between the module and its definition.
  - **Save in Library:** Indicates whether or not to save the module in a library.
  - **Library:** The library to which you are saving the module and link.
  - **Pin Designator:** The default pin designator that the Module Wizard automatically assigns.
  - **Pin Name:** Lets you change the default pin names. While you are in this box, if you press **Enter**, the program automatically updates the pin name and the box automatically cycles.
4. Enter the module and link parameters. Then, click the **Next** button. The following *Module Wizard* dialog appears:



This dialog displays the following information:

- **Placement of new link:** The name of the new link.
- **Currently defined sheets:** The sheets currently defined in the design. Defined sheets have three restrictions:
  - A defined sheet can contain circuitry for only one module.
  - The module's circuitry must be contained within one sheet.
  - If you create the defined sheet with the Module Wizard, its name is the same as the module's name.
- **Link will be placed on sheet:** The name of the sheet where the module links are to be placed. By default, the sheet name is the same as the module name.

- **The link refdes is also:** The link reference designator is automatically set to the sheet name.
5. Enter the link definition information for the target sheet and the Link RefDes. Then, click the **Next** button.

The module placement dialog appears, indicating that you are about to place the module you just created.



You can click **OK** to place the module, click **Back** to go back to the previous dialog, or click **Cancel** to abort. If you click **OK**, P-CAD Schematic returns you to the design in Place Part mode, meaning that you can now place the module in your design.

6. Select the desired location for your module, then click the left mouse button to place it.

## Reusing an Existing Module

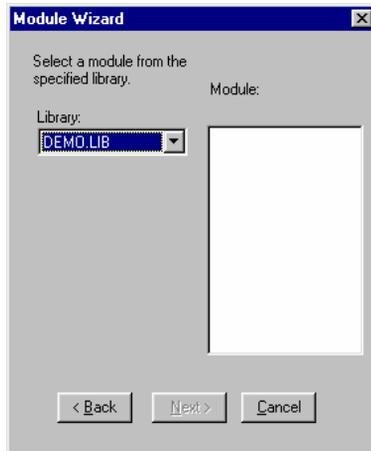
You can reuse an existing module if you don't want to create one from scratch. To reuse an existing module, follow these steps:

1. Choose the **Utils » Module Wizard** command. A *Module Wizard* dialog appears, as shown below:



2. Choose the **Reuse an existing module** button. Then, click the **Next** button.

Another dialog appears, prompting you to select the library containing the module and the module you wish to place on the schematic.

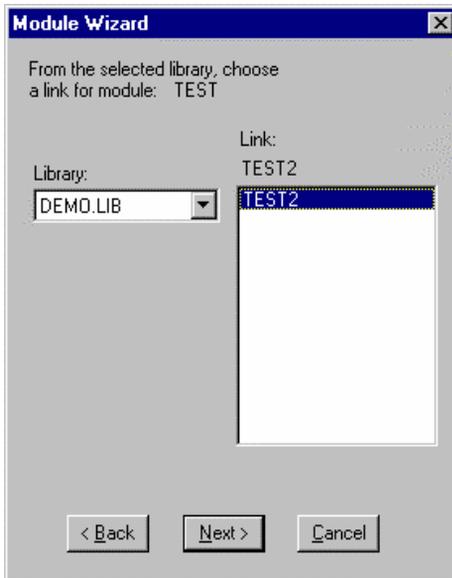


3. Select the library containing the module, then select the desired module.
4. Click **Next**. Another dialog appears, asking you if you want to place a new link, or reference an existing link:

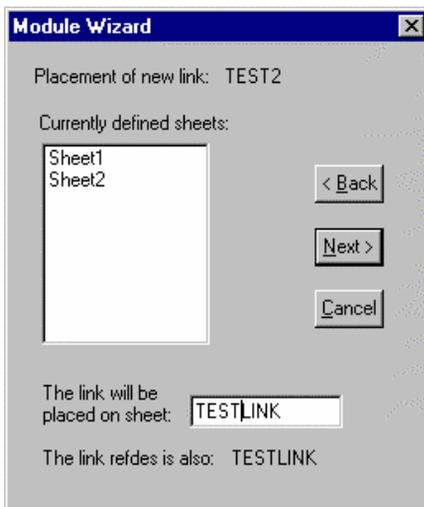


5. Choose the **Place a new link** button. Then, click the **Next** button. Another dialog appears, prompting you to select the library containing the link and the link itself.

Alternatively, select reference an existing link. P-CAD Schematic displays a dialog prompting you for a reference designator for the link. Select the desired reference designator from the list, or type the reference designator of an existing link, then click **Next**. Now skip to step 9.



6. Select the library containing the link from the Library list box, then select the desired link from the Link list box.
7. Click **Next**. Another dialog appears. The list on this dialog displays the currently defined sheets. However, P-CAD Schematic places the link on a new sheet unless you specify the name of an existing sheet in the text box.



8. Make any desired changes to the default sheet name displayed in the text box, then click **Next**. The module placement dialog appears, indicating the reference designator of the module you are about to place.



9. Make any desired changes to the reference designator. Then click **OK** to place the module, click **Back** to go back to the previous dialog, or click **Cancel** to abort.  
If you click **OK**, P-CAD Schematic returns you to the design in place part mode, meaning that you can now place the module in your design.
10. Select the desired location for your module, then click the left mouse button to place it.

You can create your own default link symbol by saving a single pin symbol with the name LINK\_PIN into your custom library. When the Module Wizard places a link, it uses that symbol.

## Utils Shortcut Directory

---

When you installed P-CAD Schematic, a sub-directory called ShortcutDirectory was created in the P-CAD directory. The ShortcutDirectory contains a list of web addresses for semiconductor manufacturers.

Choose the **Utils » Shortcut Directory** command to open the ShortcutDirectory in Windows™ Explorer. Select the desired address and choose the **Open** command from the right mouse menu, or double click on the address.

Shortcuts to any web site can be added in the ShortcutDirectory.

## Utils P-CAD PCB

---

If P-CAD PCB is installed on your computer, choose the **Utils » P-CAD PCB** command to start P-CAD PCB. Refer to your *P-CAD PCB User's Guide* for additional information.

- If P-CAD PCB is not running, choose this command to start P-CAD PCB.
- If PCB is already running, choose this command to make P-CAD PCB the active program.

## Utils P-CAD Library Executive

---

Choose this command to start P-CAD Library Executive, if P-CAD Library Executive is installed on your computer. Refer to your *P-CAD Library Executive User's Guide* for additional information.

## Utils P-CAD Pattern Editor

---

Choose this command to start P-CAD Pattern Editor, if P-CAD Pattern Editor is installed on your computer. Refer to your *P-CAD Library Executive User's Guide* for additional information.

## Utils P-CAD Symbol Editor

---

Choose this command to start P-CAD Symbol Editor, if P-CAD Symbol Editor is installed on your computer. Refer to your *P-CAD Library Executive User's Guide* for additional information.

## Utils InterPlace/PCS

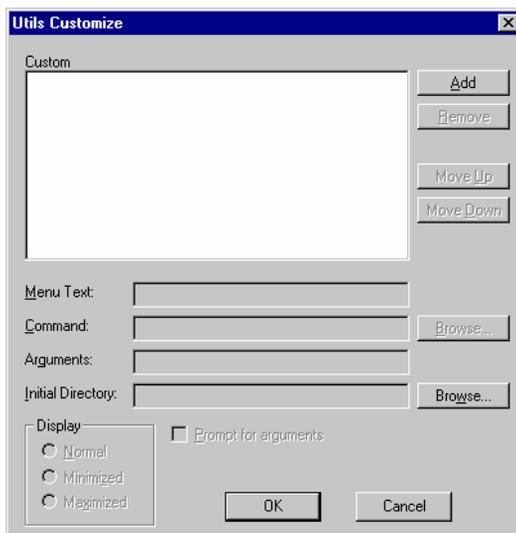
---

Choose **Utils » InterPlace/PCS** command in the Schematic **Utils** menu provides access to both P-CAD Parametric Constraint Solver and P-CAD InterPlace. P-CAD searches your license files and, depending on which license(s) are found, starts the appropriate program(s).

## Utils Customize

---

Choose **Utils » Customize** to add items to the **Utils** menu and custom toolbar, so you can gain to access other programs from P-CAD Schematic. The *Utils Customize* dialog appears.



The *Utils Customize* dialog is used to add new tools or delete and modify existing tools on the *Custom Tools* toolbar and **Utils** menu. The fields on the dialog are as follows:

- **Custom:** Provides a list of the custom tools that currently exist.
- **Menu Text:** Enter the description of the tool being added. The Menu Text field allows up to 40 characters. The buttons on the toolbar display as many letters as can fit on the button. If the name of a tool being added to the custom toolbar is too long to be displayed in its entirety, you can change the display by using lower case letters or shortening the name of the new tool.
- You may insert an ampersand (&) anywhere in the text string to designate a menu shortcut key. For instance, if the Menu Text entry is &Notepad, the menu shortcut key for the tool is the letter **N**.
- The **Utils** menu displays the list of custom tools as shown to the left.
- **Command:** Enter the path to the executable file of the new tool. Click the **Browse** button to display an *Open* dialog where the desired file can be chosen. A warning is issued if a non-existent path is entered, but the entry will still be added.
- **Arguments:** Optional entry used to pass information into the targeted program, if desired.
- **Initial Directory:** Sets the initial working directory for the program.
- **Display:** Selects the way the program appears on the screen when initialized. Normal (the default) to display the program as a window in the workspace, **Minimized** to start the program and display it as an icon at the bottom of the screen, or **Maximized** to start the program and display it across the full screen.
- **Prompt for arguments:** Check this box to automatically display the *Arguments* dialog, which provides a way to enter input that must be passed to the program at execution time. The entry is saved and recalled the next time the program is run.
- **Add:** Click the **Add** button to begin adding a new tool. You can add a maximum of 16 tools to the *Custom* toolbar.
- **Remove:** Select a tool from the custom area and click **Remove** to delete it from the customized toolbar.
- **Move Up/Move Down:** Select a tool from the Custom list and change its position in the list by clicking the **Move Up** or **Move Down** buttons.

Once all selections and entries have been made, click **OK** to apply them.

## Displaying the Custom Toolbar

The *Custom* toolbar is not displayed in the workspace until a custom tool has been added. Once a tool has been added, the toolbar automatically appears with the other Schematic toolbars.



You may control the appearance of the *Custom* toolbar in the workspace using the **View » Custom Toolbar** command. See *View Custom Toolbar* (page 256) for more information.

## Executing a Custom Tool

To launch an program added as a Custom Tool, you may choose any of these methods:

- Click the desired button on the *Custom* toolbar
- Select the tool from the list at the bottom of the **Utils** menu (shown above).

While the **Utils** menu is active select a custom tool by pressing the menu shortcut key assigned to it. For instance, select Notepad by pressing the **N** key.

## Simulate Commands

### Simulate Run

---

Choose the **Simulate » Run** command to run a simulation on your schematic. The command will only become available if you have a schematic design open. If you have created your schematic design using parts that have simulation models associated with them, clicking **Simulation » Run** will invoke the Mixed-Signal Circuit Simulator (from within the Design Explorer 99 SE application) and run a simulation on the design.

If you want to specify analysis criteria prior to running a simulation on your design, you can choose the **Simulate » Setup** command. This command will invoke the Mixed-Signal Circuit Simulator (from within the Design Explorer 99 SE application) and launch the *Analyses Setup* dialog, from where you can specify your simulation criteria before executing the simulation.

If you choose to run a simulation on a schematic that includes parts not supported by simulation models, the Mixed-Signal Circuit Simulator will not invoke. Instead a notepad window will launch, displaying an error log containing all simulation-related errors that are preventing the schematic from being simulated.

### Simulate Setup

---

Choose the **Simulate » Setup** command to allow you to specify analysis criteria prior to a simulation of your design being executed. The command will only become available if you have a schematic design open. If you have created your schematic design using parts that have simulation models associated with them, clicking **Simulation » Setup** will invoke the Mixed-Signal Circuit Simulator (from within the Design Explorer 99 SE application) and bring up the *Analyses Setup* dialog. From this dialog, you can determine which analyses to run on your design. Analyses available include: Operating Point, Transient/Fourier, AC Small Signal, DC Sweep, Noise, Transfer Function, Temperature Sweep, Parameter Sweep and Monte Carlo.

If you want to run a simulation on your design straight away, choose the **Simulate » Run** command. For more information, refer to the online help documentation.



