

PAIZI

SailWind Logic Guide

Release SailWind 3.0
Document Revision 1.2

Copyright and Disclaimer of SailWind Software
Copyright © 2023-2025 Chengdu Paizi Interconnect Electronics Technology Co., Ltd.

Copyright Information

All copyrights, patent rights, trademark rights, trade secrets, and other related intellectual property rights of the SailWind software (hereinafter referred to as the "Software"), including but not limited to its source code, object code, user interface design, graphics, images, audio, video, algorithms, data models, documentation, etc., belong to Chengdu Paizi Interconnect Electronics Technology Co. Ltd. (hereinafter referred to as the "Copyright Owner").

Installation and Use License

Users should clearly agree to all terms of this copyright and disclaimer before installing and using this software. By running this installation or software, the user indicates that they have read and agree to be bound by this copyright and disclaimer.

The copyright owner grants users a non exclusive, limited, and revocable installation license, allowing them to install the software on their designated computer devices and use its related features to complete their design tasks under the guidance of the software.

Users are not allowed to copy, distribute, modify, sell, rent, lend, transfer, reverse engineer, decompile, create derivative works, or otherwise use this software in any form, except with the explicit written permission of the copyright owner.

Disclaimer During Installation and Use

This software is provided as is, and the copyright owner does not guarantee that it is error free, defect free, and does not guarantee that the installation and use process will be successfully completed, nor does it make any commitment to the applicability, stability, security, or reliability of the installation and use process.

Users should bear the risk of using this software themselves. The copyright owner shall not be liable for any direct or indirect losses, data loss, business interruption, system damage, or other damages caused by the use or inability to use this software.

Limitations and Reservations of Rights

The use of this software is subject to the limitations and constraints of this copyright and disclaimer. The copyright owner reserves all rights not explicitly granted to users. Users are not allowed to perform any form of reverse engineering, decompilation, disassembly, decryption, modification, creation of derivative works, or use to create similar software on this software.

Other

The copyright owner has the right to modify the terms of this copyright and disclaimer at any time, and the modified terms will be notified to users through appropriate means. If the user continues to use this installation and software, it means that they have accepted the modified terms.

If any part of this copyright and disclaimer is deemed invalid or unenforceable for any reason, that part shall be deemed separate from the whole, but shall not affect the validity of other parts.

Based on the permanently authorized PADS® software of Siemens Industry Software Inc.

Contact Information

If you have any questions or suggestions about this installation or software, please contact:

Email: market@pzeda.com
Phone: 0755-86703052
Website: www.pzeda.com

Revision History

Revision	Changes	Date
1.0	Initial release, corresponding to SailWind V3.0	2024-03-25
1.1	<ul style="list-style-type: none">Added Section "Comparing Part Attributes for Consistency Checking", and updated descriptions in Chapter "CIS".Added descriptions on suite configuration in "Help > Installed Options" menu.Updated configurations for exporting the BOM report.	2024-09-27
1.2	Updated software UI figure and field descriptions for "SailWind Layout Link Dialog Box, Design Tab".	2025-03-21

Table of Contents

Chapter 1

SailWind Logic QuickStart.....**23**

Step 1 - Start a New Design.....	23
Step 2 - Select the Sheet Size.....	23
Step 3 - Add Parts and Connector Symbols.....	24
Adding a Part.....	24
Adding a Connector Symbol.....	24
Step 4 - Add Buses.....	25
Step 5 - Add Connections to Parts, Connectors, and Buses.....	26
Step 6 - Add Off-Page Symbols.....	26
Step 7 - Add Power and Ground Symbols.....	27
Step 8 - Print the Schematic.....	27
Step 9 - Generate Reports.....	27
Step 10 - Create a Layout Netlist.....	28

Chapter 2

Getting Started With SailWind Logic.....**29**

SailWind Logic Flow.....	29
Startup Options.....	30
Adding Startup Options to a Shortcut.....	31
SailWind Updates.....	33
Downloading the Update.....	33
Disabling the Check for Updates.....	33
Checking for Updates Manually.....	33
Migrating User Settings.....	34
Customizing SailWind Logic Default Settings.....	34
Mouse Button Operations.....	34
Using the Numeric Keypad to Control the View.....	35

Chapter 3

User Interface.....**37**

View Control.....	38
View Commands and Scroll Bars.....	38
Middle Mouse Button.....	38
Numeric Keypad.....	39
Saving and Restoring Views.....	40
Saving a View.....	40
Restoring a View.....	40
Project Explorer.....	41
Output Window.....	42
Session Log Management.....	43
Session Log.....	43
Navigating Pages in the Status Tab.....	43
Filtering the Status Tab Display.....	44

Table of Contents

Searching in the Status Tab.....	44
Printing Session Log Messages.....	45
Viewing and Printing Reports.....	45
Saving a Session Log to File.....	45
Clearing the Session Log Display.....	46
Macros.....	47
Creating a New Macro.....	47
Recording Mouse Movements.....	47
Opening an Existing Macro File.....	48
Viewing Multiple Open Macros.....	48
Editing a Macro.....	48
Changing the Default Text Editor.....	49
Saving the Macro.....	49
Macro Playback.....	50
Playing Back a Macro.....	50
Pausing a Playing Macro.....	50
Stopping a Playing Macro.....	50
Macro Script Debug.....	51
Setting or Removing Breakpoints.....	51
Debugging the Macro Scripts.....	52
Play a Single Line of the Macro.....	52
Perform a Subroutine Call on the Current Line.....	52
Play Back a Macro to a Point.....	52
Return From the Subroutine to the Point From Which it was Called.....	52
Continue the Execution From the Current Point.....	53
Run-Time Error Correction.....	53
Accessing Help on the Macro Language.....	53
CIS.....	54
Adding a New Configuration.....	54
Adding Parts from CIS.....	55
Comparing Part Attributes for Consistency Checking.....	56
Opening a File That is Already in Use.....	57

Chapter 4 File Operations..... 59

Creating a New Schematic File.....	59
Saving a Schematic File.....	59
Opening a Schematic File.....	60
File Import and Export.....	61
Import File Types.....	61
Importing a File.....	62
Exporting Files.....	63
OLE Object Import and Export.....	65
Importing OLE Files.....	65
Exporting OLE Files.....	65
ASCII File Format.....	66
Exporting to ASCII Output.....	66
Archiving Your Schematic.....	67

Chapter 5

Design Setup.....	69
Setting Options.....	70
Setting Schematic Editor Options.....	70
Creating a Backup File.....	70
Setting Part Editor Options.....	71
Preserving Reference Designators.....	72
Work Area and Grid Settings.....	74
Display Grid.....	74
Origin and Design Grid.....	74
Labels and Text Grid.....	74
Setting Display Colors.....	74
Setting Fonts.....	77
Choosing Stroke Font or System Fonts.....	78
Stroke Font.....	78
System Fonts.....	78
Setting Stroke Font Options.....	79
Setting System Font Options.....	80
Setting Line Widths.....	80
Converting Stroke-to-System Fonts.....	81
Converting System-to-Stroke Fonts.....	82
Managing Font Replacement.....	83
Automatic Font Replacement.....	83
Manual Font Replacement.....	84

Chapter 6

Managing Libraries and Library Data.....	85
Convert PADS Libraries to the Current Format.....	85
Creating a Library.....	86
Displaying Items in a Library.....	86
Modifying Library Data.....	88
Adding Items to a Library.....	88
Deleting Items From a Library.....	89
Copying a Library Item.....	90
Editing Items in a Library.....	91
Deleting All Items in a Library.....	92
Transferring Library Data.....	93
Setting Library Availability and Search Options.....	94
Adding Libraries to the Library List.....	94
Removing Libraries From the Library List.....	94
Library Content and the Search Order.....	95
Setting the Library List Order.....	95
Sharing a Library Across a Network.....	96
Controlling Library Search Access.....	96
Protecting Library Files.....	96
Synchronizing With SailWind Layout.....	97
Managing Library Attributes.....	98

Table of Contents

Adding an Attribute to Multiple Library Items.....	98
Deleting Attributes From Library Items.....	99
Renaming Attributes of Library Items.....	99
Importing and Exporting Libraries.....	101
Importing Library Data.....	101
Exporting Library Data.....	102
Creating Library Reports.....	103
Creating a Report of the Parts in a Library.....	103
Creating a Report of Decals, Lines or Logic Symbols in a Library.....	104
Wildcards and Expressions.....	105
Library Search Order.....	106

Chapter 7 **Library Parts.....** **109**

Part Editor Operations.....	109
Object Selection Control in the Decal Editor.....	111
Controlling Object Selection Using Preset Filter Settings.....	111
Controlling Object Selection Using the Selection Filter Dialog Box.....	111
Changing and Updating Library Parts.....	112
Creating New Parts from Existing Parts.....	112
Save the Part/Decal to the Library.....	113
Selecting Multiple Objects in the Decal Editor.....	113
CAE Decals.....	115
Constructing the New CAE Decal.....	115
Using the Decal Wizard.....	116
Manually Construct the New Part.....	117
Creating a New CAE Decal.....	119
Creating Single Gate Parts.....	119
Creating Multigate Parts.....	121
Adding Terminals.....	121
Changing Objects in the Decal Editor.....	121
Setting a Pin Number.....	122
Setting a Pin Name.....	123
Setting a Pin Type.....	124
Setting a Pin Type Using Verb Mode.....	124
Setting a Pin Type Using Object Mode.....	124
Setting Pin Swaps.....	126
Setting Pin Swaps Using Verb Mode.....	126
Setting Pin Swaps Using Object Mode.....	126
Changing a Pin Number.....	127
Changing Pin Numbers Using Verb Mode.....	127
Changing Pin Numbers Using Object Mode.....	127
Changing a Pin Name.....	128
Changing Pin Names Using Verb Mode.....	128
Changing Pin Names Using Object Mode.....	128
Changing a Pin Decal.....	129
Changing Pin Decals Using Verb Mode.....	129
Changing Pin Decals Using Object Mode.....	129

Table of Contents

Changing Sequence Numbers.....	129
Attribute Labels.....	130
Creating Attribute Labels.....	130
Modifying Terminals.....	131
Setting the Origin for a Part.....	131
Getting Gate Decals From the Library.....	132
Assigning Pin Information to the CAE Decal.....	132
Saving a Modified Decal With a Different Name.....	133
Part Types.....	134
Modifying Electrical Information for a Part.....	134
Viewing and Setting General Part Information.....	136
Viewing Part Statistics.....	136
Setting the Logic Family.....	136
Setting General Part Information.....	137
Editing Logic Families.....	139
Adding a Logic Family.....	139
Changing the Name or Prefix of a Logic Family.....	139
Deleting a Logic Family.....	139
Assigning PCB Decals.....	141
Assigning an Existing Decal.....	141
Assigning a New Decal.....	142
Unassigning a Decal.....	142
Changing the Default Decal.....	143
Resetting the Tab Data.....	143
Assigning Gates to Parts.....	144
Gate Decal and Alternates.....	144
Gate and Pin Swap Information.....	144
Assigning Gates to a Part.....	144
Assigning CAE Decals to Gates.....	145
Part Information - Pins.....	147
Adding One or More Pins to a Part.....	148
Adding a Single Pin.....	148
Adding a Series of Pins.....	148
Assigning a Decal.....	149
Pasting Pin Information.....	149
Importing Pins Using a CSV File.....	149
Editing Pin Data.....	150
Assigning a Signal Pin.....	151
Assigning an Unused Pin.....	152
Sorting Table Data.....	152
Renumbering Pins.....	152
Deleting Pins.....	153
Error Checking.....	153
Signal Pin Nets.....	153
Managing Attributes.....	154
Adding an Attribute.....	154
Modifying an Attribute.....	154
Pasting Attribute Information.....	155
Deleting an Attribute.....	155

Table of Contents

Setting Default Attributes.....	155
Adding an Attribute From the Attribute Library.....	156
Resetting the Attribute List.....	156
Browsing Library Attributes.....	157
Part Information - Pin Mapping.....	158
Mapping Alphanumeric Pin Numbers to Numeric Decals.....	158
Unmapping Pins.....	159
Checking the part.....	159
Assigning Alternate Logic Decals for Connector Symbols.....	159
Saving Part Types.....	160
Library Management for Saved Part Types.....	161
Saving a Modified Part Type With a Different Name.....	162
Creating a New Connector.....	162
Browsing for Connectors.....	163
Creating a New Pin Decal.....	164
Create a New Pin From an Existing Pin Decal.....	164
Create a New Pin Decal.....	164
Editing Objects in the Decal Editor.....	165

Chapter 8

Special Schematic Symbols..... **167**

Special Symbol Naming Conventions.....	167
Assigning Alternative Symbols for the Ground Part.....	168
Assigning Alternative Symbols for the Off-Page Part.....	169
Assigning Alternative Symbols for the Power Part.....	170
Creating New Special Symbols.....	170
Creating New Special Symbols From Existing Symbols.....	172
Updating Special Symbol Mappings.....	173
Management of Special Symbols in the Library.....	174

Chapter 9

Design and Editing Basics..... **175**

Design Operations.....	175
Modes of Operation.....	176
Editing Basics.....	177
Verb Mode (Select Command First).....	177
Object Select Mode (Select Object First).....	177
Zooming.....	179
Zooming In.....	179
Zooming Out.....	179
Specify the Zoom Area.....	179
Using Duplicate Mode.....	179
Using Delete Mode.....	180
Using Move Mode.....	180
Selecting Objects.....	182
Selecting One Object.....	182
Selecting Several Objects.....	182
Selecting All Objects in an Area.....	183

Selecting All Objects on a Sheet.....	183
Selecting an Object on All Sheets in a Schematic.....	183
Controlling Selections.....	183
Filtering Object Selections.....	184
Filtering With the Selection Toolbar.....	184
Filtering With the Popup Menu.....	184
Filtering With the Selection Filter Dialog Box.....	184
Using the Selection Filter.....	185
Selection List.....	186
Searching for an Object by Typing Information.....	186
Selecting an Object With Area Selection.....	186
Find Objects.....	187
Step and Repeat.....	187

Chapter 10

Schematic Parts..... **189**

Adding Parts.....	189
Adding Connector Pins.....	190
Adding Single Gate Parts.....	190
Adding Multigate Parts.....	190
Controlling Text Visibility for a Part.....	191
Using Alternate Symbols.....	192
Swapping Reference Designators.....	193
Swapping Reference Designators Using Verb Mode.....	193
Swapping Reference Designators by Selecting One Part at a Time.....	193
Swapping Reference Designators by Selecting Both Parts First.....	193
Saving Part Types to a Library.....	195
Saving One or More Part Types in the Schematic.....	195
Saving All of the Part Types in the Schematic.....	195
Saving Off-Page to Library.....	196
The Update From Library Function.....	197
Undoing an Update.....	198
Updating a Schematic From the Library.....	198
Updating Selected Part Types From the Library.....	199
Updating Selected CAE Decals From the Library.....	200
Updating Selected Pin Decals From the Library.....	202
Updating Selected Pin Decals Using the Update From Library Dialog Box.....	202
Updating Selected Pin Decals Using the Schematic Editor.....	202
The Compare/Update Process.....	203
How to Read the Update Report.....	206
Deleting a Part.....	213
Deleting a Part and Its Connections.....	213
Cut, Copy, and Paste.....	214
Copy as Bitmap.....	214
Creation of Groups.....	215
Creating a Group.....	215
Setting the Origin of Groups.....	216
Group Anchor Points.....	216

Table of Contents

Management of Groups.....	219
Duplicating an Existing Group.....	219
Moving Groups.....	219
Deleting Groups.....	220
Saving Groups.....	220
Pasting Groups From a File.....	221
Automatic Connection When Pasting.....	221
Rotate or Mirror a Group.....	222
Attributes Overview.....	222
Manage Attributes in a Schematic.....	224
Creating Attributes.....	224
Renaming Attributes.....	224
Deleting Attributes.....	225
Resistor Values Defined on Parts.....	225
 Chapter 11	
Sheets.....	227
Editing Sheets.....	227
Creating a Custom Sheet Border.....	227
Adding a Field.....	228
Changing a Text String Into a Field.....	229
Managing Fields.....	230
Managing All Fields in the Schematic.....	230
Managing the Fields in a 2D Line Item.....	230
 Chapter 12	
Non-Electrical Objects.....	233
Creating 2D Line Items.....	233
Adding Text.....	234
Moving Text.....	235
Adding Circles.....	235
Adding Polygons or Paths.....	235
Adding Rectangles.....	236
Modifying 2D Line Items.....	236
Pulling Arcs.....	237
Combining 2D Lines and Text.....	237
Uncombining 2D Lines and Text.....	239
Exploding Combinations.....	239
Adding Drafting Items to a Library.....	240
Adding Drafting Items From a Library.....	240
Modifying Objects in a 2D Lines Library.....	240
 Chapter 13	
Connections.....	243
Adding Connections.....	243
Naming a Connection.....	244
Net Attributes Overview.....	245

Table of Contents

Creating Net Attributes.....	245
Default Attributes for Nets.....	246
Adding Power and Ground Connections.....	247
Working With Floating Connections.....	249
Enabling Floating Connections.....	249
Disabling Floating Connections and Running the Connectivity Report.....	249
Adding Floating Connections.....	251
Creating a Dangling Connection.....	251
Creating a Floating Connection.....	251
Editing Floating Connections.....	251
Duplicating Floating Connections.....	252
Creating Floating Connections When Deleting Objects.....	252
Attaching Objects to Floating Connections.....	252
Adding Off-Page References.....	252
Editing Off-Page Symbol Sheet Numbers Per Line.....	253
Changing a Connection.....	253
Moving Connections.....	254
Splitting a Connection.....	254
Splitting Segments.....	254
Swapping Pins.....	256
Swapping Pins Using the Swap Pins Button in Verb Mode.....	256
Swapping Pins by Selecting One Pin at a Time.....	256
Swapping Pins by Selecting Both Pins First.....	257
Managing Buses.....	258
Choosing the Bus Type.....	258
Naming Buses.....	258
Adding Buses.....	259
Adding Connections to Buses.....	260
Adding Nets to a Mixed Net Bus.....	260
Extending Buses.....	260
Moving Bus Segments.....	261
Splitting Buses.....	261
Deleting Bus Segments.....	261
Deleting Buses.....	263
Delete a Bus Only.....	263
Delete a Bus and Its Connections.....	263
Deleting Connections.....	264
Detach.....	264

Chapter 14

Hierarchical Design..... 267

Hierarchical Design Overview.....	267
Creating a Top-Down Hierarchy.....	267
Creating a Bottom-Up Hierarchy.....	268
Pushing Into the Hierarchy.....	269
Popping Up the Hierarchy.....	270
Modifying a Hierarchical Symbol.....	270
Copying a Hierarchical Symbol.....	270

Table of Contents

Deleting a Hierarchical Symbol.....	271
-------------------------------------	-----

Chapter 15

Schematic Object Modification..... **273**

Modifying Drafting Objects.....	273
Modifying Fields.....	274
Modifying Parts.....	276
Changing the Reference Designator.....	276
Changing the Part Type.....	276
Changing Part Information.....	277
Changing the Visibility of Text.....	277
Changing Part Attributes.....	278
Changing PCB Decals.....	278
Assigning Unused Pins as Signal Pins.....	279
Reference Designator Renumbering.....	280
Automatically Renumbering Reference Designators.....	280
Setting Reference Designators by Sheet in a New Schematic.....	281
Setting Reference Designators by Sheet in Completed Schematics.....	282
Modifying Part Attributes.....	283
Modifying Part Attribute Labels.....	284
Modifying Part Type Labels.....	285
Searching the Library for a Decal.....	285
Rename Part.....	286
Rename Gate.....	286
Modifying Reference Designator Labels.....	287
Modifying Pins.....	288
Modifying Pin Label Fonts.....	288
Modifying Nets.....	289
Modifying Net Name Labels.....	290
Modify Buses.....	292
Changing the Name of a Bus.....	292
Changing the Bus Type.....	292
Managing Bus Nets.....	293
Modifying Bus Name Labels.....	294
Modifying Off-Page Labels.....	295
Modifying Label Font Sizes.....	295
Modifying Text.....	296
Modifying Hierarchical Components.....	296

Chapter 16

Rules..... **299**

Rules Setup.....	299
Setting Up Clearance Rules.....	300
Same Net Matrix.....	301
Routing Rules.....	302
Setting Up High-Speed Rules.....	302
Setting Up Rules.....	304
Rules Hierarchy.....	305

Table of Contents

Setting Up Default Rules.....	306
Setting Up Class Rules.....	306
Setting Up Net Rules.....	307
Setting Up Conditional Rules.....	307
Creating Differential Pairs.....	308
Differential Pair Layer Hierarchy.....	310
Creating a Rules Report.....	310
Import Rules from PCB.....	311
Export Rules to PCB.....	311
Chapter 17	
Reports.....	313
Generating Reports.....	313
The Bill of Materials Report.....	314
Setting Up the Bill of Materials Attributes.....	314
Setting Up the Bill of Materials Format.....	315
Setting Up the Bill of Materials Configuration.....	316
Chapter 18	
Working With SailWind Layout and SailWind Router.....	319
Creation of a New PCB Layout from a SailWind Logic Design.....	320
Automatic Netlist Process Using the SailWind Layout Link.....	320
Manual Netlist Process Between SailWind Logic and SailWind Layout.....	321
Interpreting and Resolving the Netlist Process Error Report.....	322
Cross-Probe Between Sailwind Products.....	324
Cross-Probing With SailWind Layout.....	324
Cross-Probing With SailWind Router.....	324
Forward Annotation From SailWind Logic to SailWind Layout.....	326
Automated Forward Annotation Process.....	326
Generating the ECO File in SailWind Layout.....	327
Generating the ECO File in SailWind Logic.....	328
Forward Annotation Results.....	330
Backward Annotation From SailWind Layout to SailWind Logic.....	332
Automated Backward Annotation Process.....	332
Creating the ECO File in SailWind Layout.....	333
Creating the ECO File in SailWind Logic.....	334
Backward Annotation Results.....	337
Attribute Level Backward Annotation.....	337
Part Level Backward Annotation.....	337
Gate Level Backward Annotation.....	338
Net Level Backward Annotation.....	339
Pin Level Backward Annotation.....	339
Contents of the Differences Report.....	340
ECO File Format.....	342
Chapter 19	
Plotting and Printing.....	347

Table of Contents

Setting Printing and Plotting Output Options.....	347
Previewing Your Output.....	348
Creating a PDF.....	348
Plotting Output.....	350
Setting Up a Pen Plotter.....	350
Adding a New Pen Plotter.....	351
Setting Up a Photo Plotter.....	352
Printing Output.....	353
Printing to PDF.....	354
Setting the Printer to Print to a File.....	354

Chapter 20

Object Linking and Embedding..... 357

Insertion of OLE Objects in SailWind Logic.....	358
Inserting a New Embedded OLE Object.....	358
Inserting an Existing File as an Embedded Object.....	359
Inserting an Existing File as a Linked Object.....	360
Embedding a Text Document.....	360
Turning Off the Display of OLE Objects.....	361
OLE Object Selection.....	361
Moving and Sizing OLE Objects.....	362
Changing an OLE Object's Icon or Label.....	362
Converting an OLE Object to Another Type.....	362
Edits of OLE Objects.....	364
Cut, Copy, and Paste SailWind Logic OLE Objects.....	364
Edit OLE Links.....	366
Changing a Linked OLE Object's Source File.....	366
Breaking the Link to a Linked OLE Object's Source File.....	366
Setting the Update Mode For a Linked OLE Object.....	367
Manually Updating a Linked OLE Object.....	367
Open, Edit, Convert OLE Objects.....	368
Editing an OLE Object's Content in SailWind Logic.....	368
OLE and Print/Plot.....	370
Deleting OLE Objects.....	370
Redraws of a Screen Containing OLE Objects.....	370
OLE and View Menu Commands.....	370
Changing the OLE Object Background Color.....	371
Saving OLE Objects.....	371

Chapter 21

Using Advanced Procedures..... 373

Spice Simulation.....	374
Analog Schematics for Simulation.....	375
Add SPICE Attributes to Library Parts Versus Schematic Parts.....	375
Adding SPICE Attributes to Schematic Parts or Nets.....	375
Creating a SPICE Netlist.....	376
Setting Up AC Analysis.....	376
Setting Up DC Source Sweep Analysis.....	377

Table of Contents

Setting Up the SPICE Netlister.....	378
Setting Up Transient Analysis.....	378
Apply Attributes to Your Analog Design.....	379
Basic Scripting.....	380
Managing Scripts.....	381
Opening an Existing Script.....	381
Manage Open Scripts.....	381
Editing a Script.....	382
Editing a User Dialog Box.....	382
Finding an Automation Statement.....	382
Watching a Variable.....	383
Creation of Scripts.....	384
Creating a Script.....	384
Inserting an Automation Statement Using the Object and Procedure Lists.....	384
Inserting an Automation Statement Using the ActiveX Automation Members Dialog.....	384
Setting the Next Statement.....	385
Showing the Next Statement.....	385
Run Scripts.....	386
Running a Script.....	386
Pausing a Running Script.....	386
Stopping a Running Script.....	386
Debug Scripts.....	387
Setting or Removing the Breakpoints.....	387
Debug the Scripts.....	387
Removing All Breakpoints in the Script.....	388
Correction of Run-Time Errors.....	388
Accessing Help on the Basic Language.....	388
Using the Basic Scripts Dialog Box.....	388
Manage the Sax Basic Engine.....	389
Basic Sample Scripts.....	391
Basic Sample Scripts 00 Through 11.....	391
Basic Sample Scripts — RGL Reports.....	392
Basic Sample Scripts — Advanced.....	392
Managing Licensed Options.....	394
Viewing a License File or License Status.....	394
License File Definition.....	394

Chapter 22 Custom Interface..... **397**

Customizing the SailWind Interface.....	398
Customizing Toolbars.....	399
Creating a Custom Toolbar.....	399
Showing or Hiding a Toolbar.....	400
Deleting a Custom Toolbar.....	400
Renaming a Custom Toolbar.....	400
Resetting Toolbars to Defaults.....	401
Creation of Custom Commands.....	402
Creating a Custom Command.....	402

Table of Contents

Defining Properties for a New Command.....	403
Editing a Custom Command.....	403
Deleting a Custom Command.....	404
Creating a Custom Menu.....	404
Adding Items to Toolbars and Menus.....	405
Moving Buttons on Toolbars.....	405
Moving Items on Menus.....	406
Removing Items From Toolbars and Menus.....	406
Customizing Shortcut Keys.....	408
Creating a New Shortcut Key.....	408
Listing Shortcut Keys.....	409
Expressions in Shortcut Keys.....	409
Deleting a Shortcut Key.....	410
Resetting Default Shortcut Keys.....	411
Assigning Shortcut Keys to Macros.....	411
Creating a Command From a Macro and Adding it to a Menu.....	412
Customized Appearance of the Screen.....	413
Resizing the Sheets List.....	413
Organizing Windows.....	414
Showing Windows.....	414
Hidden Windows.....	415
Closing Windows.....	415
Hiding Windows Automatically.....	415
Detaching Windows From the Current View.....	416
Attach Windows to the Current View.....	417
Docking to the Last Location.....	417
Docking to a New Location.....	417
Embed Windows Within Other Windows.....	419
Two Windows Sharing One Window Space.....	419
Creating Tabs Within Windows.....	420
Managing Window Tabs.....	423
Rearranging Tabs in a Window.....	423
Moving Tabs Between Windows.....	423
Converting Tabs to Windows.....	423

Chapter 23	
Crash Detection, BMW and BLT.....	425
Crash Detection.....	425
BMW and BLT.....	426
Creation of Session Playback Media With BMW.....	426
Creating Session Playback Media for a Normal Session.....	426
Automatically Creating Session Playback Media for a Crashed Session.....	427
Manually Creating Session Playback Media for a Crashed Session.....	428
Session Log Files.....	428
Session Media Files.....	429
Replaying Session Playback Media With BLT.....	429
The /BMW Command Line Switch.....	430
Scripting and Macros.....	430

Chapter 24

SailWind Logic GUI Reference.....	431
AC Analysis Dialog Box.....	435
Add Attribute Label Dialog Box.....	436
Add Bus Dialog Box.....	438
Add/Edit Command Dialog Box.....	440
Add Field Dialog Box.....	442
Add Free Text Dialog Box.....	444
Add Net to Class Dialog Box.....	446
Add New Attribute Dialog Box.....	447
Add New Attribute to Library Dialog Box.....	448
Add Part From Library Dialog Box.....	449
Add Pins Dialog Box.....	451
Archiver Dialog Box.....	453
Archiver Additional Files Dialog Box.....	455
Archiver Libraries Dialog Box.....	456
ASCII Output Dialog Box.....	458
Assign Alternatives for Ground Part Dialog Box.....	460
Assign Alternatives for Off-Page Part Dialog Box.....	462
Assign Alternatives for Power Part Dialog Box.....	464
Assign Decal to Gate Dialog Box.....	466
Assign New Gate Decal Dialog Box.....	468
Assign New PCB Decal Dialog Box.....	469
Assign Shortcut Dialog Box.....	470
Attribute Properties Dialog Box.....	471
Auto Renumber Parts Dialog Box.....	473
Basic Script Editor.....	474
Basic Scripts Dialog Box.....	476
Bill of Materials Setup Dialog Box.....	478
Bill of Materials Setup Dialog Box, Attributes Tab.....	479
Bill of Materials Setup Dialog Box, BOM Config Tab.....	481
Bill of Materials Setup Dialog Box, Format Tab.....	483
Browse for Connectors Dialog Box.....	486
Browse for Special Symbols Dialog Box.....	487
Browse Library Attributes Dialog Box.....	489
Bus Name Properties Dialog Box.....	490
Bus Properties Dialog Box.....	492
CAE Decal Wizard Dialog Box.....	494
Capture a New View Dialog Box.....	496
Change Part Type Dialog Box.....	497
Check for Updates Dialog Box.....	499
Class Rules Dialog Box.....	500
Clearance Rules Dialog Box.....	502
Compare/ECO Tools Dialog Box.....	504
Compare/ECO Tools Dialog Box, Documents Tab.....	505
Compare/ECO Tools Dialog Box, Comparison Tab.....	507
Conditional Rule Setup Dialog Box.....	510

Table of Contents

Connect to SailWind Layout Dialog Box.....	512
Connect to SailWind Router Dialog Box.....	513
Crash Detected Dialog Box.....	514
Create PDF Dialog Box.....	515
Customize Dialog Box.....	517
Customize Dialog Box, Commands Tab.....	518
Customize Dialog Box, Keyboard and Mouse Tab.....	520
Customize Dialog Box, Macro Files Tab.....	522
Customize Dialog Box, Options Tab.....	523
Customize Dialog Box, Toolbars and Menus Tab.....	525
DC Source Sweep Analysis Dialog Box.....	527
Default Rules Dialog Box.....	528
Differential Pairs Dialog Box.....	529
Display Colors Dialog Box.....	531
Display Colors Dialog Box - Part Editor.....	533
Drafting Properties Dialog Box.....	535
Edit Button Image Dialog Box.....	537
Fields Dialog Box.....	538
Field Properties Dialog Box.....	540
Font Replacement Dialog Box.....	542
Fonts Dialog Box.....	544
Get Drafting Items From Library Dialog Box.....	545
Get Gate Decal From Library Dialog Box.....	546
Get PCB Decal From Library Dialog Box.....	547
Hierarchical Symbol Wizard Dialog Box.....	548
HiSpeed Rules Dialog Box.....	550
Installed Options Dialog Box.....	553
Library List Dialog Box.....	555
Library Manager Dialog Box.....	557
Log Test Dialog Box.....	560
Logic Families Dialog Box.....	561
Make Field Dialog Box.....	562
Manage Library Attributes Dialog Box.....	564
Manage Schematic Attributes Dialog Box.....	566
Media Wizard Dialog Box.....	568
Modeless Commands and Keyboard Shortcuts.....	569
Net Attributes Dialog Box.....	580
Net Name Properties Dialog Box.....	581
Net Properties Dialog Box.....	583
Net Rules Dialog Box.....	585
Netlist to PCB Dialog Box.....	587
Off-Page Properties Dialog Box.....	589
Options Dialog Box.....	590
Options Dialog Box, Design Category.....	591
Options Dialog Box, General Category.....	595
Options Dialog Box, Line Widths Category.....	599
Options Dialog Box, Text Category.....	600
Options Dialog Box - Part Editor, General Category.....	603
Options Dialog Box - Print/Plot.....	606

Table of Contents

Output Window.....	607
Library Config Dialog Box.....	609
Part Manager Dialog Box.....	611
SailWind Layout Link Dialog Box.....	613
SailWind Layout Link Dialog Box, Design Tab.....	614
SailWind Layout Link Dialog Box, Document Tab.....	617
SailWind Layout Link Dialog Box, ECO Names Tab.....	618
SailWind Layout Link Dialog Box, Preferences Tab.....	620
SailWind Layout Link Dialog Box, Selection Tab.....	622
SailWind Router Link Dialog Box.....	623
SailWind Router Link Dialog Box, Document Tab.....	624
SailWind Router Link Dialog Box, Selection Tab.....	625
SailWind Suite Configuration Dialog Box.....	626
Part Attributes Dialog Box.....	628
Part Information Dialog Box.....	630
Part Information Dialog Box, Attributes Tab.....	631
Part Information Dialog Box, Connector Tab.....	633
Part Information Dialog Box, Gates Tab.....	635
Part Information Dialog Box, General Tab.....	637
Part Information Dialog Box, PCB Decals Tab.....	640
Part Information Dialog Box, Pin Mapping Tab.....	643
Part Information Dialog Box, Pins Tab.....	645
Part Properties Dialog Box.....	649
Part Signal Pins Dialog Box.....	651
Part Text Visibility Dialog Box.....	653
Part Type Label Properties Dialog Box.....	655
Pen Plotter Advanced Setup Dialog Box.....	657
Pen Plotter Setup Dialog Box.....	658
PCB Decal Assignment Dialog Box.....	660
Photo Plotter Advanced Setup Dialog Box.....	662
Photo Plotter Setup Dialog Box.....	664
Pin Decal Browse Dialog Box.....	666
Pin Decal List Management Dialog Box.....	667
Pin Label Fonts Dialog Box.....	669
Pin Properties Dialog Box.....	672
Plot Dialog Box.....	673
Print Dialog Box.....	674
Project Explorer.....	674
Query Hierarchical Component Dialog Box.....	677
Reference Designator Properties Dialog Box.....	678
Remap Special Symbols Dialog Box.....	680
Rename Gate Dialog Box.....	682
Rename Part Dialog Box.....	683
Renumber Pins Dialog Box.....	684
Report Manager Dialog Box.....	686
Reports Dialog Box.....	688
Routing Rules Dialog Box.....	691
Rules Dialog Box.....	696
Rules Report Dialog Box.....	697

Table of Contents

Save CAE Decal to Library Dialog Box.....	699
Save Configuration Dialog Box.....	700
Save Drafting Item to Library Dialog Box.....	701
Save Off-Page to Library Dialog Box.....	702
Save Part and Gate Decals As Dialog Box.....	703
Save Part Types to Library Dialog Box.....	704
Save Part Type to Library Dialog Box.....	705
Save PCB Decal to Library Dialog Box.....	706
Save View Dialog Box.....	707
Select Gate Decal Dialog Box.....	708
Select Pin Decal Dialog Box.....	709
Select Type of Editing Item Dialog Box.....	710
Selection Filter Dialog Box.....	712
Selections Preview Dialog Box.....	714
Server Busy Dialog Box.....	716
Sheets Dialog Box.....	717
Signal Pin Nets Dialog Box.....	719
Simulation Setup Dialog Box.....	720
SPICEnet Dialog Box.....	721
Step and Repeat Dialog Box.....	722
Text Properties Dialog Box.....	723
Transient Analysis Dialog Box.....	725
Update From Library Dialog Box.....	726
Update Selected CAE Decals From Library Dialog Box.....	730
Update Selected Part Type From Library Dialog Box.....	733
Update Selected Pin Decals From Library Dialog Box.....	736

Glossary

Third-Party Information

Chapter 1

SailWind Logic QuickStart

SailWind Logic is a robust, multi-sheet, schematic capture solution that builds an effective front-end environment for SailWind Layout.