## DocTool Commands

### Using the DocTool Commands

---

With the documentation tools in P-CAD Schematic, you can detail the fabrication, assembly, and testing of the printed circuit board. Using the commands in the **DocTool** menu, you can perform the following tasks:

- Create versatile zoned borders and custom title blocks.
- Include drawing and revision notes.
- Insert tables that include design data such as: net locations, power and ground net data, as well as last used, not used, and spare gate information.
- Place annotated sheet connectors.
- Track nets between schematic sheets.

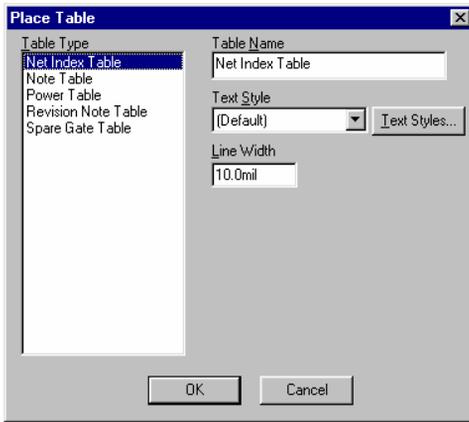
This chapter contains a detailed explanation of each command in the **DocTool** menu.

### DocTool Place Table

---

Choose **DocTool » Place Table** to place a table in your schematic design. To learn how you can use tables in a design, see one of the following topics.

When you choose this command and click the workspace, the following dialog appears:



The *Place Table* dialog contains the following options:

- **Table Type** (list box). Select the type of table you want to place from this list.
- **Table Name** (text box). Type a name for your table in this box. When you place a table in the design, this name appears above the table.
- **Text Style** (list). Select a text style for the table from this list.
- **Text Styles** (button). Click this button to open the *Options Text Style* dialog. Use the controls in this dialog to modify text styles.

### Note Numbering Frame

The options in this frame are only available when you place a Note Table or Revision Note Table. For more information, see, *Note Tables (page 112)* or *Revision Note Tables (page 114)*.

- **Top to Bottom** (option button). Choose this button to increment note numbers from the top to the bottom of the table.
- **Bottom to Top** (option button). Choose this button to increment note numbers from the bottom to the top of the table.
- **Width of Note Column** (text box) Type a value in this box to specify the column width in number of characters. When a note exceeds the column width, the text wraps to the following line.

Note number or symbols do not appear in a revision notes table. The *Note Numbering* option specifies the order of the placed notes, with or without the numbers displayed.

### Pins to Include Frame

The options in this frame are only available when you place a Power Table. For more information, see *Power and Ground Tables (page 113)*.

- **All Pins** (option button). Choose this button to include all pins in the table.

- **Hidden Pins Only** (option button). Choose this button to include only hidden pins in the table.

### Components to Include Frame

The options in this dialog are only available when you place a Power Table. For more information, see *Power and Ground Tables* (page 113).

- **All Components** (option button). Choose this button to display all components in the table
- **Only RefDes Prefix** (option button). Choose this button to only display components of a particular value.

## DocTool Titles

---

Choose **DocTool » Titles** to create a custom title sheet for your schematic design.

When you choose this command, the *Options Sheets* dialog appears with the **Titles** tab selected. You use the options in this dialog build title sheets that can include any or all of the following elements: design border, zones, and a title block.

In Schematic, zones are intelligent and can be used to mark the locations of sheet connectors, as well as cross- reference these connected nets between schematic sheets. For details, see *Sheet Connector Cross Referencing* (page 107).

For more information, see *Title Sheets* (page 98) and *Options Sheets* (page 311).

## DocTool Notes

---

Choose **DocTool » Notes** to add or import notes that you can place in your schematic design.

When you choose this command, the *Design Info* dialog appears with the **Notes** tab selected. You use the options in this dialog to add or import notes into your design.

For more information, see *Placing a Net Index Table* (page 111) and *File Design Info* (page 183).

## DocTool Update

---

Choose **DocTool » Update** to recalculate the design data and update the selected tables or diagrams embedded in the design.

## DocTool Update All

---

Choose **DocTool » Update All** in P-CAD Schematic to recalculate the design data and updates all tables or diagrams embedded in the design.



## Macro Commands

### Using the Macro Commands

The Macro menu lists the commands available to record and run macros. A series of actions that is often repeated during a design session can be recorded, assigned to a key sequence and played back at any time, automating the design process.

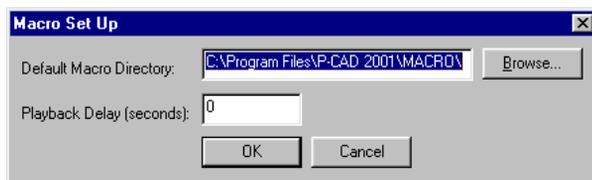
### Macro Setup

Choose **Macro » Setup** to specify the directory in which macros are stored and the playback delay interval.

Before you start recording the set of actions that will make up a new macro, you should choose **Macro » Setup** if you want the macro to be stored in a directory other than the default directory, or if you need to change the playback interval.

#### Setting Up a Macro

1. Choose **Macro » Setup** to open the following dialog.



2. Click **Browse** to open the *Open* dialog.
3. Select the directory you want to use as the default macro storage location. Then, click **Open** to return to the *Macro Setup* dialog.

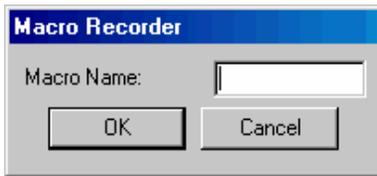
4. Enter the Playback Delay interval in seconds. The number you enter here controls how fast each recorded event appears in the workspace when the macro is run. You can set the interval to the thousandths (.001 of a second).
5. Click **OK** to commit your setup selections or click **Cancel** to exit without changing the settings.

## Macro Record

---

Choose **Macro » Record** to record a macro and to assign a name to that macro.

1. Choose **Macro » Record** to open the following dialog:



2. Type a name for the macro you will record in the Macro Name text box.
3. Click **OK** to close the dialog. The Macro Recording Tool appears and the background of the **M** button in the status line is red, to indicate that you are in recording mode. Any actions you perform are recorded.
4. To stop recording, click **M** on the status line or press the **M** key.

After you record a macro, you can assign it to a shortcut key by choosing **Options Preferences**. Alternatively, you can add it to the *Utils* menu by choosing **Utils » Customize**. For additional information, see *Options Preferences (page 301)* and *Utils Customize (page 352)*.

## Macro Recording Tool

While recording events in a macro, the following controls are available in the Macro Recording Tool:



The buttons in the Macro Recording Tool perform the following functions:

	<p><b>Stop Recording:</b> Terminates the recording process and stores the events in the designated file. The recording process can also be stopped by pressing <b>M</b> or clicking <b>M</b> on the status bar.</p>
--	---

	<b>Pause Recording:</b> Temporarily disables the recording process. When you pause a recording this button changes to the <b>Resume Recording</b> button.
	<b>Recording Suspend:</b> Inserts a suspend that creates a pause during playback. You can also insert a suspend by pressing the <b>Pause/Break</b> key.
	<b>Resume Recording:</b> This button appears in the Macro Recording Tool instead of the <b>Pause Recording</b> button whenever you pause recording. It also appears during playback whenever the playback reaches a recorded suspend activity.
	<b>Recording Origin:</b> Places an origin in the macro. When a recorded origin is encountered, you can click the left mouse button in the workspace to set a relative playback location during playback.
	<b>Cancel Recording:</b> Terminates the recording process and does not save the file.

## Temporary Macro Record (M Toggle Button)

The Macro toggle button (**M** button located on the status line) allows you to create temporary (filename is `Sch_default.mac`) macros on the fly. Typically, you would use this macro for a short time within a design process (e.g., repeatedly placing a combination of lines, duplicating the same lengths and angles, etc.).

Only one temporary macro is available at a time; each time you record a temporary macro, it overwrites the previous one.

### Recording a Temporary Macro

To record a temporary macro, follow these steps:

1. To begin recording, click **M** on the status line or press **M**; the button background changes to red.
2. Perform the actions you wish to record.
3. To stop recording, click **M** or press **M** again; the colored background disappears.
4. Press **E** (execute) to playback the temporary macro. The actions you recorded repeat each time you press **E**.

The entire group of macro functions is available to a temporary macro, including pause and suspend.

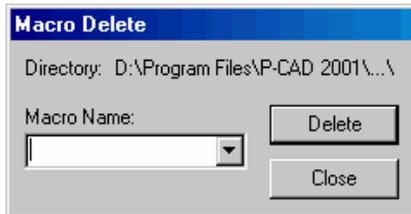
The name of the temporary macro is `Sch_default.mac`, so don't create any macros by that name.

If you want to create a more permanent macro, then you need to name it and record it with **Macro Record**. You can rename any macro with **Macro Rename**.

## Macro Delete

---

Choose **Macro » Delete** to delete a macro. When you choose this command, the following dialog appears:



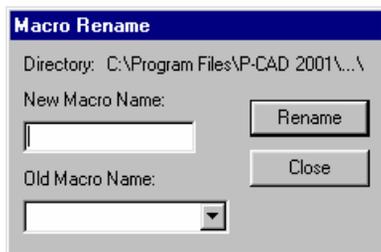
Select the macro you want to delete from the Macro Name combo box, or type the name of the macro. The combo box lists all the \*.mac files found in the default macro directory set in the *Macro Setup* dialog. Click **Delete** and the macro disappears from the list.

## Macro Rename

---

Choose **Macro » Rename** to rename a saved macro or the temporary macro *Sch\_default.mac*. When you rename *Sch\_default.mac*, it is no longer a temporary macro, but a saved, named macro.

When you choose this command, the following dialog appears:



Select the file to rename from the Old Name combo box. This combo box lists all the \*.mac files found in the default macro directory, which is set in the *Macro Setup* dialog. For instructions, see *Macro Setup*, (page 361).

The Macro menu lists the commands available to record and run macros. A series of actions that is often repeated during a design session can be recorded, assigned to a key sequence and played back at any time, automating the design process.

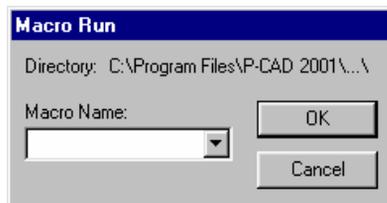
Enter a new name for the selected macro in the New Name edit box. Next, click **Rename** and then click **Close**.

## Macro Run

---

Choose **Macro » Run** to playback a saved macro. While the macro is running, the **M** button on the status line has a green background.

When you choose this command, the following dialog appears:



To run the macro, type a name or select one from the Macro Name combo box. This combo box lists all the \*.mac files found in the default macro directory set in the *Macro Setup* dialog. Click **OK** to run the selected macro.

This command is also handy for viewing your list of macros before you run one.

If you use a macro frequently, you can assign it to a key as a shortcut by choosing **Options » Preferences**. Or, you can add it to the *Utils* menu by choosing **Utils » Customize**. For information, see *Options Preferences* (page 301) and *Utils Customize* (page 352).

### Running Temporary Macros

To run the temporary macro, press **E**. To record a temporary macro, press **M**, do a series of actions, and press **M** again to end recording.

To clear the temporary macro, press **M** twice to record nothing.

### Recording Efficient Macros

The macro utility records every keystroke and mouse click that occurs in the workspace. Because the macro utility focus is toward pressing keys and clicking the mouse, it is advantageous to use the menu commands and the keyboard.

By understanding how the macro utility operates, you can make the most of this powerful tool when recording your macros. The following list provides some helpful hints that will maximize the effectiveness of your recordings:

- If you click a button in the toolbar while recording, the toolbar must be in the same location during playback. By using the shortcut keystrokes instead of toolbar buttons, the toolbar can be in any location without affecting the playback.
- When choosing commands, use the menu commands by clicking on them with the left mouse button instead of the commands in the shortcut menu. The right mouse menu changes location based on mouse location.

- When information dialogs appear while recording a macro, and you need to interact with the dialog during playback, insert a suspend command by pressing the **Pause/Break** key.
- There are times when you may select an item from a drop down list during an event recording. If the order of the items in the list is subject to change due to additions or deletions, the most effective way to eliminate choosing the wrong item is to type the desired value in the edit box instead of selecting from the list. Then you are assured that the correct item is incorporated into the event.
- Whenever options on the status line are recorded, such as changing a layer, you must assure that the status line view is enabled when running the macro. If the status line is not visible during macro playback, the location of the recorded command cannot be identified nor executed.
- When working with text it is best to make changes using keys such as **DEL**, **BACKSPACE**, etc.
- Retain the same size workspace and focus during playback as you used when recording. Changing the workspace size or zoom factor causes adjustments in location coordinates. For instance, if you place a component in the outer regions of a design and change the zoom so that those coordinates are no longer visible in the workspace, the component cannot be placed during playback.
- The resolution of the system in which you are running a recorded macro should match the resolution of the system in which the macro was recorded.

## Macro Features

The macro tool features extend beyond the record and play functions. This section describes these additional features:

- **Running a Macro:** You can launch the Schematic program and a particular macro from the command line by entering a fully qualified path to the executable and then the macro files name.

```
<full path>.SCH.exe /e
```

```
<full path><macro name>
```

If Schematic is already running, this command opens the macro in the current Schematic session. If Schematic is not currently running it is launched and the macro opened.

- **Status Line Recording Indicators:** When the status line is enabled, the Macro Toggle button (**M**) displays different colors to indicate the type of activity being performed. When recording, the background is red. During playback, the background is green. Yellow indicates a pause in the recording, or shows that a suspend or origin command has been encountered during playback.
- **Automatic Delays:** The macro recording automatically inserts a delay into a macro whenever a *Print Setup*, *Font* or *Custom Color* dialog has been displayed. This allows the dialog time to appear in the workspace. If the interval is not large enough for the playback to accommodate the next event, you can change it directly by editing the \*.mac file or by editing the Sch.ini file and changing the "MacroCommonDialogDelay" to a larger number.

- **Editing Macro Files:** You can directly edit a macro file to add or change delays, launch other programs, etc. Each additional command that you add must use the correct syntax.

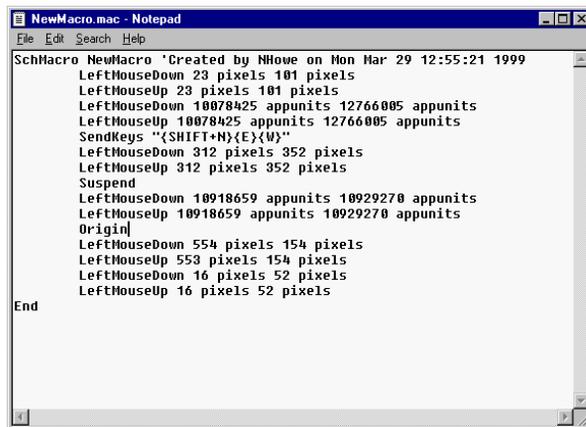
The following section describes the commands that can be added and the proper syntax for each.

## Macro File Syntax

If you have a recorded macro that needs to be modified slightly and you do not want to record the entire chain of events again, you can edit the macro directly. This section explains the syntax used to record the events in the macro file.

Syntax is very important in the macro files. Each macro command is case sensitive and each line must contain a carriage return.

A sample macro, opened in a text editor, is shown below:



```

SchMacro NewMacro 'Created by NHowe on Mon Mar 29 12:55:21 1999
LeftMouseDown 23 pixels 101 pixels
LeftMouseUp 23 pixels 101 pixels
LeftMouseDown 10078425 appunits 12766005 appunits
LeftMouseUp 10078425 appunits 12766005 appunits
SendKeys "{SHIFT+N}(E)(M)"
LeftMouseDown 312 pixels 352 pixels
LeftMouseUp 312 pixels 352 pixels
Suspend
LeftMouseDown 10918659 appunits 10929270 appunits
LeftMouseUp 10918659 appunits 10929270 appunits
Origin|
LeftMouseDown 554 pixels 154 pixels
LeftMouseUp 553 pixels 154 pixels
LeftMouseDown 16 pixels 52 pixels
LeftMouseUp 16 pixels 52 pixels
End
  
```

There are a variety of event types: mouse, keyboard, special and edit. Each has its own keywords and syntax format as shown below. Syntax descriptions show a comma to separate the parts of the format for readability. The commas are not to be inserted into the actual format of the command. An example of an actual syntax is provided as well.

### Mouse Events

Mouse events occur while the cursor is over the workspace and also when the cursor is over the program, but not in the workspace (i.e., over a menu, toolbar, dialog, etc.). Events that occur when the cursor is over the workspace are recorded as appunit locations. Events occurring when the cursor is over a menu, toolbar, etc., are recorded in pixels.

**Syntax:** Keyword, white space, positive integer, white space, unit string, white space, positive integer, unit string.

```

LeftMouseDown 23 Pixels 101 pixels
LeftMouseUp 18 pixels 583 pixels
LeftMouseDown&KeyStroke 179 appunits 50 appunits SHIFT
  
```

Keywords: LeftMouseDown, LeftMouseUp, LeftMouseDoubleClick, RightMouseDown, RightMouseUp, LeftMouseDown&Keystroke<appunits or pixels>Ctrl,LeftMouseDown&Keystroke<appunits or pixels>SHIFT,LeftMouseDown&Keystroke<appunits or pixels>Alt.

**Unit Strings:** A unit string in a mouse event is either pixels or appunits. In the example above, 23 and 101 are the X and Y coordinates of the left mouse down action, and pixels indicates that the mouse click was made on a toolbar command. If the unit string contains appunits, then the mouse click was made in the workspace at the indicated X and Y coordinates.

### Keyboard Events

**Syntax:** Keyword, white space, quoted character string.

```
SendKeys "{SHIFT+N} {e} {w}"
```

Keywords: SendKeys

**Character String:** Each keystroke, and combinations thereof, must be enclosed in curly brackets and the string of curly bracketed strokes enclosed in quotations. In the example above, the word New is recorded using the keyboard syntax format.

**Keyboard Codes:** The following list defines the recordable keys and how they are represented in the macro file.

Keys	Codes
a – z	A – Z
A – Z	SHIFT+A – SHIFT+Z
0 – 9	0 – 9
F1 - F12	F1 – F12
Alt	ALT
Back Space	Backspace
Ctrl	CTRL
Delete	Delete
End	End
Escape	ESC
Home	Home
Insert	Insert
Num Lock	Not recordable
Page Down	PageDown

Keys	Codes
Page Up	PageUp
Pause Break	Not recordable
Print Screen	Not recordable
Scroll Lock	Not recordable
Space	Space
Tab	TAB
Down Arrow	Down
Left Arrow	Left
Right Arrow	Right
Up Arrow	Up
Numpad 0	Numpad0
Numpad 1	Numpad1
Numpad 2	Numpad2
Numpad 3	Numpad3
Numpad 4	Numpad4
Numpad 5	Numpad5
Numpad 6	Numpad6
Numpad 7	Numpad7
Numpad 8	Numpad8
Numpad 9	Numpad9
Numpad /	NumpadDivide
Numpad -	NumpadMinus
Numpad *	NumpadMultiply
Numpad +	NumpadPlus
Numpad Clear	NumpadClear
`	Backquote
-	Minus
=	Equal

Keys	Codes
\	Backslash
;	Semicolon
'	Singlequote
/	Slash
.	Period
,	Comma
[	LeftBracket
]	RightBracket
~	SHIFT+Backquote
!	SHIFT+1
@	SHIFT+2
#	SHIFT+3
\$	SHIFT+4
%	SHIFT+5
^	SHIFT+6
&	SHIFT+7
*	SHIFT+8
(	SHIFT+9
)	SHIFT+0
-	SHIFT+Minus
+	SHIFT+Equal
	SHIFT+Backslash
{	SHIFT+RightBracket
}	SHIFT+LeftBracket
"	SHIFT+Singlequote
:	SHIFT+Semicolon
?	SHIFT+Slash
<	SHIFT+Comma

Keys	Codes
>	SHIFT+Period
Special Windows™ keys	Not recordable

### Special Events

**Syntax:** Keyword.

Origin

Suspend

Delay 1.001

Keywords: Origin, Suspend and Delay.

**Exception:** The Delay keyword is generated by the macro utility whenever a print setup, font or custom color dialog is invoked in the macro. Inserting a one-time delay interval pauses the playback so that there is time to set up the next recorded event. The Delay keyword is followed by a positive floating number. This delay time is added to the global playback interval set using the Macro Setup command.

### Edit Events

**Syntax:** Keyword, qualifier.

Wait 4

ExecuteCommand "C:\ACCEZZL\PATED.EXE"

Keywords: Wait, ExecuteCommand.

**Qualifier:** Each of these keywords has its own type of qualifier. The Wait keyword is followed by a positive number indicating the pause interval in seconds. During playback the wait interval is counted down in the Status Bar and the playback resumes when the interval is complete.

The ExecuteCommand launches another program and requires the program name enclosed in quotes. It may also required the path to the program name or file.

### File Syntax

**Beginning:** The macro file begins with the keyword SchMacro, followed by the name of the macro and a comment string. The comment string, which can also be added to any line in the macro, must begin with a single quote.

**Ending:** Every macro must end with the keyword End.

**Other:** Every line must begin with one of the keywords. Although optional, a tab at the end of each line helps increase a macro's readability when it is edited.



## Window Commands

### Window New Window

---

Choose **Window » New Window** to open additional windows for the active design. You can move independently in each window, making it easy to compare different parts of the same design.

A number identifying the window is added to the file name in the title bar and at the bottom of the **Window** menu.

### Window Cascade

---

Choose **Window » Cascade** to arrange all open windows so that the window tiles 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

---

Choose **Window » Tile** to open all open 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

---

Choose **Window » Arrange Icons** to arrange all minimized design windows in the P-CAD Schematic window. This command positions these icons so that they are evenly spaced and don't overlap.

To minimize a window, click the  button. When you do, the window becomes minimized, as shown in the following illustration:

To restore a minimized window to the maximum window size, double-click the window.

## Window 1,2,

---

The bottom of the **Window** menu lists all open windows. Select the name of the window you want to make active from the **Window** menu. Designs appear on the **Windows** menu in the order that you opened them.

If there are more than nine windows opened, the option **More Windows** appears. Select it to view additional windows.

## Help Commands

### **P-CAD Schematic Help Topics**

---

Choose this command to open the P-CAD Schematic Help file.

### **How to Use Help**

---

Choose this command to connect to the Windows Help file, which shows you how to use the online Help system.

### **Series II Commands**

---

Choose this command to view a list that maps the Tango Schematic Series II commands to its equivalent P-CAD commands and features.

### **About P-CAD Schematic**

---

Choose this command to open a dialog box that contains information about the P-CAD Schematic program, such as the product license number, version number, release date, memory used, and available memory.



## Keyboard Reference

This appendix is a reference of commands and functions accessed through P-CAD shortcut keys.

Standard Windows key combinations are functional for all of the menu commands; use the normal combination **ALT, X, Y** where X equals the underscored menu character, and Y equals the underscored command character.

You can use the *Options Preference* dialog to change shortcut keys for commands and macros. For details, see *Options Preferences* (page 301).

### Schematic Keyboard Reference

Press this key combination	To perform this action
<b>ALT+F4 (File Exit)</b>	A shortcut for the <b>File » Exit</b> command. Choose this command to quit the P-CAD Schematic program. If any open design has been modified since the last save, you are prompted whether you want to save the changes to the file. The program writes configuration information to the <code>Sch.ini</code> file when you exit. For details, see <i>File Exit</i> (page 199).
<b>ALT + MOUSE CLICK</b>	For any click-and-drag or drag-and-drop operations, you can hold down the <b>ALT</b> 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 <b>ALT</b> key, you would normally have to click and hold the button down while you are dragging.
<b>ARROW KEYS</b>	Press an <b>arrow</b> key to move the cursor to the next grid point. Press <b>CTRL+arrow</b> to move the cursor 10 grid points. This is useful to pan the window.

<b>CTRL+MOUSE CLICK</b>	You can copy-and-drag an object by selecting the object, holding down <b>CTRL</b> and clicking the left mouse button in the selected object region, and dragging a copy of the object to a location and releasing to paste it.
<b>CTRL+C (Edit Copy)</b>	A shortcut for the <b>Edit » Copy</b> command. For details, see <i>Edit Copy (page 203)</i> .
<b>CTRL+N (File New)</b>	A shortcut for <b>File » New</b> . This command opens a new, untitled design window in the workspace. For details, see <i>Using the File Commands, (page 169)</i> .
<b>CTRL+O (File Open)</b>	A shortcut for <b>File » Open</b> . When you choose this command, the <i>File Open</i> dialog appears, where you can choose a design file to open. For details, see <i>File Open (page 169)</i> .
<b>CTRL+P (File Print)</b>	A shortcut for <b>File » Print</b> . When you choose this command, the <i>File Print</i> dialog appears, where you can print sheets from your design and set options for your output. For details, see <i>File Print (page 172)</i> .
<b>CTRL+S (File Save)</b>	A shortcut for <b>File » Save</b> . Choose this command to save changes to the current, active design without closing it. For details, see <i>File Save (page 170)</i> . To save the design to a different file, choose <b>File » Save As</b> . For details, see <i>File Save As (page 171)</i> .
<b>CTRL+V (Edit Paste)</b>	A shortcut for the <b>Edit » Paste</b> command. For details, see <i>Edit Paste (page 204)</i>
<b>CTRL+X (Edit Cut)</b>	A shortcut for the <b>Edit » Cut</b> command. For details, see <i>Edit Cut (page 203)</i> .
<b>CTRL+Z (Edit Undo)</b>	A shortcut for <b>Edit » Undo</b> , which reverses a completed action. If you have not completed your action (e.g., you are in Place Line mode and have not finished a series of segments), Undo will not reverse the action. Use the <b>BACKSPACE</b> key to unwind the unfinished actions when you are in a placement mode. For details, see <i>Edit Undo (page 201)</i> .
<b>CTRL+F4</b>	Closes the active window.
<b>CTRL+F6 and CTRL+TAB</b>	Switches focus to the next open window. Use <b>CTRL+SHIFT+F6</b> to switch focus to the previous window.
<b>SHIFT+F4 (Window Tile)</b>	A shortcut for the <b>Windows » Tile</b> command. Windows are resized and arranged side-by-side so that all windows are visible and none overlap. For details, see <i>Window Tile (page 373)</i> .