- Step 1 - Start a New Design
- Step 2 - Select the Sheet Size
- Step 3 - Add Parts and Connector Symbols
- Step 4 - Add Buses
- Step 5 - Add Connections to Parts, Connectors, and Buses
- Step 6 - Add Off-Page Symbols
- Step 7 - Add Power and Ground Symbols
- Step 8 - Print the Schematic
- Step 9 - Generate Reports
- Step 10 - Create a Layout Netlist

Step 1 - Start a New Design

Launch SailWind Logic and start a new design from the Welcome Screen.

Procedure

1. Click the Windows **Start > SailWind [version] > SailWind Logic** menu item.
2. On the Welcome Screen, click **New**.

Step 2 - Select the Sheet Size

SailWind Logic starts with an empty schematic sheet of the default size. You can change the sheet size.

Procedure

1. Click the **Tools > Options** menu item.
2. In the Options dialog box, click the **Design** category.
3. Select the size you want from the Sheet area:
 - a. Choose the size of the sheet by clicking in the **Size** box and selecting from the dropdown list.
 - b. Choose a corresponding sheet border by clicking **Choose** and selecting the appropriate border from the “Drafting Items” list of the “Get Drafting Item from Library” dialog box.
 - c. Click **OK** to close the Get Drafting Item from Library dialog box.
 - d. Click **OK** to close the Options dialog box.
4. To maximize the sheet size within your viewing area, on the main toolbar, click the **Sheet** button.

Step 3 - Add Parts and Connector Symbols

You can add parts and connector symbols to your design as needed.

[Adding a Part](#)

[Adding a Connector Symbol](#)

Adding a Part

You can add parts to your design individually, or you can add multiple instances.

Procedure

1. Click the **Schematic Editing Toolbar** button, then click the **Add Part** button.
2. In the Filter section of the Add Part from Library dialog box, set the Library to “All Libraries.”
3. In the Items box, type the * wildcard character, and click **Apply**.
4. Select a part and notice the symbol of the part in the preview window. Click **Add**.

The part in outline form attaches to the pointer and is ready for placement.

5. Move the pointer to move the part to an appropriate location and click to place the part.

Another copy of the selected part automatically attaches to the pointer for placement.



Tip

SailWind Logic automatically adds reference designators to parts as they are added.

-
6. Click to place another part or press the Esc key to cancel the placement of the same part type.
 7. Click **Close** to close the Add Part from Library dialog box.

Adding a Connector Symbol

Each pin of a connector has a unique symbol for the schematic. This results in multiple symbols that make up one part. All connector pins have the same reference designator to show that they are symbol pins of the same part.

Procedure

1. On the Schematic Editing toolbar, click the **Add Part** button.
2. In the Filter area of the “Add Part from Library” dialog box, select the “connect” library from the Library list, then click **Apply**.
3. Select a connector symbol to add to the schematic and click **Add**.



Tip

If the symbol faces the wrong direction when attached to the pointer, right-click and click the **Alternate** popup menu item. Right-clicking and clicking the **Alternate** popup menu multiple times enables you to choose one of any available symbols to represent the connector.

4. Click to place the connector symbol.
5. When you have finished placing all of the desired symbols, right-click and select **Cancel** or press the Esc key to end the placement mode.
6. Close the dialog box.

Step 4 - Add Buses

Buses are used to consolidate a group of nets into one connection. This eliminates the potential clutter of many nets going from, or to, the same location in the design.

Procedure

1. On the Schematic Editing toolbar, click the **Add Bus** button.
 2. Click to start the bus, click to add the corner, and double-click to finish the shape.
-
-
- Tip**
- To remove the corner you just placed, press the Backspace key. To cancel the bus placement, press the Esc key.

3. In the Add Bus dialog box, type a name for the bus in the Bus Name box.
 4. If the bus is to be used for strictly memory or data arrays, select Bit Format in the Bus Type area; otherwise, select Mixed Net.
 - **Bit Format** — The name of the bus must follow the bit format shown for arrayed connections.



Tip

[NN:NN] represents [LSB:MSB].

- **Mixed Nets** — The name of the bus must be unique. Add the names of all subnets to the Bus Nets area. The mixed net bus can also contain bit format connections. To enter values for a range, double-click the Start and End cells.

5. Click **OK**.
6. Click to place the bus name.

Step 5 - Add Connections to Parts, Connectors, and Buses

Add connections to parts, connectors, and buses to define the connectivity of your design.

Procedure

1. Click the **Schematic Editing Toolbar** button and then click the **Add Connection** button.
2. At the endpoint of a component pin, click to start a connection and draw the connection.

The connections are orthogonal by default, and you must click to create corners.



Tip

To remove the corner you just placed, press Backspace. To cancel the connection, press Esc.

3. End the connection using one of the following methods:

- Click at the endpoint of another pin on a part or connector.
- If you create a connection that ends in a bus, in the Add Bus Net Name dialog box, select the net from the Net name list and click **OK**.

The connection and its name remain selected in the design.

4. Place the label using one of the following:

- If you are satisfied with the location of the net name label, click in empty space.
- If the net name label is not in the correct location, right-click and click the **Move** popup menu item. The label attaches to the pointer. Click to place the label.

Step 6 - Add Off-Page Symbols

When connections need to connect to a circuit on another schematic sheet in the design, an off-page symbol is used. The off-page symbol has multiple representations, similar to a connector.

Procedure

1. Create a connection from the pin of a part in the design.
2. Right-click and click the **Off-page** popup menu item.

The default symbol appears on the pointer for placement, attached to the end of the connection.

3. If the symbol is not facing the correct direction, right-click and click the **Alternate** popup menu item until you find the correctly oriented symbol.

4. Click to place the symbol.
5. In the Add Net Name dialog box, type the net name and click **OK**.

Step 7 - Add Power and Ground Symbols

Power and ground symbols are added to the end of a connection and have different representations according to the ground or power name.

Procedure

1. Create a connection from the pin of a part.
2. Click to make a corner and drag up for power or drag down for ground.
3. Right-click and click the **Ground** popup menu item or **Power** depending on the connection and the direction it has been drawn. The default ground or power symbol appears on the pointer.
Notice the name that is annotated to the connection is displayed on the status bar in the lower left corner. If you clicked Power, it is +5V; if you clicked Ground, it is GND.
4. If you want a different voltage or ground, right-click and click the **Alternate** popup menu item for different graphical symbols and different names.

Step 8 - Print the Schematic

When you have finished editing your design, you can print it for checking and documentation purposes.

Procedure

1. Click the **File > Print** menu item.
2. In the Print dialog box, click **Options**.
3. In the Options dialog box, examine the Preview window. The outline within the larger area represents the area of the paper that will be taken up by the schematic. This plot area is often oriented perpendicular to the printed sheet. If this is the case, to maximize the design area on the printed sheet, click on the Orientation box (in the "Positioning" area) and select 90 as the orientation angle; then click **OK**.
4. Click the **Properties** button to make any printer changes before printing/plotting. Do not change the orientation of the sheet (Portrait to Landscape) as this will conflict with the orientation set previously in the Options dialog box.
5. After completing this setup, click **OK**.

Step 9 - Generate Reports

You can generate a number of different reports for design analysis.

Procedure

1. Click the **File > Reports** menu item.
 2. In the Reports dialog box, click what you want to report on, and click **OK**.
-



Tip

You can report on more than one item at a time, with each item generating a report (TXT file). These reports open in the default text editor and are formatted with a default set of customizable options.

Results

The report generates and a link to the report displays in the Output window. Click the link to view the report.

Step 10 - Create a Layout Netlist

You can create a Layout netlist to examine the database and to pass the design data to SailWind Layout.

Procedure

1. Click the **Tools > Layout Netlist** menu item.
The default name of the netlist is *default.asc*, unless you saved the schematic.
 2. Give the file a unique name and click **OK**.
The netlist opens in the default text editor for viewing.
-



Tip

If you saved the schematic, the default name of the netlist (the Output File Name) is the name of the schematic followed by an .asc file extension.

Chapter 2

Getting Started With SailWind Logic

The SailWind Logic workflow guides you through the steps necessary to create a schematic design.

[SailWind Logic Flow](#)

[Startup Options](#)

[Adding Startup Options to a Shortcut](#)

[SailWind Updates](#)

[Migrating User Settings](#)

[Customizing SailWind Logic Default Settings](#)

[Mouse Button Operations](#)

[Using the Numeric Keypad to Control the View](#)

SailWind Logic Flow

Creating a schematic design of your system requires a number of steps to get from initial concept to completed design. Each step is presented in a logical order so that you can manage the entire process of creating the schematics for your system.

1. Set up a new design
 - Create a new design (See [Creating a New Schematic File](#))
 - Select a sheet size (See [Setting Schematic Editor Options](#))
 - Add 2D line objects (See [Adding Drafting Items From a Library](#))
 - Set up options
 - Global (See [Options Dialog Box, General Category](#))
 - Design (See [Options Dialog Box, Design Category](#))
2. Set up rules and constraints
 - Decide which design rules you need (See [Setting Up Rules](#))
 - Set up design rules (See [Rules Setup](#))
3. Place parts

- Locate parts in the library (See [Managing Libraries and Library Data](#))
 - Place parts (See [Schematic Parts](#))
 - Set attributes (See [Attributes Overview](#))
4. Wire the schematic
- Connect the components (See [Connections](#))
 - Edit the connections (See [Changing a Connection](#))
 - Name the connections (See [Naming a Connection](#))
 - Assign net constraints (design rules) (See [Setting Up Net Rules](#))
5. Prepare the Design for Layout
- Output a netlist to SailWind Layout (See [Creation of a New PCB Layout from a SailWind Logic Design](#))
 - Generate reports (See [Reports](#))
6. Perform Design Annotations
- Forward annotate design changes to SailWind Layout (See [Forward Annotation From SailWind Logic to SailWind Layout](#))
 - Backward annotate design changes from SailWind Layout (See [Backward Annotation From SailWind Layout to SailWind Logic](#))

Startup Options

You can use startup options, known as command line switches, to control the initial SailWind Logic configuration. Use command line switches to enable different options, to open a file, start macros, and record a SailWind Logic session. You can type multiple command line options.

For instructions to add options to a program shortcut, see “[Adding Startup Options to a Shortcut](#)”.

Table 1. SailWind Logic Command Line Options

Option	Description
filename	Opens the specified design file when you start SailWind Logic. Type the folder path and filename. Use quotation marks for directories or filenames with spaces, for example: “C:\SailWind Projects\Samples\previewpart.sch”  Restriction: Do not use a forward slash (/) before the filename in the command line.

Table 1. SailWind Logic Command Line Options (continued)

Option	Description
/BMW[<i>initials</i>]	Opens the Media Wizard. Use the Media Wizard to start recording a session log or to convert the previous session log to media that can be replayed by Basic Log Test. To create session media files for the current SailWind Logic session, use the BMW modeless command. To use the BMW command line switch, type /BMW or /BMWxx, where xx is your initials, in the command line. Capitalization. [] represents optional text. This option is associated with another modeless command, BLT. BLT is the Log Test; it finds and runs the session media created by BMW to play back a recorded SailWind Logic session. For information, see Modeless Commands and Keyboard Shortcuts on page 569.
/l	Opens the last file you had open when you start SailWind Logic.
/mmacro name	Runs the specified macro in the default macro file. For example, to run the macro MyMacro, type /mMyMacro.
/Mmacro file	Specifies the file to use as the default macro file. For example, to run the macro MyMacro contained in the file <i>user1.mcr</i> , type /Muser1.mcr /mMyMacro. Note the required capitalization.
/nc	Starts SailWind Logic without displaying the splash screen that includes copyright information.
/sXXX	Starts a Basic script when you start SailWind Logic. Use quotation marks for filenames with spaces, for example:/s"C:\<install_folder>\SailWind<version>\Samples\Scripts\Logic\Unsupported\Attributes to Excel.bas"

Adding Startup Options to a Shortcut

If you repeatedly start your design sessions with the intention of launching a specific design file or specifying a particular design environment setting, then you can add startup options to the properties of a shortcut.



Tip

If you create your own shortcuts, copy the **Start** menu shortcuts instead of generating them from the executables in the install directory. **Start** menu shortcuts contain a "wrapper" that enables the proper environment variables to be defined as the program launches.

Procedure

1. In the shortcut properties, click in the box with the pathname.
2. Press End, press Spacebar, and then type the command line switch you want to use, and enclose with double quotes " " each string that contains a space.

When specifying a file to start, do not use a / before the filename. You can specify multiple command line switches. For example, to start the program with *preview.sch*, the command line might read:

```
"\<install_folder>\<version>\Programs\SailWindLogic.exe" "C:\SailWind  
projects\Samples\preview.sch"
```

SailWind Updates

The SailWind products automatically check for a new software version when you launch an application. If a new version is detected, a tooltip is displayed in the system tray.

[Downloading the Update](#)

[Disabling the Check for Updates](#)

[Checking for Updates Manually](#)

Downloading the Update

When a new version of SailWind is detected, you can download the update.

Prerequisites

An Internet connection is required for the check.

Procedure

1. Right-click the icon in the system tray.
2. Click the **Open download page** popup menu item.
3. Follow the instructions on the download page.

Disabling the Check for Updates

If you do not want to check for updates automatically, disable the Check for Updates functionality. You can enable the check at any time, or you can manually check for updates.

Procedure

1. Click the **Help > Check for Updates** menu item.
2. In the [Check for Updates Dialog Box](#), select the “Disable ‘Check for Updates’ functionality” check box.

Checking for Updates Manually

Do not check for updates manually unless you have disabled the automatic check.

Procedure

1. Click the **Help > Check for Updates** menu item.
2. In the [Check for Updates Dialog Box](#), click **Check for Updates**.

Migrating User Settings

You can use the SailWind User Settings Migration tool to extract your settings from one installation of SailWind Logic, Layout, and Router and import them into another installation or version.

For information on how to do this, see User Settings Migration in the *SailWind User Settings Migration Guide*.

Customizing SailWind Logic Default Settings

Several settings define the default ASCII parameters of operation of SailWind Logic. You can customize the user interface by modifying these system settings and saving the files.

The file *default.txt* is a SailWind Logic ASCII file that contains default option settings. It is read into memory when you start SailWind Logic or click the **File > New** menu item to begin a new design.

Procedure

1. Click the **Tools > Options** menu item.
2. Change the settings as desired using the **General** and **Design** categories to change the sheet size and design grid, and so forth.
3. Click **OK** to save the changes.
4. Click the **File > Export** menu item, and then set the folder to *C:\<install_folder>\SailWind<version>\Settings*.
5. Select *default.txt* from the list of files. Type the name in the File name area if it does not exist.
6. Click **Save**.
This displays the ASCII Output dialog box.
7. In the ASCII Output dialog box, click **Select All**.
8. Leave the output format at the current setting and click **OK**.

Mouse Button Operations

SailWind Logic follows Microsoft® Windows® conventions for two-button mouse operations. SailWind Logic also supports the use of a three-button mouse. The middle button provides quick access to the pan and zoom commands.

Use the middle mouse button to pan (to move the view from side to side or up and down) without changing the size. Click the middle mouse button where you want to center the work area. The screen repaints, placing the point you chose at the center.

To define a specific area that you want to enlarge, hold the middle mouse button down, and move the mouse diagonally and up across the area you want to zoom. A dynamic rectangle expands with the cursor movement. When you release the middle mouse button, the view zooms to the rectangle.

To zoom out, press the middle mouse button and drag diagonally and down. When you zoom out, a solid rectangle appears at the cursor. This represents the current view size. The thin line that expands from the

solid box represents the new view size in proportion to the old. The zoom-out ratio also displays with the cursor.

Using the Numeric Keypad to Control the View

You can control the view using the extended keypad or the numeric keypad, located on the far right of most keyboards. The NumLock light can be on or off except where specified.

Table 2. Numeric Keypad Functions

Click	To
Home	Fits board to the view.
End	Redraws current view.
Arrows	With Num Lock On, pans the viewing window. Moves one-half the screen width in the direction of the arrow. With Num Lock Off, moves cursor on grid unit.
5	With Num Lock On, draws zoom rectangle.
Pg Up	Zooms In centered at cursor location.
Pg Dn	Zooms Out centered at cursor location.
Ins	Centers the view at current cursor location, without zooming.

Getting Started With SailWind Logic
Using the Numeric Keypad to Control the View

Chapter 3

User Interface

The software interface is robust and configurable. It offers numerous methods for viewing and navigating your designs, as well as extensive capabilities for generating custom macro scripts for playback and debugging.

[View Control](#)

[Project Explorer](#)

[Output Window](#)

[Opening a File That is Already in Use](#)

View Control

You can use several methods to control which portion of the database is visible on the screen.

[View Commands and Scroll Bars](#)

[Middle Mouse Button](#)

[Numeric Keypad](#)

[Saving and Restoring Views](#)

View Commands and Scroll Bars

There are a number of different ways to interact with the design environment to control the view of the design. SailWind Logic provides a comprehensive set of commands to enable you to pan and zoom within the design including using the mouse buttons/wheel, keyboard shortcuts and various items on the **View** menu.

To control the view, use the scroll bars or the following commands in the **View** menu:

- Click the **View > Zoom** menu item, or the **Zoom** button on the toolbar to enter Zoom mode.
 - To zoom in, position the cursor at the desired view center, and click the left mouse button.
 - To zoom out, position the cursor at the desired view center, and click the right mouse button.
 - To define a specific view area, position the cursor at the desired view center, click and drag to define the extents, and release.
- Click the **View > Redraw** menu item, or the **Refresh** button on the toolbar to redraw the current view.
- Click the **View > Sheet** menu item, or the **Sheet** button on the toolbar to display the entire sheet.
- Click the **View > Extents** menu item to resize the view to display all objects in the design.
- Use the scroll bars to pan the view.

Middle Mouse Button

You can use the middle mouse button to perform a number of viewing operations.

Use the middle mouse button to perform panning and zooming operations:

- To pan, click the middle mouse button. The view is centered at the cursor.
- To zoom in, click and hold the middle mouse button. Drag the cursor diagonally up. To adjust the bounding box, move the mouse. When you release the middle mouse button, the area within the bounding box displays.

- To zoom out, click and hold the middle mouse button. Drag the cursor diagonally down. This draws an inner box, representing the current view. Moving the cursor adjusts the outer box, which represents the new view. The relative size of the outer box to the inner box determines how far the view zooms out. Release the middle mouse button to complete the zoom out operation.
- To view the entire design, click and hold the middle mouse button, drag the cursor horizontally, and release.

Numeric Keypad

Use the numeric keyboard to control the view.

For more information, see [Using the Numeric Keypad to Control the View](#).

Saving and Restoring Views

If you repeatedly pan or zoom to a particular area of your design, you can save time by saving a work area view so that you can instantly restore it when needed.



Restriction:

Capture is not available while in the Part Editor.

[Saving a View](#)

[Restoring a View](#)

Saving a View

You can save a view of your design for later recall.

Procedure

1. Arrange the work area to display the view you want to capture.
 2. Click the **View > Save View** menu item.
-



Tip

In the Save View dialog box, the preview shows you the location of the selected view in relation to the extents of the design.

3. In the Save View dialog box, click **Capture**.
4. In the “Capture a new view” dialog box, type a new name for the view and then click **OK**.

The new view name is listed in the View Name list.



Tip

You can create up to nine views. The view names appear at the bottom of the **View** menu.

5. Click **Close**.

Restoring a View

You can restore a previously saved view.

Procedure

1. Click the **View > Save View** menu item.
 2. Select a name from the View Name list and click **Apply** to apply a previously selected view to the work area.
-



Tip

When you apply a view, the previous view saves automatically. Click the **View > Previous View** menu item to restore the view.

3. Select a name from the View Name list and click **Delete** to remove it from the View Name list.

Project Explorer

The Project Explorer lists the objects in your design in a hierarchical structure. When you update your design, the hierarchical structure also updates automatically.



Restriction:

The Project Explorer is unavailable in the Part Editor.



Note:

To open the Project Explorer, click the **Project Explorer** button.

You can configure the Project Explorer so you can click on a listed object to select it in the workspace, or select it and zoom to it.

Procedure

1. Right-click in Project Explorer, and click the **Allow Selection** menu item.
A check mark indicates that the functionality is enabled.
2. If you want to Zoom to the object you select, click the **Zoom to selection** menu item.
A check mark indicates that the functionality is enabled.

Output Window

Use the Output window for viewing reports and session logs, macro editing and debugging, and custom programming and debugging.

The Output window is located in the lower left section of the display window. You can dock or float the Output window. You can also open or close the Output window.

The Output window has two tabs:

- [Status Tab](#) — Displays information on the current session.
- [Macro Tab](#) — Enables you to run, edit, and debug scripts.

[Session Log Management](#)

[Macros](#)

[Macro Playback](#)

[Macro Script Debug](#)

[Accessing Help on the Macro Language](#)

[CIS](#)

Session Log Management

Various methods are available for managing the session logs, including filtering, viewing, printing and saving the log data.

Session Log

- [Navigating Pages in the Status Tab](#)
- [Filtering the Status Tab Display](#)
- [Searching in the Status Tab](#)
- [Printing Session Log Messages](#)
- [Viewing and Printing Reports](#)
- [Saving a Session Log to File](#)
- [Clearing the Session Log Display](#)

Session Log

A session log, which appears in the **Status** tab of the Output window, contains all program output for the current session, including names of open and saved files, integrity test results.

The session log file messages in the **Status** tab are color coded by subject. Underlined items are links. The color codes are shown in the following table:

Table 3. Session Log Text Color Meanings

Color	Meaning
Red	Errors
Green	Warnings
Black	Messages
Blue	Links to files, Web pages, and database objects

Navigating Pages in the Status Tab

Use the **Status** tab toolbar buttons in the Output window to navigate to the previous or next page, and to refresh a display of reports and other pages. You can also stop updates to pages, and return to the session log display.

To perform these functions, use the following **Status** tab toolbar buttons:

Table 4. Status Tab Toolbar Buttons

Command	Description
Back	Displays the previous page.
Forward	Displays the next page.

Table 4. Status Tab Toolbar Buttons (continued)

Command	Description
Stop	Stops page updates.
Refresh	Refreshes the display of reports and other pages.
Home	Returns to the session log.

Filtering the Status Tab Display

The session log file messages in the **Status** tab are color coded by subject. You can choose to view any combination of the color coded messages.

Use the following procedure to filter the display in the **Status** tab.

Procedure

1. Right-click in the Output window and click one of the following commands from the **Filter** popup menu item:

Table 5. Filter Submenu Commands

Command	Description
Error	Displays errors.
Warning	Displays warnings.
Message	Displays messages.
Show all	Displays all items (errors, warnings, and messages).

2. Select an item to show it; clear the item to hide it.

Searching in the Status Tab

If you want to locate a specific item or term in the log, you can search for text in the **Status** tab. There are options available to search for whole word only and specific case matches.

Procedure

1. Right-click and click the **Find** popup menu item.
2. In the Find dialog box, type the text you want to find in the dialog box and complete any other dialog box options.
3. Click **Next**.

The tab scrolls to the occurrence of the word, highlighting the word.

Printing Session Log Messages

You can print a hard copy of the session log for review purposes.

Procedure

1. Right-click and click **Print**.
2. In the Windows standard Print dialog box, set options as needed.
3. Click **Print**.

Viewing and Printing Reports

The session log contains links to reports that you can view and print.

To view the report, click the link. The report appears replacing the session log as the active page in the **Status** tab.

You can print the displayed report.

Procedure

1. Right-click the displayed report and click **Print**.



Tip

As an alternative, on the **Status** tab toolbar, click the **Print** button.

2. In the Windows standard Print dialog box change any Print dialog box options as needed.
3. Click **Print**.

Saving a Session Log to File

There are times when you might want to refer to something that was documented in the session log (such as when debugging a file or submitting a Support Center request). You can save a session log to a file for future reference.

Procedure

On the Status toolbar, click the **Log to File** button to save the session log for future reference.

If a session log file already exists, new information is appended. If a session log file does not exist, a new file is created.

The default path for the session log comes from the *.ini* file entry:

```
FileDir=C:\SailWind Projects
```

When you first start the software, the location is set in the following registry key:

```
HKEY_CURRENT_USER\Software\Mentor Graphics\\SailWind Layout>Status Window\LastLogName
```

Clearing the Session Log Display

You can clear a session log when it is no longer needed.

Procedure

Right-click and click **Clear** to clear the session log display each time you open a file.

Results

This prevents you from viewing information from a previously opened file. It does not delete the log file.

Macros

You can create macros to simplify redundant activities. You can record any set of procedural steps for replay as a single action. You can also nest macros.



Tip

Dialog box actions are recorded as results rather than actions, so when you replay, you don't see the dialog boxes in the replay process. Because of this you can't create a macro that stops on an open dialog box; it must follow through to some result or action. For example, you can create a macro that selects the **File > Open** menu item, selects a file, and selects **OK**. The macro, when played back, opens a file.

[Creating a New Macro](#)

[Recording Mouse Movements](#)

[Opening an Existing Macro File](#)

[Viewing Multiple Open Macros](#)

[Editing a Macro](#)

[Changing the Default Text Editor](#)

[Saving the Macro](#)

Creating a New Macro

Macros can be very flexible and helpful beyond your current design session. You can create a new macro by recording your keystrokes, mouse movements and commands and then save it for later recall.

Procedure

1. On the main toolbar, click the **Output Window** button.

2. On the **Macro** tab, click the **New** button.

New macros are given a name of Macro#, where # is a numeric sequence such as Macro1 or Macro2.

3. If desired, click the **Compress mouse moves** and/or **Relative mouse moves** buttons.

See [Recording Mouse Movements](#) for more information.

4. On the **Macro** tab toolbar, click the **Record** button.

5. Perform the keystrokes, commands, and mouse clicks to include in the macro.

6. On the **Macro** tab toolbar, click the **Stop** button.

Alternatively, you can also script a macro instead of recording mouse actions.

Recording Mouse Movements

Mouse movements are recorded in macros. You can record compressed or uncompressed mouse movements and relative or absolute movements.

- **Compress Mouse Mode** — Compress mouse mode records only the start point and endpoint of a mouse movement. It does not record any of the intermediate coordinates between the start and end points. Compression is recommended under most circumstances because it significantly reduces the size of your macro file. Recording intermediate mouse movements increases the file size, but documents coordinate information if required for a special application.
- **Relative Mouse Mode** — Relative mouse mode records the start point and endpoint of a movement in incremental coordinates instead of absolute coordinates.

Opening an Existing Macro File

Macros are created in and stored in macro files that have a *.mcr* extension. To open an existing macro file (*.mcr*), you can use the menus or the toolbar.

You can open multiple macros in the macro editor. The macro editor also supports nested macros.

Procedure

1. On the main toolbar, click the **Output Window** button; then, in the Output window, click the **Macro** tab.
2. Click the **Open** button, or select the macro file in the Open File dialog box and click **Open**.

Viewing Multiple Open Macros

You can switch your view between multiple open macros. This enables you move back and forth between more than one macro without having to open and close them individually.

Procedure

Click a macro in the List of Open Macros area (left-hand pane) of the **Macro** tab to switch between open macros.

Editing a Macro

You can copy or cut selected text to the Clipboard. You can also paste the selection from the Clipboard into the text window. You can paste text from the Clipboard into other applications. You can also switch between open macros to edit multiple macros.