<b>SHIFT+F5 (Window Cascade)</b>	A shortcut for the <b>Windows » Cascade</b> 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. For details, see <i>Window Cascade (page 373)</i> .
<b>DEL (Edit Delete)</b>	A shortcut for the <b>Edit » Delete</b> command, which deletes all selected objects. For details, see <i>Edit Delete (page 234)</i> .
<b>F1 (Help)</b>	Press the <b>F1</b> key to display context-sensitive help. If you put focus on a command or dialog (by mouse or keyboard) and press <b>F1</b> , the Help window appears containing information specific to the focus item.
<b>PAGE DOWN</b>	Scrolls one page down in the workspace. <b>CTRL+PAGE DOWN</b> scrolls one page right.
<b>PAGE UP</b>	Scrolls one page up in the workspace. <b>CTRL+PAGE UP</b> scrolls one page left.
<b>SPACEBAR</b>	<p>The <b>SPACEBAR</b> 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 <b>SPACEBAR</b> twice. Therefore, to simulate the click-and-hold mouse action, you press and release the <b>SPACEBAR</b> once.</p> <p>As the left mouse button is used in such a variety of ways throughout the P-CAD program, this <b>SPACEBAR</b> keystroke can become a regular part of your work.</p>
<b>BACKSPACE (Unwind)</b>	Used as unwind command while placing objects with multiple segments (e.g., lines, polygons). Each <b>BACKSPACE</b> stroke unwinds the previously placed segment.
<b>ESC (Escape)</b>	Terminates placement of objects with multiple segments; it also cancels a redraw in progress. It is often equivalent to the right mouse button. <b>ESC</b> also exits from dialogs (equaling the <b>Close</b> or <b>Cancel</b> button).
<b>+ PLUS KEY (Zoom In)</b>	Press the <b>+</b> key as a shortcut for choosing the <b>View » Zoom In</b> command. The plus key causes a zoom in to occur at the cursor location. The plus key does not change the cursor to a zoom cursor (as do the zoom commands from the <i>View</i> menu).
<b>- MINUS KEY (Zoom Out)</b>	Press the <b>-</b> key as a shortcut for choosing the <b>View » Zoom Out</b> 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). For details, see <i>View Zoom Out (page 252)</i> .
<b>A KEY (Grid Toggle)</b>	Press the <b>A</b> key to switch between absolute and relative grid settings.

<b>C KEY (View Center)</b>	A shortcut for the <b>View » Center</b> 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 <b>C</b> . For details, see <i>View Center (page 252)</i> .
<b>D and SHIFT+D (Change Reference Designators)</b>	Press <b>D</b> to switch to the next available reference designator; <b>SHIFT+D</b> changes the reference designator to the highest unused preceding reference designator.
<b>E KEY (Play Macro)</b>	Press <b>E</b> to play back (executes) the temporary macro (named <b>_DEFAULT</b> ). Temporary macros are recorded by clicking the <b>Macro</b> button or by pressing the <b>M</b> key. Each successive temporary macro overwrites the previous one. See <i>Macro Record (page 362)</i> for more information.
<b>F KEY (Flip Object)</b>	Press the <b>F</b> key to flip an object during Place and Select operations. Not all objects can be flipped. Refer to <i>Edit Select (page 247)</i> for more information about flipping objects.  The <b>F</b> key also switches between orthogonal mode pairs; see <i>Options Configure (page 292)</i> for details on orthogonal modes.
<b>G KEY (Grid Select)</b>	This key scrolls forward through the list of grid settings. Use <b>SHIFT+G</b> to scroll back through the list.
<b>J KEY (Enter Coordinate)</b>	Gives focus to the X coordinate box in the status line. From there, you can enter new X and Y coordinates.
<b>L and SHIFT+L Keys (Sheets)</b>	<b>L</b> sequences forward through the sheets in the design file of the active window; <b>SHIFT+L</b> sequences backward through the sheets.
<b>M KEY (Record Macro)</b>	Duplicates the macro toggle button ( <b>M</b> button on the Status line) for starting/stopping recording of the temporary macro (see <b>E</b> key description). See <i>Macro Record (page 362)</i> for more information.
<b>O and SHIFT+O Keys (Orthogonal Mode)</b>	<b>O</b> sequences forward through the orthogonal modes during placement of wires and lines; <b>SHIFT+O</b> sequences backward throughout the modes. Press the <b>F</b> key to switch between orthogonal mode pairs. Orthogonal modes are set in the <i>Options Configure</i> dialog box. See <i>Options Configure (page 292)</i> for more information.
<b>P and SHIFT+P (Change Reference Designators)</b>	Press the <b>P</b> key to change the reference designator to the next gate; Press <b>SHIFT+P</b> to change the reference designator to the previous gate.
<b>Q KEY (Draft Mode)</b>	Press the <b>Q</b> key to switch between draft mode ON and OFF.
<b>R KEY (Rotate)</b>	<b>R</b> Rotates objects 90 degrees during Place and Select operations. For more information about rotating objects with Place and Select operations <i>Edit Select (page 247)</i> .

<b>S KEY (Select)</b>	<b>S</b> key (Select)
<b>U KEY (Undo)</b>	A shortcut for the <b>Edit » Undo</b> command, which reverses a completed action. See <i>Edit Undo (page 201)</i> for more information. For the unwind feature, refer to the <b>BACKSPACE</b> key description in this section.
<b>W KEY (Line Width Scroll)</b>	Scrolls forward through the list of line widths established in the design. Use <b>SHIFT+W</b> to scroll back through the list.
<b>X KEY (Cursor Style)</b>	Press the <b>X</b> key to switch between the three cursor styles: <b>Arrow</b> , <b>Small Cross</b> , and <b>Large Cross</b> .
<b>Z KEY (Zoom Window)</b>	A shortcut for the <b>View » Zoom Window</b> command. Just press <b>Z</b> and then draw the zoom window; a zoom cursor (magnifying glass) appears until you draw the zoom window. Whatever you surround with the zoom window fills the screen. For details, see <i>View Zoom Window (page 252)</i> .



# Translating Tango-Schematic Designs

This appendix provides information on translating Tango-Schematic designs into P-CAD Schematic.

If you choose the **Archive Library** command in Tango-Schematic Version 1.30a or earlier, the Pattern Attribute is not included in each part. If you want pattern information saved in your translated design, use version 1.40 to archive your parts or add pattern data manually with a text editor.

Also, verify that all parts have a RefDes, in particular power parts and logos. The **Archive Library** command does not save parts without a RefDes. You need to copy these parts to the archived library.

## Translation Process

---

Before translating a Tango-Schematic design (using the **File » Open** command in P-CAD Schematic) you should prepare your designs by performing the following steps:

1. Choose the **Post Cleanup** command to remove any collinear (overlapping) wire segments.
2. Choose the **Post Compile** command and the DRC report to detect existing errors. Errors may be fixed in Tango-Schematic before translation or after translation using P-CAD Schematic.
3. Convert any hierarchical designs into a flat format. All ports and modules should be replaced with the necessary schematic information in order to retain all net connectivity between sheets. All files of the design must have the same root name, with extensions of .S01, .S02, etc. Each file is translated to sheets in your P-CAD Schematic design, with corresponding default names Sheet1, Sheet2, etc.
4. Save your design files in ASCII format. P-CAD Schematic does not accept Tango-Schematic designs in binary format.

After translation you can generate an ERC report in P-CAD Schematic and compare the results to your Tango- Schematic report. You can also use the NETCOMP netlist-compare utility to compare netlists generated by Tango-Schematic and P-CAD Schematic. This utility compares nets on a node-by-node basis, disregarding net names since they can be different, and lists all nets not having identical matches. These differences are often caused by problems described throughout this section.

You can fix errors discovered during these tests in Tango-Schematic before translation or after translation using P-CAD Schematic.

The Tango-Schematic libraries associated with the design must be translated prior to opening a Tango-Schematic design in P-CAD. Refer to the P-CAD Library User's Guide for instructions on how to do this. After translating the libraries, run the Library Setup command in P-CAD Schematic to open all libraries needed for the translation.

P-CAD Schematic will use the first occurrence of a part found in the open libraries. If a part occurs in more than one library, the first open library with the desired part format will be used before any other library containing the same part. You may wish to insure that your archived library is first in the list of open libraries.

## Considerations

---

For a complete and correct translation, the translated Tango-Schematic libraries must contain the parts exactly as they exist on your schematic design. If the library part definition is different from the part on the design, you must either alter the library part to have the correct definition, or replace all instances of the part on the Tango-Schematic design with the part from the library and correct any errors caused by the change. Some common differences that become evident after translation are:

- The part in the library does not have the same orientation as the part on the sheet. These parts appear rotated on the translated sheet.
- The part in the library has a reference point, which is different from the reference point of the part on the design. These parts appear offset from the correct location.
- The pin positioning and definition are not the same. If the pins are out of position, having differing lengths, or have different pin designators, the translation reflects these differences.
- The RefDes, type, or value locations for a library part are different from the part on a sheet. This can happen when these attributes are moved on the sheet for cosmetic reasons; these parts can appear different in P-CAD Schematic, but these differences are only cosmetic and do not affect normal processing.

## Design Practices Considered

---

Tango-Schematic interprets some ambiguous design practices differently from P-CAD Schematic.

- Placing a wire so that it crosses over the endpoints of non-connected pins. In Tango-Schematic, a wire touching pin endpoints without stopping over them is not connected. P-CAD includes these pins in the net.
- Global Hidden Pins made visible on the sheet, and connected pin-to-pin to another part. These occurrences are flagged as an error and must be modified to include a wire.
- Parts that do not have a "Type" flag an error in P-CAD. These parts must be modified to include a Type field.

- Placing junctions over wires or pins that do not require a junction for connectivity. Tango-Schematic allows the placement of extraneous junctions. Using P-CAD Schematic, junctions are necessary only when an intersection of three or more wires and/or pins need to be connected together.

The default net names given to nets that were not specifically named on your Tango-Schematic designs are renamed using P-CAD default net names. These are not necessarily the same as those used in Tango-Schematic. When you compare netlists generated by the two products, take these different defaults into account.

If you use wires instead of lines for bus entries in Tango-Schematic, you may get wires that appear to short together after translating the design to P-CAD Schematic. This can be changed by replacing the wires used for bus entries by lines in Tango-Schematic, or by manually moving the translated wires in P-CAD Schematic.



## P-CAD System Messages

This appendix documents the messages that you might encounter when importing or exporting a PDIF file in P-CAD PCB or P-CAD Schematic.

### Error Messages

Error Message	Cause	Solution
<b>A reference designator is required in instance &lt;name&gt; near line &lt;line number&gt;.</b>	The reference designator is missing.	Enter the reference designator and try again
<b>Bad PIN_DEF record #&lt;number&gt; near line &lt;line number&gt;. Pad ignored.</b>	A syntax error was encountered. A PIN_DEF entry is malformed.	Check the PDIF file at the designated line number to determine the cause. Then consult your PDIF manual to correct the syntax.
<b>Component instance &lt;name&gt; (&lt;ref des&gt;) pin &lt;pin des&gt; references an unknown net &lt;name&gt; near line &lt;line number&gt;.</b>	The net does not exist.	Either create it in Master Designer or reset the pin's net name.
<b>Data in this design extends beyond 60 square inches. Load aborted.</b>	P-CAD PCB allows a maximum of 60 square inches for a design. This design is too big.	Either make it smaller or break up the design.
<b>Device expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	Each entry in a cross-reference file has a device, but one was not found.	Add the device name at the specified line number.

Error Message	Cause	Solution
<b>Duplicate pin designator &lt;pin des&gt; in instance &lt;name&gt; near line &lt;line number&gt;. Pin ignored.</b>	P-CAD PCB and Schematic do not allow duplicate pin designators. The duplicate pins will not show up in the resulting component.	Edit the PDIF file to change the pin designators.
<b>Duplicate reference designator &lt;reference designator&gt; in instance &lt;name&gt; near line &lt;line number&gt;. Instance ignored.</b>	A duplicate reference designator was found. P-CAD PCB and Schematic do not support duplicate reference designators. This instance was ignored.	Either change the reference designator in Master Designer or edit the PDIF file and change the Rd statement.
<b>Equal sign expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	A syntax error was encountered. An entry in a cross-reference file may have the power and ground pins listed.	Enter an equal sign. The format is (<pin des> = <net name>,...). The system logs this error if it encounters a pin designator, which is not followed by an equal sign.
<b>File revision number is unrecognized.</b>	The PDIF version number was not recognized.	Use a PDIF file from Master Designer version 6.0 or later.
<b>Gate number expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	A syntax error was encountered. Each entry in a cross-reference file has a gate number, but one was not found.	Check the PDIF file at the designated line number to determine the cause. Then consult your PDIF manual to correct the syntax.
<b>Instance &lt;name&gt; referenced an undefined component (COMP_DEF=&lt;name&gt;) near line &lt;line number&gt;.</b>	A syntax error was encountered. An instance references the name of a COMP_DEF that was not defined prior to the instance.	Check the PDIF file at the designated line number to determine the cause. Then consult your PDIF manual to correct the syntax.
<b>Keyword expected on line &lt;line number&gt;.</b>	A syntax error was encountered.	Check the PDIF file at the designated line number to determine cause. Then consult your PDIF manual to correct the syntax.
<b>Left paren expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	A syntax error was encountered.	Correct the syntax. The format is (<pin des> = <net name>,...). The system logs this error if it encounters a net name, which is not followed by a left paren or comma.

Error Message	Cause	Solution
<b>Load failed near line &lt;line number&gt;.</b>	A syntax error was encountered.	Check the PDIF file at the designated line number to determine the cause. Then consult your PDIF manual to correct the syntax.
<b>Net name expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	A syntax error was encountered.	Correct the syntax. The format is (<pin des> = <net name>,...). The system logs this error if it encounters an equal sign, which is not followed by a net name.
<b>No {Lystr...} section found.</b>	A syntax error was encountered.	Check the PDIF file at the designated line number to determine cause.
<b>No pad stacks defined near line &lt;line number&gt;.</b>	PDIF PCBs require an attached PAD_STACK record to load properly into P-CAD PCB.	Edit the PDIF file at the designated line number to include the name of the attached pad stack.
<b>Not enough memory.</b>	This means the system could not allocate enough memory to do its work.	Close some applications and try again.
<b>Pad stack &lt;name&gt; has shape &lt;pdif object&gt; (&lt;number&gt;X&lt;number&gt;) that is smaller than both the pad hole size (&lt;number&gt;) and the via hole size (&lt;number&gt;) near line &lt;line number&gt;. Hole size set to 0.</b>	A pad type (Pt) is trying to set too big a hole. The hole is assigned either through the tool table file or the pad/via default sizes specified in the PCB.INI file.	Either change the default pad/ via hole sizes in the PCB.INI file, or change the specified tool table.
<b>Part file name expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	Each entry in a cross-reference file has a part file name, but one was not found.	Check the PDIF file at the designated line number to determine the cause.
<b>P-CAD attribute near (x, y) has no value.</b>	This message means that the system found a Master Designer attribute that does not have a value.	N/A
<b>PDIF item &lt;pdif object&gt; is not supported in nets.</b>	Some PDIF objects are not supported in P-CAD PCB or Schematic nets.	Remove the object and try again.
<b>Pin &lt;pin des&gt; of part &lt;part number&gt; of component &lt;name&gt; could not be added near line &lt;line number&gt;.</b>	A memory or resource limit has been reached.	N/A

Error Message	Cause	Solution
<b>Pin number expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	A syntax error was encountered. An entry in a cross-reference file may have the power and ground pins listed.	Correct the syntax. The format is (<pin des> = <net name>,...). The system logs this error if it encounters an equal sign, which is not preceded by a pin number.
<b>Power or ground pin &lt;pin des&gt; for component &lt;name&gt; does not have a net name or a pin name.</b>	A power pin was found that does not have a net name or does not have a pin name.	Check the PDIF file at the designated line number to determine the cause.
<b>Right paren expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	An entry in a cross-reference file may have the power and ground pins listed. The format is (<pin des> = <net name>,...). The system logs this error if it encounters an equal sign, which is not preceded by a pin number.	Check the PDIF file at the designated line number to determine the cause.
<b>Symbol file name expected on line &lt;line number&gt; of &lt;file name&gt;.</b>	A syntax error was encountered. Each entry in a cross-reference file has a symbol file name, but one was not found.	Check the PDIF file at the designated line number to determine the cause.
<b>Too many power and ground pins for error file.</b>	If a component is heterogeneous, the system writes the cross-reference file lines to the error file. The cross-reference file line consists of the number of gates, the device, the part file name, the power and ground pins, and the symbol file name. There is a limit to how many power and ground pins can be written to the error file. This message means that limit was exceeded.	Edit the cross-reference file and add any pins that are missing.

Error Message	Cause	Solution
<b>Too many symbols on this sheet to create P-CAD symbol name.</b>	P-CAD symbol instances require a name. This name's format consists of NCssssxxxx; where ssss is sheet number and xxxx is a symbol number. This only allows for 9999 symbols on a sheet. If the system detects that there are more than 9999 symbols on a sheet, which require a symbol name, it logs this error.	Create a new sheet and move some of the parts onto it.
<b>Unable to open file &lt;file name&gt;.</b>	This could mean that the file does not exist, or that the system is trying to open for writing a read-only file.	Make sure the specified file exists and that it is not a read- only file.
<b>Unable to rename file &lt;file name&gt; to &lt;file name&gt;.</b>	This could mean that the file the system is trying to rename does not exist, or that a file already exists that it is trying to rename it to.	Make sure the specified file exists and that it is not a read- only file.
<b>Unable to translate text object near (x, y).</b>	The system was unable to get the data from a text object so that it could be translated.	N/A
<b>Unable to translate text style for text object near (x, y).</b>	The system found a text object that is using an unrecognizable text style.	N/A
<b>Unrecognized CN format in instance &lt;name&gt; near line &lt;line number&gt;.</b>	A syntax error was encountered.	Make sure the CN syntax is correct. The CN format can be of 2 styles: {CN <net name> ...} Or {CN <pin name> <net name> ...}.
<b>Unrecognized justification style.</b>	You are using a justification style that is currently unrecognizable to the system.	Use a justification style that the system recognizes.

## Warning Messages

Warning Message	Explanation
<b>Attribute key too long near (x, y).</b>	PDIF format allows only 23 characters for an attribute key. The system found a key with more than 23 characters.
<b>Attribute object near (x, y) has no value.</b>	The system found an attribute with no value. PDIF format requires a value so this attribute was ignored by the system.
<b>Attribute value too long near (x, y).</b>	PDIF format allows only 255 characters for an attribute value. The system found an attribute value with more than 255 characters. The attribute value is translated truncated to 255 characters.
<b>Cannot open file &lt;file name&gt;.</b>	This could mean that the file does not exist, or that the system is trying to open for writing a read only file.
<b>COMP_DEF &lt;name&gt; has a package pin number (&lt;number&gt;) that is out of range near line &lt;line number&gt;.</b>	The pin number specified in the SPKG Sp record does not map to any pad in the PIN_DEF record. Either the pin des number is greater than the number of pads or a pad with the same name (number) could not be found.
<b>COMP_DEF &lt;name&gt; has no SPKG section near line &lt;line number&gt;. Pin names will start at '1'.</b>	The COMP_DEF has no pin to pad mapping (packaging) information. The components pins created will have pin designators that match the pad's position in the PIN_DEF array. The component is most likely a PCB only component.
<b>COMP_DEF &lt;name&gt; references unknown pad stack &lt;number&gt;. Style &lt;name&gt; used instead.</b>	A pad references a padstack number (Pt) that is not defined in the PAD_STACK record. The PCB "(Default)" padstack will be used instead.
<b>Component &lt;name&gt; has had its name changed to &lt;name&gt;. This is required to maintain uniqueness in P-CAD.</b>	A component's type in Master Designer is used as the file name for the symbol data. This limits the component's type to 8 characters. In P-CAD PCB and Schematic, the component's type can be 17 characters. This means that the type is truncated to 8 characters and slightly modified to make it unique if there is more than one component that has that 8 character prefix.
<b>Component &lt;name&gt; is heterogeneous and will be written as &lt;number&gt; separate COMP_DEFs. Cross reference file data follows:</b>	Master Designer does not allow heterogeneous components. During translation the heterogeneous components are separated into homogeneous components. The cross-reference file data is included in the error file for your convenience.

Warning Message	Explanation
<b>Component &lt;Ref Des&gt;, type &lt;type&gt; was given a PRT attribute of &lt;PRT attribute value&gt; which may cause packaging errors in P-CAD.</b>	The PRT attribute value is the file name for the PCB part data. The value is created by combining the type with “.prt”. In P-CAD PCB and Schematic, the type can be longer than 8 characters.  If the type is longer than 8 character, this is an illegal file name. You will need to fix this attribute in Master Designer before you can package the schematic.
<b>Copper Pour not allowed in pattern &lt;name&gt; (&lt;ref des&gt;) near line &lt;line number&gt;. Pour demoted to polygon.</b>	PCB does not support copper pours in patterns. The pour was converted to a polygon, which is supported.
<b>Could not create pad/via style for pad &lt;number&gt; near line &lt;line number&gt;.</b>	A memory or resource limit has been reached. Reduce the number of padstack in the design or simplify their construction.
<b>Cross reference file missing entry for &lt;name&gt;. Unable to attach symbol name for multi- part component.</b>	The attached symbol name could not be added to the component because there was no cross-reference file entry for the part.
<b>Heterogeneous component &lt;name&gt; is missing gates &lt;part number&gt;. Parts &lt;ref des&gt; cannot be placed. Place the missing gates as spares and reload.</b>	Heterogeneous components are found in the cross-reference file but the PDIF data is not found in the PDIF file. If you place the missing gates in Master Designer, re-create the PDIF file, the PDIF data will be included and a complete heterogeneous component can be created in Schematic.
<b>Homogeneous part &lt;name&gt; has non-constant gate equivalencies.</b>	Master Designer does not store gate equivalence. The system is warning you that it found a homogeneous part with non- constant gate equivalencies.
<b>lat value too long at line &lt;line number&gt;.</b>	PDIF format allows only 255 characters for an lat value. The system found an lat value with more than 255 characters. The attribute value is translated truncated to 255 characters.
<b>Instance &lt;name&gt; does not specify a location, (0,0) assumed, near line &lt;line number&gt;.</b>	The instance PI record is missing.
<b>Instance &lt;name&gt; has an illegal IPT record near line &lt;line number&gt;.</b>	The number of entries in the instance Ipt record does not match the number of entries in the COMP_DEF PIN_DEF record.

Warning Message	Explanation
<b>Instance &lt;name&gt; referenced a package number (&lt;part number&gt;) that does not exist in COMP_DEF &lt;name&gt; near line &lt;line number&gt;.</b>	An instance of a heterogeneous component references a part number that is not defined for the number of parts in the component. The heterogeneous information in the cross-reference file may be incorrect.
<b>Layer name &lt;name&gt; has been truncated to &lt;name&gt;.</b>	The layer name is used as the PDIF file name. File names are limited to 8 characters. The system truncates the layer name to 8 characters and if necessary, slightly modifies it to make it unique if there are more than one layer that have the same first 8 characters.
<b>Missing pad stack name for pad/ via &lt;number&gt; near line &lt;line number&gt;.</b>	A syntax error was encountered. The padstack name is missing from the PAD_STACK record.
<b>Name &lt;name&gt; truncated to &lt;number&gt; characters near line &lt;line number&gt;.</b>	The object's name was too long and truncated so that it would be a valid PCB or Schematic name.
<b>Net &lt;name&gt; attribute lost. &lt;key&gt; = &lt;value&gt;.</b>	Master Designer does not have net attributes. When the system warns you when it detects a net attribute.
<b>&lt;number&gt; pin(s) were created for a &lt;number&gt; pin symbol (COMP_DEF=&lt;name&gt;, I=&lt;name&gt;) near line &lt;line number&gt;.</b>	The number of entries in the PIN_DEF record does not match the number of Sp records in the SPKG record.
<b>Object near line &lt;line number&gt; failed to load.</b>	A syntax error was encountered. Check the PDIF file at the designated line number to determine cause.
<b>Pad stack &lt;name&gt; does not have consistent enough shapes to set the predefined '(Signal)' layer near line &lt;line number&gt;. Pad/via style layer '(Signal)' set to 0.</b>	The padstack shape defined for the Top layer does not match the shape defined for the Bottom layer.  The padstack shapes defined for internal signal layers are not all the same.
<b>PDIF item &lt;pdif object&gt; is not supported in nets.</b>	Some PDIF objects are not supported in PCB or Schematic nets. It could be a DRC error indicator.
<b>PDIF item &lt;pdif object&gt; is not supported in pad stacks.</b>	Some PDIF objects are not supported in PCB pad stacks. These include polygons, text, flashes and lines.