Restriction:

If you have Notepad as the default text editor, longer macro files may not be loaded because of size constraints in Notepad. See [Changing the Default Text Editor](#) for information on how to change the default text editor.

Procedure

1. Select the text you want to copy or cut.
2. On the **Macro** tab toolbar, click the **Copy** or **Cut** buttons.
3. Place the pointer at the insertion point where you want to place the copied text.

4. On the **Macro** tab toolbar, click the **Paste** button.

You will see that your selection has been pasted in the Output window at the insertion point.

As an alternative, right-click in the Output window and click **Copy**, **Cut**, or **Paste**.

Changing the Default Text Editor

To access large files using Edit, you must install an ASCII text editor with a suitable file size capacity.

Procedure

1. Open the *SailWindlogic.ini* file in a text editor.
2. Modify the [general] section, specifying a new text editor executable name. Include the drive and folder if the new editor is not in your Windows folder.
3. Save the *.ini* file and close the text editor.

Saving the Macro

Macros are not limited to your current design session. You can save a macro for future recall.

Procedure

1. Click the **Save** button.
2. In the standard Windows “Save As” dialog box, enter a filename, if necessary, and click **Save**.

Related Topics

[Using Command Line Switches with Macros \[SailWind Logic Command Reference Manual\]](#)

Macro Playback

You can play back an existing macro using Run. Run also resumes the playback of a paused macro. When you play a macro, you cannot use the mouse in the workspace.

[Playing Back a Macro](#)

[Pausing a Playing Macro](#)

[Stopping a Playing Macro](#)

Playing Back a Macro

Once a macro has been recorded, you can play it back.

Procedure

1. On the **Macro** tab, click the **Open** button and open a macro (.mcr) file.

Recent macros can be found by clicking the **Tools > Macros** menu item, and then selecting the desired macro from the list.

2. On the **Macro** tab toolbar, click the **Run** button.

As an alternative, right-click in the **Macro** tab and click **Run**.

Pausing a Playing Macro

You can pause a macro during playback.

Procedure

1. On the **Macro** tab toolbar, click the **Pause** button to pause a playing macro at any time.
2. Click the **Play** button to resume playing the macro.

Stopping a Playing Macro

You can stop a macro that is currently playing.

Procedure

Right-click in the **Macro** tab and click **Stop** or on the **Macro** tab toolbar, click the **Stop** button to stop a playing macro.

You cannot resume the playback of the macro once you have stopped it. When you click **Run**, the macro starts from the beginning.

Related Topics

[Using Command Line Switches with Macros \[SailWind Logic Command Reference Manual\]](#)

Macro Script Debug

When playing back a macro, you can run it step-by-step, or to a certain location in the script. To perform these debugging tasks, insert breakpoints in the macro at the points at which you want the macro to stop.

[Setting or Removing Breakpoints](#)

[Debugging the Macro Scripts](#)

[Run-Time Error Correction](#)

Setting or Removing Breakpoints

The ability to set or remove breakpoints is useful when you debug a macro. If the macro engine encounters a breakpoint when playing back a macro, it pauses the macro.

Procedure

1. Place the cursor on the line in which to add a breakpoint.
2. Right-click in the **Macro** tab and click **Toggle Break**, or as an alternative, on the **Macro** tab toolbar, click the **Toggle Breakpoint** button.

This inserts a breakpoint at the current cursor location. A breakpoint marker appears in the gutter area.

When the macro engine encounters a breakpoint while playing back a macro, it pauses the macro. The next line in the macro is marked with the instruction pointer.

Debugging the Macro Scripts

Once breakpoints are inserted, you can debug macros.

- [Play a Single Line of the Macro](#)
- [Perform a Subroutine Call on the Current Line](#)
- [Play Back a Macro to a Point](#)
- [Return From the Subroutine to the Point From Which it was Called](#)
- [Continue the Execution From the Current Point](#)

Play a Single Line of the Macro

You can play a single line of a macro by using a button on the **Macro** tab toolbar or the right mouse button menu command.

Procedure

Right-click in the **Macro** tab and click the **Step Over** menu item, or on the **Macro** tab toolbar, click the **Step Over** button.

Perform a Subroutine Call on the Current Line

You can perform a subroutine call on the current line by using a button on the **Macro** tab toolbar or the right mouse button menu command.

Procedure

Right-click in the **Macro** tab and click **Step Into** menu item, or, on the **Macro** tab toolbar, click the **Step into** button.

Play Back a Macro to a Point

You can play back a macro to a point by using a button on the **Macro** tab toolbar or the right mouse button menu command.

Procedure

Right-click in the **Macro** tab and click the **Step to Cursor** menu item, or click the **Step to cursor** button on the **Macro** tab toolbar.

Return From the Subroutine to the Point From Which it was Called

You can return from the subroutine to the point from which it was called by using a button on the **Macro** tab toolbar or the right mouse button menu command.

Procedure

Right-click in the **Macro** tab and click the **Step Out** menu item, or, on the **Macro** tab toolbar, click the **Step out** button.

Continue the Execution From the Current Point

You can continue the execution of a macro from the current point by using a button on the **Macro** tab toolbar or the right mouse button menu command.

Procedure

Right click in the **Macro** tab and click the **Run** menu item, or, on the **Macro** tab toolbar, click the **Run** button.

Run-Time Error Correction

If run-time errors occur, the macro debugger switches to step-by-step mode and displays a detailed message on the status bar. The instruction pointer is set on the line that produced the error. After fixing the error, you can resume playback of the macro.

Related Topics

[Using Command Line Switches with Macros \[SailWind Logic Command Reference Manual\]](#)

Accessing Help on the Macro Language

You can access help on the macro language at any time.

Procedure

Click in the edit area of the **Macro** tab and press F1 for information on the term and a sample script.

CIS

The **CIS** tab displays part information from CIS as indicated. You can:

1. [Specify a new configuration for data source to use and choose what to display in the CIS tab.](#)
2. [Search and add part\(s\) from CIS.](#)
3. [Compare part attributes in design with those in CIS to check the consistency.](#)

[Adding a New Configuration](#)

[Adding Parts from CIS](#)

[Comparing Part Attributes for Consistency Checking](#)

Adding a New Configuration

After launch, SailWind Logic automatically connects to the specified data source, from which to load data into the **CIS** tab.

Follow the steps below to specify a new configuration and choose what to display in the **CIS** tab.

Procedure

1. On the **CIS** tab toolbar, click the **New** button.
2. In the Library Config dialog box, add and connect to the target data source as follows:
 - a. Click **ODBC Config**.
 - b. Add an ODBC data source to use in the pop-up window.
 - c. Click **Update** to make the newly added data source active and available in the **ODBC DSN** list.
 - d. Select the target database from the list to establish connection and load its table list in the Database Tables area.
3. Select the database table(s) to use by checking the **To CIS** checkbox.

Only those selected will be available in the **CIS** tab.

4. In the "Table Configuration" area, specify what and how table fields are displayed in the **CIS** tab. For more information, see the table below.

Table 6. Table Configuration Description

Name	Description
Field Name	Lists all fields in the table selected on the left.
Field Type	Specifies what type the table fields belong to from the drop-down list, wherein:

Name	Description
	<ul style="list-style-type: none"> Part_Type is mandatory, based on which to load data into the CIS tab. Besides, you can see schematic symbol and PCB decal assigned to the part type in the local libraries from the CIS preview window. Part_Number is mandatory, based on which to check whether part attribute values in design are identical with those in CIS. Category allows to show table structure hierarchically by subcategories in the CIS tab. All field types except Normal must be unique.
Field Alias	<p>Specifies the table heading for each field to display in the CIS tab.</p> <ul style="list-style-type: none"> Field aliases corresponding to Field Type "Part_Type" and "Part_Number" are defined by default, and no modification is allowed. If nothing is set, Field Name will be used instead.
Transfer to Design	<p>Specifies to add the Field Name to the part attributes. If yes, you can see it in the Part Attributes list by checking part properties in the design.</p> <p> Tip When set, Field Alias will be used instead.</p>
Visibility in CIS	Specifies to display Field Name in the CIS tab. When set, Field Alias will be used instead.
Key	Reserved
Browsable	Specifies to add hyperlinks to the field contents in the CIS tab, which often links to such reference files as datasheets and drawings.
Property Checking	Specifies attribute(s) to compare in the Part Manager Dialog Box , checking whether the attribute values in design are identical with those in CIS.

5. Click **Save** to save the configuration.

SailWind Logic automatically connects to the data source and loads specified information into the **CIS** tab.

Adding Parts from CIS

After configuring data source as needed, you can check and use CIS data in the design. To locate parts efficiently, SailWind Logic provides you with filter feature. This section describes how to use CIS data.

Procedure

1. Connect to the target data source and specify the table fields to display, as described in "[Adding a New Configuration](#)".
2. In the **CIS** tab, select a table on the left, and view its part information on the right.

3. Use the filter to locate parts, and click **Search** to activate the filter. Parts that match the search filter settings are displayed.
Filtering by field name and keyword is supported, whereas wildcard or expression is not currently supported.
4. Select a part and check its part type (CAE decal) and PCB decal in the preview window.
5. Add a part to the design as follows:
 - a. Double-click the target part in the table. The part attaches to and follows the cursor movement.
 - b. Click on the schematic to place the part; another instance attaches to the cursor automatically.
 - c. Press the Esc key when you are done adding the part(s).
6. Select the part in the design, right-click and click the Attributes popup menu item. In the attributes list, you can see attribute(s) added by "Transfer to Design" feature in the [Library Config Dialog Box](#).

Comparing Part Attributes for Consistency Checking

Use the Part Manager to compare part attributes in design with those in CIS to check the consistency. For inconsistent attributes, you can update from CIS with multiple options. You can also specify the attributes to compare.

Procedure

1. Check that all attributes to compare are selected in the [Library Config Dialog Box](#).
 2. On the **CIS** tab toolbar, click the **Part...** button to activate the comparison.
-

**Note:**

The comparison is conducted on the basis of CIS attribute Part_Number.

3. Check the search results in the Part Attribute Info. area, where errors are highlighted in red. You can also:

- Use the filter to search for specific part(s) by the reference designator, schematic sheet, part number, error type, or the **Show Error Only** button.
-

**Note:**

Filtering by Component Name or Part Number is case-sensitive and no wildcard or expression is currently supported.

- Click on any item in the table, and in the "Comparison Results" area see respective attribute values assigned in design and CIS, with differences highlighted in red.

4. Update inconsistent schematic part attribute(s) from CIS in either of the following ways, which takes effect only on parts found with "Attribute is not equal" error.

- Click **Update All** to update all inconsistent part attributes from CIS.
- Select one or more items in the table, and click **Update the Selected** to update the inconsistent attributes of the selected parts only. Use Ctrl for multiple selections.
- Update attribute(s) for a specific part in the "Comparision Results" area, with two available options:
 - Update the selected attribute: Right-click on the attribute cell and click the **Update Selected Attribute From CIS** popup menu item
 - Update all attributes: Right-click and click the **Update Selected Part From CIS** popup menu item

Opening a File That is Already in Use

The SailWind products help you avoid making changes to a file that is already opened by another user.

The first user to open a file in a shared location becomes the owner of the file for the duration the file is open; the file is locked to all other users. If you try to open a file that someone else has already opened, you will get a warning message letting you know the current owner and the name of the computer from where the file is locked. You have the option to view a read-only version of the file but you will not be able to update it while the owner still has it open. You can save the file with another name using **Save As**.

Chapter 4

File Operations

Use the file operations to create and save a new schematic file. You can also open and save existing designs, import and export files from other file formats, and archive your schematics.

[Creating a New Schematic File](#)

[Saving a Schematic File](#)

[Opening a Schematic File](#)

[File Import and Export](#)

[Archiving Your Schematic](#)

Creating a New Schematic File

You can create a new blank schematic file for your design. SailWind Logic uses previously-defined default settings to define the starting design configuration.

Prerequisites

- Use the **File > New** menu item to clear the current schematic from memory and start a new schematic.

Procedure

1. On the main toolbar, click the **New** button.

The prompt “Save old file before resetting?” displays.

2. If you click **No**, a blank drawing sheet representing sheet 1 of the new schematic displays in the work area. If you click **Yes**, the [Saving a Schematic File](#) dialog box appears if the current schematic is untitled.

As an alternative, you can also create a new file from the popup menu in Microsoft® File Explorer.

- a. In File Explorer, go to the folder where you want to create the file.
- b. Right-click, and then click the **New > SailWind Logic Schematic** menu item. This action creates a new design file in the current folder.
- c. Type a name for the file and click Enter. Make sure the file has a *.sch* extension. This creates a zero byte file.

When you open this file in SailWind Logic, SailWind Logic recognizes the zero byte file and performs a **File > New** command using *default.txt*.

Saving a Schematic File

Use the **File > Save** (or **Save As**) menu item to write design information to a file. The File Save As dialog box displays the schematic files contained in the default *\SailWind Projects* folder. The list of displayed files includes those created in PowerLogic and SailWind Logic.



Restriction:

Files opened by another user are locked to any edits. See [Opening a File That is Already in Use](#).

The name of the currently open design is displayed in the File Name text box at the bottom of the dialog box. Click **Save** to save the file. If the filename exists, a prompt is displayed to overwrite the existing file. The filename *default* is displayed for new designs.

You can also save a file by clicking the **Save** button on the Standard toolbar.

Related Topics

[Opening a Schematic File](#)

[Import File Types](#)

[Exporting Files](#)

Opening a Schematic File

You can open a schematic file in different ways.

Restrictions and Limitations

Files opened by another user are locked to any edits. See [Opening a File That is Already in Use](#) on page 57.

Procedure

1. On the toolbar, click the **Open** button.

The File Open dialog box displays the schematic files contained in the default *\SailWind Projects* folder. The list displays files according to the Files of type setting.

2. Select “Schematic Files (*.sch)” from the “Files of type” dropdown list.

3. Select a schematic file and then click **Open** to load the design into SailWind Logic.

As an alternative, you can also open a file by dragging the file from Windows Explorer and dropping it into the SailWind Logic window.

Related Topics

[Creating a New Schematic File](#)

[Import File Types](#)

[Exporting Files](#)

File Import and Export

Using SailWind Logic, you can import different file types, including data from schematics created with other tools. You can extract design information from an open schematic and save it in an ASCII format compatible with the previous or current software version.

- [Import File Types](#)
- [Importing a File](#)
- [Exporting Files](#)
- [OLE Object Import and Export](#)
- [ASCII File Format](#)
- [Exporting to ASCII Output](#)

Import File Types

SailWind Logic enables you to import different file types, including data from various formats and schematics created with other tools.

You can use the import functions for the following:

- Insert data from various formats into the current schematic.
- Translate schematics created with other tools, and open them as new SailWind Logic schematics.

This function invokes a non-GUI instance of the Symbol and Schematic Translator to translate a single design, using the default mapping files located in `C:\<install_folder>\<version>\Settings\cadstar2pl.cnv, orcad2pl.cnv, pcad2pl.cnv, and protel2pl.cnv`.

For information on more complex translations, such as translating only selected files from a `.DDB` container, or translating with special mappings, see the *SailWind Symbol and Schematic Translator Guide*.

Table 7. File Types Imported into the Current Schematic

File Type	Description
ASCII File Format (*.txt)	PADS-format ASCII. The ASCII files you can import include those created from PowerLogic and SailWind Logic.  Note: Beginning with PADS 9.0, die parts and flip chips are identified by the Special Purpose settings in the Part Type rather than being designated by the DIE and FLP logic families. When you import an ASCII file created by a previous PADS version, these Special Purpose settings are automatically set for parts having the logic family DIE or FLP. The part's family designation remains the same.
OLE Import/Export on page 65(*.ole)	SailWind Logic enables you to embed files from other applications as OLE objects in your schematic using Edit/Insert New Object. Once you have an OLE object in your design, you can export the object as a singular item to an <code>.ole</code> file using File Export (see

Table 7. File Types Imported into the Current Schematic (continued)

File Type	Description
	Exporting Files). You can then import the .ole file into other SailWind Logic schematics.
ECO files (*eco)	Similar to PADS-format ASCII. Each type of data begins with a header line with a key word surrounded by asterisks (*).
(.asc) files	SailWind Layout rules.

Table 8. File Types Opened as New Schematics

File Type	Description
CAD .csa files	CADSTAR Archive files (ASCII)
CADSTAR .scm files	CADSTAR Schematic files (binary)
OrCAD .dsn files	OrCAD Capture files
P-CAD .sch files	P-CAD Schematic files (ASCII & binary) generated in P-CAD 2002 & newer
Protel .sch files	Protel 99 Schematic files (ASCII & binary)
Protel .schdoc files	Protel DXP/Altium Designer Schematic files (ASCII & binary)
Protel .prjpcb files	Protel combined design/schematic files

Importing a File

SailWind Logic supports the importing of different file types into the design environment.

Procedure

1. Click the **File > Import** menu item.
2. In the displayed prompt, Click **Yes** to save the existing design before importing the file, or **No** to import the file without saving changes.
3. From the “Files of type” dropdown list, select the type of file you want to import.
4. Select the file you want to import, then click **Open**.

Import progress is reported in the Status bar, and logging messages and links to log files are displayed in the Output Window.

Related Topics

[Exporting Files](#)

[Opening a Schematic File](#)

[Import Rules from PCB](#)

Exporting Files

Extract design information from an open schematic file and save it in an ASCII format compatible with the previous or current software version. You can also use Export to create default startup conditions for SailWind Logic.



Note:

You can save all or some of the open design in ASCII format.

Procedure

1. Click the **File > Export** menu item.
2. From the “Save as type” dropdown list, select the type of file you want to export to.
3. Type a name for the ASCII file in the File Name text box.
The default is the name of the currently open design.
4. Click **Save**.

This opens the [Exporting to ASCII Output](#) Dialog Box. Select the appropriate check boxes to indicate the information that you want to write to the ASCII file.



Note:

SailWind Logic enables you to embed files from other applications as OLE objects in your design by clicking the **Edit > Insert New Object** menu item. After you have an OLE object in your design, you can export the object as a singular item to an .ole file.

5. Select the SailWind Logic output version for your exported file.

Beginning with PADS 9.0, die parts and flip chips are identified by the Special Purpose settings in the Part Type rather than being designated by the DIE and FLP logic families. With this change, the following changes occur when you export a design to an ASCII file of a previous PADS version:

- The Special Purpose settings of any die parts and flip chips are cleared.
- Die parts and flip chips having a family designation other than DIE and FLP lose their die part or flip chip special purpose and become normal parts.
- Any normal parts that have the DIE or FLP family designation are treated as die parts or flip chips in the previous PADS version.

For more information, see [OLE Object Import and Export](#).

Related Topics

[Import File Types](#)

[Opening a Schematic File](#)

[Saving a Schematic File](#)

[Export Rules to PCB](#)

OLE Object Import and Export

You can import and export .ole files in SailWind Logic to add images, text files, and other objects created in other programs and link to or embed them into your design. You can also export images and objects from SailWind Logic so that they can be included in documentation or reports.

[Importing OLE Files](#)

[Exporting OLE Files](#)

Importing OLE Files

You can import an .ole file containing OLE objects into a SailWind Logic schematic, and you can export an OLE object from a schematic to an .ole file.

Procedure

1. Click the **File > Import** menu item.
2. Click **Yes** at the prompt to save your current design changes or click **No** to discard them.
3. In the File Import dialog box, select “OLE Files (*.ole)” as the file type to import.
4. Browse to and select the file you want to open.
5. Click **Open** and then click **Yes** at the prompt.

The OLE objects import into the current schematic.

Results

OLE objects are placed on the sheet on which they were created. For example, if you originally placed an OLE object on Sheet 1 and a second object on Sheet 2, these OLE objects are placed on Sheets 1 and 2 when imported.

If some of the OLE objects you want to import exist on sheets that do not exist in the current schematic, these objects are deleted. For example, you originally created OLE objects on Sheets 1, 2, and 3. The schematic you are importing the OLE objects into only has Sheets 1 and 2. The OLE objects from Sheet 3 are deleted.

Exporting OLE Files

You can export OLE objects from SailWind Logic so that they can be utilized by other programs or other SailWind Logic design sessions.



Tip

Handles appear around the object when it is selected.

Procedure

1. Select an OLE object and click the **File > Export** menu item.
2. In the File Export dialog box, select OLE (*.ole) as the file type.

3. Name the file and select a location in which to save the file.
 4. Click **Save**. The *.ole* file is created and is available for import into other SailWind Logic schematics.
-



Tip

You can also import application files from other sources using the **Edit > Insert New Object** menu item. You cannot import an *.ole* file.

ASCII File Format

You can export SailWind design files to the ASCII file format to edit information outside of the SailWind design environment. The ASCII files for SailWind Logic are organized into sections, beginning with a keyword enclosed with an asterisk.

The first section starts with *SCH, and gives the system setup parameters for the circuit.

The second section, starting with *CAM, lists the default settings for the plot and printing outputs.

Following this, all of the data for each sheet is grouped together in the *SHT* section. This data starts with *CAE*, listing the window zoom setting and cursor location for the sheet.

Table 9. ASCII File Formatting Sections

Option	Description
TXT	Free text items in the sheet.
LINES	2D line items, including library entry.
CAEDECAL	Description of the CAE symbol decals for all parts in the sheet.
PARTTYPE	Description of the part types that appear in the sheet.
BUSES	Description of buses in the sheet, including the name and location.
PART	List of all parts, including their attributes, which appear in the sheet. Gates and connector pins are each listed separately.
OFFPAGE REFS	List of all off-page flags in the sheet. This includes power and ground symbols and bus connections.
TIE-DOTS	List of all tie-dots in the sheet.
CONNECTION	List of the connections in the sheet, with the signal name and path.
NETNAMES	List of all net names that are displayed in the schematic.

After all sheets are listed, the file ends with *END*.

Exporting to ASCII Output

Define what information you want to write to an ASCII file.

Procedure

1. Click the **File > Export** menu item.
 2. Type a name for the output file, and then click **Save**.
-



Tip

The filename and path appear at the bottom of the ASCII Output dialog box.

3. In the [ASCII Output Dialog Box](#), select items in the “Select Sections to Output” area to appear in the ASCII file.
You can click **Select All** to select all items, or click **Unselect All** to clear all selections.
4. In the SailWind Logic Output Version area, select the appropriate version of the software you are using from the list.
5. Click **OK**.

Archiving Your Schematic

You can create a folder, a PDF, and/or a .zip file that contains all of your schematic files and supporting files. This includes the schematic itself, a design file, libraries, and any additional files or folders you want. You choose what to archive; all fields are optional.

Procedure

1. Open the schematic you want to archive.
2. Click the **File > Archive** menu item.
3. Select the files and folders you want to archive in the [Archiver Dialog Box](#).
4. Click **OK**.

Results

An archive folder that contains the design and/or schematic files, the libraries, and any additional files and folders you have indicated is created.

If you chose to compress the files, the .zip file is the only file in this folder.

If you chose to create a PDF, the file is created using the schematic name and placed in the archive folder.

If you chose to compress the files using the zip format, a .zip file is created in the following format:

```
<project_name>YYYYMMDDHHMMSS.zip
```

File Operations

Archiving Your Schematic

Where YYYY is the year, MM is the month, DD is the day, HH is the hour (using a 24-hour format), MM is the minute, and SS is the second of the exact time you created the file. The file contains the same folder structure as the archive folder.

Chapter 5

Design Setup

You can define various options for the initial setup of your design. These include the settings for the schematic editor, the part editor, file backups, the work area and grids. You can also specify colors settings, and fonts for your SailWind Logic schematic design.

- [Setting Options](#)
- [Setting Display Colors](#)
- [Setting Fonts](#)
- [Managing Font Replacement](#)

Setting Options

Using the Options dialog box, you can preset options for commands in SailWind Logic, setting up how those SailWind Logic commands will work and overriding the default settings in the *default.txt* file. Setting options enables you to set up a working environment that suits your design and the way you work.

You can set options for the [Schematic Editor](#) on page 70 and for the [Part Editor](#) on page 71.

- [Setting Schematic Editor Options](#)
- [Creating a Backup File](#)
- [Setting Part Editor Options](#)
- [Preserving Reference Designators](#)
- [Work Area and Grid Settings](#)

Setting Schematic Editor Options

SailWind Logic includes an extensive set of options for the Schematic Editor. These options enable you complete control over general design properties as well as the appearance of text and line widths.

Procedure

1. While you are in the Schematic Editor, click the **Tools > Options** menu item.
2. In the Options dialog box, select the appropriate category—“[General](#)” on page 595, “[Design](#)” on page 591, “[Text](#)” on page 723, or “[Line Widths](#)” on page 599.

See also [Setting Fonts](#).
3. Set the options you want to change.
4. Click another category to set its options.
5. When you finish setting options, click **OK**.

Creating a Backup File

SailWind Logic automatically creates backup files based on the settings you choose. This enables you to choose backup intervals, file naming, the number of backup files desired as well as the storage location.

Procedure

1. Click the **Tools > Options** menu item, and then select the **General** category.
2. In the Interval box (in the “Automatic backups” area), type the time in minutes between automatic backups to a file.
3. In the Number of backups box, type the quantity (1-9) of different backup files to create.

**Tip**

Backup files are named <filename>#.sch, where # is a sequential number. For example, *logic1.sch*, *logic2.sch*, and so on.

4. Perform one of the following:

- Click **Backup File** to change the folder or name of the backup file, and then, in the Backup File dialog box, browse to the folder, type the filename, and click **Save**.
 - Select the “Use design name in backup file name” check box to use the design name instead of the product name as the filename, for example *preview_logic1.sch*, *preview_logic2.sch* instead of *logic1.sch*, *logic2.sch*
5. Select the “Create backup files in design directory” check box to place all of your backup files in the same directory as the design.

**Tip**

Click to clear if you want your backup files in one, common backup directory.

6. Click **OK**.

Results

Table 10. Backup File Creation

If this is selected:		The Backup File is Saved
Create backup files in design directory	Use design Name in backup file name	
		in one common directory without the design name.
X	X	in the design directory using the design name.
X		in the design directory without the design name.
	X	in one common directory using the design name.

Setting Part Editor Options

You can set the options for the Part Editor including specific design properties such as cursor style, grids, text heights and line widths.

Procedure

1. While you are in the Schematic Editor, click the **Tools > Part Editor** menu item.
2. In the Part Editor, click the **Edit > CAE Decal Editor** menu item.

3. In the CAE Decal Editor, click the **Tools > Options** menu item.
4. In the Options dialog box, select the appropriate category— [General](#) on page 595, [Text](#) on page 600, or [Line Widths](#) on page 599.
See also [Setting Fonts](#).
5. Set the options you want to change.
6. Click another category to set its options.
7. If you want to save the options you have set as defaults for creating new parts, click the **Save As Default** button.

These defaults are used for any new parts (Part Types, Connectors, CAE Decals and Pin Decals) you create, using either the **File > New** menu item in the Part Editor, or from the Library Manager dialog. (When you edit existing parts, SailWind Logic begins with the options stored with the part.)

[Table 11](#) lists the options that are saved and set as defaults.

Table 11. Part Editor Options Saved and Set as Defaults

From the Options Dialog General Category	From the Options Dialog Text Category	From Setup menu > Fonts Dialog
Design grid Labels and Text grid Display grid Snap to Grid	Font, style and size of: Pin Number Pin Name Ref-Des Part Type Attr Label 1 Attr Label 2	Current font mode (system or stroke)

8. When you finish setting options, click **OK**.

Preserving Reference Designators

When you paste a group in a design, SailWind Logic uses the reference designators current in the saved group file. You have the option of enabling or disabling the preservation of group reference designators.



Note:

When reference designators are preserved, if conflicts with current parts exist, SailWind Logic renames the duplicate reference designators and generates an error report containing the renamed parts. The report displays in the default text editor.

Procedure

1. Click the **Tools > Options** menu item. In the Options dialog box, select the Design category.
2. In the Options area, select the “Preserve Ref Des on Paste” check box to preserve group reference designators.

Results

If you enable preservation of group reference designators, SailWind Logic retains all reference designation when you copy the group. If you disable preservation, after pasting a group into a new design, reference designation starts at the first number; for example, U1. See also [Options Dialog Box, Design Category](#).

Work Area and Grid Settings

The maximum work area is a square 56 inches by 56 inches. The current grid settings appear on the message line at the bottom of the work area, between the current default line width and the current cursor X and Y location. When you move an object or use a drafting command, the grid readout is replaced by a Delta X and Y reading, calculated from the cursor selection point when the command starts. Minus numbers mean left and downward.

- [Display Grid](#)
- [Origin and Design Grid](#)
- [Labels and Text Grid](#)

Display Grid

SailWind Logic uses a dot grid as a drafting aid. You can set this field of white dots called the display grid to match your design grid, or set it at larger multiples of the design grid.

Use the General category in the Options dialog box to set the display grid or use the [GD Modeless Command](#) on page 569. If you do not want to display the display grid, set it to 10.

Origin and Design Grid

When you start a new file, the default drawing format is centered in the work area with the origin, or 0,0 point, in the lower left corner. The origin appears as a large white dot. As you move the cursor, its position relative to the origin displays in the lower right corner of the screen. The numbers change in the multiples of the design grid. The minimum design grid increment is 2 mils.

The design grid intervals begin in all directions from the origin. To set the design grid, use the General category in the Options dialog box or use the G modeless command.

During design, if you move a part across the board, the cursor may move smoothly but the part snaps from grid point to grid point. When Snap to Grid is checked in the status window, you cannot place a part off of the current design grid.

To set the origin while in the Part Editor, select [Setup/Set Origin](#) on page 131 from the menu. Position the cursor where you want the new origin located and click.

Labels and Text Grid

All labels, fields, names, attributes, and text use the labels and text grid setting. Like the design grid, the minimum grid increment is 2 mils. To set the labels and text grid, use the General category in the Options dialog box.

Setting Display Colors

Use the Display Colors dialog box to control the colors of design objects and the design area. SailWind Logic saves setup information with the schematic.

Procedure

1. Click the **Setup > Display Colors** menu item.
 2. In the Selected Color area, select a color to assign to items.
 3. To change the available colors, click **Palette**. Palette opens the Color dialog box where you can specify new colors or customize colors that appear in the Selected Color area.
-



Tip

Click **Default Palette** to restore the default color settings in the Selected Color area.

Refer to the Microsoft Windows Help for more information on changing the Color Palette.

4. Click the color box of an object to change the color.
-



Note:

Two color box columns appear next to the items in the Titles area. “Frg” indicates the foreground color of the text item. “Box” indicates the color of the box that is drawn around the text item. This box serves two purposes:

- It indicates the exact size of the text item when it is plotted, thereby helping you avoid overlaps while moving the item.
 - It provides visibility of the text item at very small zoom levels.
-

5. To make an item invisible, set the object color to the background color.
-



Tip

When working with the color configuration, note the following:

- The color selection for displaying items or making them invisible does not affect plotting of the schematic.
 - Background sets the color of the work area surface.
-

6. To save your color assignments to a file, click **Save**, and then type the configuration name in the Save Configuration dialog box.
-



Tip

When saving the color configuration, observe the following:

- SailWind Logic saves configuration files in the `C:\<install_folder>\<version>\Settings` folder with a `.ccf` extension.
 - The configuration names also appear at the bottom of the **Setup** menu.
 - To modify an existing configuration with the current color settings, click **Save**. The current configuration filename appears in the Save Configuration dialog box. You can overwrite the settings in this file, or specify a different `.ccf` file, and click **OK**.
-

7. To change the default palette, complete the modifications to the Selected Color area and click **Save**. Type `default` in the Save configuration dialog box to override the existing default configuration, and click **OK**.
-

Related Topics

[Display Colors Dialog Box - Part Editor](#)

Setting Fonts

You can set up your designs to use stroke font or the system fonts that ship with the Windows operating system. The system fonts installed on your system are available for use.