Warning Message	Explanation
<b>PID too long at line &lt;line number&gt;. Truncated.</b>	Master Designer's PID is equivalent to PCB's type attribute. The PID is limited to 15 characters. If PCB's type attribute is longer than 15 characters, it is truncated to 15 characters.
<b>Power and ground attribute is too long for symbol &lt;name&gt;.</b>	If you get this warning and are not using Master Designer Version 8.0 software, switch to Version 8.0 if possible. Version 8.0 uses multiple PWGDi attributes to handle case. If you are using Version 8.0 and still get this message the PWGDi attribute is truncated. If you cannot use Version 8.0 and are getting this message, the PWGD attribute is truncated.
<b>Power pin &lt;pin des&gt; of component &lt;name&gt; could not be added near line &lt;line number&gt;. Probably duplicate.</b>	A duplicate power pin was detected in the component. Check the cross reference file or the PWGD(i) attribute(s) and try again.
<b>Rotation rounded to &lt;angle&gt; at line &lt;line number&gt;.</b>	Master Designer Schematic can only be rotated to 0, 90, 180, 270. If a rotation other than those values is detected, the object is rotated to the nearest allowable angle.
<b>Symbol &lt;name&gt; already exists in table on line &lt;line number&gt; of &lt;file name&gt;.</b>	A duplicated symbol file name was found in the cross-reference file. First one found is used. Others are discarded.
<b>Text height is less than 2 at line &lt;line number&gt;.</b>	P-CAD PCB and Schematic's text height is used as the text size for PDIF translation. Master Designer's text size cannot be less than 2. If a text height is found to be less than 2, 2 is used.
<b>Text object near (x, y) has no string, object ignored.</b>	An empty text object was found and ignored.
<b>Text too long near (x, y).</b>	Text is limited to 255 characters. Text is truncated to 255.
<b>The number of components pins created for &lt;name&gt; does not match the number of PIN_DEF entries near line &lt;line number&gt;. Power pins might be missing.</b>	The number of P statements in the COMP_DEF's PIN_DEF section determines the number of pins.
<b>The number of pins in section &lt;part number&gt; (COMP_DEF=&lt;name&gt;) is not equal to the number of pins defined (PIN_DEF) near line &lt;line number&gt;.</b>	A PKG part section does not have the same number of entries (pins) as the PIN_DEF record.

Warning Message	Explanation
<b>Translating TangoPRO polygon object near (x, y) as P-CAD lines.</b>	Master Designer Schematic does not have a filled polygon object. If the system detects a polygon object is translates it as lines for Master Designer.
<b>Unrecognized keyword &lt;key&gt; near line &lt;line number&gt;. Keyword ignored.</b>	A syntax error was encountered. Check the PDIF file at the designated line number to determine cause.
<b>Via references unknown pad stack &lt;number&gt;. Style &lt;name&gt; used instead.</b>	The padstack number referenced by a via does not exist. The via style used will be <name>, probably '(Default)'.

# The Tango Netlist Format

A Tango netlist file is an ASCII text file, with a carriage return and a line feed at the end of every line. The netlist is divided into two sections: the component section and the net section.

## The Component Section

---

This section describes each component in the netlist, such as the component type, reference designator, cost, ordering code, etc.

There can be any number of component descriptions in the component section. Each component description is contained within square brackets [ ], with each square bracket being the first and only character on a line.

The component description has the following format:

```
[  
  refdes  
  package  
  type  
  (blank line)  
  (blank line)  
]
```

The `refdes` is the reference designator, which can be up to 16 alphanumeric characters in length. All letters in the reference designator must be upper-case. (For example: U1, R3, XTAL2.)

The `package` is the component package, which must be the name of a PCB library pattern. The package can be up to 16 alphanumeric characters in length. All letters in the package must be upper-case. (For example: DIP40, RES400, DB9.)

The `type` is the component type. The type can be up to 16 alphanumeric characters in length. (For example: Z80ACPU, 47K, 4.00 MHz, DB9.)

The following example is a netlist description for Z80ACPU:

```
[
U5
DIP40
Z80ACPU
]
```

## The Net Section

---

This section contains one or more net definitions. A net is a group of nodes to be connected together. A node consists of a component label, a delimiter and a pin designator.

There can be any number of nets in the net section. Each net is contained within parentheses ( ), with each parenthesis being the first and only character on a line.

Each net in the net section has the following format:

```
(
net
node
node
.
.
.
node
)
```

The net is the net name, which can be up to 16 alphanumeric characters. You can use upper-case and lower-case characters. Since case is not significant when comparing net names, P-CAD PCB forces all characters entered to upper-case. Therefore, do not use case alone to distinguish net names. Do not use parentheses or square brackets for the first character in a net name.

The lines after the net name contain the nodes in the net, with one node per line. Each node consists of:

- A component name (up to 16 alphanumeric characters long)
- A single-character delimiter (a hyphen or a comma)
- A pin designator (up to 16 alphanumeric characters long)

Each node is scanned from left to right until the first occurrence of a delimiter (a hyphen or a comma) is found. Everything to the left of the delimiter is considered the component name;

everything to the right is considered the pin designator. Do not use any of the delimiter characters in your component names, since this will cause the program to incorrectly read the netlist.

A node must appear only once in the netlist, since by definition a node can only belong to one net. A node appearing in more than one net would imply that those nets should be joined into one net, since they have the node in common. If a node appears in more than one net, P-CAD PCB issues an error message and halts the netlist load.

A sample net section is shown as follows:

```
(
A0
U1-10
U2-10
U3-10
U4-10
U5-30
U6-34
)
(
CPUCLK
R5-2
U6-20
U9-6
)
(
U10.1_J2.2
J2-2
U10-1
)
```



## -A-

About the User's Guide.....	2
absolute grid toggle .....	296
adding	
custom fields .....	104
field sets.....	106
net attributes .....	60
Note Table notes .....	112
object values.....	55
Revision Note Table.....	115
title sheets .....	19, 293, 312
wires .....	264
Alias, Library command.....	320
Align Parts, Edit command.....	237
aligning parts	
horizontally or vertically .....	87
All, View command .....	252
altering .....	see editing or modifying
arc	
changing centerpoint.....	274
modifying size.....	225
resizing .....	89, 274
rotating/flipping.....	274
width.....	225
Arc, Place command.....	273
Archive Library, Library command.....	327
Arrange Icons, Window command.....	373
Ascend, View command.....	255
ASCII files	
opening.....	22
saving .....	171
assigning	
field sets.....	107
Attribute Properties.....	228
Attribute, Place command .....	279
attributes	
adding.....	160
deleting .....	160
editing .....	160
name .....	279
net	
adding .....	95
deleting.....	95
editing.....	95
Part Number.....	279
Part Properties .....	218
RefDes .....	279
reference .....	229
reference link	
launching.....	95
rotating/flipping .....	280
Type .....	279
Value.....	279
autopanning	
adjusting .....	295
percent display, defined.....	17
autosave .....	294
compress .....	294
AutoSave	
defined .....	17
<b>-B-</b>	
backup file, creating.....	170
block selection	
defined .....	82
error indicators.....	137
items .....	288
modes defined .....	83
Block Selection, Options command .....	287
bounding outline .....	253
bus	
connections .....	266
definition .....	240, 265

displaying name .....	220
highlight.....	246
line length, measuring.....	266
orthogonal modes.....	266
resizing.....	89
unhighlight.....	246
Bus, Place command .....	265
buttons	
Command Toolbar .....	256
Placement Toolbar.....	256

**-C-**

Cascade, Window command.....	373
Center, View command.....	252
changing .....	see editing or modifying
checked command, defined.....	7
circles, placing.....	273
class editor.....	310
clear all overrides	
ERC.....	336
clearance rules.....	304
Clipboard	
copying objects to .....	91
defined.....	89
pasting objects from .....	93
close button .....	13
Close, File command.....	170
collocated objects, defined .....	80
colors	
custom colors .....	298
defining preferences .....	28
display colors.....	298
Display Colors frame .....	175
highlight.....	240
item colors .....	298
Item Colors frame.....	175
Options Display .....	297
print .....	172, 174
print setup.....	142
unhighlight.....	240
Command Toolbar, View command...8,	256
component	
copying .....	322
deleting from a library.....	323
renaming.....	324
replacing types .....	233
Compress binary designs.....	295

Configure, Options command.....	292
connecting	
wires to a bus .....	40
Copy Matrix, Edit command .....	236
Copy to File, Edit command.....	203
Copy, Edit command .....	203
Copy, Library command .....	321
copying.....	see also pasting
a matrix of objects.....	92
circuitry .....	56
multiple items between libraries.....	322
nets .....	91
objects .....	248
objects to a file .....	91
objects to the Clipboard.....	91
single items between libraries .....	322
creating	
backup files .....	17
connections between files.....	18
custom reports .....	120
DXF files .....	195
field sets .....	106
global nets .....	42
macros .....	12
matrix of objects.....	92
title sheets.....	101
CTRL/SHIFT behavior, setting.....	81
Current Line, Options command .....	313
current sheet, defined.....	12
Current Wire, Options command .....	312
cursor	
free floating.....	257
snappy .....	257
custom title sheets .....	98
Custom Toolbar .....	9
displaying.....	353
Custom Toolbar, View command.....	256
custom tools	
executing .....	354
Customize, Utils command.....	352
Cut, Edit command .....	203
cutting.....	see also deleting or pasting
objects from a design.....	90

**-D-**

DataTips	
defined .....	15

showing or hiding .....	15	Edit Net Class To Net Class Definitions .....	308
DDE Hotlinks .....	295	Edit Nets .....	241
about .....	127	Edit Part .....	239
Edit Parts .....	129	Field Properties .....	230
enabling .....	127	Find Errors .....	232
highlighting parts .....	129	Force Update .....	333
selecting highlighted objects.....	131	Hierarchy Navigator .....	255
unhighlighting parts .....	129	Library Alias.....	320
default file viewer .....		Library Archive Library.....	327
selecting a viewer .....	16	Library Browes .....	320
default pin designators.....	332	Library Copy.....	322
Delete, Edit command .....	234	Library Delete .....	324
Delete, Library command .....	323	Library Setup .....	325
Delete, Macro command .....	364	Line Properties .....	224
deleting .....		Line Selection Mask .....	289
ERC errors .....	137	Net Classes .....	306, 309
items in a block.....	288	Net Name.....	244
nets .....	246	Options Block Selection .....	287
objects.....	234	Options Configure .....	292
objects from nets.....	94	Options Current Line.....	313
ports .....	269	Options Current Wire .....	312
reports.....	178	Options Design Rules.....	303, 304, 307
the current sheet .....	20	Options Display .....	297
Descend, View command .....	255	Options Grid.....	295
Deselect All, Edit command .....	238	Options Preferences .....	301
design borders .....		Options Sheets.....	311
adding a design border .....	102	Options Text Style .....	314
border dimensions.....	98	Part Properties .....	233
design border, defined .....	98	Part Selection Mask .....	290
zones .....	99	Pin Properties .....	222
Design Info, File command .....	183	Pin Selection Mask .....	290
Design Rules, Options command .....	303	Place Attribute .....	243, 279
Design Technology Parameters .....		Place Field.....	281
dialog.....	188	Place IEEE Symbol .....	282
Design Technology Parameters, File .....		Place Part.....	260
command.....	188	Place Pin .....	269
dialogs .....		Place Port .....	267
Add Text Style.....	314	Place Properties.....	229
Arc Properties.....	225	Polygon Properties .....	226
Attributes.....	243	Port Properties.....	221
Attributes Properties .....	228	Port Selection Mask.....	291
Bus Connection Properties .....	266	Properties .....	229
Bus Name .....	245	Rename Style .....	315
Bus Properties .....	220	Save Component As .....	327
Change Attribute Value .....	306	Set By Attribute .....	305
Edit Copy Matrix .....	236	Symbol Save As .....	326