Restriction:

If the schematic uses fonts or character sets that are not installed on your system, a font substitution process begins automatically when the file is loaded. During this process, you are asked to select fonts to substitute for those that are missing from your system.

Use the Fonts dialog box to set up or change the fonts to be used in your design.

[Choosing Stroke Font or System Fonts](#)

[Setting Stroke Font Options](#)

[Setting System Font Options](#)

[Setting Line Widths](#)

[Converting Stroke-to-System Fonts](#)

[Converting System-to-Stroke Fonts](#)

Choosing Stroke Font or System Fonts

Depending upon your design and visual style objectives, you can set up your design to use the stroke font or system fonts.

[Stroke Font](#)

[System Fonts](#)

Stroke Font

Use the Fonts dialog box to set or change the font type in your design. The “stroke” font is a simple font that is represented in SailWind Logic using lines and arcs. It does not depend upon any specific Windows font being on the system. There are some limitations imposed by the use of the stroke font.

Restrictions and Limitations

- Graphical symbols and special characters, such as arrows, check boxes, and bullets are not available in a stroke font.
- Other symbols such as mathematical, technical, and geometrical symbols, plus arrows and dingbat blocks are not available in a stroke font.

Procedure

1. Click the **Setup > Fonts** menu item.
2. In the Fonts dialog box, select “Stroke” to use stroke font in your design then click **OK**.
3. Click **Yes** when asked to verify that you want to change to stroke font.



Note:

When changing the font type, the following conditions may apply:

- After you confirm your selection, all text in your design changes to the new font type.
 - You cannot undo the changes after you switch font types in your schematic. To revert to a prior font, you must change it to that type.
 - Size and width of text characters in a stroke font are specified in mils; points are used exclusively for system fonts.
 - Bold, underline, and italic font styles cannot be assigned to a stroke font.
-

System Fonts

You can choose which font to use as the system font. “System” fonts are typically fonts that are resident in your Windows environment and include a large selection of font styles. Be careful to use a font selection that is a standard font so that users on other systems can view your files using the same font.

Restrictions and Limitations

- Be sure the non-ASCII symbols, such as +/-, ohm, and degrees are available on your system for the fonts you select. If the character sets you select are not available, a blank space or blank text box appears where the symbols should be. In this case, select character sets or fonts that are available on your system, and the symbols will display in your schematic.

Procedure

1. Click the **Setup > Fonts** menu item.
2. In the Fonts dialog box, select “System” to use a system font in your design.
3. Select the name of the font you want to use from the Default Font dropdown list.
The font in use is shown at the top of the list.
4. Click the button for the font style you want: **B** for Bold, **I** for Italic, or **U** for Underlined.



Tip

You can select any combination of the available styles:

- For example, you can select Bold and Italic or Italic and Underlined.
- By default, the style is regular—neither Bold, Italic, nor Underlined.

5. When you have finished selecting the desired font, click **OK**.
6. Click **Yes** when asked to verify that you want to change the system font.



Note:

When changing the font type, the following conditions may apply:

- The default font and style in use are displayed in the Default Font area.
- If you change to system fonts, a verification message displays, asking for confirmation that you want to change to system fonts. After you click **OK**, all fonts in your design change to the new selection.
- You cannot undo the change once you switch font styles in your schematic. To revert to a prior font, you must change it to a new font.
- Non-letter symbols, such as ampersand (&), pound sign (#), and copyright (©), registered trademark (®), and trademark (™) symbols, as well as the Euro, Pound, Yen, and Cent are supported in system fonts.

Related Topics

[Setting Fonts](#)

Setting Stroke Font Options

You can set options for the text and line widths of objects in the workspace. The features of these tabs change depending on your use of either stroke font or system fonts for design objects.

Procedure

1. Click the **Tools > Options** menu item, and then click the **Text** category.
 2. To change the size or width of a text item, select a text Type and click the **Edit** button, and then type a new value in mils in the size or width lists.
-



Tip

You must select a Type and click the **Edit** button for each attribute you want to change; you cannot edit both size and width simultaneously.

3. When you have finished making changes, click **OK**.

Setting System Font Options

You can set the system font options to control how text objects in your design will appear. You can choose the font style, size, and other visual characteristics of the selected font.

Procedure

1. Click the **Tools > Options** menu item, then click the **Text** category.
 2. To change the font for a type of text, select a text Type and click the **Edit** button, and then select the font you want from the list.
-



Note:

Fonts are displayed in the list in the following order:

- The fonts currently used in the design are listed at the top.
 - The available system fonts are listed below the dividing line.
-

3. To change the style, select the check boxes in the B, I, or U columns.
 4. To change the text size, select the Size box for the type you want to change, click the **Edit** button, and type a new value.
-



Note:

System font sizes are whole point sizes.

5. When you have finished making changes, click **OK**.

Setting Line Widths

Use the Line Widths category to change the size of line widths in the workspace. This enables the capability for you to add visual emphasis to objects in your design.

Procedure

1. Click the **Tools > Options** menu item, and then click the **Line Widths** category.
2. Select a line Type and click the **Edit** button, and then in the Width column, type a new value for the size you want in mils.
3. When you have finished making changes, click **OK**.

Converting Stroke-to-System Fonts

If your design or visual style objectives change, you can convert text from stroke to system fonts.

Restrictions and Limitations

Be sure the non-ASCII symbols, such as +/-, ohm, and degrees are available on your system for the fonts you select. If the character sets you select are not available, a blank space or blank text box appears where the symbols should be. In this case, select character sets or fonts that are available on your system, and the symbols will display in your schematic.

Procedure

1. Click the **Setup > Fonts** menu item.
2. In the Fonts dialog box, click System to use a system font in your design.
3. Select the name of the font you want to use from the list.
The font in use is shown at the top of the list.
4. Click the button for the font style you want: **B** for bold, **I** for Italic, or **U** for Underlined.



Tip

You can select any combination of the available styles:

- For example, you can select Bold and Italic or Italic and Underlined.
- By default, the style is regular—neither bold, italic, nor underlined.

5. When you have finished selecting the desired font, click **OK**.



Tip

Certain restrictions apply during the conversion:

- All text is converted to the nearest whole point size; there is no fractional point size conversion.
- Stroke font line widths are ignored during conversion.

6. Click **Yes** when asked to verify that you want to change the system font.

Examples

A stroke font text string with a size of 125 mils becomes a system font text string with a size of 9 points. However, a text string with a size of 100 is rounded to 7 points (97 mils).

Converting System-to-Stroke Fonts

If your design or visual style objectives change, you can convert text from system to stroke fonts.

Restrictions and Limitations

- Graphical symbols and special characters, such as arrows, check boxes, and bullets are not available in a stroke font.
- Other symbols such as mathematical, technical, and geometrical symbols, plus arrows and dingbat blocks are not available in a stroke font.

Procedure

1. Click the **Setup > Fonts** menu item.
2. In the Fonts dialog box, select “Stroke” to use stroke font in your design, and then click **OK**.
3. Click **Yes** when asked to verify that you want to change to stroke font.



Note:

When changing the font type, the following conditions may apply:

- After you confirm your selection, all text in your design changes to the new font type.
- You cannot undo the changes once you switch font types in your schematic. To revert to a prior font, you must change it to that type.
- Size and width of text characters in a stroke font are specified in mils; points are used exclusively for system fonts.
- Bold, underline, and italic font styles cannot be assigned to a stroke font.

Examples

A system font text string with a size of 9 points becomes a stroke font text string with a size of 125 mils, and a text string with a size of 7 points is rounded to 100 mils.

Related Topics

[Setting Fonts](#)

Managing Font Replacement

When you open a design created with fonts that are not installed on your system, the Font Replacement dialog box opens automatically.

**Tip**

If the schematic uses fonts or character sets that are not installed on your system, empty boxes appear where you expect to find text or symbols. Once font replacement process completes, the symbols display properly.

There are two types of font replacement:

- [Automatic Font Replacement](#)
 - [Manual Font Replacement](#)
-

**Tip**

You can select some fonts for automatic replacement, and select others for manual replacement.

[Automatic Font Replacement](#)

[Manual Font Replacement](#)

Automatic Font Replacement

If the design you are editing uses a font that is not resident on your system, you can use the automatic font replacement feature to substitute the font with one of your available system fonts. This feature uses the standard Windows font substitution routine to propose a suitable replacement font.

Procedure

1. When a design is opened that contains a font that is not resident on your system, SailWind Logic presents the Windows Font Replacement dialog box. In the Font Replacement dialog box, click in the Mode column and select “Skip” to preserve the original font settings, or select “Auto” to use a standard Windows font substitution.
-

**Tip**

The Design Font column lists the fonts in use.

2. In the Font column, select the name of the font you want to replace.
3. Click **OK**.

Manual Font Replacement

If the design you are editing uses a font that is not resident on your system, you can manually replace it with one of the resident fonts. You can also use this capability to change an existing font if you need to improve legibility or use a compressed font to save space on your schematic sheet.

Procedure

1. When a design is opened that contains a font that is not resident on your system, SailWind Logic presents the Windows “Font Replacement” dialog box. In the Font Replacement dialog box, click in the Mode box and select “Manual.”
2. In the Font column, select the name of the font you want to replace.



Note:

If you need to replace only one font, only one font displays; clicking in the box has no effect.

3. In the Replacement column, select the font you want to use as a replacement.
4. Repeat steps 1-3 for all fonts identified for replacement.
5. Click **OK**.

Related Topics

[Setting Fonts](#)

Chapter 6

Managing Libraries and Library Data

With SailWind Logic you can create and modify libraries, set searchability, manage library attributes, import and export libraries, report on contents, and use search parameters.

The parts, symbols and other items you use to create a schematic in SailWind reside in one or more SailWind *libraries*. A single SailWind library consists of four files, each containing items of a specific type identified by the filename extension.

Table 12. SailWind Library Files

File Extension	File Contents
.pt	Parts — Data about a part, such as a 74LS02, including logic family, attributes, pins, and gates.
.pd	Decals — The graphical representation of the part when it is drawn. It is often referred to as the footprint.
.ld	Logic — The graphical representation of a schematic part, such as a NOR gate. This section functions as a part list reader only. Use SailWind Logic to create and modify Logic decals (sometimes called CAE Decals).
.ln	Lines — General graphical data you can store in the library, such as a company logo, to use in any design file.

For information on creating PCB decals, see Creation of a New Decal in the *SailWind Layout Guide*.

[Convert PADS Libraries to the Current Format](#)

[Creating a Library](#)

[Displaying Items in a Library](#)

[Modifying Library Data](#)

[Setting Library Availability and Search Options](#)

[Managing Library Attributes](#)

[Importing and Exporting Libraries](#)

[Creating Library Reports](#)

[Wildcards and Expressions](#)

[Library Search Order](#)

Convert PADS Libraries to the Current Format

You can convert libraries from a previous version of PADS to the current format. This enables you to reuse library content from previous designs and to edit the content in the newer design environment.

For information on how to convert your older PADS libraries to the current SailWind format, see [Converting Your Libraries](#) in the *SailWind Netlist Library Converter Guide*.

Creating a Library

To add a new library to your library list, you first create a new empty library, and then populate it with content. After adding it, you can reposition it in the library list to the preferred location in the search order.

For more information, see [Adding Libraries to the Library List](#).

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager dialog box, click **Create New Lib**.
3. In the New Library dialog box, specify the folder and library filename, and then click **Save**.
4. Click **Close**.

Results

Your library is added to the bottom of the Library list, which is also the last library in the [Library Search Order](#). To move your library up in the library list and search order, see [Setting the Library List Order](#).

Displaying Items in a Library

You can display the items that exist in a library in the Library Manager dialog box. You can choose to view PCB decals, part types, drafting items, or logic CAE decals.

Procedure

1. Click the **File > Library** menu item.
2. In the [Library Manager Dialog Box](#), select a library in the Library list or select All Libraries.
3. Click one of these buttons to display categories of items in the library:

- **Decals** — PCB decals (component footprints)
-



Tip

Use SailWind Layout to edit PCB decals.

- **Parts** — Parts
- **Lines** — Drafting objects
- **Logic** — CAE decals (schematic symbols)

4. To filter the list, type [wildcards or expressions](#) on page 105 in the Filter box, and then click **Apply**.
-



Tip

An empty filter box yields no results. If you do not want to restrict the results with a filter, but display all items, type * (asterisk) and click **Apply**.

Results

Library item names are displayed in the PCB Decals, Part Types, Line Items, or CAE Decals list. (The list name changes based on the Filter you've selected.) The preview window displays a graphic of the library object.



Note:

Since library Parts have no visual representation, the preview window displays the first CAE decal associated with the part.

Related Topics

[Creating a Report of the Parts in a Library](#)

[Creating a Report of Decals, Lines or Logic Symbols in a Library](#)

Modifying Library Data

You can modify library data by adding, deleting, copying, editing, or transferring data to meet the requirements of your specific design.

- [Adding Items to a Library](#)
- [Deleting Items From a Library](#)
- [Copying a Library Item](#)
- [Editing Items in a Library](#)
- [Deleting All Items in a Library](#)
- [Transferring Library Data](#)

Adding Items to a Library

You can add new items to a library. When added to the library, the items are available for the current and future design sessions.

Procedure

1. Click the **File > Library** menu item.
2. In the [Library Manager Dialog Box](#), select a library in the Library list.



Restriction:

If you select (All Libraries), the **New** button is unavailable.

3. Click one of these buttons to display categories of items in the library:

- **Decals** — PCB decals (component footprints)
- **Parts** — Parts
- **Lines** — Drafting objects
- **Logic** — CAE decals (schematic symbols)

4. Click the **New** button.

- **Decals** — Unavailable. The currently assigned PCB decal displays. Use the Library Manager in *SailWind Layout* to add PCB decals to a library.

See also [Creation of a New Decal in the *SailWind Layout Guide*](#).

- **Parts** — You will enter the Part Editor. Click the **Edit Electrical** button. The Part Information dialog box appears so you can define a new part.

See also [Modifying Electrical Information for a Part](#).

- **Lines** — **New** and **Edit** are unavailable. Any 2D drafting items appear, for example, drawing formats and title blocks, and you can only copy or delete them. There is no special library lines editor. Use drafting tools to create or edit lines and save them to the library.



Tip

To add new drafting items to the library, create and [combine](#) on page 237 the items in SailWind Logic, then right-click and click **Save to Library**.

See also [Creating 2D Line Items](#), [Adding Drafting Items to a Library](#).

- **Logic** — You will enter the CAE Decal Editor.

See also [Creating a New CAE Decal](#).

Related Topics

[Copying a Library Item](#)

Deleting Items From a Library

You can delete one or more selected items from a library. This enables you to purge old or unwanted data from your library and improve library search times.

Procedure

1. Click the **File > Library** menu item.
 2. In the [Library Manager Dialog Box](#), select a library in the Library list.
-



Restriction:

If you select (All Libraries), the **Delete** button is unavailable.

3. Click the one of these buttons to display categories of items in the library:
 - **Decals** — PCB decals (component footprints)
 - **Parts** — Parts
 - **Lines** — Drafting objects
 - **Logic** — CAE decals (schematic symbols)
 4. To filter the list, type [wildcards or expressions](#) on page 105 in the Filter box, and then click **Apply**.
-



Tip

An empty filter box yields no results. If you do not want to restrict the results with a filter, but display all items, type * (asterisk) and click **Apply**.

5. Select one or more items in the PCB Decals, Part Types, Line Items, or CAE Decals list. (The list name changes based on the Filter you have selected.)
-



Tip

Use Ctrl+click to select multiple non-sequential items. Use Shift+click or drag the cursor to select a range of items.

6. Click **Delete**.

Related Topics

[Deleting All Items in a Library](#)

Copying a Library Item

You can copy a selected item to another name or another library.

Procedure

1. Click the **File > Library** menu item.
 2. In the [Library Manager Dialog Box](#), select a library in the Library list.
-



Restriction:

If you select (All Libraries), the **Copy** button is unavailable.

3. Click one of these buttons to display categories of items in the library:
 - **Decals** — PCB decals (component footprints)
 - **Parts** — Parts
 - **Lines** — Drafting objects
 - **Logic** — CAE decals (schematic symbols)

4. To filter the list, type [wildcards or expressions](#) on page 105 in the Filter box, and then click **Apply**.
-



Tip

An empty filter box yields no results. If you do not want to restrict the results with a filter, but display all items, type * (asterisk) and click **Apply**.

5. Select an item from the list and click **Copy**.
 6. In the dialog box that opens, select another library to receive the copy of the item, and/or type a new item name, and then click **OK**.
-

- **Decals** — Opens the Save PCB Decal to Library Dialog Box
- **Parts** — Opens the Save Part Type to Library Dialog Box
- **Lines** — Opens the Save Drafting Item to Library Dialog Box
- **Logic** — Opens the Save CAE Decal to Library Dialog Box

Related Topics

[Importing and Exporting Libraries](#)

Editing Items in a Library

As library content is frequently changing, you can use the Library Manager to edit items in a library. Select the item that you want to edit, make the desired edits, and then save the edits to the current library item or rename the item as a new library object.

Procedure

1. Click the **File > Library** menu item.
2. In the [Library Manager Dialog Box](#), select a library from the Library list.



Tip

If you select (All Libraries), the **Edit** button is unavailable.

3. Click the one of these buttons to display categories of items in the library:
 - **Decals** — PCB decals (component footprints)
 - **Parts** — Parts
 - **Lines** — Drafting objects
 - **Logic** — CAE decals (schematic symbols)
4. To filter the list, type [wildcards or expressions](#) on page 105 in the Filter box, and then click **Apply**.



Tip

An empty filter box yields no results. If you don't want to restrict the results with a filter, but display all items, type * (asterisk) and click **Apply**.

5. Select an item in the list and click **Edit**.

- **Decals** — Unavailable. The currently assigned PCB decal appears and you can only copy or delete it. Use the Library Manager in the PCB design program to modify the PCB decal.
See [Editing a Library Decal](#) in the *SailWind Layout Guide*.
- **Parts** — You enter the Part Editor. Click the **Edit Electrical** button. The Part Information dialog box appears so you can edit the part.
See [Modifying Electrical Information for a Part](#), [Part Editor Operations](#).
- **Lines** — Unavailable. Any 2D drafting items appear, for example, drawing formats, title blocks, etc. and you can only copy or delete them. There is no special library lines editor. Use drafting tools to create or edit lines and save them to the library.



Tip

To add new drafting items to the library, create and [combine](#) on page 237 the items in SailWind Logic, then right-click and click **Save to Library**.

- **Logic** — Opens the CAE Decal Editor with the selected decal.
See [Creating a New CAE Decal](#).

Deleting All Items in a Library

When you choose to delete all of the items in a library, the software replaces the existing library with a new empty one of the same name. By doing so, the software ensures that there are no leftover items in your library.



Restriction:

You cannot delete the contents of a read-only library.

Procedure

1. Click the **File > Library** menu item.
2. Click **Create New Lib**. The New Library dialog box displays.
3. Select the library file whose contents you want to delete, click **Save**, and then click **Yes** when prompted.
4. Click **Close**.

Related Topics

[Deleting Items From a Library](#)

Transferring Library Data

There may be times when it is desirable to transfer library objects from one library to another. This may be useful when you want to archive a specific library with a design or for creating libraries with a specific mixture of content for new designs.



Tip

To copy over single items, see [Copying a Library Item](#).

Procedure

1. Follow the steps to [export library data from the library](#) on page 102.
2. Follow the steps of the Importing Library Data topic of the *SailWind Layout Guide* to import the library data to another library.

Setting Library Availability and Search Options

Specify the libraries available to the design, the library search order, and other search-related options. Operations affect the contents of the Library list in the Library Manager dialog box.

- [Adding Libraries to the Library List](#)
- [Removing Libraries From the Library List](#)
- [Library Content and the Search Order](#)
- [Setting the Library List Order](#)
- [Sharing a Library Across a Network](#)
- [Controlling Library Search Access](#)
- [Protecting Library Files](#)
- [Synchronizing With SailWind Layout](#)

Adding Libraries to the Library List

You add libraries to the library list to make their contents available. The library must be listed in the Library Manager to search it and use its parts, decals, or line items in your design.

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
2. In the [Library List Dialog Box](#), click **Add**.
3. In the Add Library dialog box select a file type:
 - Select Library Files (*.pt9) from the file type list to add a SailWind Logic library.
 - Select OrCAD Symbol Library Files (*.olb) from the file type list to add an OrCAD library. The OrCAD library translation may take some time. See the progress indicator in the Status Bar.

Any errors, warnings, or messages created during the translation display in the *Logic.err* file, which opens in the default text editor.

4. Specify the folder and filename of the library to add, and then click **Open**.

Results

The library is added beneath the currently selected library in the Library list. If a library is not selected, the new library is added to the bottom of the list.

Removing Libraries From the Library List

If you want to prevent the contents of a library from being used in a design, remove the library from the library list.

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
2. In the [Library List Dialog Box](#), select one or more libraries from the Library list and then click **Remove**.

Results

The library files are removed from the Library List but are not deleted from the computer.

Library Content and the Search Order

Parts, symbols, and decals you create do not have to be located in the same library together. When your part refers to a symbol or decal in a different library, the software automatically picks up the symbol or decal when needed.

If you have multiple library items of one type and with the same name, the first occurrence of the item is chosen when the library is searched.

For example, during import of a schematic netlist, the libraries are searched for a 0603 decal. But there are two, each in a different library. The libraries are searched from the first library in the list, to the last library in the list. The first occurrence found will be selected for use to represent the part in the design.

When you have multiple libraries available, they are processed in their order in the library list whenever the libraries are searched. The libraries are searched during the following procedures:

- Searching for library items using various dialogs
- Importing a netlist from the schematic
- Updating your design from the library
- Annotating your design with new components from the schematic

You can change the search order of libraries using the “[Library List Dialog Box](#)” on page 555.

Setting the Library List Order

You can specify the order in which libraries are searched. The libraries are searched in the order in which they are listed in the Library list.

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
2. Select the library from the Library list in the [Library List Dialog Box](#), and then click **Up** or **Down** as needed.

Results

With each click, the library moves up or down one place in the library list. The libraries are searched top to bottom.

Related Topics

[Library Search Order](#)

Sharing a Library Across a Network

You can share a library across a networked environment, to enable more than one user to access the library simultaneously.

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
2. In the [Library List Dialog Box](#), in the Library list, select the library.



Tip

You can select multiple libraries using the Shift and Ctrl keys.

3. Select the Shared check box.

Results

More than one user can access the library file at the same time.

Controlling Library Search Access

You can enable or disable the searching of a particular library when performing operations that involve libraries, such as adding parts.

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
2. In the [Library List Dialog Box](#), in the Library list, select the library.



Tip

You can select multiple libraries using the Shift and Ctrl keys.

3. Select the “Allow Search” check box.

Protecting Library Files

The Read Only check box is only a status indicator. It is always shaded and unavailable. You can set library read-only status only in the Microsoft Windows File Manager.

Restrictions and Limitations

To ensure file protection, the system administrator who owns the files is the only one who can control the process.

Procedure

1. In Windows File Explorer, locate your library files.
-



Tip

By default, libraries installed with the software are located at `C:\<install_folder>\<version>\Libraries`

2. Select all four library files, right-click and click **Properties**.
3. In the Properties dialog box, click the **General** tab and select the “Read-only” check box.
4. Click **OK**.

Results

The library Read-Only check box in the [Library List Dialog Box](#) will not update until you reopen the dialog box.

Synchronizing With SailWind Layout

You can enable or disable the synchronizing of library settings between SailWind Logic and SailWind Layout. When the Synchronize with SailWind Layout check box is checked in SailWind Logic, all changes made in the libraries within SailWind Logic are pushed to SailWind Layout.



Tip

To ensure a round-trip synchronization, select the same check box in SailWind Layout.

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
2. In the [Library List Dialog Box](#), select the Synchronize with SailWind Layout check box.

Managing Library Attributes

Use the Manage Library Attributes dialog box to manage attributes on a library-by-library basis. You can add, delete, and rename attributes for all parts or decals in an individual library or in all libraries. You can also display all the attributes in a library, whether the attributes apply to all items or to individual items.



Tip

This dialog box does not manage attributes in the current design. Use the [Manage Schematic Attributes Dialog Box](#) to manage attributes in an open design file.

[Adding an Attribute to Multiple Library Items](#)

[Deleting Attributes From Library Items](#)

[Renaming Attributes of Library Items](#)

Adding an Attribute to Multiple Library Items

You can add an attribute to all parts and decals, or just to parts or decals individually, in one or all libraries.

Restrictions and Limitations

This process will yield no result or warning if the library you are working with is read-only. Check the status of the library in the [Library List Dialog Box](#).

Procedure

1. Click the **File > Library** menu item, then, in the Library Manager dialog box, click the **Attr Manager** button.
2. In the [Manage Library Attributes Dialog Box](#), in the Select Library list, select an individual library or the (All Libraries) item at the top of the list.
3. In the Item Types list, choose whether to apply the new attribute to All Items, or only either Part Types or PCB Decals.
4. Click **Add Attr**.

The [Add New Attribute to Library Dialog Box](#) appears.

5. For the Attribute Name, do one of the following:
 - Type an attribute name in the box.
 - Click **Browse Lib. Attr** to search all libraries for an existing attribute name.
6. (Optional) Type a value in the Attribute Value box.
7. Click **OK**.

The Attribute Name appears in the Attributes in Library list.

8. Click **Close**.

Results

Your new attribute is added. Check for the new attribute in the Decal Attributes dialog box (for PCB decals) or on the [Attributes tab of the Part Information dialog box](#) on page 631 (for Part Types).

Related Topics

[Manage Attributes in a Schematic](#)

Deleting Attributes From Library Items

You can delete one or more attributes from all parts and decals, or just from parts or decals individually, in one or all libraries.

Restrictions and Limitations

This process will yield no result or warning if the library you are working with is read-only. Check the status of the library in the [Library List Dialog Box](#).

Procedure

1. Click the **File > Library** menu item, then, in the Library Manager dialog box, click the **Attr Manager** button.
2. In the [Manage Library Attributes Dialog Box](#), in the Select Library list, select an individual library or the (All Libraries) item at the top of the list.
3. In the Item Types list, choose whether to delete the attribute(s) from All Items, or from either Part Types or PCB Decals.
4. In the Attributes in Library list, select one or more attributes to delete, and then click **Delete Attrs**.
5. In the prompt that appears, click **OK**.
6. Click **Close**.

Results

Your attribute(s) is deleted. Check for the deleted attribute in the Decal Attributes dialog box (for PCB decals) or on the [Attributes tab of the Part Information dialog box](#) on page 631 (for Part Types).

Related Topics

[Manage Attributes in a Schematic](#)

Renaming Attributes of Library Items

You can rename an attributes of all parts and decals, or just of parts or decals individually, in one or all libraries.

Restrictions and Limitations

This process will yield no result or warning if the library you are working with is read-only. Check the status of the library in the [Library List Dialog Box](#).

Procedure

1. Click the **File > Library** menu item, then, in the Library Manager dialog box, click the **Attr Manager** button.
2. In the [Manage Library Attributes Dialog Box](#), in the Select Library list, select an individual library or the (All Libraries) item at the top of the list.
3. In the Item Types list, choose whether to rename the attribute(s) from All Items, or from either Part Types or PCB Decals.
4. In the Attributes in Library list, select one or more attributes to rename, and then click **Add**.
5. Double-click in the New Name cell, type the name, and then click **Rename Attrs**.



Tip

You can specify the name of an existing attribute. No error message appears when you do this. The only time this may have an adverse effect is if both attributes are assigned to a single item, in which case the error is reported in the error file and the rename is not performed for those items where there are conflicts.

-
6. Click **Close**.

Results

Your attribute is renamed. Check for the renamed attribute in the Decal Attributes dialog box (for PCB decals) or on the [Attributes tab of the Part Information dialog box](#) on page 631 (for Part Types).

Related Topics

[Manage Attributes in a Schematic](#)

Importing and Exporting Libraries

Use the Library Manager dialog box to import or export library data in ASCII format.

[Importing Library Data](#)

[Exporting Library Data](#)

Importing Library Data

You can import library data from a previously-exported library ASCII file. This provides a convenient method for importing library data from another design or from another designer's library.



Tip

Beginning with PADS 9.0, die parts and flip chips are no longer identified by their family designations (DIE or FLP), but instead by the Special Purpose settings in the [General tab of the Part Information dialog box](#) on page 637. When you import an ASCII file created by a previous PADS version, these Special Purpose settings are automatically set for parts having the logic family DIE or FLP. The part's family designation remains the same.

Procedure

1. Click the **File > Library** menu item.
 2. In the [Library Manager Dialog Box](#), in the Library list, select the library to receive the library data.
 3. To import one of the four file types, you must select the matching filter:
 - If the file type is **.c**, select the Logic filter. This ASCII file contains CAE decals (logic symbols).
 - If the file type is **.l**, select the Lines filter. This ASCII file contains drafting objects.
 - If the file type is **.d**, select the Decals filter. This ASCII file contains PCB decals (component footprints).
 - If the file type is **.p**, select the Parts filter. This ASCII file contains part types.
 4. Click **Import**.
-



Tip

Import fails if the library to receive imported items is read-only.

5. In the Library Import File dialog box, specify the folder and the filename, and then click **Open**.

Related Topics

[Transferring Library Data](#)

Exporting Library Data

You can export library data into an ASCII file for importing into a library. This provides a convenient method for exporting library data to another design or to another designer's library.

Procedure

1. Click the **File > Library** menu item.
 2. In the [Library Manager Dialog Box](#), in the Library list, select the library whose data you want to export.
 3. Click any of the following:
 - **Decals** — Exports PCB decals (component footprints)
 - **Parts** — Exports Components
 - **Lines** — Exports drafting objects
 - **Logic** — Exports CAE decals (schematic symbols)
 4. To filter the list, type [wildcards or expressions](#) on page 105 in the Filter box, and then click **Apply**.
-



Tip

To display all items in the library, type an asterisk (*) and click **Apply**.

5. Select one or more items in the list, and then click **Export**.
6. In the Library Export File dialog box, specify the folder, type the filename, and then click **Save**.

Results

- The Special Purpose settings of any die parts and flip chips are cleared.
- Die parts and flip chips having a family designation other than DIE and FLP lose their die part or flip chip special purpose and become normal parts.
- Any normal parts that have the DIE or FLP family designation are treated as die parts or flip chips in the previous PADS version.

Creating Library Reports

You can create reports from the Library Manager to list any number of library objects. The Parts report can be configured to list the values of attributes that you choose to include in the report.

[Creating a Report of the Parts in a Library](#)

[Creating a Report of Decals, Lines or Logic Symbols in a Library](#)

Creating a Report of the Parts in a Library

From the Library Manager, you can generate a report about the parts in a single library or all libraries. The report (an ASCII file) lists each part and its associated attributes.

You can specify the attributes you want reported. [Figure 1](#) and [Figure 2](#) show example parts reports.

Procedure

1. Click the **File > Library** menu item.
2. In the [Library Manager Dialog Box](#), select a library from the Library list, or select All Libraries.
3. In the Filter area, click **Parts**. A list of parts in the library (or in all libraries) appears in the Part Types area.



Tip

To refine the list, use the filter field. Type a part name in the field or use wildcards (*) to specify a group of parts. Then click **Apply**.

4. When you have the list of parts you want to report on, click **List to File**.
5. In the [Report Manager Dialog Box](#), specify the part attributes you want to include in the report.
 - a. In the Available attributes list, click an attribute to select it.
 - b. Click **Include>>**.

The attributes appear in the Selected attributes list.

To remove attributes from the Selected attributes list, select them and click **<<Exclude**.

6. (Optional) You can refine the list of parts to report on. In the Part Filter field, type a part name in the field or use wildcards (*) to specify a group of parts, and then click **Apply**.
7. Click **Run**.
8. In the Library List File dialog box, select a folder and file format for the report.

You can select either of two formats:

- Library List format (*./lst*): Information is formatted in columns for viewing or printing. ([Figure 1](#)).
- Comma-separated values format (*.csv*): format recognized by Microsoft® Excel® ([Figure 2](#)).

9. Click **Save**.

10. In the Report Manager dialog box, click **Close**.

Results

The Report Manager generates the report and displays a link to it in the Output window. To view or print the report, click the link. Notepad opens and displays the report.

Figure 1. Parts Report in .lst Format

PADS LIBRARY (analogdev Part Types) DIRECTORY LISTING						
Library:	analogdev	Part Name	Part Number	Description	Manufacturer #1	PCB Decal
		AD1315	AD1315KZ	HIGH SPEED ACTIVE LOAD WITH INHIBIT	ANALOG DEVICES	Z-16A
		AD1321	AD1321KZ	HIGH SPEED PIN DRIVER WITH INHIBIT	ANALOG DEVICES	Z-16A
		AD1322	AD1322KZ	ULTRAHIGH SPEED PIN DRIVER WITH INHIBIT	ANALOG DEVICES	Z-16A
		AD1376	AD1376 (J,K)D	HIGH SPEED, 16-BIT A/D CONVERTER	ANALOG DEVICES	DH-32E
		AD1377	AD1377 (J,K)D	HIGH SPEED, 16-BIT A/D CONVERTER	ANALOG DEVICES	DH-32E
		AD1378	AD1378 (S,T)D	WIDE TEMPERATURE, 16-BIT A/D CONVERTER	ANALOG DEVICES	DH-32E
		AD1380	AD1380 (J,K)D	16-BIT SAMPLING ADC	ANALOG DEVICES	DH-32E
		AD1382	AD1382KD	16-BIT, 500KHZ, SAMPLING ADC	ANALOG DEVICES	DH-48A

Figure 2. Parts Report in .csv Format

```
"Library","Part Name","Part Number","Description","Manufacturer #1","PCB Decal 1",
"analogdev","AD1315","AD1315KZ","HIGH SPEED ACTIVE LOAD WITH INHIBIT","ANALOG DEVICES","Z-16A",
"analogdev","AD1321","AD1321KZ","HIGH SPEED PIN DRIVER WITH INHIBIT","ANALOG DEVICES","Z-16A",
"analogdev","AD1322","AD1322KZ","ULTRAHIGH SPEED PIN DRIVER WITH INHIBIT","ANALOG DEVICES","Z-16A",
"analogdev","AD1376","AD1376 (J,K)D","HIGH SPEED, 16-BIT A/D CONVERTER","ANALOG DEVICES","DH-32E",
"analogdev","AD1377","AD1377 (J,K)D","HIGH SPEED, 16-BIT A/D CONVERTER","ANALOG DEVICES","DH-32E",
"analogdev","AD1378","AD1378 (S,T)D","WIDE TEMPERATURE, 16-BIT A/D CONVERTER","ANALOG DEVICES","DH-32E",
"analogdev","AD1380","AD1380 (J,K)D","16-BIT SAMPLING ADC","ANALOG DEVICES","DH-32E",
"analogdev","AD1382","AD1382KD","16-BIT, 500KHZ, SAMPLING ADC","ANALOG DEVICES","DH-48A",
```

Related Topics

[Report Manager Dialog Box](#)

[Creating a Report of Decals, Lines or Logic Symbols in a Library](#)

[Managing Libraries and Library Data](#)

Creating a Report of Decals, Lines or Logic Symbols in a Library

From the Library Manager, you can generate a report listing the decals, lines, or logic symbols in a single library. The report is an ASCII file listing each item's name and the date and time the item was modified.

Procedure

1. Click the **File > Library** menu item.
 2. In the [Library Manager Dialog Box](#), select a library from the Library list.
-



Restriction:

The **List to File** button is unavailable for Decals, Lines, and Logic Symbols if you select All Libraries.

3. In the Filter area, click either the Decals, Lines, or Logic filter.

A list of corresponding items from the library appears in the dialog box (in the “PCB Decals,” “Line Items,” or “CAE Decals” area, depending on your filter selection).



Tip

To select one or more specific item, use the filter field. Type an item name in the field or use wildcards (*) to specify a group of items. Then click **Apply**.

4. When you have the list you want to report on, click **List to File**.
5. In the Library List File dialog box, specify a folder and filename for the report and click **Save**.

Results

Notepad appears, displaying a list of the item names and the date and time when each was last modified. You can print the list from Notepad.

Related Topics

[Creating a Report of the Parts in a Library](#)

[Managing Libraries and Library Data](#)

Wildcards and Expressions

You can use wildcards and expressions to filter the information that is displayed. This promotes rapid and precise browsing of library content.

The filter supports the wildcards and expressions listed in [Table 13](#). [Table 14](#) gives examples of wildcard usage.

Table 13. Wildcards and Expressions

Expression:	Use to:
*	Match any number of characters.
?	Match any one character.
[set]	Match any character in the specified set.

Table 13. Wildcards and Expressions (continued)

Expression:	Use to:
	Tip: A set is composed of characters or a range of characters; for example, A-Z or 0-9 or a-z.
[!set] or [^set]	Match any character not in the specified set.
\	Match a special syntactic character exactly, suppressing the special character's syntactic significance. Tip: The following characters need the \ before them: `[]*?!^`

Table 14. Usage Examples of Wildcards and Expressions

Expression:	Results in all items that:
74*	Start with 74: 7404, 74LS04, 74622.
74??	Start with 74 followed by any two characters: 7404, 74T2, 74TP.
74??08	Start with 74, followed by any two characters, and end with 08: 74LS08, 74HC08, 744608.
*08	Start with any number of characters and end with 08: 2146108, 5408, 54HCT08, 744608.
08	Start with any number of characters, followed by 08, and end with any number of characters: 5408, 5408BE, 54HCT08AE, 74ABT08CE2, 941M70839.
[57]*	Start with 5 or 7 with any number of characters after: 54HCT244, 5968BAE4, 74ACT44.
[5-7]*	Start with 5, 6, or 7 followed by any number of characters: 54LS08, 6225BE, 69TF77, 74ALS02.
[57]4HCT??	Start with 5 or 7, followed by 4HCT, and end with any two characters: 54HCT04, 54HCT74, 74HCT27, 74HCT84.
74A[CH]*	Start with 74A, followed by C or H, and end with any number of characters: 74AC244, 74AHCT27.
74A[!C-H]*	Start with 74A, followed by any character except the letters C through H, and end with any number of characters: 74ABT44, 74ALS244, 74ABF365.
*08	Start with the character \, followed by any number of characters, and end with 08: \LS08, \HCT08, \ABT08.

Library Search Order

The list of libraries in the Library Manager dialog box displays the library search order. When you have multiple libraries available, they are processed in their order in the library list whenever the libraries are searched.

The libraries are searched during the following procedures:

- Searching for library items using various dialogs
- Importing a schematic netlist into SailWind Layout
- Updating your design from the library
- Annotating SailWind Layout with new components from the schematic

You can change the search order of libraries using the [Library List Dialog Box](#).

See also [Setting the Library List Order](#).

Chapter 7

Library Parts

The processes associated with creating, managing and editing library parts contain many levels of detail. You can use the Decal Editor to create CAE decals (schematic symbols) including the editing of gates, terminals, pins, pin numbers, sequence numbers, attribute labels, and then use the Part Editor to incorporate the data into part types.

[Part Editor Operations](#)

[Object Selection Control in the Decal Editor](#)

[Changing and Updating Library Parts](#)

[Creating New Parts from Existing Parts](#)

[Save the Part/Decal to the Library](#)

[Selecting Multiple Objects in the Decal Editor](#)

[CAE Decals](#)

[Part Types](#)

Part Editor Operations

Use the Part Editor to create and modify standard schematic symbols such as gates, resistors, capacitors, connectors, etc., and special symbols such as ground/power and off-page references. You also use the Part Editor to create new CAE decals, pin decals, and assign PCB decals, the physical representation of the part in a PCB design system.

You can access the Part Editor by clicking the **Tools > Part Editor** menu item, or clicking the **Edit** button in the Library Manager dialog box.

Unneeded menu commands and toolbar buttons are removed or replaced with menu commands and toolbar buttons specific to creating and modifying parts.

A SailWind Logic part is comprised of the three elements shown in the following table:

Table 15. SailWind Logic Part Elements

Element	Description
Part Type	The electrical information for a part. This includes the logic family, CAE Decal assignment, part name, connection pins, gate swapping information, attributes, etc. See also Modifying Electrical Information for a Part .
Logic or CAE Decal	The CAE decal is the logic symbol that appears on the schematic, for example, a NAND gate. See also Creating a New CAE Decal .
PCB Decal	The graphical representation of the part when it is drawn in the PCB design program. It is often referred to as the footprint. Assigning a PCB Decal is optional and only required if you are passing a netlist to SailWind Layout.

- [Changing and Updating Library Parts](#)
- [Creating Single Gate Parts](#)
- [Creating Multigate Parts](#)
- [Creating New Parts from Existing Parts](#)
- [Creating a New Connector](#)
- [Creating a New CAE Decal](#)
- [Creating a New Pin Decal](#)
- [Special Schematic Symbols](#)

Object Selection Control in the Decal Editor

You can control object selection using the preset filter settings, or you can use the Selection Filter dialog box.

[Controlling Object Selection Using Preset Filter Settings](#)

[Controlling Object Selection Using the Selection Filter Dialog Box](#)

Controlling Object Selection Using Preset Filter Settings

You can control object selection using the preset filter settings that are available from the right mouse button menu. This enables quick access for selecting terminals, labels, drafting items or anything.

Procedure

1. Right-click in the Decal Editor.
2. Click **Select Terminals**, **Select Labels**, or **Select Drafting Items**. To select all of these options, click **Select Anything**.

Controlling Object Selection Using the Selection Filter Dialog Box

You can control which objects you can select by using the Selection Filter dialog box. This provides quick access for selecting parts, gates, nets, pins, buses, connections, 2D lines, text and other design objects.

Procedure

1. Do one of the following:
 - Click the **Edit > Filter** menu item.
 - Right-click in the workspace and click **Filter**.
 - Right-click in the Decal Editor and click **Filter**.

The Selection Filter dialog box appears.

See also[Using the Selection Filter](#).

2. Select one or more of the items, for example: **Pins**, **Labels**, **2D Lines**, or **Text**. To select all of these options, click **Anything**. To clear all of the options, click **Nothing**.
3. Click **Close**.

Related Topics

[Selecting Multiple Objects in the Decal Editor](#)

Changing and Updating Library Parts

When you add a part to a schematic, SailWind Logic creates a copy of the library part and adds it to the schematic database. Thereafter, the schematic representation is uncoupled from the library. Modifying a part in the Part Editor will only affect the library entry of that part, not the current version of the part type in the schematic.

Related Topics

[Modifying Parts](#)

[The Update From Library Function](#)

Creating New Parts from Existing Parts

You can create a new part by adapting an existing part. Make a copy of an existing CAE decal if pins need to be added or deleted. Modifying an existing CAE Decal without first making a copy will modify all parts that use that decal.

Prerequisites

- Verify which CAE decal is referenced by the part you are going to copy.
- Modify that CAE decal and save it with a new name.

Procedure

1. In the Part Editor, click the **File > Open** menu item, or click the **Open** button on the toolbar.
2. In the [Select Type of Editing Item Dialog Box](#), click CAE Decal and then click **OK**.
The Get Gate Decal from Library dialog box displays.
3. In the Items box, type a [wildcard or expression](#) on page 105 to filter the symbols, and click **Apply**.
4. Click **OK**.
You enter the Decal Editor and the decal appears.
5. Modify the decal as required.
6. Click the **File > Save As** menu item.
The Save CAE Decal to Library dialog box displays.
7. Type a name for the new CAE decal and select a library folder.
8. Click **OK**.
9. Click the **File > Exit** menu item.



Note:

Making a copy of the part uses the same process as making a copy of the CAE decal, except you select Part Type instead of selecting CAE Decal from the Select type of editing item dialog box.

- Change both the CAE decal and the electrical information, then save it as a new part.
 - You can rename a modified decal that you accessed through the part type.
-

See also [Saving Part Types](#).

Save the Part/Decal to the Library

From within the Part Editor, you can save a part/decal to the library. After adding the part to the library, you can then place it into a design.

Procedure

1. Click the **File > Save As** menu item.

The Save Part and Gate Decals As dialog box appears.

2. Select a library folder for the new part. The default is `C:\<install_folder>\<version>\Libraries`.
 3. Type a name for the new part or decal.
 4. Click **OK**.
 5. Click the **File > Exit Part Editor** menu item.
-



Tip

A checking routine is executed when you save a part. Resolve all errors before exiting the Part Editor; parts with errors cannot be added to the schematic.

Selecting Multiple Objects in the Decal Editor

You can select multiple objects in the Decal Editor. This enables you to move, rotate or mirror multiple objects simultaneously.

Procedure

1. Use the Filter Selection to determine which objects you can select.

See also [Object Selection Control in the Decal Editor](#).

2. Do one of the following:

- Right-click in the Decal Editor and select a filter, or click **Select All** to select all of the objects that are selected in the Filter Selection dialog box.
 - Click and drag over the objects you want to select.
 - Ctrl+click on each object you want to select.
-



Tip

When moving multiple objects in the Decal Editor, right-click to access the Rotate 90, X Mirror, and Y Mirror commands.

CAE Decals

Use the CAE Decal Editor to create and edit CAE Decals, otherwise known as schematic symbols or logic decals.

- [Constructing the New CAE Decal](#)
- [Using the Decal Wizard](#)
- [Manually Construct the New Part](#)
- [Creating a New CAE Decal](#)
- [Creating Single Gate Parts](#)
- [Creating Multigate Parts](#)
- [Adding Terminals](#)
- [Changing Objects in the Decal Editor](#)
- [Setting a Pin Number](#)
- [Setting a Pin Name](#)
- [Setting a Pin Type](#)
- [Setting Pin Swaps](#)
- [Changing a Pin Number](#)
- [Changing a Pin Name](#)
- [Changing a Pin Decal](#)
- [Changing Sequence Numbers](#)
- [Attribute Labels](#)
- [Creating Attribute Labels](#)
- [Modifying Terminals](#)
- [Setting the Origin for a Part](#)
- [Getting Gate Decals From the Library](#)
- [Assigning Pin Information to the CAE Decal](#)
- [Saving a Modified Decal With a Different Name](#)

Constructing the New CAE Decal

Using the Part Editor you can construct a new CAE decal by adding gates and defining the logic family.

Procedure

1. Click the **Tools > Part Editor** menu item.
2. In the Part Editor, on the toolbar, click the **Edit Electrical** button.

This displays the “[Part Information for Part dialog box](#)” on page 134.
3. In the Part Information for Part dialog box, select the **PCB Decals** tab to assign the PCB decal.

The pin count for the assigned PCB decal must be equal to or greater than the number in the electrical description.
4. Select the **Gates** tab.

5. Click **Add**, once for each gate in the part.

The gates are listed alphanumerically in the Gates multicolumn list box.

6. Double-click on the CAE Decal column for a gate, and then select the **Browse (...)** button.

The Assign Decal to Gate dialog box appears.

7. To filter the decals, type a [wildcard or expression](#) on page 105 in the Filter box, and click **Apply**.

8. Click **Assign**. Click **OK** when you finish assigning decals for the gate.

9. (Optional) Assign gate swapping information for the part.

Refer to the [Gates Tab](#) on page 144 topic for additional information.

10. Repeat steps 5 through 8 for the other gates.

11. Click the **Pins** tab to use Pin Count to indicate the number of pins in the part. Enter the number of pins for the entire package including signal pins, not for an individual gate.

12. On the **General** tab, select a Logic Family and Prefix from the list and other electrical information as required.

See also [Modifying Electrical Information for a Part](#).

13. Click **OK**.

Results

The Part Editor now displays the assigned CAE Decal outlines. Note that no pin information appears.

Using the Decal Wizard

Use the Decal Wizard dialog box to automatically create a new CAE decal. You must be in the Decal Editor mode of the Part Editor, and creating gate information, to use this feature.

Procedure

1. Click the **Tools > Part Editor** menu item.

2. In the Part Editor, on the standard toolbar, click the **New** button.

3. In the Select Type of Editing Item dialog box, click CAE Decal, and click **OK**.

You are now in the Decal Editor and ready to create a new CAE decal.

4. Click the **Decal Editing Toolbar** button.

5. Click the **CAE Decal Wizard** button.

6. To set the horizontal and vertical distance from the terminal connection point to the decal outline, type a value or use the arrows to indicate the pin length in the Pin Length area.

7. To set the horizontal and vertical distance between pins, type a value or use the arrows to specify the distance in the Pin Spacing area.

8. To set the width of the decal outline, type a value or use the arrows in the Min Width box.



Tip

Pin decals are moved left or right to accommodate the box width.

9. To set the minimum height of the decal outline, type a value or use the arrows in the Min Height box.
-



Tip

If you enter a value larger than needed to accommodate the number of input or output pins, space is added to the bottom of the decal.

10. In the Left Pins area, select the pin decals for the left, or input, side of the part from the Pin Decal list, and then type the number of pins in the Pin Count box or use arrows to indicate the pin count.
11. In the Right Pins area, select the pin decals for the right, or output, side of the part from the Pin Decal list, and then type the number of pins in the Pin Count box or use arrows to indicate the pin count.
12. In the Upper Pins area, select the pin decal to use for the upper pins, and then type the number of pins in the Pin Count box or use arrows to indicate the pin count.
13. In the Lower Pins area, select the pin decal to use for the lower pins, and then type the number of pins in the Pin Count box or use arrows to indicate the pin count.
14. Click **OK**.

Results

The Decal Wizard creates a rectangle of the correct size, adds the pins, and creates place holders for the sequence number, pin number, pin type, and swap information.



Tip

Use the Part Editor Drafting toolbar to modify the basic information created by the decal wizard. You can add additional terminals, change the pin decal, or change the pin sequence number.

Manually Construct the New Part

You can manually construct a new part for use in your design by creating the 2D line body shape, adding terminals, assigning pin numbers and pin names.

Procedure

1. Click the **Tools > Part Editor** menu item.
2. In the Part Editor, on the Part Editor toolbar, click the **New** button.
3. In the “Select type of editing” dialog box, click Part Type, and click **OK**.
4. On the Part Editor toolbar, click the **Edit Graphics** button. When prompted, click **OK** to confirm the creation of a new part.

5. On the Part Editor toolbar, click the **Decal Editing Toolbar** button to open the Symbol Editing toolbar.

6. Click the **Create 2D Line** button, then construct the body of the decal.

Refer to the [Creating 2D Line Items](#) topic for additional information.

7. Click the **Add Terminal** button.

See also [Adding Terminals](#).

8. Indicate the locations for the terminals.

9. Right-click and click **Cancel** when finished adding terminals.

10. Click the **Set Pin Number** button.

11. In the Set Pin Number dialog box, in the Start pin number area, type values in the Prefix and/or Suffix boxes.

A preview of pin numbers based on your input is displayed below the boxes.



Tip

Exceptions can be applied:

- Alphabetic and numeric values can be used in either box. For example, A1 or 1A.
 - For a single numeric, use either Prefix or Suffix box, and void the other box.
-

12. In the Increment options area, choose what to increment by clicking either Increment prefix or Increment suffix.

13. In the Step value box, type a positive or negative number to increase or decrease the pin numbers with consecutive or stepped values.

14. If using alphanumerics, you can select the Use JEDEC pin numbering check box to ensure legal alphanumeric values are used.

15. Click **OK**.

16. Select each pin or pin text in ascending sequence to assign a number.

You can click twice on a pin to skip a number.



Note:

You can use the **Change Pin Number** button to change a pin number if necessary.

17. Use the other options of the Symbol Editing toolbar to modify the pin numbers and add pin names, pin swapping information, pin type, etc.

18. On the **File** menu, select **Return to Part**, and then click **Yes** when the keep changes to gate message appears.

Creating a New CAE Decal

Creating a CAE Decal follows the same basic steps as creating a new packaged part. Start by placing SailWind Logic directly into decal editing mode, and then create the new CAE decal.

Procedure

1. Click the **Tools > Part Editor** menu item.
2. In the Part Editor, on the Part Editor toolbar, click the **New** button.
3. In the [Select Type of Editing Item Dialog Box](#), select CAE Decal.
4. Click **OK**. Four text entries are displayed in the working area

Table 16. CAE Decal Editing Mode - Text Entries In Working Area

Entry	Description
REF	The default space reserved for the reference designator in the schematic.
PART-TYPE	The default location of the part type name in the schematic.
* Free Label 1	The default location for the first displayed attribute.
* Free Label 2	The default location for the second displayed attribute.

5. Construct the new CAE decal for the part manually or with SailWind-Logic's Decal Wizard automatic part creation tool.
 - Use the [Decal Wizard](#) on page 116 to create the outline and add terminals.
 - (Optional) Use the Part Editor Symbol Editing Toolbar to add additional terminals, change the pin decal, or change the pin sequence number.
 - (Optional) Add additional attribute labels. See the [Creating Attribute Labels](#) topic for more information.
6. [Save the part](#) on page 113 to the library.

Creating Single Gate Parts

Single gate parts are parts that are represented by a single schematic symbol, such as an application-specific integrated circuit. Perform the basic procedure for creating a single gate part in the Part Editor.

Creating a new part involves multiple steps, with dialog boxes and information windows to for assistance. See the referenced documentation in the steps that follow for additional information.

In the following steps, you create the CAE Decal before entering electrical information. This causes the part type and the CAE decal to share the same name. If you want to use an existing CAE decal instead of creating a new one, specify the electrical information first.

See also [Creating Multigate Parts](#).

Procedure

1. Click the **Tools > Part Editor** menu item.
2. In the Part Editor, click the **Edit > CAE Decal Editor** menu item, or on the Part Editor toolbar, click the **Edit Graphics** button.
3. In the Select Gate Decal dialog box, enter a name for the decal and click **OK** when prompted to create a decal for the new part.

Four text entries are displayed in the working area: REF is the default space reserved for the reference designator in the schematic. PART-TYPE is the default location of the part type name in the schematic. *Free Label 1 is the default location for the first displayed attribute. *Free Label 2 is the default location for the second displayed attribute.

See also [Attribute Labels](#).

4. Construct the new CAE decal for the part [manually](#) on page 117 or with the automatic part creation tool in the [Decal Wizard](#) on page 116.
5. Within the Part Editor, if you created the CAE decal in step 4 using the Decal Wizard, use the Decal Editing Toolbar to modify the pin numbers created by the decal wizard and add pin names, gate swapping information, and so forth.
6. On the **File** menu, select **Complete**.
7. Assign electrical information to the part.
 - a. On the Part Editor toolbar, click the **Edit Electrical** button.
The “[Part Information for Part](#)” on page 134 dialog box displays.
 - b. Set the pin count to indicate the total number of pins in the part.
This is mandatory if there are signal pins in the part that were not added when you created the CAE decal in the steps above.
(Optional) Click the **Decals** tab to assign the PCB decal.
The pin count for the assigned PCB decal must be equal to or greater than the electrical description.
 - c. On the **General** tab, select a Logic Family from the list.
See also [Viewing and Setting General Part Information](#).
(Optional) Select the **Pins** tab and assign power and ground signal information.
 - d. Select the **Attributes** tab and assign bill of material and part list information.
 - e. Click **OK**.
8. [Save the part](#) on page 113 to the library.

Creating Multigate Parts

Multigate parts are parts that use more than one symbol or gate to represent a complete part in the PCB design system, for example, a 7400. This topic discusses the basic procedure for creating a multigate part in SailWind Logic.

See the referenced documentation in the topics below for additional information.

The steps required to create a multigate part are as follows:

1. [Construct the new CAE decal](#) on page 115 for the part.
2. [Assigning Pin Information to the CAE Decal](#)
3. [Save the Part/Decal to the Library](#).

Adding Terminals

A terminal consists of a pin decal and a series of text strings that define the terminal's number, swap data, etc.

Procedure

1. On the Decal Editing Toolbar, click the **Add Terminal** button.
2. In the Pin Decal Browse dialog box, select a decal, then click **OK**.

The pin decal attaches to the cursor.

3. Move the cursor into position and click to anchor the pin.

Before placing the terminal, you can use the popup menu to modify the orientation of the terminal and modify terminal information, including the pin number, pin name, pin type, and swap class.

After a terminal is placed, a new terminal, identical to the first, is attached to the cursor for placement.

If you defined a pin number or pin name, these are automatically incremented to the next number and name for the new terminal.

If you assigned pin type and swap class, the values are copied to the new terminal.

4. When finished, right-click and click **Cancel**.

Changing Objects in the Decal Editor

Depending upon the object type that you select, you can choose between a number of different methods to change one or more objects in the Decal Editor. These include such operations as query/modify, move, copy, delete and combine or explode.

Procedure

1. Select the objects you want to modify.

See also [Selecting Multiple Objects in the Decal Editor](#).

2. Right-click in the Decal Editor for editing options.

3. Refer to the following table to see which editing options are available, depending on the objects selected.

Table 17. Decal Editor - Editing Options Based on Selected Object

Objects Selected	Menu Commands Available
Drafting Items	Query/Modify, Move, Copy, and Delete. Explode and Combine are available when multiple objects are selected. (See Modifying Drafting Objects)
Labels	Move
Terminals	Query/Modify, Move, Copy, and Delete (See Modifying Terminals)
Combination	Move is available for any combination of objects. Copy and Delete are available for all objects except labels.

Setting a Pin Number

Use Set Pin Number to assign pin numbers to several pins. The pin number is incremented each time you select a pin.

Procedure

1. On the Decal Editing toolbar, click the **Set Pin Number** button.
2. In the Set pin number dialog box, in the Start pin number area, type values in the Prefix and/or Suffix boxes.

A preview of pin numbers based on your input is displayed below the boxes.



Tip

When entering pin numbers, observe the following:

- Alphabetic and numeric values can be used in either box. For example, A1 or 1A.
- For a single numeric, use either Prefix or Suffix box, and void the other box.

3. In the Increment options area, choose what to increment by clicking either Increment prefix or Increment suffix.
4. In the Step value box, type a positive or negative number to increase or decrease the pin numbers with consecutive or stepped values.
5. If using alphanumeric, you can select the “Use JEDEC pin numbering” check box to ensure legal alphanumeric values are used.
6. Click **OK**.

7. Select each pin or pin text in ascending sequence to assign a number.

You can click twice on a pin to skip a number.

Use the **Change Pin Number** button in the Editing Toolbar within the Part Editor to change a pin number if necessary.

8. Right-click and click **Cancel** when finished.

Setting a Pin Name

Use Set Pin Name to add or change the name for several pins. Pin names label the function of a pin, for example, CLK, DATA0, and so forth. The suffix of the name increments each time you select a pin. This command does not check for duplicate naming conventions.

Precede the text with the backslash character (\) to add a bar over the text characters.

Procedure

1. On the Decal Editing toolbar, click the **Set Pin Name** button.
2. Type the pin name in the information window.
3. Click **OK**.
4. Select each pin to assign names.

The pin name increments each time you click a pin, enabling you to assign pin names to a part sequentially (for example, D0, D1, D2, and so forth). To skip a particular pin name in the sequence, select the same pin until the desired pin name appears, then continue selecting other pins.
5. Right-click and click **Cancel** when finished.
6. Use the **Change Pin Name** button from the Part Editor Decal Editing toolbar to change the name of a single pin.

Setting a Pin Type

Use Set Pin Type to change the type of pin the terminal represents, for example, load, source, and so on. You can choose between verb mode or object mode when performing this task.

- [Setting a Pin Type Using Verb Mode](#)
- [Setting a Pin Type Using Object Mode](#)

Setting a Pin Type Using Verb Mode

You can use verb mode to set pin types. Start the command and then select one or more objects to which you want to apply the command.

Procedure

1. On the Decal Editing toolbar, click the **Set Pin Type** button.
The Pin Type information window appears.
2. Select a pin type from the dropdown list box
[Table 18](#) lists the available pin types and their corresponding letters.
3. Select the pins to which to apply the pin type information.
The text string associated with the pin is modified to reflect the new pin type.
4. Repeat steps 2 and 3 if necessary.
5. Right-click and click the **Cancel** popup menu item to exit the verb mode.

Setting a Pin Type Using Object Mode

You can use object mode to set pin types. Select one or more objects and then select the command you want to use to perform an action on them.

Procedure

1. Select the decal pins to which to apply the pin type information.
2. On the Decal Editing toolbar, click the **Set Pin Type** button.
The Pin Type information window appears.
3. Select a pin type from the dropdown list box.
[Table 18](#) lists the available pin types and their corresponding letters.
4. Click **OK**.

**Table 18. Characters Used
to Represent Pin Types**

Letter	Represents this pin type
B	Bidirectional Pin
C	Open Collector Pin
G	Ground Pin
L	Load Pin
O	Or-Tieable Source Pin
P	Power Pin
S	Source Pin
T	Tristate Pin
U	Undefined
Z	Terminator Pin

Setting Pin Swaps

Use Set Pin Swap to define a swap class for a terminal. The PCB layout software uses swapping information for length minimization and routing optimization. The swap class is assigned by a numeric value. The number for classes is 0, if the pin is not to be swapped under any circumstances, and 1 through 99. A pin can be swapped with any other pin within the gate that has the same swap class number.

- [Setting Pin Swaps Using Verb Mode](#)
- [Setting Pin Swaps Using Object Mode](#)

Setting Pin Swaps Using Verb Mode

You can set pin swaps using verb mode. Start the command and then select one or more objects to which you want to apply the command.

Procedure

1. On the Decal Editing toolbar, click the **Set Pin Swap** button.
2. Type the number that identifies the swap class in the information window.
3. Click **OK**.
4. Select the pins to assign as this swap class.
5. Right-click and click the **Cancel** popup menu item to exit the verb mode.

Setting Pin Swaps Using Object Mode

You can set pin swaps using object mode. Select one or more objects and then select the command you want to use to perform an action on them.

Procedure

1. Select the decal pins to assign as the swap class.
2. On the Decal Editing toolbar, click the **Set Pin Swap** button.
3. Type the character that identifies the swap class in the information window.
4. Click **OK**.

Changing a Pin Number

This command changes the number assigned to a pin. It does not check for duplicate numbering.



Tip

To assign numbers to several pins in an ascending sequence, use the **Set Pin Number** button from the Editing Toolbar in the Part Editor.

[Changing Pin Numbers Using Verb Mode](#)

[Changing Pin Numbers Using Object Mode](#)

Changing Pin Numbers Using Verb Mode

You can change pin numbers using verb mode. Start the command and then select one or more objects to which you want to apply the command.

Procedure

1. On the Decal Editing toolbar, click the **Change Pin Number** button.
 2. Type the new number in the information window.
-



Tip

Alphanumeric pin numbers are valid.

3. Click **OK**.
4. Select the terminal requiring a new number.
5. Right-click and click the **Cancel** popup menu item to exit verb mode.

Changing Pin Numbers Using Object Mode

You can change pin numbers using object mode. Select one or more objects and then select the command you want to use to perform an action on them.

Procedure

1. Select the decal terminal.
 2. On the Decal Editing toolbar, click the **Change Pin Number** button.
 3. Type the new number in the information window.
-



Tip

Alphanumeric pin numbers are valid.

4. Click **OK** to update the selected terminal.
-

Changing a Pin Name

Changes the name assigned to a single pin. Pin names are used to label the function of a pin, for example, CLK, DATA0, and so on. This command does not check for duplicate naming conventions.

Precede the text with the back slash character (\) to add a bar over the text characters.



Tip

To assign names to several pins in an ascending sequence, use the **Set Pin Name** button from the Editing Toolbar in the Part Editor.