Text Properties .....	224
Text Selection Mask .....	292
Text Style Properties .....	315
Utils Electrical Rules Check .....	334
Utils Renumber .....	329
View Jump Location .....	253
View Jump Text .....	254
Wire Properties .....	218
Wire Selection Mask .....	292
Display, Options command .....	297
DocTool Commands .....	357
Notes .....	359
Place Table .....	357
Titles .....	359
Update .....	359
Update All .....	359
DocTool Toolbar .....	9
double-click display properties .....	94
draft mode for polygons .....	275
drag and drop .....	248
Drawing Interchange Format .....	190
DRC errors .....	176, 300
DTPs	
about .....	147
adding	
groups .....	150
items .....	153
sections .....	151
browsing .....	153
copying data .....	156
creating .....	148
deleting information .....	154
item specific info .....	158
modifying properties .....	159
opening .....	148
renaming	
groups and items .....	154
setting up attributes .....	158
tree, defined .....	189
updating design info .....	155
viewing item statistics .....	155
duplicating objects .....	236
DXF Files	
loading .....	190
supported items .....	192
DXF In, File command .....	190
DXF Out	

considerations .....	196
DXF Out, File command .....	195
DXF, defined .....	195

**-E-**

ECOs .....	294
exporting .....	340
formatting defined .....	17
importing .....	339
recording .....	338
types of changes .....	339
Edit Commands .....	201
Align Parts .....	237
Copy .....	7, 203
Copy Matrix .....	92, 236
Copy to File .....	203
Cut .....	7, 203
Delete .....	234
Deselect All .....	238
Explode Part .....	237
Highlight .....	238
Measure .....	246
Move By RefDes .....	209
Nets .....	240
Parts .....	239
Paste .....	7, 204
Circuit .....	206
Circuit From File .....	208
From Clipboard .....	205
From File .....	205
Properties .....	211
Redo .....	202
Select .....	247
Select All .....	238
Select Highlight .....	238
Undo .....	201
Unhighlight .....	238
Unhighlight All .....	238
editing .....	see also modifying
Arc properties .....	225
attributes .....	228
DTP attributes .....	160
DTP properties .....	159
Field properties .....	230
field results .....	105
fields .....	231
items in a block .....	288

Line properties.....	224	Export ECOs, Utils command.....	340
macros.....	367	exporting	
net attributes.....	95	Note Table notes.....	112
nets.....	94	Revision Note Table.....	115
Pin Designator.....	224	Extent, View command.....	251
Pin properties.....	222		
Polygon properties.....	226	<b>-F-</b>	
properties.....	52	features.....	1
reports.....	119	Field, Place command.....	280
tables.....	117	fields	
Text properties.....	227	code, defined.....	103
electrical pin type.....	217	Field Properties.....	230
Electrical Rule Check flags.....	232	field sets, defined.....	106
electrical rule checking.....	334	fields defined.....	103
Enabling		result, defined.....	103
DDE Hotlinks.....	127	rotating/flipping.....	281
Engineering Change Order (ECO).....	338	user-defined fields.....	183
ERC		file	
annotate errors.....	134	compression	
Bus/Net Errors.....	134	autosave.....	294
clear all overrides.....	336	binary designs.....	295
Component Errors.....	134	Design Info	
configuration.....	133	Attributes tab.....	185
Hierarchy Errors.....	336	Fields tab.....	184
Net Connectivity Errors.....	335	General tab.....	183
No Node Nets.....	134, 335	Revisions Tab.....	187
setting up checks.....	133	Statistics tab.....	188
severity levels.....	336	pasting from.....	93
Single Node Nets.....	134, 335	statistics.....	183
summarize ignored checks.....	135	viewer.....	16
summarize ignored errors.....	336	File Commands	
summarize overridden errors.....	336	Close.....	170
Unconnected Pins.....	335	Design Info.....	183
Unconnected Wires.....	134, 335	Design Technology Parameters.....	188
ERC Error, View command.....	232	DXF In.....	190
ERC errors		DXF Out.....	195
controlling display of.....	136	Exit.....	199
finding.....	135	New.....	169
fixing and deleting.....	137	Open.....	6, 169
overriding.....	137	PDIF In.....	197
ERC, Utils command.....	334	PDIF Out.....	198
error indicators		Print.....	172
overriding.....	137	Printer Setup.....	177
Existing module		Reports.....	177
reusing.....	348	Save.....	170
Exit, File command.....	199	Save As.....	171
Explode Part, Edit command.....	237	file viewer.....	295

Find Errors, Utils command .....337  
 fixing  
   ERC errors.....137  
   errors in a design.....66  
 flip object key .....380  
 Force Update, Utils command .....333

**-G-**

gate equivalence .....217  
 GateEq .....217  
 gates  
   last used .....116  
   replacing component types of.....233  
   spare .....116  
 Generate Netlist, Utils command....124, 342  
 generating  
   reports.....119  
 global net.....108  
 grid  
   snap cursor to.....257  
 grid options  
   absolute and relative settings .....296  
   Grid Combo box .....11  
   Grid Spacing .....296  
   Grid Style .....296  
   Grid Toggle button.....11  
   Mode .....297  
   Relative Grid Origin .....296  
 Grids, Options command .....295  
 Ground Table .....113

**-H-**

hardware requirements.....3  
 Help Commands .....375  
 help key.....379  
 help, online .....375  
 Hiding  
   DataTips .....15  
   toolbars .....7  
 Hierarchical schematics  
   ascend .....255  
   black box .....345  
   descend.....255  
   introduction.....345  
 highlight.....130, 240  
   an attached net.....130, 240  
   parts .....129  
 Highlight, Edit command .....238

hotlinks,enabling..... 295

**-I-**

icons  
   Command Toolbar..... 256  
   Placement Toolbar ..... 256  
 IEEE Symbol, Place command ..... 282  
 ignored errors  
   ERD ..... 336  
 imperial units ..... 293  
 Import ECOs, Utils command ..... 339  
 importing .....see also loading  
   Note Table notes..... 112  
   Revision Note Table..... 115  
 inside block  
   defined ..... 83  
 Installation and Setup  
   installing P-CAD products..... 4  
   system requirements..... 3  
 InterPlace/PCS, Utils command ..... 352

**-J-**

Jump Location, View command ..... 253  
 Jump Text, View command ..... 254  
 jumper pins.....262, 264, 268, 344

**-K-**

keyboard shortcut  
   drag-and-drop cut/paste..... 377  
   Schematic Reference..... 377  
   view center ..... 252  
   zoom out..... 252  
   zoom window ..... 252  
 Keyboard tab ..... 302

**-L-**

Last, View command..... 251  
 library  
   create an alias..... 320  
   creating new..... 319  
   definition ..... 319  
   deleting symbol/component from ..... 323  
   opening a ..... 325  
   opening with drag-and-drop file load 325  
   purpose ..... 319  
   renaming a symbol or component..... 324  
 Library Commands..... 319  
   Alias ..... 320

Archive Library .....	327
Copy .....	321
Delete.....	323
New.....	319
Rename .....	324
Setup .....	325
Symbol Save As.....	326
Line, Place command .....	272
lines	
line length, measuring.....	272
line shapes, orthogonal modes.....	273
properties.....	224
resizing .....	89
selection criteria .....	290
using orthogonal modes.....	88
linking	
references .....	229
loading .....	see also opening
DXF files.....	190
reports.....	178
Loading PDFIF files .....	197
local net .....	108
location, jumping to.....	253

**-M-**

Macro Commands .....	361
Delete.....	364
Record.....	12, 362
Rename .....	364
Run.....	365
Setup .....	361
macros	
edit events.....	371
editing .....	367
executing.....	363
features .....	366
file syntax.....	367, 371
keyboard codes .....	368
keyboard events .....	368
mouse events .....	367
recording tool .....	362
running.....	365, 366
special events.....	371
status line indicators.....	366
temporary.....	363
temporary macro name.....	363
using effectively.....	365

Manual, Rewire command .....	285
matrix, defined.....	92
maximize button .....	13
Measure, Edit command.....	246
menu bar	
defined .....	6
menu commands .....	6
merging	
net attributes.....	207
metric units.....	293
MillimeterPrecision setting.....	293
minimize button.....	13
modes	
changing absolute and relative grid ...	297
pairs, defined.....	88
modifying.....	see also editing
arcs .....	274
field results.....	105
objects .....	50
reports .....	119
tables .....	117
Module Wizard	
creating a link .....	346
creating a module.....	346
introduction .....	345
reusing an existing module .....	348
Module Wizard, Utils command .....	345
Mouse tab .....	302
Mouse-Click Behavior	
double-click displays properties.....	94
Move By RefDes, Edit command.....	209
moving	
a toolbar .....	7
an object .....	84
items in a block .....	288
objects .....	50
objects during placement.....	84
parts by RefDes .....	84, 209
ports.....	268
rotating and flipping objects.....	86
multiple selection, defined.....	79
Multisheet designs	
loading .....	21

**-N-**

net	
attributes	

merge when pasting .....	207
copy .....	91
creating named net classes .....	310
Delete command .....	246
deleting objects from .....	235
highlight.....	245
increment .....	294
increment value, defined.....	17
local vs. global .....	108
managing net connections;.....	95
netlist format	
Tango.....	397
netlist, generating.....	70, 124, 342
renaming.....	208, 244
Select command .....	246
unhighlight.....	245
net attribute selection.....	57
Net Classes, Options command .....	309
Net Index Tables .....	111
placing.....	111
Nets, Edit command .....	240
New, File command .....	169
New, Library command .....	319
New, Window command .....	373
Note Tables .....	112
adding notes .....	112
exporting notes.....	112
importing notes .....	112
placing.....	113
numbering pads.....	329

**-O-**

objects	
nets	
deleting.....	94
pasting.....	204
properties.....	94
selecting preferences .....	81
Open, File command .....	169
opening..... see also loading	
ASCII Files .....	22
menus.....	6
multisheet designs .....	21
Schematic Files (*.sch).....	21
Tango Series II Files .....	21
Options Commands	
Block Selection.....	287

Configure.....	292
Current Line .....	313
Current Wire .....	312
Design Rules .....	303
Display.....	297
Grids.....	295
Net Classes .....	309
Preferences.....	301
Sheets .....	311
Text Style.....	314
Options Design Rules	
Class to Class tab.....	308
Design tab.....	303
Net tab .....	307
Orthogonal Modes .....	273
45/90 Line-Line Mode .....	88, 294
90/90 Line-Line Mode .....	88, 294
enabling .....	294
enabling ortho modes .....	88
mode pairs, defined .....	88
ortho modes,defined .....	88
output	
netlist .....	124
outside block	
defined .....	83
overridden errors.....	336
Overriding	
error indicators.....	137

**-P-**

pad number .....	217
panning	
screen, adjusting .....	295
setting the autopan % display .....	17
part	
exploding.....	237
highlight an attached net .....	240
highlight/unhighlight .....	240
jump to .....	239, 240
list box .....	239
number.....	217
placing .....	32
power components .....	262
properties .....	239
selection criteria.....	290
splitting a net .....	262
Part Properties.....	212