[Changing Pin Names Using Verb Mode](#)

[Changing Pin Names Using Object Mode](#)

Changing Pin Names Using Verb Mode

You can change pin names using verb mode. Start the command and then select one or more objects to which you want to apply the command.

Procedure

1. On the Decal Editing toolbar, click the **Change Pin Name** button.
2. Select the decal terminal requiring a new name.
3. Type the new name in the information window.
4. Click **OK**.
5. Right-click and click the **Cancel** popup menu item to exit verb mode.

Changing Pin Names Using Object Mode

You can change pin names using object mode. Select one or more objects and then select the command you want to use to perform an action on them.

Procedure

1. Select the decal terminal.
2. On the Decal Editing toolbar, click the **Change Pin Name** button.
3. Type the new name in the information window.
4. Click **OK** to update the selected terminal.

Changing a Pin Decal

You can select a different pin decal for existing terminals in your schematic symbol. This is especially useful when defining connector pin usage or when redefining a symbol in the library.

- [Changing Pin Decals Using Verb Mode](#)
- [Changing Pin Decals Using Object Mode](#)

Changing Pin Decals Using Verb Mode

You can change pin decals using verb mode. Start the command and then select one or more objects to which you want to apply the command.

Procedure

1. On the Decal Editing toolbar, click the **Change Pin Decal** button. The [Pin Decal Browse Dialog Box](#) appears.
2. Select the new pin decal from the list to add to the gate.
3. Click **OK**.
4. Select the terminals requiring new decals to complete the change.
5. Right-click and click the **Cancel** popup menu item to exit the verb mode.

Changing Pin Decals Using Object Mode

You can change pin decals using object mode. Select one or more objects and then select the command you want to use to perform an action on them.

Procedure

1. Select one or more decal terminals.
2. Click the **Change Pin Decal** button.
3. Select the in decal required from the list of pin decals.
4. Click **OK** to update the selected terminals.

Changing Sequence Numbers

Use Change Sequence Number from the Part Editor Decal Editing Toolbar to change the sequence for pins in the part. The pin sequence number is used to set a corresponding number for the pins in alternate CAE decals. When you assign pin numbers with Set Pin Number or change a pin number with Change Pin Number, the same number is assigned to any alternate decals.

Procedure

1. On the Decal Editing toolbar, click the **Change Sequence Number** button.
2. Select the pin requiring a new sequence number.
3. Type the new sequence number in the information window.
4. Click **OK**.

Attribute Labels

Attribute Labels are placeholders for attributes within a CAE Decal. When you add an attribute to a part, it is placed in this reserved location. SailWind Logic lets you create an unlimited number of attribute labels.

See also [Attributes Overview](#).

Use attribute labels to customize attribute locations in the design. You can create and place attribute labels so that when a part is added to a design, assigned attributes are less likely to exist where a connection or other design information exists. You can also specify a unique justification or orientation for the label.



Tip

When working with attribute labels, observe the following:

- If you add an attribute and do not have an available attribute label, the attribute is added just below the insertion point of the part. Additional attributes are added below the last one.
- Attribute labels do not display in the Schematic Editor or the Part Editor. You must be viewing a decal to view an attribute label. Use Display Colors to control the display of attribute labels when creating or edit a decal. See also [Setting Display Colors](#).
- You can create an attribute label with a specific name; one that matches the actual attribute's name. This makes placement of specific attribute information easier. You can also use a generic attribute label name, or free label, for any attribute assigned to the part. The name used is *Free Label N, where N is the number of the label
- As you add attributes, they are placed at either the location that matches the attribute name, if specified, or the first available free label location. See also [Creating Attribute Labels](#).

Creating Attribute Labels

You can create attribute labels while editing or creating a CAE Decal.

Procedure

1. On the Decal Editing toolbar, click the **Add Attribute Label** button.
2. Define the attribute label using one of the following methods:
 - Type a name in the Attribute Name list box for the label that corresponds to an attribute.
 - Select a name from the Attribute Name list box. When you select an attribute name, it is removed from the list box.

- Select *Free Label to use a free label. When you Select Free Label, the number increments to the next available number.
 - Select **Browse Lib Attr** to open the Browse Library Attributes dialog box, select an attribute name from the list and then click **OK**.
3. (Optional) Set the text size, width, or justification.
4. Click **OK**.

See also [Setting Display Colors](#).

Modifying Terminals

Use the Terminal Properties dialog box to view or modify the properties of the terminals selected in the Decal Editor.

Procedure

1. In the Decal Editor, right-click a decal terminal and click the **Properties** popup menu item.

This opens the Terminal Properties dialog box.
2. To change the pin decal for all the terminals selected in the Decal Editor, click **Change Pin Decal**.
3. In the Pin Decal Browse dialog box, select a pin decal, and then click **OK**.
4. To change the terminal number, type the new value in the Number box.

If you select multiple terminals in the Decal Editor, this option is unavailable.
5. To change the terminal name, type the new name in the Name box.

If you select multiple terminals in the Decal Editor, this option is unavailable.
6. To change the swap class value, type a new value in the Swap class box.

If the values of the terminals differ, this text box is blank.
7. Click **OK**.

Setting the Origin for a Part

From within the Part Editor, use the **Setup > Set Origin** menu item to specify a new origin for a CAE Decal. When you add a part in SailWind Logic, the point of insertion is determined by the origin of the decal. You can perform this task while creating or modifying a decal.

Procedure

1. Open a CAE Decal in the Decal Editor and click the **Setup > Set Origin** menu item.
2. Indicate a new origin.

An information window with the new X,Y coordinate appears.
3. Click **Yes** to accept the new origin.

Related Topics

[Creating a New CAE Decal](#)

Getting Gate Decals From the Library

Use the “Get Gate Decal from Library” dialog box to open an existing CAE Decal in the Part Editor.

Procedure

1. In the Part Editor, click the **Open** button.
2. In the “Select Type of Editing Item” dialog box, click CAE Decal and then click **OK**.
3. Type a [wildcard or expression](#) on page 105 in the Items box to filter the symbols, and click **Apply**.
4. Select the part from the decals area.
5. Click **OK**. The selected decal displays.

Assigning Pin Information to the CAE Decal

You can view and modify pin information for a CAE decal in the Part Editor. You can also assign JEDEC pin numbering if your symbol requires you to use that standard.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Graphics** button.
2. In the Select Gate Decal dialog box, select a gate from the Gate list, or select a gate from the work area.
The decal displays in the CAE Decal Editor.
3. Click the **Decal Editing Toolbar** button to display the Decal Editing Toolbar, then click the **Set Pin Number** button.
4. In the “Set pin number” dialog box, in the “Start pin number” area, type values in the Prefix and/or Suffix boxes.
A preview of pin numbers based on your input is displayed below the boxes.



Tip

When specifying the pin numbers, note the following:

- Alphabetic and numeric values can be used in either box. For example, A1 or 1A.
- For a single numeric, use either Prefix or Suffix box, and void the other box.

5. In the Increment options area, choose what to increment by clicking either Increment prefix or Increment suffix.
6. In the Step value box, type a positive or negative number to increase or decrease the pin numbers with consecutive or stepped values.

7. If using alphanumerics, select the “Use JEDEC pin numbering” check box if you want to ensure legal alphanumeric values are used.

8. Click **OK**.

9. Select each pin or pin text in ascending sequence to assign a number.

You can click twice on a pin to skip a number.

Use **Change Pin Number** to change a pin number if necessary. Refer to the Terminal Toolbox topic for additional information.

The software assigns the same pin numbers to any alternate CAE decals.

10. Set other pin information such as pin type and pin swapping, as needed.

11. Right-click and click the **Cancel** popup menu item to exit the process.

12. Repeat the steps for the other gates.

Saving a Modified Decal With a Different Name

If you modify a decal associated with a part type, use the Save Part and Gate Decals As dialog box to rename the decal. After modifying the decal, you can save it with a different name.

Procedure

1. Click the **File > Return to Part** menu item, and then click **Yes** to the “Returning to Part level - Keep changes to Gate?” prompt.

2. Click the **File > Save As** menu item.

The “Save Part and Gate Decals As” dialog box appears.

3. Select the decal in the Name of Gate Decal multicolumn list box and click **Edit**.

4. Type a new name for the Decal and click **OK**.

5. Click **Yes** to the “Part Type item exists. Overwrite item?” prompt to replace the current library part with the one containing the modified decal.

Related Topics

[Saving Part Types](#)

Part Types

You use the Part Editor to work with part information. This is a collection of the electrical and attribute information that represents the physical characteristics of a part as opposed to the graphical representation contained in the CAE decal.

- [Modifying Electrical Information for a Part](#)
- [Viewing and Setting General Part Information](#)
- [Editing Logic Families](#)
- [Assigning PCB Decals](#)
- [Assigning Gates to Parts](#)
- [Assigning CAE Decals to Gates](#)
- [Part Information - Pins](#)
- [Managing Attributes](#)
- [Browsing Library Attributes](#)
- [Part Information - Pin Mapping](#)
- [Assigning Alternate Logic Decals for Connector Symbols](#)
- [Saving Part Types](#)
- [Library Management for Saved Part Types](#)
- [Saving a Modified Part Type With a Different Name](#)
- [Creating a New Connector](#)
- [Browsing for Connectors](#)
- [Creating a New Pin Decal](#)
- [Editing Objects in the Decal Editor](#)

Modifying Electrical Information for a Part

Use the Part Information dialog box to modify electrical information for a part. Electrical information includes the part statistics, the logic family, the PCB decal assignment, the CAE decal assignment, gate and signal pins, gate swapping information, and various other attributes.

You must identify the electrical or part type information, before assigning a CAE Decal to the part.

You access this information by clicking the **Tools > Part Editor** menu item, and then, on the toolbar, click the **Edit Electrical** button.

You can perform the following actions in the Part Information dialog box:

- [Viewing and Setting General Part Information - General tab](#)
- [Assigning PCB Decals to library parts - PCB Decals tab](#)
- [Assigning Gates to Parts - Gates tab](#)
- [Part Information - Pins tab](#)
- [Managing Attributes - Attributes tab](#)

- [Part Information - Pin Mapping tab](#)
- [Assigning Alternate Logic Decals for Connector Symbols - Connector tab](#)

Related Topics

[Managing Libraries and Library Data](#)

Viewing and Setting General Part Information

Use the **General** tab in the Part Information dialog box to view part statistics and set family information.

- [Viewing Part Statistics](#)
- [Setting the Logic Family](#)
- [Setting General Part Information](#)

Viewing Part Statistics

Information about the part you are creating or editing is listed under part statistic categories.

Table 19. Part Statistic Categories

Category	Description
Pin Count	Displays the total number of pins in the part. Includes gate pins, signal pins, and unused pins. If multiple decals are assigned with different pin counts, a range of smallest to largest decal pin counts is shown.
Decal	Displays the name of the decal, as chosen on the PCB Decals tab.
Gate Count	Displays how many gates exist in the part.
Signal Pin Count	Displays the number of signal pins in the part.

Setting the Logic Family

You can set the logic family to define a component type and assign an alphanumeric Reference Designator (RefDes) prefix for part recognition in your design.



Note:

Beginning with PADS 9.0, die parts and flip chips are identified by the Special Purpose settings in the Part Type rather than by the DIE and FLP logic families. With this change, any reference designator (logic family) can be assigned to a die part or flip chip.

Procedure

1. In the Part Editor, click the **Edit Electrical** button and in the Part Information for Part dialog box, click the **General** tab.
2. To identify the logic family, select a logic family from the list in the Logic Family area.

When you select a logic family, the default reference designator prefix for this part appears next to Ref. Prefix.



Tip

The default is UND, or undefined. If you leave the logic family undefined, the software prompts you for the reference designator prefix when you add a part. You can specify an alpha prefix or an alphanumeric prefix.

3. Alternatively, if you want to add, delete, and edit logic families and default reference designator prefixes, click **Families**.

The Logic Families dialog box opens.

- a. Make your changes and then click **OK** to return to the Part Information for Part dialog box.
- b. Select the new or updated logic family from the list.

See also [Editing Logic Families](#).

Setting General Part Information

You can set the general part information such as logical to physical pin mapping, special purpose part definitions and ECO registration for a part.

Procedure

1. In the Part Editor, click the **Edit Electrical** button and in the Part Information for Part dialog box, click the **General** tab.
 2. Select the “Define mapping of Part Type pin numbers to PCB Decal” check box to activate the **Pin Mapping** tab where you can map logical pin numbers to different physical pin numbers.
-



Restriction:

Note the following restrictions:

- The check box is unavailable after you add one or more alphanumeric decals to the part type. Remove the assigned alphanumeric decal to make the check box available. The check box also becomes available if you assign a numeric decal. However, you will still need to remove the alphanumeric decal from the list to make the part valid.
 - You must assign a decal to use the **Pin Mapping** tab.
 - Only decals with sequential numerical pin numbers can be used with pin mapping.
-

3. Select the Special Purpose check box, then select the appropriate radio button to identify the part as one of the following types:

- **Connector** — In contrast to other part types, connectors do not require a prefix list, or gate definitions.
-



Restriction:

When working with connectors, note the following restrictions:

- This check box is automatically selected when you create or modify connectors. It is unavailable if you open a part other than a connector.
 - This check box is unavailable when the part already has gate or pin assignments.
 - The **Gate Decals** tab is unavailable when the Connector check box is selected.
 - Some **Pins** tab controls not applicable to connector parts are disabled.
-

- **Die Part** — See the following note.
 - **Flip Chip** — See the following note.
-



Note:

Beginning with PADS 9.0, die parts and flip chips are identified by the Special Purpose settings in the Part Type rather than by the DIE and FLP logic families. With this change, any reference designator (logic family) can be assigned to a die part or flip chip.

4. To enable a part to be passed between the design and schematic file for forward and backward annotation, click **ECO Registered Part**. By default, all existing part types in your design are ECO registered.
-



Tip

Typically, you do not select this check box for non-electrical parts. For example, if you create and add a mounting hole to your design in the layout software, you would not need the part (mounting hole) to pass back to your schematic when you perform a backward annotation of the design.

5. To apply the part information edits to other parts in the library, type prefixes and wildcards into the Prefix List box matching the names of the other parts to update.
-



Note:

Examples of prefix and wildcard usage:

- Question mark ? in a prefix acts as a wildcard for one character. The prefix "?4" is the equivalent of "54" or "74".
 - If you type "\02" as the suffix, the edits are applied to all parts ending in 02.
-



CAUTION:

The software applies the contents of the Prefix List box when you click **OK** or **Save As** from other tabs in the Part Information dialog box.

6. Click the **Check Part** button to check the part for missing or inconsistent information while the Part Information dialog box is open.
7. Click **OK**.

Editing Logic Families

To add, delete, or modify logic family names and default reference designator prefixes, use the Logic Families dialog box.

[Adding a Logic Family](#)

[Changing the Name or Prefix of a Logic Family](#)

[Deleting a Logic Family](#)

Adding a Logic Family

If you are creating a new part for your design that requires a logic family definition that does not exist in the predefined list, you can add a new logic family to the database for use in current and future designs.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
2. In the Part Information dialog box, click the **General** tab and then click the **Families** button.
3. In the Logic Families dialog box, click **Add**. A new entry opens in the Logic Families list.
4. In the Family text box, type the new logic family name.
5. Click in the Prefix text box, and type the prefix you want to use for the new family.

You can specify an alphabetic prefix or an alphanumeric prefix. The maximum length for the prefix is six characters. The last character must be an alphabetic character.
6. Click **OK**.

Changing the Name or Prefix of a Logic Family

If your design requirements require it, you can change the name or prefix of an existing logic family.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
2. In the Part Information dialog box, click the **General** tab and then click the **Families** button.
3. In the Logic Families dialog box, select the logic family from the list.
4. In the Family text box, type the name of the new family.
5. In the Prefix text box, type the new prefix to associate with the new family.
6. Click **OK**.

Deleting a Logic Family

If your design requirements change, or you no longer need or support a particular logic family, you can delete a logic family.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
2. In the Part Information dialog box, click the **General** tab and then click the **Families** button.
3. In the Logic Families dialog box, select the logic family that you want to delete from the list.
4. Click the **Delete** button.
5. Click **OK**.

Related Topics

[Modifying Electrical Information for a Part](#)

Assigning PCB Decals

To assign decals, or footprints, to library parts, use the **PCB Decals** tab in the Part Information dialog box. You must assign a decal before assigning gate information, signal pin names, or pin mapping to a part. The decal also specifies the number of pins in the part.

[Assigning an Existing Decal](#)

[Assigning a New Decal](#)

[Unassigning a Decal](#)

[Changing the Default Decal](#)

[Resetting the Tab Data](#)

Assigning an Existing Decal

There are situations where you might want to define more than one PCB decal to a part. This enables one symbol to represent different versions of a part that exist in different physical packages. You can assign up to sixteen PCB decals to a part.

Procedure

1. Open a Part Type.
2. On the toolbar in the Part Editor, click the **Edit Electrical** button and then, in the Part Information dialog box, click the **PCB Decals** tab.
3. In the Library dropdown list, select the library from which you want to assign decals.

The available decals from the selected library appear in the Unassigned Decals list box.

4. To filter the contents of the Unassigned Decals list use any of the following:

- a. Type [Wildcards and Expressions](#) into the Filter box.



Tip

Type asterisk * in the Filter box to display all decals.

- b. Type a number in the Pin Count box, and then click **Apply**.



Tip

Delete all numbers in the Pin Count box to display all decals. This box is always available as a filter to enable decals of differing pin counts to be assigned.

- c. Click the “Show only Decals with pin numbers matching Part Type” check box to filter out decals that do not have pin numbers matching existing gate and signal pins on the **Pins** tab, or the physical pin numbers on the **Pin Mapping** tab.

5. Select a decal name from the Unassigned Decals list box.

6. Click **Assign**. The selected decal is assigned to the part and moved to the Assigned Decals list box.



Tip

As a shortcut, double-click on a decal name to assign it to the part.



Restriction:

Note the following restrictions:

- You must assign a decal with enough pins for all the defined gate pins and signal pins on the **Pins** tab.
 - Only decals with sequential numerical pin numbers can be used with pin mapping.
-

7. To assign additional decals to the part, repeat Steps 4 and 5 above.
8. You can check the part for missing or inconsistent information while the Part Information dialog box is open. Click the **Check Part** button.

The software checks the assigned decals when you exit the tab (regardless of whether you click the **Check Part** button) to ensure they contain physical pin numbers for all the gate and signal pins defined in the **Pins** tab.

Assigning a New Decal

You can specify a decal that does not exist in the library yet, but may exist in another PCB designer's library, or may be created later.

Procedure

1. Open a Part Type.
2. In the Part Editor, on the toolbar, click the **Edit Electrical** button and then in the Part Information dialog box, click the **PCB Decals** tab.
3. Click **Assign New**.
4. In the New PCB Decal dialog box, type a name and click **OK**.

The name is added to the end of the Assigned Decals list.

Unassigning a Decal

As your design needs change, you may need to remove a current decal assignment so that you can reassign a different decal to a part.

Procedure

1. Open a Part Type.
2. In the Part Editor, on the toolbar, click the **Edit Electrical** button and then in the Part Information dialog box, click the **PCB Decals** tab.
3. Select a decal name from the Assigned Decals list box.
4. Click **Unassign**.

The decal is unassigned from the part, and moved to the Unassigned Decals list box.



Tip

As a shortcut, double-click on a decal name to remove it from the Assigned Decals list.

Changing the Default Decal

The first decal listed in the Assigned Decals list box is the default decal used when you add the part to Layout. You can make any decal in the Assigned Decals list the default decal by moving it to the top of the list.

Procedure

1. Open a Part Type.
2. In the Part Editor, on the toolbar, click the **Edit Electrical** button and then in the Part Information dialog box, click the **PCB Decals** tab.
3. In the Assigned Decals list, select the decal that you want to make the default.
4. Click **Up** until the decal is at the top of the list.

The decal becomes the default decal.

Resetting the Tab Data

You may sometimes want to discard changes you have made in the current session with the **PCB Decals** tab, and return the tab's data to its original state.

Procedure

1. Open a Part Type.
 2. In the Part Editor, on the toolbar, click the **Edit Electrical** button and then in the Part Information dialog box, click the **PCB Decals** tab.
 3. Click **Reset**.
-



Tip

Clicking **Reset** affects the current tab only.

Related Topics

[Modifying Electrical Information for a Part](#)

Assigning Gates to Parts

Use the **Gates** tab to assign gate information to a part, including the number of gates, gate swap information, and CAE Decals for the part.



Note:

A space and a period (.) are illegal characters for pin names.

[Gate Decal and Alternates](#)

[Gate and Pin Swap Information](#)

[Assigning Gates to a Part](#)

Gate Decal and Alternates

For each gate, you can enter the CAE Decal name which is the name of the logic symbol that is used to display the part in the schematic. Alternate decal assignments must have the same number of pins. One primary and three alternate decals may be defined for each gate. Assigning a CAE Decal is optional in SailWind Layout, but required in SailWind Logic.

Gate and Pin Swap Information

If at least one decal is assigned to a part, you can enter or modify its gate information. This includes enabling or disabling swaps for gates within a part or between similar parts. This information lets SailWind Logic know which gates it can substitute for connection length minimization after placement.

To gate swap, assign a number to the gate on the **Gates** tab. Gates with like numbers can swap within the part, or to other similar parts. A value of one (1) indicates that the gate is swappable with gates of the same part type in the schematic database. If a part contains more than one type of swappable gate, then identify the second type with the number 2, the third type with 3, etc. The number 0 indicates that no swapping can occur.

Assigning Gates to a Part

You can assign gates to a part.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Gates** tab.
2. To add a gate to the part, click **Add**.

The gate appears in the Gate list. The first gate is automatically designated A, the second gate B, and so forth.

When you select a gate, the Pins list shows the number of pins for the selected gate.

3. To swap gates with the same Swap ID number, select the gate click **Edit**, and then type a value between 1 and 100 in the Swap column.

To disable swap for the gate, type a value of 0.

4. To define Logic decals for the gate, select the cell under CAE Decal 1 click **Edit**, and then type a decal name in the CAE Decal box or click the **Browse** button to search for a decal from a library.

The **Browse** button opens the Assign Decal to Gate dialog box.



Tip

You can define up to 4 alternates; you do not have to define alternates.

See also [Assigning CAE Decals to Gates](#).

5. To remove the selected gate from the Gates list, click **Delete**.

The gates following the removed gate are automatically renamed in sequential order.



Tip

If you delete a gate, the pins information in the **Pins** tab is reset. You can add a gate and move the pins from one gate to the other on the **Pins** tab to retain pin information before you delete the gate.

6. To return the options on the tab to the original settings when the tab first appeared, click **Reset**. This resets only the current tab.

Gate pins are added on the **Pins** tab.

See also [Part Information - Pins](#).

Assigning CAE Decals to Gates

The Assign Decal to Gate dialog box is used to assign a decal to each of the gates in a part. Some parts may have multiple gates of different types, so you can assign individual specific decals to each of the gates.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Gates** tab.
2. Select the cell under CAE Decal 1 and click the **Edit** button or double-click the cell under CAE Decal 1.
3. Type a decal name in the CAE Decal box or click the “...” **Browse** button to search for a decal from a library.

The **Browse** button opens the Assign Decal to Gate dialog box.

4. Select the library from which you want to assign decals.

The available decals from the selected library appear in the Unassigned Decals list box. The list changes according to the library you select.

5. To limit the items that appear in the Unassigned Decals list box, type a [wildcard or expression](#) on page 105 in the Filter box and click **Apply**.
6. Select the decals you want to assign to the gate from the Unassigned Decals list and click **Assign**.



Tip

When assigning decals, observe the following:

- The decal you select in the Unassigned Decals list box is displayed in the preview box.
 - You can assign up to four decals. Assigned decals must have the same number of pins.
The first decal is the default decal, the decal used when you add a part to the schematic.
-

7. To change the order, select the decal in the Assigned Decals list and click **Up** or **Down**.
8. To remove a decal from the gate, select it from the Assigned Decals list and click **Unassign**.
9. To assign a decal name of a nonexistent decal in the library, click **Assign New**.

The Assign New Gate Decal dialog box appears.



Tip

You can specify a decal that does not exist in your library, but may exist in another designer's library, or that you may create later.

10. Click **OK** to apply your changes.

Part Information - Pins

Use the **Pins** tab in the Part Information dialog box to assign gate pins, signal pins, and unused pins to the part. Pin numbers added to the **Pins** tab, must match those of the PCB Decal. Use the **Pin Mapping** tab to overlay logical (schematic) alphanumeric pin numbers onto the physical numeric PCB decal.



Tip

To undo all changes for this tab only, click **Reset**.

[Adding One or More Pins to a Part](#)

[Editing Pin Data](#)

[Assigning a Signal Pin](#)

[Assigning an Unused Pin](#)

[Sorting Table Data](#)

[Renumbering Pins](#)

[Deleting Pins](#)

[Error Checking](#)

[Signal Pin Nets](#)

Adding One or More Pins to a Part

You can add pins to a part using several methods. You can add all pins automatically by assigning a decal. You can add a single pin to the part, add a series of pins, and paste pins from a database; or you can import pins using a comma separate value (CSV) file.

- [Adding a Single Pin](#)
- [Adding a Series of Pins](#)
- [Assigning a Decal](#)
- [Pasting Pin Information](#)
- [Importing Pins Using a CSV File](#)

Adding a Single Pin

You can use the **Pins** tab on the Part Information dialog box to add a single pin to a part.

Procedure

1. Open a Part Type, then, on the toolbar, click the **Edit Electrical** button.
2. On the **Pins** tab of the Part Information dialog box, click the **Add Pin** button.



Note:

You must add a pin number to make the pin valid, and then change any other fields as needed.

If this is the first pin to be added, it takes the default of belonging to Gate-A. If pins already exist, the new pin takes the Pin Group of the currently selected pin.

Adding a Series of Pins

You can use the **Pins** tab on the Part Information dialog box to add a series of pins to a part.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Pins** tab in the Part Information dialog box.
2. On the **Pins** tab of the Part Information dialog box, click the **Add Pins** button.
3. In the Add Pins dialog box, type the number of pins to add in the Number of pins box.



Tip

Total pins for the part can not exceed 32,767.

4. In the Start pin number area, type values in the Prefix and/or Suffix boxes.

A preview of pin numbers based on your input is displayed below the boxes.

**Tip**

When entering pin numbers, observe the following:

- Alphabetic and numeric values can be used in either box. For example, A1 or 1A. If you type alphanumerics and the decal uses numerics, you must use the **Pin Mapping** tab to map the alphanumerics onto the decal.
- For a single numeric, use either the Prefix or Suffix box, and void the other box.

5. In the Increment options area, choose what to increment by clicking either Increment prefix or Increment suffix.
6. In the Step value box, type a positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
7. If using alphanumerics, you can select the “Use JEDEC pin numbering” check box to ensure that legal alphanumeric values are used.
8. Click **OK**.

The new pins display in the list on the **Pins** tab of the Part Information dialog box.

Assigning a Decal

When you assign a decal on the **PCB Decals** tab, the pin numbers from the decal are automatically populated into the **Pins** tab table. PCB Decal pin numbers can be alphanumeric or numeric and the pin numbers in the PCB Decal must match the pin numbers listed in the **Pins** tab table.

Pasting Pin Information

You can copy selected table data from the **Pins** tab or from Microsoft Excel and paste it into the Pins table. The selected cell in the table is the paste origin. Data is pasted below and to the right of the paste origin.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Pins** tab in the Part Information dialog box.
2. If you are copying from Excel, select data and use the Excel Copy command.
If you are copying from the **Pins** tab, select data and click the **Copy** button on the **Pins** tab.
3. Click the **Paste** button to paste the data into the table starting at the paste origin.

**Restriction:**

When the pasted data includes either Pin Group or Pin Number data, the software adds extra pin rows automatically, otherwise the paste will fail if the number of rows and columns in the pasted data does not match those available in the table below and to the right of the paste origin.

Importing Pins Using a CSV File

You can import data from a comma separated value file into a pins table.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Pins** tab in the Part Information dialog box.
2. Click the **Import CSV** button.
3. In the File Open dialog box, browse and select the CSV file.
4. Click **Open** to begin the import.

The entire contents of the **Pins** tab table is replaced with the data of the CSV file.



Note:

CSV field names must correspond to the column headers in the **Pins** tab table. Only the first two characters of the header must match. For example, "Pi" for the Pin Group column. Gate or "Ga" are acceptable alternatives to the Pin Group header.



Tip

A sample CSV file is located in your `\SailWind projects\Samples` folder.

Editing Pin Data

You can click a cell in the row of the pin you are editing to edit the cell contents, or select one or more cells of the same column and click the **Edit** button.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Pins** tab in the Part Information dialog box.
2. Click the Pin Group cell and choose gate, signal pin, or unused pin from the list.

Gates listed in the Pin Group cell list are added using the **Gates** tab. Signal pins require a signal name in the Name cell.

See [Assigning a Signal Pin](#), [Assigning an Unused Pin](#).

3. Click the Number cell and type a pin number for the pin.

The Number cell must contain at least 1 character and can contain up to 15 characters.



Note:

The pin number must match the PCB decal. For example, alphanumeric to alphanumeric.



Tip

Pin numbers can be either numeric or alphanumeric. Prior to PADS2007, alphanumeric pin numbers were not legal on the PCB decal but overlaid the numeric decal numbers, and were stored within the Part Type. You can continue to keep numeric PCB decals and use the **Pin Mapping** tab to overlay different pin numbers onto the numeric PCB decal pin numbers. See [Part Information - Pin Mapping](#).

4. Click the Name cell and type a pin signal or function name. For example, "Clock" or "CLK." A pin name is not required. The Name column is not used for unused pins.
-



Restriction:

Gate pin names can contain up to 40 characters, while signal pin names can contain up to 47. All alphanumeric characters are accepted. In signal pin names, you cannot use special characters such as question marks ?, curly braces {}, asterisks *, periods ., commas , , or spaces. But in gate pin names, you can use any character except a space.

5. Click the Type cell and choose a pin type from the list.

The type column is only used with gate pins.

6. Click the Swap cell and type a swap number, or use the up/down arrows.
-



Tip

You swap pins within gates to uncross connections and facilitate routing. Pins with like numbers can swap within a gate. Type 0 to disable swapping.

7. Click the Sequence (Seq.) cell and type the gate sequence number. The sequence number determines the mapping of CAE gate pins to PCB decal pins. The sequence is automatically shared with alternate CAE decals. For example, the sequence number shows how pin numbers appear on the CAE gate decal; therefore, in Gate A, sequence number 1 could be pin 1, but in Gate B, sequence number 1 would be pin 4.
-



Note:

Exception: When editing pin data for connectors, only the Pin Group and Number columns are relevant. Data entered in other columns is rejected. Connectors do not have gates, so the Pin Group column just indicates whether a pin is a connector pin or an unused pin.

Assigning a Signal Pin

Assign signal names to implicit pins—pins which are not displayed on any gate in the schematic. Typically, ground and power pins are the only implicit pins. You are not required to use Signal Pins. Instead, you can add power and ground pins to a gate or create a separate gate for power and/or ground pins. For the parts in the libraries shipped with SailWind Logic, the standard ground signal name is GND. The standard power signal name is +5V.

Assigning an Unused Pin

You can assign a pin to be an unused pin. An unused pin is a pin that is defined in a PCB decal but has no electrical function in the part type. The unused pin information is not saved in the part type, but is derived automatically based on the number of assigned gate and signal pins to the number of pins in the assigned PCB decal.

Sorting Table Data

You can sort the columns in a pins table in ascending order.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Pins** tab in the Part Information dialog box.
2. Double-click a column header to sort the column.

Renumbering Pins

You can renumber pins in a pins table on the **Pins** tab of the Part Information dialog box. If required, you can use JEDEC pin numbering.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Pins** tab in the Part Information dialog box.
2. Select one or more cells in the Number column.
3. Click the **Renumber** button. In the Renumber Pins dialog box, the Number of pins box displays the number of pins selected for renumbering.
4. In the Start pin number area, type values in the Prefix and/or Suffix boxes.

A preview of pin numbers based on your input is displayed below the boxes.



Tip

When entering pin numbers, observe the following:

- Alphabetic and numeric values can be used in either box. For example, A1 or 1A.
 - For a single numeric, use either the Prefix or Suffix box, and void the other box.
-

5. In the Increment options area, choose what to increment by clicking either Increment prefix or Increment suffix.
6. In the Step value box, type a positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
7. If using alphanumerics, you can select the “Use JEDEC pin numbering” check box to ensure that legal alphanumeric values are used.

8. Click **OK** to apply your changes.

The renumbered pins display on the **Pins** tab of the Part Information dialog box.

Deleting Pins

If you need to modify the part, you can delete pins in the Pin table on the **Pins** tab.

Procedure

1. Open a Part Type, on the toolbar, click the **Edit Electrical** button, and then click the **Pins** tab in the Part Information dialog box.
2. Select one or more cells in the Pin column you want to delete and then click the **Delete Pins** button.

Error Checking

When you click **Check Part**, **OK**, or **Save As**, or when you click a different tab, validation occurs and checks are run for error conditions.

The following conditions are checked:

- Empty pin numbers, pin numbers with embedded spaces, duplicated Pin numbers or special characters such as question marks ?, curly braces {}, asterisks *, periods ., commas , , or spaces.
- Empty, duplicated, or non-sequential sequence numbers within a single gate.
- Non-empty Type cells for signal pins or unused pins, or empty Type cells for gate pins.
- Pin names with illegal characters for gate pins, net names with illegal characters for signal pins, empty name for signal pins. Blank pin names are permitted for gate pins.
- Empty pin swap for gate pins, non-empty pin swap for signal and unused pins. Pin swap values for gate pins outside of the range 0 to 100.

Signal Pin Nets

If signal pin nets are invisible, they do not belong to any particular sheet; therefore, they cannot be individually selected using the mouse.

See also [Signal Pin Nets Dialog Box](#).

Managing Attributes

Use the **Attributes** tab in the Part Information dialog box to add an attribute to a part. A large number of predefined attribute types are available for selection.

See also “[Attributes Overview](#)” on page 222.

- [Adding an Attribute](#)
- [Modifying an Attribute](#)
- [Pasting Attribute Information](#)
- [Deleting an Attribute](#)
- [Setting Default Attributes](#)
- [Adding an Attribute From the Attribute Library](#)
- [Resetting the Attribute List](#)

Adding an Attribute

You can add attribute names and values to a part using the Part Editor.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
2. In the Part Information dialog box, click the **Attributes** tab.
3. Click **Add**.
4. Type an attribute name in the Attribute column.
5. Type an attribute value in the Value column.
6. Click **OK** to assign the new attribute.

Modifying an Attribute

You can modify attribute names and values for a part using the Part Editor.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
2. In the Part Information dialog box, click the **Attributes** tab.
3. Double-click an attribute from the Attribute multi-column list box, or select the attribute and click **Edit**.

4. Type a new attribute name or value in the appropriate text box.
-



Tip

Modifying attributes in the **Attributes** tab modifies the attribute only in the part being edited. Use the [Manage Schematic Attributes Dialog Box](#) and the [Library Manager Dialog Box](#) to manage attributes design-wide or in all libraries.

Pasting Attribute Information

You can copy selected table data from the **Attribute** tab or from Microsoft Excel and paste it into the Attributes table. The selected cell in the table is the paste origin. Data is pasted below and to the right of the paste origin.

Procedure

1. In Excel, select data and use the **Copy** command in Excel, or on the **Attributes** tab, select data and click the **Attributes** tab **Copy** button.
2. Click the **Paste** button to paste the data into the table starting at the paste origin.
3. Click **OK**.

Deleting an Attribute

You can delete attribute names and values for a part using the Part Editor.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
2. In the Part Information dialog box, click the **Attributes** tab.
3. Select an attribute.
4. Click **Delete**.
5. Click **OK**.

Setting Default Attributes

You can save a set of attributes as default attributes to be added automatically each time a new part is created.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
2. In the Part Information dialog box, click the **Attributes** tab.
3. Add all desired default attributes to the list of attributes for a part.
4. Click **Save As Default**.

**Restriction:**

Values are not saved along with the default attributes.

**Tip**

When the default attribute list is created, it is saved in a *defaultattribute.txt* file that is shared between SailWind Logic and SailWind Layout.

5. Click **OK**.

Adding an Attribute From the Attribute Library

You can search through all of the other attribute lists in the library for attributes to add to your current list.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
2. In the Part Information dialog box, click the **Attributes** tab.
3. Click **Browse Lib. Attr** to search for an attribute.
See also [Browsing Library Attributes](#).
4. In the Browse Library Attributes dialog box, select an attribute group from the Group dropdown list to limit the display of attributes to only those of a particular group.
5. Locate the attribute you want to add from the list and double-click it or select it and click **OK** to add it to the **Attributes** tab in the Part Information dialog box.
6. Click **OK** to close the Part Information dialog box.

Resetting the Attribute List

If you make changes to the attribute list and then later decide you do not want to keep them, you can reset the Attribute list for the current session back to the state it was in before the tab was accessed.

Procedure

1. In the Part Editor, open a Part Type, and click the **Edit Electrical** button.
 2. In the Part Information dialog box, click the **Attributes** tab.
 3. Click the **Reset** button to undo all changes made in the current editing session (returns the attribute list to before the tab was accessed).
-

**Restriction:**

Only the current tab is reset.

Browsing Library Attributes

You can browse and list all of the attribute names from libraries specified in the Library List dialog box.

Procedure

1. Click the **Browse Lib. Attr** button to open the Browse Library Attributes dialog box.
 2. In the Filter list, select a category of attributes to match your search.
-



Note:

Click **Refresh** to update the Attributes in library list.

3. Select an attribute from the list.

4. Click **OK**.
-



Tip

The Attributes in library list includes part type and decal attributes.

Part Information - Pin Mapping

Use the **Pin Mapping** tab in the Part Information dialog box to overlay alphanumeric pin numbers onto numeric PCB decal pins. Prior to PADS 2007, alphanumeric pin numbers could not be saved in PCB decals.

Requirements:

- On the **General** tab, select the “Define mapping of Part Type pin numbers to PCB Decal” check box to make the **Pin Mapping** tab available.
- On the **PCB Decals** tab, assign a decal with sequential numerical pin numbers to use the **Pin Mapping** tab. The decal determines the number of pins in the part.



Tip

To undo all changes for this tab only, click **Reset**.

[Mapping Alphanumeric Pin Numbers to Numeric Decals](#)

[Unmapping Pins](#)

[Checking the part](#)

Mapping Alphanumeric Pin Numbers to Numeric Decals

You can use the **Pin Mapping** tab to map alphanumeric pin numbers to numeric decals.

Prerequisites

- On the **General** tab of the Part Information dialog box, select the “Define mapping of Part Type pin numbers to PCB Decal” check box to make the **Pin Mapping** tab available.
- On the **PCB Decals** tab, assign a decal with sequential numerical pin numbers to use the **Pin Mapping** tab. The decal determines the number of pins in the part.

Procedure

1. Click the **Pin Mapping** tab of the Part Information dialog box. In the decal list above the preview window, select the assigned decal to which you want to map alphanumeric pins.
2. Map the pins using one of the following methods:
 - In the Unmapped Pins list, select one or more alphanumerics. Select one or the starting row in the Part Type column if you have a consecutive list to map. Click **Map**.
 - Select a cell in the Part Type column and click the **Edit** button, or simply double-click the cell.
 - Select an alphanumeric in the Unmapped Pins list and double click the pin in the decal preview window to map the alphanumeric to the pin. The next row in the Unmapped Pins list becomes the next selected alphanumeric for mapping.

- Click **Copy Map** to copy both columns of the mapping table. Paste the mapping table into Excel. Make mass edits. Copy the data from Excel and click **Paste Map** on the **Pin Mapping** tab.



Restriction:

Copy Map and **Paste Map** only work with the whole pin mapping table and not selective rows.

3. Repeat as necessary.

4. Click **OK**.

Unmapping Pins

You can use the **Pin Mapping** tab to unmap pins from a decal.

Prerequisites

- On the **General** tab of the Part Information dialog box, select the “Define mapping of Part Type pin numbers to PCB Decal” check box to make the **Pin Mapping** tab available.
- On the **PCB Decals** tab, assign a decal with sequential numerical pin numbers to use the **Pin Mapping** tab. The decal determines the number of pins in the part.

Procedure

1. On the **Pin Mapping** tab of the Part Information dialog box, select a Decal pin number.
2. Click **Unmap**.

Checking the part

After you have finished entering or editing part information, you can check to ensure the information entered in the Part Information dialog box is correct.

Procedure

Click the **Check Part** button to check for missing or inconsistent information entered in the Part Information dialog box.

Assigning Alternate Logic Decals for Connector Symbols

Use the **Connector** tab to define the alternate Logic decals to display in a schematic. Decals are referred to as Special Symbols. You can associate a logical Pin Type with each alternate so that you can have a graphical indication of the connector pin function in the schematic.



Restriction:

The **Connector** tab is available only when you open an existing connector or create a new connector.



Tip

Many users like to use a different symbol, or decal, to distinguish between input (Source) and output (Load) pins. You may define multiple symbols for each of the ten different pin types.

You can define Special Symbols, or connector decals, for different pin types in the part.

Procedure

1. Open a Part Type, click the **Edit Electrical** button on the toolbar and then click the **Connector** tab.
 2. Click **Add**.
 3. Type a Special Symbol or select one from a library by clicking the **Browse** button.
The **Browse** button opens the [Browse for Special Symbols Dialog Box](#).
 4. Double-click the new Pin Type entry, or select the new entry and click **Edit**.
 5. Select a Pin Type from the Pin Type dropdown list.
 6. If you want to remove the special symbol from the list, click **Delete**.
 7. To return the options on the tab to the original settings when the tab first appeared, click **Reset**.
-



Tip

Reset only resets the current tab.

Saving Part Types

After you finish defining or editing your Part Types, you can save them to the library for further use in the current and future design sessions.

Use the [Save Part and Gate Decals As Dialog Box](#) to save part types to the library. You can also rename the associated decal during the save process. If you change the pin count or other information in the decal, this prevents other parts that use the same decal from being affected.



Tip

The Change Type option of the Part Properties dialog box will not update the schematic copy of the part with the modified version in the library if you decrease the number of pins in the associated decal.

Procedure

1. Click the **File > Save As** menu item.
2. In the “Save Part and Gate Decals As” dialog box, type a name in the Name of Part text box.
3. Select a library location from the Library dropdown list box.

To change the name of the associated decal, select the decal from the CAE Decal column of the Names of Decals multicolumn list box and click **Edit**.
Type a new name for the decal.
4. Click **OK**.
5. If the part type already exists, the message “Part Type item exists. Overwrite item Y/N?” appears. Click **Yes** to overwrite the part or **No** to cancel the save.
6. If you modified the part using Edit Part/Hierarchical Symbol to enter the Part Editor, the message “Update all parts of Part Type” appears. Click **Yes** to update parts on the schematic with the revised library version or **No** to leave the schematic version of the part unchanged.

Library Management for Saved Part Types

Knowing where parts are located can help you to better manage your library content. Part Type and decals are saved in specific locations so that they can be logically arranged and found when needed.

The library folder used for part types and associated decals is described below:

- Part type data is saved under the name and library chosen in the Save Part and Gate Decals As dialog box.
- Decals created when editing the part type are saved in the same library as the part type.
- Decals that are modified but not renamed are saved to the library from which they were opened.
- Renamed decals, whether modified or not, if unique across all libraries, are saved in the same library as the part type.
- Renamed decals that are not unique across all libraries display the overwrite warning prompt. If you click **Yes** to overwrite an existing decal with the renamed decal, it replaces the library copy in its current library. If you answer **No** to the overwrite prompt, you can select a different decal name and folder.

Related Topics

[Save Part and Gate Decals As Dialog Box](#)

[Saving a Modified Decal With a Different Name](#)

[Saving a Modified Part Type With a Different Name](#)

Saving a Modified Part Type With a Different Name

If you modify a part type and want to keep the existing library copy, use the Save Part and Gate Decals As dialog box to rename the part. After modifying the electrical information you can save the modified part.

Procedure

1. Click the **File > Save As** menu item.

The Save Part and Gate Decals As dialog box appears.

2. Type a new name for the part type in the Name of Part text box.

3. (Optional) Use the Library dropdown list box to specify a new library location.

See also [Library Management for Saved Part Types](#), [Saving a Modified Decal With a Different Name](#).

4. Click **OK**.

Related Topics

[Saving Part Types](#)

Creating a New Connector

Connectors differ from other parts in that a connector is typically shown as a number of individual pins, rather than a single complete part. When a connector is used on a schematic, the pin number rather than the reference designation is incremented, for example, P1-1, P1-2, P1-3, not P1, P2, P3.

Also, you can assign several CAE Decals to a connector for left and right side locations on the schematic and for alternate decals. The connectors supplied with SailWind Logic use the prefix CON in their name. You can open an existing connector to see how they are constructed.

Procedure

1. In the Part Editor, click the **New** button.
2. In the Select type of editing item dialog box, select “Connector” and then click **OK**.
3. Click the **Edit Electrical** button. The [Part Information for Part -- New Connector dialog box](#) on page 134 displays.
4. On the **Pins** tab, add pins.

For more information, see [Part Information - Pins](#).

5. Select the **Connector** tab.
6. Add the schematic symbols for the connector in the Special Symbols area.
See also [Connector Tab](#) on page 159.
7. Select the **PCB Decals** tab to assign PCB decals for the connector.
See also [PCB Decals Tab](#) on page 141.

8. Select the **Attributes** tab to assign attribute information.
See the [Managing Attributes](#) topic for more information.
9. Click **OK** to close the Part Information dialog box. The connector symbols display in the Part Editor.
10. Click the **File > Save As** menu item.
11. Enter a name and library folder location.
12. Click **OK**.
13. Click the **File > Exit Part Editor** menu item.

Browsing for Connectors

Use the Browse for Connector dialog box to access the library and open an existing connector.

Procedure

1. Type a [wildcard or expression](#) on page 105 in the Items box to filter the connectors, and click **Apply**.
2. Select a connector from the list.
3. Click **OK**. The connector decal(s) display in the Part Editor.

Related Topics

[Creating a New Connector](#)

Creating a New Pin Decal

Pin Decals are used to represent the terminal pins on a part. You can make a copy of an existing pin decal and modify it or you can create a new pin decal.

[Create a New Pin From an Existing Pin Decal](#)
[Create a New Pin Decal](#)

Create a New Pin From an Existing Pin Decal

To save design time and promote reuse, you can create a new pin from an existing pin decal.

Procedure

1. In the Part Editor, click the **Open** button.
2. In the [Select Type of Editing Item Dialog Box](#), select “Pin Decal” and then click **OK**.
The Pin Decal Browse dialog box displays.
3. Select an existing pin decal from the Pins area.
The highlighted decal is displayed in the Picture area.
4. Click **OK**.
5. Modify the graphics for the decal and position the text strings as required.
6. Click the **File > Save As** menu item.
The Save Item to Library dialog box displays.
7. Enter a name and library location for the new pin decal.
8. Click **OK**.

Create a New Pin Decal

When you create a new Pin Decal, SailWind Logic automatically creates the necessary text strings for the associated pin information. You only need to specify the 2D lines that represent the terminal pin, then reposition the text strings.

Procedure

1. In the Part Editor, click the **New** button.
2. In the [Select Type of Editing Item Dialog Box](#), select Pin Decal.
3. Click **OK**.

Four text entries are displayed in the working area:

- #E is a placeholder for the pin number.
 - PNAME is a placeholder for the pin name.
 - NETNAME is a placeholder for the net name.
 - #0:TYP=U SWP=0 are placeholders for the sequence number, pin type and swap class.
4. Use the Create 2D Line button in the Decal Editing toolbar to create the pin decal.
5. Reposition the text strings, if necessary.
6. Click the **File > Save** menu item.
- The Save Item to Library dialog box appears.
7. Enter a name and library location for the new pin decal.
8. Click **OK**.



Tip

To control which decals are displayed in the Pin Decal List, use the Pin List Manager

Editing Objects in the Decal Editor

The Decal Editor offers a flexible environment for editing your decals. You can create new parts, edit existing parts and also create derivative parts from existing parts.

To edit objects in the Decal Editor, refer to the following topics:

- [Object Selection Control in the Decal Editor](#)
- [Selecting Multiple Objects in the Decal Editor](#)
- [Changing Objects in the Decal Editor](#)
- [Modifying Terminals](#)

Chapter 8

Special Schematic Symbols

Special Symbols in SailWind Logic include power symbols, ground symbols, and off-page reference symbols. SailWind Logic enables only one part definition in the library for Special Symbols. For this reason, the options for these types are grayed out and unavailable when you are in the Part Editor and you select **File menu > New**. You can only modify the existing symbols or add new symbols to the part definition.

- [Special Symbol Naming Conventions](#)
- [Assigning Alternative Symbols for the Ground Part](#)
- [Assigning Alternative Symbols for the Off-Page Part](#)
- [Assigning Alternative Symbols for the Power Part](#)
- [Creating New Special Symbols](#)
- [Creating New Special Symbols From Existing Symbols](#)
- [Updating Special Symbol Mappings](#)
- [Management of Special Symbols in the Library](#)

Special Symbol Naming Conventions

The special symbols in the SailWind Logic library use specific names or a unique naming prefix. This topic contains tables that provide definitions that can help you to locate a special symbol in the library.

Table 20. Off-Page Reference Symbols

Symbol	Description
REFIN	Off -page reference for left side
REFOUT	Off -page reference for right side

Table 21. Power Symbols

Symbol	Description
+	+5 volt and +12 volt symbols, arrow pointing up
-	-5 volt and 12 volt symbols, arrow pointing down
BUBBLE	+5 volt symbol with a circle
Y	+5 volt symbol with a 'Y' shape

Table 22. Ground Symbols

Symbol	Description
GND	Standard ground

Table 22. Ground Symbols (continued)

Symbol	Description
AGND	Analog ground
CHGND	Chassis ground

Assigning Alternative Symbols for the Ground Part

If your design requires the use of multiple ground symbols, use the Assign Alternatives for Ground Part dialog box to assign ground symbols.

Procedure

1. In the Part Editor, on the toolbar, click the **Open** button.
2. In the [Select Type of Editing Item Dialog Box](#), click **Ground** and then click **OK**.
The current symbols are displayed.
3. Click the **Edit Electrical** button.
4. Click **Add**. This creates a new entry for the symbol.



Tip

To add a new ground symbol, follow the procedure in [Creating New Special Symbols](#).

5. In the Special Symbol column, click the button to the right of the newly created text box.
The “[Browse for Special Symbols dialog box](#)” on page 487 appears.
6. To filter the symbols, type a [wildcard or expression](#) on page 105 in the Items box, and click **Apply**.
7. Select a symbol decal.
8. Click **OK**.
9. In the [Assign Alternatives for Ground Part Dialog Box](#), select the text box under Pin Type.
10. Click **Edit**.
11. Select a pin type from the dropdown list box.
This is normally set to **Ground**.
12. Select and edit the text box for the Signal Name.
13. Click **OK**.

Assigning Alternative Symbols for the Off-Page Part

If your design requires the use of multiple different off-page symbols, use the Assign Alternatives for Off-Page Part dialog box to assign additional off-page reference symbols.



Note:

See “[Special Schematic Symbols](#)” on page 167 for additional information on the creation and use of special schematic symbols.

Procedure

1. In the Part Editor, on the toolbar, click the **Open** button.
2. In the [Select Type of Editing Item Dialog Box](#) click Off-page and then click **OK**.
The current symbols are displayed.
3. Click the **Edit Electrical** button.
4. Click **Add**.

This creates a new entry for the symbol.



Tip

To add a new off-page reference symbol, follow the procedure in [Creating New Special Symbols](#).

5. In the Special Symbol column, click the button to the right of the newly created text box.
The [Browse for Special Symbol dialog box](#) on page 487 appears.
 6. To filter the symbols, type a [wildcard or expression](#) on page 105 in the Items box, and click **Apply**.
 7. Select a symbol decal.
 8. Click **OK**.
 9. In the [Assign Alternatives for Off-Page Part Dialog Box](#), select the text box under Pin Type.
 10. Click **Edit**.
 11. Select a pin type from the dropdown list box.
-



Tip

For left side off-page references, select Source, for right side, select Load.

12. Click **OK**.
-

Assigning Alternative Symbols for the Power Part

If your design requires the use of multiple different power symbols, use the Assign Alternatives for Power Part dialog box to assign additional power symbols.

Procedure

1. In the Part Editor, on the toolbar, click the **Open** button.
2. In the [Select Type of Editing Item Dialog Box](#), click Power, and then click **OK**.
The current symbols appear.
3. Click the **Edit Electrical** button.
4. Click **Add**.

This creates a new entry for the symbol.



Tip

To add a new power symbol, follow the procedure in [Creating New Special Symbols](#).

-
5. In the Special Symbol column, click the button to the right of the newly created text box.
The “[Browse for Special Symbols dialog box](#)” on page 487 appears.
 6. To filter the symbols, type a [wildcard or expression](#) on page 105 in the Items box, and click **Apply**.
 7. Select a symbol decal.
 8. Click **OK**.
 9. In the [Assign Alternatives for Power Part Dialog Box](#), select the text box under Pin Type.
 10. Click **Edit**.
 11. Select a pin type from the dropdown list box.
This is normally set to Power.
 12. Select and edit the text box for the Signal Name.
 13. Click **OK**.

Creating New Special Symbols

Special symbols in SailWind Logic include power symbols, ground symbols, and off-page reference symbols. SailWind Logic enables only one part definition in the library for special symbols. For this reason, the options for these types are unavailable when you are in the Part Editor and you select the **File > New** menu item.

See also [Creating New Special Symbols From Existing Symbols](#).

Procedure

1. In the Part Editor, click the **Open** button.
2. In the **Select Type of Editing Item Dialog Box**, select the type of symbol to add: Off-page, Power, or Ground.
3. Click **OK**.

The currently defined symbols display for the chosen type.

4. Click the **Edit > Part Type Editor** menu item, or on the toolbar, click the **Edit Electrical** button.

The Assign Alternatives dialog box for the type of Special Symbol appears.

Refer to the following topics for additional information:

- **Off-page symbols** — Refer to [Assigning Alternative Symbols for the Off-Page Part](#).
- **Power symbols** — Refer to [Assigning Alternative Symbols for the Power Part](#).
- **Ground symbols** — Refer to [Assigning Alternative Symbols for the Ground Part](#).

5. Click **Add**. The software creates a new entry.

6. Type a name for the new symbol and enter the appropriate information.



Tip

For left side off-page references, select Source, for right side, select Load.

-
7. Click **OK**.

8. Click the **Edit > CAE Decal Editor** menu item, or on the toolbar, click the **CAE Decal Editor** button.

The **Select Pin Decal Dialog Box** appears.

9. Select the new symbol name and when prompted, click **OK** to create the new symbol.

10. Click the **Decal Editing Toolbar** button, click the **Create 2D Line** button, and then right-click and use the available commands to create the symbol.



Tip

In the schematic, the net connection point will be the origin of the symbol.

-
11. Reposition the text strings as required.

12. Click the **File > Return to Part** menu item, and when prompted, click **Yes** to keep the gate changes.

The new symbol and existing symbols are displayed.

13. Click the **File > Save** menu item.

If the [Save Part Type to Library Dialog Box](#) displays, accept the defaults and click **OK**.

14. Click the **File > Exit Part Editor** menu item.
-

**Tip**

For SailWind Logic to recognize the new special symbol, perform the instructions in [The Update From Library Function](#).

Creating New Special Symbols From Existing Symbols

Special symbols in SailWind Logic include power symbols, ground symbols, and off-page reference symbols. SailWind Logic enables only one part definition in the library for off-page reference symbols, so this option is unavailable when you click the **New** button to create a new part. You can modify the existing symbols and/or add new symbols.

See also [Creating New Special Symbols](#).

Procedure

1. Do one of the following:

- Copy an Existing Special Symbol
 - a. In the Library Manager (**File > Library** menu item), click the **Logic** button.
 - b. To filter the symbols, type a [wildcard or expression](#) on page 105 in the Filter box, and click **Apply**.

See also [Special Symbol Naming Conventions](#).

- c. Click **Copy**. The Save CAE Decal to Library dialog box displays.
-

**Tip**

Copy is unavailable if the Library list is set to All Libraries. (The library name is listed before the item in the CAE Decals list.) Double-click on the symbol to enter the library and then click **Copy**.

- d. Type a name for the new special symbol.

- e. Click **OK**.

- Modify an Existing Special Symbol

- a. In the Library Manager, click the **Logic** button.

- b. To filter the symbols, type a [wildcard or expression](#) on page 105 in the Items box and click **Apply**.

- c. Select the symbol in the CAE Decals area.

- d. Click **Edit**.
-



Tip

Edit is unavailable if the Library list is set to All Libraries. (The library name is listed before the item in the CAE Decals list.) Double-click on the symbol to enter the library and then click **Edit**.

- e. Minimize the Library Manager dialog box or move it to one side.

- f. Modify the graphics and rearrange the text strings as required.

- g. Click the **Save** button.

2. Close the Library Manager dialog box or use it for another operation.

Refer to the following topics for information on assigning the new decal as an alternate decal:

Table 23. Topic List Regarding Assigning New Decal as Alternate Decal

Alternate symbols	Topic to see
Off-page symbols	Assigning Alternative Symbols for the Off-Page Part
Power symbols	Assigning Alternative Symbols for the Power Part
Ground symbols	Assigning Alternative Symbols for the Ground Part

Updating Special Symbol Mappings

You can change the assignment or “mapping” of a special symbol using the Update from Library function. For example, you can assign a new ground symbol to both a common ground and an analog ground.

Procedure

1. Click the **Tools > Update from Library** menu item.
2. In the Mode area of the [Update From Library Dialog Box](#), choose “Update design from library.”
3. In the Items area, select one or more of the following check boxes:
 - Off-page symbols
 - Ground symbols
 - Power symbols
4. Click **OK**.
5. In the Schematic Symbol column of the [Remap Special Symbols Dialog Box](#), locate the power, ground, or off-page symbol that you want to re-map double-click on the corresponding Library Symbol box and then select the new symbol that you want to assign to the schematic symbol.



Note:

Only symbols currently associated with the special symbols in your library appear in the Library Symbol dropdown list.

6. Repeat step 4 for every symbol you want to re-map.

7. Click **OK**.

SailWind Logic updates the symbols and generates the [Update From Library report](#) on page 206.

The updated symbols appear in the “Mapped to” column of the report.

Management of Special Symbols in the Library

Special Symbols are those used to create off-page reference, power and ground symbols, and connectors. The default location of these special symbols is the `C:\<install_folder>\<version>\Libraries`.

Reinstalling the library will overwrite the common library files. If you modify a special symbol and want to keep the change, use Library/Export to transfer the symbol from the `C:\<install_folder>\<version>\Libraries` folder to the `C:\<install_folder>\<version>\Libraries\usr` folder or to another library name.

Chapter 9

Design and Editing Basics

A wide selection of methods are available for design editing and navigation. This enables a high level of flexibility for selecting, moving, duplicating, and deleting design objects. A robust selection filter allows precise selection of design objects individually and as groups. There are also operations available for finding design objects as well as a very powerful step and repeat function.

- [Design Operations](#)
- [Modes of Operation](#)
- [Selecting Objects](#)
- [Controlling Selections](#)
- [Filtering Object Selections](#)
- [Using the Selection Filter](#)
- [Selection List](#)
- [Find Objects](#)
- [Step and Repeat](#)

Design Operations

When you start SailWind Logic or select **New** from the **File** menu, a drawing format is automatically added to the work area. The drawing format is the representation of the sheet on which you will begin to create your schematic design.

See [Options Dialog Box, Design Category](#) for information about changing the default drawing format. Use Options to set the default working grid, display grid, text size, etc.

To add new design information:

- On the toolbar, click the **Schematic Editing toolbar** button
- Add and edit parts from libraries
- Create hierarchical symbols
- Add connections
- Swap pins and reference designators
- Define bus structures
- Create and edit non-electrical information; text, charts, notes, etc.

Related Topics

- [Part Editor Operations](#)

Modes of Operation

You can use the different modes of operation to edit designs in SailWind Logic. These include verb mode and object mode. There are also multiple choices available for zooming to various areas of your design using your pointer device or keyboard commands. To accommodate repetitive operations, you can use duplicate mode, delete mode, or move mode

[Editing Basics](#)

[Zooming](#)

[Using Duplicate Mode](#)

[Using Delete Mode](#)

[Using Move Mode](#)

Editing Basics

Most design work involves editing the database—adding, modifying, and deleting items. This section describes the modes of operation in SailWind Logic that enable you to edit your designs.

SailWind Logic has two basic modes of operation: [Object Select Mode \(Select Object First\)](#), and [Verb Mode \(Select Command First\)](#). In addition to these operational modes, SailWind Logic has Zoom mode. Zooming overrides all other modes until you specify another mode.

[Verb Mode \(Select Command First\)](#)

[Object Select Mode \(Select Object First\)](#)

Verb Mode (Select Command First)

Commands that operate on a selected object are considered verb commands. In Verb mode, select the mode, and then select the objects on which to perform the command.

Procedure

1. To put SailWind Logic in Verb mode, select one of the following commands from the standard toolbar.
 - [Using Duplicate Mode](#)
 - [Using Move Mode](#)
 - [Using Delete Mode](#)
 - Query Mode — (see [Schematic Object Modification](#))
2. Then select an object. When you move the cursor off the toolbar, a small V appears on the cursor to show that SailWind Logic is in Verb mode.
3. To exit Verb mode, on the Schematic Editing toolbar, click the **Select** button or press the Esc key.

Object Select Mode (Select Object First)

In Object Select mode, select the object, and then select a command to perform an action.

Procedure

1. Position the cursor over the object and click once.
The selected item highlights.
2. Select a command, using one of the following:

- Right-click and click a command.
- Click a command from a menu.
- Click a button.

Zooming

The **Zoom** button acts as a toggle. You can zoom in, out or to a specific area of the design.

[Zooming In](#)

[Zooming Out](#)

[Specify the Zoom Area](#)

Zooming In

You can zoom into the view to see more detail in your design.

Procedure

1. On the standard toolbar, click **Zoom** to enter zoom mode.
2. Point to the new view center and click.

Zooming Out

You can zoom out from the current view to see more of the objects in your design.

Procedure

1. On the standard toolbar, click **Zoom** to enter zoom mode.
2. Point to the new view center and right-click.

Specify the Zoom Area

To view a specific area of your design, you can drag a rectangle to specify the zoom area.

Procedure

1. On the standard toolbar, click the **Zoom** button to enter zoom mode.
2. Press and hold the left button and drag diagonally upward to zoom in.
3. Press and hold the left button and drag diagonally downward to zoom out.

See also [Creation of Groups](#).

Using Duplicate Mode

Use Duplicate Mode to replicate existing parts, text, and drafting objects. When you copy parts, SailWind Logic automatically increments the reference designator and gate modifier.

Procedure

1. On the toolbar, click the **Duplicate** mode button.
2. Select the item to copy.
A duplicate of the item follows the cursor movement.
3. (Optional) Depending on the available options for the selected item, you can adjust the orientation and other settings from the popup menu before placement. Position the item and indicate its location to complete the copy.
4. When you finish placing copies of this item, right-click and click the **Cancel** popup menu item.
SailWind Logic remains in duplicate mode until you select another mode or the **Select** button from the toolbar. You can also press the Esc key to exit the mode.