Attributes Pin tab.....	218	display characteristics.....	271
Component Pin tab.....	216	Place Attributes dialog.....	243
Component tab.....	215	Place Commands.....	259
Symbol Pins tab.....	214	Arc273	
Symbol tab.....	213	Attribute.....	279
Text Styles button.....	214	Bus.....	265
Part, Place command.....	259	Field.....	280
Parts, Edit command.....	239	IEEE Symbol.....	282
Paste, Edit command.....	204	Line.....	78, 272
pasting		Part.....	259
behavior.....	204	Pin 78, 269	
circuits.....	206	Polygon.....	274
circuits from a file.....	208	Port.....	78, 267
Clipboard objects.....	93	Ref Point.....	275
from a file.....	93, 205	Text.....	275
from clipboard.....	205	Wire.....	78, 263
P-CAD Binary Files		Place Table, DocTool command.....	357
loading multisheet designs.....	21	Placement Toolbar.....	8
opening .SCH files.....	21	Placement Toolbar, View command.....	256
P-CAD CFG Files (*.cfg).....	21	placing.....	see also pasting
P-CAD Library Executive, Utils command.....	352	arcs.....	273
P-CAD Pattern Editor, Utils command.....	352	attributes.....	279
P-CAD PCB, Utils command.....	351	buses.....	40
P-CAD Schematic window, defined.....	5	custom fields.....	105
P-CAD Symbol Editor, Utils command.....	352	fields.....	280
P-CAD system messages.....	387	IEEE symbols.....	282
PDF files.....	197	inline ports.....	268
PDF In, File command.....	197	lines.....	272
PDF Out, File command.....	198	Net Index Tables.....	111
pin		Note Tables.....	113
designator.....	217	objects.....	78
modifying text.....	223	pins.....	269
moving.....	224	polygons.....	274
display.....	223	ports.....	41, 267
display characteristics.....	224	Power Table.....	114
equivalence.....	217	reference points.....	275
name.....	217	Revision Note Table.....	116
modifying text.....	223	Spare Gate Table.....	117
moving.....	224	text.....	45, 275
pin des, renumbering default.....	332	wires.....	36
renumbering.....	331	play macro key.....	380
selecting length.....	223	polygon	
selection criteria.....	290	altering the shape.....	275
Pin, Place command.....	269	complex.....	275
PinEq.....	217	draft mode.....	275
pins		query/modify.....	226
		resizing a polygon.....	89

rotating/flipping .....275  
 Polygon, Place command .....274  
 port  
   deleting.....269  
   moving .....268  
   placing inline .....268  
   renaming.....269  
   selection criteria .....291  
 Port Properties  
   Net tab.....221  
 Port, Place command .....267  
 Power Table.....113  
   placing.....114  
 Preferences, Options command .....301  
 preview ECO file .....340  
 previous view.....251  
 print  
   colors.....172, 174  
   colors setup .....142  
   Image Options, defined .....140  
   Image Scale, defined.....140  
   print job setup.....140  
   Print Region, defined .....140  
   scaling to fit page.....145  
   selecting a printer.....139  
   set up.....173  
   setting up.....139  
 Print Setup, File command.....177  
 Print, File command.....172  
 prompt line, defined .....11  
 Prompt Line, View command .....256  
 properties  
   arc 225  
   bus.....220  
   double clicking to access.....212  
   field .....230  
   line .....224  
   part.....212  
   pin 222  
   polygon .....226  
   pop-up menu .....211  
   port .....221  
   replacing component types .....233  
   text .....226  
   View ERC Errors.....232  
   wire .....218  
 Properties dialog

opening the dialog..... 94  
 Properties, Edit command ..... 211

**-R-**

Record ECOs, Utils command ..... 338  
 Record, Macro command ..... 362  
 recording a macro ..... 362, 363  
 Redo, Edit command..... 202  
 redraw, interrupting ..... 251  
 Redraw, View command..... 251  
 Ref Point, Place command..... 275  
 reference  
   launching a link ..... 160  
   locations ..... 229  
 reference designators ..... 217  
   renaming..... 207  
   renumbering ..... 54, 329  
 relative grid toggle ..... 296  
 removing ..... see deleting  
 Rename Net, Utils command..... 343  
 Rename, Library command ..... 324  
 Rename, Macro command ..... 364  
 renaming  
   a net..... 244, 343  
   while pasting..... 208  
   ports..... 269  
   reference designators ..... 207  
   sheet names ..... 20  
 renumber  
   pads ..... 329  
   reference designators ..... 329  
 Renumber, Utils command ..... 329  
 reordering  
   sheets in a design..... 19  
 replacing  
   component types ..... 233  
 report  
   Attributes ..... 118  
   Bill of Materials..... 119  
   custom  
     adding..... 120  
     formats ..... 179  
     selections..... 181  
     sorting ..... 182  
   extensions..... 178  
   format options ..... 121  
   generating..... 119, 123

Global Nets .....	119
Last Used RefDes .....	119
Library Content .....	119
options .....	178
origin .....	179
Parts Location .....	119
Parts Usage .....	119
selecting criteria .....	122
setting	
data selection options .....	121
sort options .....	122
style format .....	179
types .....	178
types of reports .....	118
Reports, File command .....	177
resizing	
objects .....	52, 89, 249
Resolve Hierarchy, Utils command .....	344
Revision Blocks .....	103
Revision Note Table .....	114
adding .....	115
exporting .....	115
importing .....	115
placing .....	116
Rewire Commands .....	285
Manual .....	285
rotating	
items in a block .....	288
objects .....	52
polygons .....	275
run temporary macro .....	365
Run, Macro command .....	365
Run, Simulate command .....	355
<b>-S-</b>	
Save As, File command .....	171
Save, File command .....	170
saving	
a symbol to a library .....	326
an ECO file .....	341
new name or location .....	171
Schematic Files (*.sch) .....	21
Schematic Sheets	
adding a sheet .....	19
changing a sheet name .....	20
defining the current sheet .....	19
deleting a sheet .....	20
reordering in a design .....	19
scroll bars, defined .....	13
search for text .....	254
searching	
RefDes list .....	211
Select All, Edit command .....	238
Select Highlight, Edit command .....	238
Select, Edit command .....	247
selecting	
a block .....	83
a default file viewer .....	16
a net .....	81
a printer or plotter .....	139
all objects .....	80
an object .....	78
choosing a selection color .....	82
collocated objects .....	80
contiguous net objects .....	81, 249
contiguous, defined .....	81
current highlight color .....	128
defining	
block selection criteria .....	83
selection preferences .....	81
error indicators .....	137
highlighted objects .....	80, 131
items in a block .....	288
items, status line info .....	247
multiple objects .....	79
nets .....	246
objects .....	46, 248
overlapping objects .....	248
part of an object .....	79
select actions	
copying objects .....	248
drag and drop .....	248
moving .....	248
resizing .....	249
rotating and flipping .....	249
Selection Mask	
dialog .....	83
parameters .....	290
specifying a selection point .....	82, 86, 87
setting up	
display options .....	29
ERC options .....	63
grid spacing .....	30
print jobs .....	140

printers and plotters .....139

Setup, Library command .....325

Setup, Macro command .....361

Setup, Simulate command.....355

severity levels.....134

ERC.....336

sheet

adding .....29, 312

connectors

annotating.....108

cross-referencing .....107

definition.....44, 108

example .....107

jump to.....110

viewing properties .....110

enabling/disabling.....311

re-ordering .....312

Sheets, Options command .....311

Shortcut Directory, Utils command.....351

shortcut menu, defined.....13

Showing

DataTips .....15

toolbars .....7

Simulate Commands .....355

Run.....355

Setup.....355

single selection, defined .....78

Snap to Grid, View command .....257

software requirements .....3

Spare Gate Table .....116

placing.....117

splitting a net .....59

standard title sheets .....98

status line

defined .....11

information .....247

information, defined.....12

Status Line, View command .....257

subselection, defined .....79

summarize overridden errors

ERC.....336

swapping

pin and gate .....217

switching

between sheets .....35

symbol

copying .....322

deleting from a library ..... 323

pin217

renaming..... 324

Symbol Save As, Library command ..... 326

system

requirements ..... 3

**-T-**

tables

modifying..... 117

updating ..... 118

tabs

Attributes, Design Info ..... 185

Attributes, Part Properties..... 218

Class to Class, Options Design Rules. 308

Colors, Options Display ..... 297

Colors, Print Options ..... 142, 174

Component Pins, Part Properties ..... 216

Component, Part Properties..... 215

Design, Options Design Rules ..... 303

Fields, Design Info ..... 184

Format, Customize Report..... 179

General, Design Info ..... 183

Keyboard, Options Preferences ..... 302

Miscellaneous, Options Display.... 29, 176

Miscellaneous, Print Options..... 143

Mouse, Options Preferences ..... 302

Net Class, Options Design Rules..... 304

Net, Options Design Rules ..... 307

Net, Port Properties ..... 221

Net, Wire Properties..... 219

Notes, Design Info ..... 186

Port, Port Properties ..... 221

Port, Port Selection Mask ..... 291

Revision, Design Info ..... 187

Selection, Customize Report Attributes

..... 181

Sheet Connector, Part Properties..... 110

Sheets, Options Sheets..... 311

Sort, Customize Report Attributes..... 182

Statistics, Design Info..... 188

Symbol Pins, Part Properties..... 214

Symbol, Part Properties ..... 213

Wire, Wire Properties ..... 218

Wire, Wire Selection Mask ..... 292

Tango

Tango designs

- translating ..... 383
  - Tango netlist format.....397
  - Tango Series II ASCII Files (\*.s01) .....21
  - Tango Series II Files
    - loading.....21
  - Technology Files ..... 188
  - temporary macro .....363, 365
  - text
    - flipping.....227, 278
    - jump to (searching for text) .....254
    - justification .....227, 229, 244, 277
    - location .....227
    - properties.....226
    - rotating .....227, 277, 278
    - selection criteria .....292
    - style
      - adding .....314
      - default styles .....314
      - deleting.....315
      - display options.....317
      - display TrueType .....317
      - properties .....315
      - renaming .....315
      - stroke font properties .....316
      - TrueType fonts.....316
    - zooming/panning .....278
  - Text Style, Options command .....314
  - Text, Place command .....275
  - Tile, Window command .....373
  - title sheets .....98
    - adding .....293
    - design borders .....98
    - setting up.....24
    - title blocks, defined .....99
    - using a standard title sheet .....100
    - zones, defined.....99
  - toolbars
    - Command Toolbar .....8, 256
    - Custom Toolbar .....9
    - docked toolbar, defined .....7
    - DocTool Toolbar.....9, 10
    - floating toolbar, defined .....7
    - moving a toolbar.....7
    - Placement Toolbar .....8, 256
    - showing or hiding a toolbar .....7
  - touching block
    - defined.....83
  - translating
    - Tango designs..... 383
  - tutorials
    - generating reports and netlists.....69
    - printing a design .....72
    - setting up a design .....23
    - verifying a design.....63
    - working with objects .....32
  - type swapping..... 213
- U-**
- underscored letter, defined .....6
  - Undo, Edit command ..... 201
  - Undo, shortcut command..... 381
  - unhighlight ..... 130
    - an attached net..... 130, 240
    - parts ..... 129
  - Unhighlight All, Edit command..... 238
  - Unhighlight, Edit command..... 238
  - units of measurement
    - changing ..... 293
    - choosing a scale .....28
  - User's Guide
    - about the guide .....2
  - using
    - Edit Nets dialog .....95
    - title sheets
      - custom ..... 102
      - global.....100
      - standard .....100
  - Utils Commands..... 329
    - Customize ..... 352
    - ERC ..... 334
    - Export ECO ..... 340
    - Find Errors ..... 337
    - Force Update ..... 333
    - Generate Netlist ..... 124, 342
    - Import ECOs ..... 339
    - InterPlace/PCS ..... 352
    - Module Wizard..... 345
    - P-CAD Library Executive..... 352
    - P-CAD Pattern Editor..... 352
    - P-CAD PCB ..... 351
    - P-CAD Symbol Editor ..... 352
    - Record ECOs..... 338
    - Rename Net..... 343
    - Renumber ..... 329

Resolve Hierarchy.....344  
 Shortcut Directory.....351

**-V-**

View Commands.....251  
 All 252  
 Ascend.....255  
 Center .....252  
 Command Toolbar .....8, 256  
 Custom Toolbar .....9, 256  
 Descend.....255  
 DocToolbar .....9  
 Extent.....251  
 Jump Location .....253  
 Jump Text.....254  
 Last .....251  
 Placement Toolbar.....8, 256  
 Prompt Line.....256  
 Redraw .....251  
 Snap to Grid.....257  
 Status Line .....257  
 Toolbar.....7  
 Zoom Out.....252  
 Zoom Window.....252  
 view ECO file.....341  
 viewing  
   errors in a design.....65

**-W-**

Window Commands .....373  
 1,2 (other windows).....374  
 Arrange Icons .....373  
 New Window .....373  
 Window Cascade .....373

Window Tile ..... 373  
 wire  
   adding to nets ..... 264  
   orthogonal modes ..... 264  
   properties ..... 218  
   resizing..... 89  
   selection criteria..... 292  
   setting wire width ..... 219  
   width ..... 219  
 Wire Properties  
   Net tab ..... 219  
   Text Style button ..... 219  
 Wire, Place command ..... 263  
 working with objects..... 77  
 workspace  
   defined ..... 10  
   setting size..... 23  
   setting the autopan percent display ..... 17  
   setting the current zoom factor..... 17  
   size ..... 293

**-Z-**

Zones  
   zone intelligence, defined ..... 99  
   zones, defined ..... 99  
 zoom  
   current zoom factor, defined ..... 17  
   in and out..... 31  
   zoom factor, defined..... 295  
   zoom window key..... 381  
   zooming, key..... 379  
 Zoom Out, View command ..... 252  
 Zoom Window, View command ..... 252