Related Topics

[Schematic Parts](#)

[Non-Electrical Objects](#)

[Step and Repeat](#)

Using Delete Mode

Use Delete mode to remove parts, connections, unconnected buses, text, drafting objects, and hierarchical symbols from the design. If you delete a part with connections, SailWind Logic also deletes the connections.



Tip

If you are in [Object Select Mode \(Select Object First\)](#), you can delete a part without deleting the connections.

Procedure

1. On the toolbar, click the **Delete** mode button.
2. Select the object to delete.

SailWind Logic remains in Delete mode until you select another mode or the **Select** button from the standard toolbar. You can also press the Esc key to exit the mode.

Using Move Mode

Use Move Mode to adjust the position of existing parts, connections, text, and drafting objects.

Procedure

1. On the toolbar, click the **Move** mode button.
2. Select the object to move.

The object follows the cursor movement.



Tip

(Optional) Depending on the available options for the selected object, you can adjust the orientation and other settings from the popup menu before placement. Position the object and indicate its location to complete the move.

SailWind Logic remains in move mode until you select another mode or the **Select** button from the toolbar. You can also press the Esc key to exit the mode.

Selecting Objects

You can select individual or multiple objects in your design. Selecting multiple objects enables you to apply a specific action to multiple objects simultaneously.



Note:

You can also search and select objects. For more information, see [Searching for an Object by Typing Information](#).

- [Selecting One Object](#)
- [Selecting Several Objects](#)
- [Selecting All Objects in an Area](#)
- [Selecting All Objects on a Sheet](#)
- [Selecting an Object on All Sheets in a Schematic](#)

Selecting One Object

To highlight or apply a command only to one object, you can select a single object in your design.

Procedure

Place the cursor over the object and click the left mouse button.

The object selects and highlights.

Any previously selected objects are deselected. If you click over empty space, all previously selected objects are deselected.

If you try to select an object in a dense or crowded area, use the [Using the Selection Filter](#) to disable other items for selection.

To select invisible signal pin nets, which you cannot select by clicking on them, see the [Signal Pin Nets](#) topic.

Selecting Several Objects

To highlight or apply a command to multiple objects, you can select several objects in your design.

Procedure

Press and hold the Ctrl key while you select multiple items.

If you did not select an object previously, the software adds it to the set of selected objects.

If you selected the object previously, the software removes it from the set of selected objects.

Selecting All Objects in an Area

To highlight or apply a command to all objects, you can select all of the objects in a particular area of the design.

Procedure

1. Hold the left mouse button down and drag a selection rectangle around one or more objects; start at one corner of the area and drag to the opposite diagonal corner.
2. When you release the button, the software selects all objects contained within the rectangle.

You can add additional objects to the selection or remove objects from the selection using Ctrl +click.

Selecting All Objects on a Sheet

On the sheet you are viewing, you can select all objects of a particular type, depending on the Selection Filter you are using.

For example, you could select all nets or all gates on the sheet.

Procedure

To select all objects of a particular type on the sheet you are viewing, click the **Edit > Select All on Sheet** menu item, or press Ctrl+A.

Selecting an Object on All Sheets in a Schematic

Across the entire schematic, you can select all objects of a particular type, depending on the Selection Filter you are using. For example, you could select all nets or all gates on every sheet in the schematic.

Procedure

To select all objects of a particular type on every sheet in the schematic, click the **Edit > Select All on Schematic** menu item, or press Shift + Ctrl + A.

Controlling Selections

Sometimes you cannot easily select the object you want because several objects exist at the same location. You can use the Selection Filter to specify specific object types so that only those items can be selected.

To control what information you select, use the [Using the Selection Filter](#).

See also [Selection List](#).

Filtering Object Selections

There are three ways to filter the objects you select: the Selection toolbar buttons, the popup menu when nothing is selected, and the Selection Filter Dialog Box.

[Filtering With the Selection Toolbar](#)

[Filtering With the Popup Menu](#)

[Filtering With the Selection Filter Dialog Box](#)

Filtering With the Selection Toolbar

Use the Selection toolbar buttons to enable or disable object selection. This enables you to limit your selection to only specific object types.

Procedure

1. On the toolbar, click the **Selection Toolbar** button.
2. On the Selection Filter toolbar, click the buttons for the objects for which you want to enable or disable selection.



Tip

You cannot use the Parts Filter and the Gates Filter at the same time.

Filtering With the Popup Menu

With nothing selected, you can use the popup menu that appears to set the selection filter to the desired object type.

Procedure

1. With nothing selected in the schematic, right-click to open the popup menu.
2. Select the object you want to filter from the list.

Filtering With the Selection Filter Dialog Box

You can use the Selection Filter dialog box to specify a specific filter. This enables very precise selection of objects by enabling only those object types that you want to select.

Procedure

1. Click the **Edit > Filter** menu item.
2. Select the desired item or items for your filter.

Using the Selection Filter

Use the Selection Filter dialog box to control which objects you can select. The Selection Filter offers the capability to set very precise filters enabling you to select anything, a very narrow subset of the available design objects, or nothing at all. You can use the Nothing selection to quickly reset the filter and then you can make a new filter selection.

Procedure

1. With nothing selected, right-click and click the **Filter** popup menu item.
As an alternative, click the **Edit > Filter** menu item.
2. In the Selection Filter dialog box, select the check box beside each design object that you want to turn on or off.
Click **Anything** if you want to select the check boxes for all design objects or click **Nothing** if you want to clear all check boxes.
3. Click **Close**.

Related Topics

[Filtering Object Selections](#)

[Object Select Mode \(Select Object First\)](#)

Selection List

The Selection list provides an efficient means of searching and selecting particular objects in your design.

[Searching for an Object by Typing Information](#)

[Selecting an Object With Area Selection](#)

Searching for an Object by Typing Information

You can search for one or many objects by entering search criteria in the Selection List box of the Selection toolbar.



Restriction:

You must set the selection filter to enable selection of the object type being searched.

Procedure

1. On the Standard toolbar, click the **Selection Toolbar** button.
 2. In the list box of the Selection Filter toolbar, type information about the object.
For example, type the part name, gate name, part type name, net name, or pin name of the object you want, and then press the Enter key.
- The workspace pans to the object and selects it.



Tip

When working with the Selection List, note the following:

- The Selection List supports [Wildcards and Expressions](#).
- You can search for several objects by separating entries with commas.

See also [Filtering Object Selections](#).

Selecting an Object With Area Selection

When you make an area selection, the Selection list on the Selection toolbar displays the selected objects and the sheet on which they are located. When multiple objects are selected, All in List appears in the list, making it easier for you to locate a particular object by just selecting the item, especially in a congested design.

Procedure

1. On the Standard toolbar, click the **Selection Toolbar** button.
2. Click the Selection Filter toolbar buttons to include the object you want to locate.
See also [Filtering Object Selections](#).
3. Make an area selection in the general area of the object you want to locate.

4. Select the particular object from the Selection list on the Selection Filter toolbar.
- If desired, select “All in List” to re-select all of the objects in the multiple selection.
5. Click the **Next Object** or **Previous Object** buttons to move the selection from one object to the next in the current multiple selection.



Tip

Other selection methods:

- To make multiple selections in the schematic, use Ctrl or Shift and click.
- To locate a very small object, zoom in on the current sheet, then select the object from the list.

Find Objects

You can search for objects in SailWind Logic using a number of different options. These include the modeless commands, the Selection Toolbar, the Selection Filter, and by controlling the view.

Table 24. Search for Objects in SailWind Logic Using These Options

Option	Description
Modeless Commands and Keyboard Shortcuts on page 569	Use keystrokes to locate specific objects or types of objects.
Selection Toolbar	Controls which types of objects you can select, locates specific objects from a multiple selection, or moves between sheets in the schematic.
Using the Selection Filter	Controls types of objects you can select. Enables you to select the object you want even when several objects exist at the same location.
View Control	Changes the perspective and size of the work area.

Step and Repeat

Use Step and Repeat to multiply objects as you place them during an add or duplicate operation. Step and Repeat is available in the Schematic Editor and the Decal Editor. In the Schematic Editor, the Step and Repeat command copies parts, connections, text, or drafting items. In the Decal Editor, the Step and Repeat command copies terminals, text, or drafting items.

Procedure

1. Select an object during an add or duplicate operation. While a dynamic object is attached to the cursor, right-click and click the **Step and Repeat** popup menu item.



Tip

When adding a new object in the Schematic Editor, you must place the first object manually before you can use Step and Repeat.

2. In the Step and Repeat dialog box, click the direction of placement for the array.
 3. In the Count box, type the number of objects to place or click the arrow buttons.
 4. In the Distance box, type the distance between objects or click the arrow buttons.
-



Tip

If you place a second object and then Step and Repeat, the spacing between the objects will become the default value in the Distance box and will repeat the pattern you have started.

5. Click the **Preview** button to view the placement of the multiple objects based on the options you set. The placement of the objects is based on the location of the original object selected.
-



Tip

Zoom Mode is available during Step and Repeat.

6. After achieving the desired placement preview, click **OK** to place the objects.

Results

After using Step and Repeat, the original dynamic object is still attached to the cursor. You can continue to add or duplicate using this object, or you can press the Esc key to end the operation.

When duplicating connections, only connections with valid start and end points remain after the Step and Repeat operation; even if invalid connections appear during the preview. A duplicated connection can start on a component pin or on another connection segment. A duplicated connection can end on a component pin, a connection segment, or a bus segment.

SailWind Logic automatically assigns new bus netnames to bus segment copies. The new netname is based on the original bus segment netname, plus one. For example, if you make a duplicate of bus segment D00, the duplicate is assigned the netname D01.

See also [Working With Floating Connections](#).

If you manually place a duplicate within 0.5" of the original (in both X and Y directions), the next connection is placed at the same offset. Then press the Space Bar to place a new duplicate. You can continue to use this semi-automated Step and Repeat process to place subsequent duplicates.

Chapter 10

Schematic Parts

Numerous methods are available for adding, swapping, saving, updating and controlling visibility of parts in your design. These operations can be performed on individual design objects, or groups of objects.

- [Adding Parts](#)
- [Adding Connector Pins](#)
- [Adding Single Gate Parts](#)
- [Adding Multigate Parts](#)
- [Controlling Text Visibility for a Part](#)
- [Using Alternate Symbols](#)
- [Swapping Reference Designators](#)
- [Saving Part Types to a Library](#)
- [The Update From Library Function](#)
- [Deleting a Part](#)
- [Deleting a Part and Its Connections](#)
- [Cut, Copy, and Paste](#)
- [Copy as Bitmap](#)
- [Creation of Groups](#)
- [Group Anchor Points](#)
- [Management of Groups](#)
- [Attributes Overview](#)
- [Manage Attributes in a Schematic](#)
- [Resistor Values Defined on Parts](#)

Adding Parts

Use the Add Part from Library dialog box to load a part from a library into the current schematic drawing. SailWind Logic automatically assigns a reference designator when you add the part.

Refer to the [Adding Single Gate Parts](#) and [Adding Multigate Parts](#) topics for additional information.

Refer to the [Using Duplicate Mode](#) topic for information on adding parts by making copies of parts that already exist in the schematic.

Procedure

1. On the Schematic Editing toolbar, click the **Add Part** button.
 2. Select a part in the Items list or select a part from the Part Name dropdown list box in the [Add Part From Library Dialog Box](#).
-



Tip

Use the filter to locate the part. Use the wildcard convention, with or without leading characters, to expand or narrow the filter. Click **Apply** to activate the filter.

3. Click **Add**. The part attaches to and follows the cursor movement.
(Optional) You can rotate or mirror the item before you indicate its location. Right-click and click Rotate 90, X Mirror, and Y Mirror as often as needed to set the correct orientation of the item. Click **Alternate** to place an alternate symbol, if available. Click **Next Type** to select a different gate in the part, if available.
4. Click on the schematic to place the part; another instance attaches to the cursor automatically.
5. When you are done adding the part(s), right-click and click the **Cancel** popup menu item or press the Esc key.

Related Topics

[Using Alternate Symbols](#)
[Adding Connector Pins](#)

Adding Connector Pins

Connector pins are special symbols within SailWind Logic libraries. You add them like multigate parts, whereby the pin number increments each time you add a connector symbol. SailWind Logic does not increment the reference designator until you use all the connector pins.

After adding one or more pins of a connector and establishing a reference designator, use [Using Duplicate Mode](#) to add additional pins. This causes SailWind Logic to use the same reference designator for new pins.

Adding Single Gate Parts

Single gate parts are parts that are represented by a single schematic symbol. When you add the part to the schematic, unique reference designator assignment starts with the next available reference designator number.

The [Adding Parts](#) section lists the step-by-step procedure.

You specify the reference designator prefix in the [General tab](#) on page 136 of the Part information dialog box when you create a part.

Related Topics

[Adding Multigate Parts](#)

Adding Multigate Parts

Multigate parts are parts that consist of more than one symbol or gate to represent a complete part; for example, a 7400 which contains 4 gates. You can create parts that have multiple gates of a single type or multiple gates of different types.

When you add a new multigate part, the gate with the lowest set of pin numbers is used along with the next available reference designator and an alphanumeric suffix, or gate modifier. The reference designator and suffix are separated by the dash (-) character. For example, a 7400 added to a new schematic would use the gate with pins 1 through 3 and would be assigned as part U1-A. When you add

additional gates of the same part, the next available gate and gate modifier is used, for example, the gate with pins 4 through 6 is added and assigned as U1-B.

SailWind Logic continually manages the packaging of gates in a schematic. If you delete, rename, or change the part type of a gate, SailWind Logic keeps track of these changes and fills in the gaps when a new gate is added.

Using Alphanumeric Prefixes

You can use any combination of letters and numbers in SailWind Logic to define a reference designator prefix.

You specify this prefix in the [General tab](#) on page 136 of the Part information dialog box when you create a part.

If you specify a reference designator prefix that contains alphanumeric characters, SailWind Logic maintains this prefix for multigate parts. A prefix of U1A for a part containing four gates, when added to a design, would use reference designator assignments U1A-A, U1A-B, U1A-C, and U1A-D.

- [“Adding Parts” on page 189](#)
- [Adding Single Gate Parts](#)
- [Rename Gate](#)
- [Rename Part](#)
- [Modifying Parts](#)

Controlling Text Visibility for a Part

Use the Part Text Visibility dialog box to control the display of text associated with the selected part. You can control the visibility of one part or all parts of the same type.

Procedure

1. Select a part, right-click and click the **Visibility** popup menu item.
2. In the Attributes list, select the check boxes of attributes to display in the schematic.
3. In the “Item Visibility” area, select the check boxes to display the corresponding label, such as part type or pin numbers.
4. In the “Attribute Name Display” area, set the attribute name option. Display just the attribute value or display the attribute name and its value, for example, PCB DECAL=SO14.
 - **All Off** — Makes all attribute names invisible, displays only the value.
 - **No Change** — Keeps the current attribute name visibility settings.
 - **All On** — Displays all attribute names and their values.
5. In the “Apply Update to” area, set the selected part update options. This area is unavailable if you have selected more than one part on the schematic.

- **This Gate** — Updates the selected gate.
- **This Part** — Updates a part or all associated gates of a part.
- **All Parts This Type** — Updates all matching gates or parts in the design.

6. Click **OK**.



Tip

When using [object mode selection](#) on page 177, with multiple parts selected, check boxes may be in an intermediate state (selected, yet you cannot edit them) because some selected parts have the check box selected and others have it cleared. You can select or clear the check box and update all selected parts.

Using Alternate Symbols

Some parts can be shown in the schematic with more than one representation. One example of this is the NAND gate, with its DeMorgan representation. A second example is the use of IEEE and ANSI symbol standards. Within SailWind Logic's library, it is possible to define one primary and up to three alternate symbols for a part.

While placing the part in the schematic, the Alternate command enables you to select the version you want to use in the schematic.

Power symbols, ground symbols, and off-page reference symbols are also available as alternate symbols.

When you add a new part or duplicate an existing part, and you want the alternate representation, right-click and click **Alternate** before placing the part.

Procedure

Right-click and click the **Alternate** popup menu item while in Duplicate or Move mode.

Swapping Reference Designators

To swap reference designators, you can use the **Swap Ref. Des.** button on the Schematic Editing toolbar or **Swap Ref. Des.** on the popup menu. If the reference designator prefixes of the selected parts are different, swapping also updates the reference designators of other parts in the package.

[Swapping Reference Designators Using Verb Mode](#)

[Swapping Reference Designators by Selecting One Part at a Time](#)

[Swapping Reference Designators by Selecting Both Parts First](#)

Swapping Reference Designators Using Verb Mode

You can swap reference designators using verb mode. Using verb mode enables you to initiate the command and then perform consecutive swap operations without having to restart the command each time.

Procedure

1. On the Schematic Editing toolbar, click the **Swap Ref. Des.** button.
2. Select the first part.
3. Select the second part with which to swap reference designators. The software changes or swaps the reference designators on the selected parts.

Alternatively, to use the popup menu to swap reference designators, you can either select one part at a time or select both parts at once.

Swapping Reference Designators by Selecting One Part at a Time

You can swap reference designators by selecting one part at a time.

Procedure

1. Select the first component.
2. Right-click and click the **Swap Ref. Des.** popup menu item.
3. Select another component to complete the swap.

Swapping Reference Designators by Selecting Both Parts First

You can swap reference designators by selecting both parts first, and then specifying the command.

Procedure

1. Using Ctrl+click, select the component reference designators you want to swap.
2. Right-click and click the **Swap Ref. Des.** popup menu item to complete the swap.

Saving Part Types to a Library

You can save one part type, multiple part types, or all part types in a schematic to the library.

[Saving One or More Part Types in the Schematic](#)

[Saving All of the Part Types in the Schematic](#)

[Saving Off-Page to Library](#)

Saving One or More Part Types in the Schematic

You can save one or more part types to a library. Once saved to the library, these parts are available for use in the current design as well as future design sessions.

Procedure

1. Select one or more parts in the schematic.
2. Right-click and click the **Save to Library** popup menu item.
The [Save Part Types to Library Dialog Box](#) appears.
3. Select the library where you want to save the part types.
4. Click **OK**.

Saving All of the Part Types in the Schematic

You can save all of the part types in the schematic to a library. This enables you to quickly save all of the part types and make them available for the current and future design sessions.

Procedure

1. Right-click and click the **Select Parts** popup menu item.
2. Right-click again and click the **Select All on Schematic** popup menu item.
3. Right-click and click the **Save to Library** popup menu item.
The [Save Part Types to Library Dialog Box](#) appears.
4. Select the library where you want to save the part types.

5. Click **OK**.



Tip

When saving part types, observe the following:

- If a part type or logic decal you are saving already exists in the library, a message box appears. Click **Yes** to overwrite the part type or logic decal in the library. Click **No** to save only those part types that do not already exist in the library. Click **Cancel** to return to the Save Parts to Library dialog box without saving the part types to the library.
 - When parts of the same type have attributes with the same values, the attribute names and values are saved in the library with the part type. When parts of the same type have different values for the same attribute name, only the attribute name with a blank value is saved to the library.
-

Saving Off-Page to Library

Use Save Off-page to Library to update the off-page, ground, or power symbols in the library with the current version(s) in the schematic.

Procedure

1. Click the **Tools > Save Off-page to Library** menu item.
2. Select the symbol to update.
3. Click **OK**. The selected item is opened in the Part Editor.
4. Modify the symbols as required.
5. Click the **File > Save** menu item.
6. Follow the prompts to replace the versions currently in the library and update the symbols in the schematic.

Related Topics

[Special Schematic Symbols](#)

The Update From Library Function

In SailWind Logic, when you add parts from the library to a schematic, a copy of the library content for the parts is written into the schematic. Over time, however, as updates are made to the library, the parts in the schematic can become out of sync with newer versions in the library. Use Update From Library to synchronize the parts in your design with the latest versions from the library.

Use Update from Library (UFL) to:

- Create a report of any part differences between the current schematic and the library.
- Update the schematic to reflect the current state of the library.

You can compare or update the following categories of items:

- Part types
- CAE decals
- Pin decals
- Ground symbols
- Power symbols
- Off-page symbols

You can compare or update:

- All of any or all of these categories.
- Selected part types, CAE decals, and pin decals. You cannot compare/update selected ground, power, or off-page symbols.



CAUTION:

Since the only link between library and schematic content for an item is the item's name, updating can have unforeseen and unintended consequences. When part types or gates are replaced globally in a schematic, they are replaced without regard to any local editing of the item; edited instances of parts or gates are replaced with the common instance from the library. The undesirable consequences of such changes could range from minor, like a repositioned label, to catastrophic, like deleted pins or disconnected nets.

To prevent such unintended consequences, the update function always prompts you before updating any item in the schematic that has a newer timestamp than its counterpart in the library. To prepare yourself to respond to these prompts, before updating you should:

1. Generate a comparison report (using the Generate comparison report button in the Update From Library dialog box).
2. In the comparison report, locate the status messages identifying items whose timestamps are newer than in the library.
3. Determine and write down what your response will be to each prompt when it occurs during the update.

You may also want to update a copy of the schematic, and check/test the updated copy for unwanted changes or behavior.

[Undoing an Update](#)

[Updating a Schematic From the Library](#)

[Updating Selected Part Types From the Library](#)

[Updating Selected CAE Decals From the Library](#)

[Updating Selected Pin Decals From the Library](#)

[The Compare/Update Process](#)

[How to Read the Update Report](#)

Undoing an Update

The update process creates an undo checkpoint before it begins the update. If after the update you determine that you want to reinstate the design to its previous state you can revert to the previous checkpoint.

You can undo any update (whether completed, canceled, or failed) to the last-saved checkpoint using the **Edit > Undo** menu command.

Updating a Schematic From the Library

Information in your library can be subject to changes and updates. To ensure that the content in your design is in agreement with the latest updated library content, you can update a schematic with new versions of parts from the library.

You can update all items in any or all of these categories:

- Part types
- CAE decals
- Pin decals
- Ground symbols
- Power symbols
- Off-page symbols

Procedure

1. Click the **Tools > Options** menu item, then click the Design category.
2. In the Options area, select or clear the “Allow overwriting of attribute values in design with blank values from library” check box as appropriate; click **OK** to close the Options dialog box.
3. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
4. Set the library list order that Update from Library will use when searching the libraries for items matching the items in the design. (See [General Compare/Update Rules](#).) Click **OK** then close the Library Manager dialog box.
5. Click the **Tools > Update from Library** menu item.
6. Modify the settings in the [Update From Library Dialog Box](#) as appropriate.
7. Click **OK**.



Tip

When the comparison or update operation is completed, the update report opens and a link to the report file displays in the Output Window.

Related Topics

[The Compare/Update Process](#)

[How to Read the Update Report](#)

[Update From Library Dialog Box](#)

[Updating Selected Part Types From the Library](#)

[Updating Selected CAE Decals From the Library](#)

[Updating Selected Pin Decals From the Library](#)

Updating Selected Part Types From the Library

As you make changes and updates to your library content, you may also need to selectively update parts in your design. You can select one or more parts and then update the part types of *all* parts having the same part type(s) to synchronize with the latest version in the library.

Restrictions and Limitations

All parts having the same part type as a selected part are updated, but attributes are updated only for the parts actually selected.

Procedure

1. Click the **Tools > Options** menu item, then click the **Design** category.
2. In the Options area, select or clear the “Allow overwriting of attribute values in design with blank values from library” check box as appropriate and click **OK**.

3. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
 4. Set the library list order that Update from Library will use when searching the libraries for items matching the items in the design. (See [General Compare/Update Rules](#).) Click **OK** then close the Library Manager dialog box.
 5. In the Schematic Editor, with nothing selected, right-click and click the **Select Parts** (or Select Gates) popup menu item.
 6. Select the parts whose part types you want to compare/update.
-



Tip

If you want to update all but a few part types, select all the parts/gates, then in Step 8, clear the check boxes of the part types you do not want to update.

7. Right-click and click the **Update > Part Type** popup menu item.
8. Modify the settings in the [Update Selected Part Type From Library Dialog Box](#) as appropriate.
9. Click **OK**.

Results

When the comparison or update operation completes, the update report opens and a link to the report file displays in the Output Window.

Related Topics

[The Compare/Update Process](#)

[How to Read the Update Report](#)

[Updating a Schematic From the Library](#)

[Updating Selected CAE Decals From the Library](#)

[Updating Selected Pin Decals From the Library](#)

Updating Selected CAE Decals From the Library

You can select one or more parts, and then update the CAE decals of all parts that use the same decal(s).

Restrictions and Limitations

This procedure updates the pin decal assignments in the CAE decals, but does not update the pin decals themselves. Use one of the procedures in [Updating Selected Pin Decals From the Library](#) to update the pin decals.

As a corollary, if, for instance, you update CAE decal X to use PINB instead of PINA, it uses the version of PINB geometry currently in the schematic. If the schematic does not have a PINB, it installs from the library.

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button
 2. Set the library list order that Update from Library uses when searching the libraries for items matching the items in the design. (See [General Compare/Update Rules](#).) Click **OK** then close the Library Manager.
 3. In the Schematic Editor, with nothing selected, right-click and click the **Select Parts** (or **Select Gates**) menu item.
 4. Select the parts/gates whose CAE decals you want to compare/update.
-



Tip

If you want to update all but a few CAE decals, select all the parts/gates, then in Step 6 clear the check boxes of the decals you do not want to update.

5. Right-click and click the **Update > CAE Decal** menu item.
6. Modify the settings in the [Update Selected CAE Decals From Library Dialog Box](#) as appropriate.
7. Click **OK**.

Results

When the comparison or update operation completes, the update report opens and a link to the report file displays in the Output Window.

Related Topics

- [The Compare/Update Process](#)
- [How to Read the Update Report](#)
- [Update Selected CAE Decals From Library Dialog Box](#)
- [Updating a Schematic From the Library](#)
- [Updating Selected Part Types From the Library](#)
- [Updating Selected Pin Decals From the Library](#)

Updating Selected Pin Decals From the Library

You can update selected pin decals from the library using the Update from Library dialog box or directly from within the schematic editor.

- [Updating Selected Pin Decals Using the Update From Library Dialog Box](#)
- [Updating Selected Pin Decals Using the Schematic Editor](#)

Updating Selected Pin Decals Using the Update From Library Dialog Box

As you make changes and updates to the content in your library, you may need to synchronize pin decals in your design with the updated library content. You can use the Update From Library dialog box to update all instances of one or more selected pin decal(s).

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button
2. Set the library list order that Update from Library will use when searching the libraries for items matching the items in the design. (See [General Compare/Update Rules](#).) Click **OK** then close the Library Manager dialog box.
3. Click the **Tools > Update from Library** menu item.
4. Select the Pin Decals check box, and select the decals you want to compare/update.
5. Click **OK**.

Results

When the comparison or update operation completes, the update report opens and a link to the report file displays in the Output Window.

Updating Selected Pin Decals Using the Schematic Editor

As you make changes to the content in your library, you may need to refresh your design content to reflect those updates. You can use the Update Selected Pin Decals from Library dialog box to select one or more pins, and update all instances of the pin decal(s) used by the selected pin(s).

Procedure

1. Click the **File > Library** menu item, then click the **Manage Lib. List** button.
2. Set the library list order that Update from Library will use when searching the libraries for items matching the items in the design. (See [General Compare/Update Rules](#).) Click **OK** then close the Library Manager dialog box.
3. In the Schematic Editor, with nothing selected, right-click and click the **Select Pins** popup menu item.
4. Select the pins you want to compare/update.



Tip

If you want to update all but a few pin decals, select all the parts/gates, then in Step 6 clear the check boxes of the decals you do not want to update.

5. Right-click and click the **Update > Pin Decal** menu item.
6. Modify the settings in the [Update Selected Pin Decals From Library Dialog Box](#) as appropriate.
7. Click **OK**.

Results

When the comparison or update operation completes, the update report opens and a link to the report file displays in the Output Window.

Related Topics

- [The Compare/Update Process](#)
- [How to Read the Update Report](#)
- [Update Selected Pin Decals From Library Dialog Box](#)
- [Updating a Schematic From the Library](#)
- [Updating Selected Part Types From the Library](#)
- [Updating Selected CAE Decals From the Library](#)

The Compare/Update Process

The update process begins with a comparison of the schematic content with the library content. When schematic content is found to be different from the library content, the schematic content is updated according to the options set in the Update From Library dialog box, and the results documented in the report file. The compare process is the same, except that the schematic is not updated.

General Compare/Update Rules

There are a number of rules that govern the compare/update process.

In comparing and updating, the following general rules apply for all items:

- **Matching items** — For each item in the schematic, all available libraries are searched to find a matching named entry, and the first matching entry found is used. If no match is found, comparison of the item is skipped, and a “not found” entry is made in the report file.
-



Tip

Libraries are searched in the order shown in the Library List dialog box. You can use this dialog box to change the search order.

- **Timestamps** — Each item in the schematic and library has a timestamp. Schematics of PADS versions prior to 2005 SPac2 do not have timestamps. A comparison of the timestamps determines what is compared/updated, as follows:
 - **Timestamp in schematic is older** — If the timestamp in the schematic is older than the timestamp in the library, the item in the schematic is always compared/updated.
 - **Timestamps are the same** — If the timestamps in the library and schematic are the same, comparison/update of the item depends on the setting of the Include items with identical time stamp check box in the [Update From Library Dialog Box](#). If this check box is cleared, no comparison/update is made; if the check box is selected, all items are compared/updated regardless of timestamps.
 - **Timestamp in schematic is newer** — If the timestamp in the schematic is newer than the timestamp in the library:
 - When you are comparing, a status message is written to the report.
 - When you are updating, a status message is written to the update report, and you are prompted to choose whether to update the schematic or not.
- **What is updated** — When an item is updated, the content from the library replaces the entire content of the item in the schematic.

Skipped Updates

There are 3 situations in which the update of an item is skipped.

These situations are:

- When an item in the design is newer than the equivalent item in the library, and you reply No to the “<item> in the schematic has a more recent time stamp than in the library. OK to overwrite the version in the schematic?” prompt.
- When a newer item in the library has fewer pins than the equivalent older item in the design.
- When the number of GND, POWER or OFF symbols in the library is fewer than the number of available symbols in the design.

Older Schematics

Schematics of PADS versions prior to 2005 SPac2 do not have timestamps. When one of these is opened, the timestamps on all part types, decals, pin decals, and off-page, power & ground symbols are set to zero. To determine if the content of the part types and decals in the schematic is in sync with the library, compare the schematic and library with the Hide identical results check box selected.

What is Compared

Many design objects are examined during the compare process.

The following table lists the content compared and reported for each item selected for comparison or update.

Table 25. Compared and Reported Part Structures

Item	Item Component	What is Compared & Reported
Timestamps		See “ General Compare/Update Rules ” on page 203.
Part Type		Logic Family, Number of gates, Number of connector pins, Number of Signal pins, List of attribute names
	<i>For each gate:</i>	Number of CAE Decals, CAE Decal names, Number of electrical pins on gate, Gate swap class.
	<i>For each gate pin:</i>	Pin number (alphanumeric if defined), Pin name, Pin type, Pin swap class
	<i>For each signal pin:</i>	Pin Number (alphanumeric if defined), Net name
	<i>For each attribute:</i>	<p>Value string</p> <p>Note: There can be attributes in the library that are just placeholders, with blank values that must be set for each individual part in the schematic. Any attribute in the library with a blank value is considered to be such a placeholder, and updates of corresponding attributes in the schematic is controlled by the “Allow overwriting of attribute values in design with blank values from library” check box in the Options Dialog Box, Design Category.</p>
PCB Decal assignment		<p>The part's assigned PCB Decal name is on the list of PCB Decal alternates for the Library part type.</p> <p>If it is not, a warning is written to the report, but the decal assignment is not updated.</p>
PCB Decal pin count		
	<i>For connectors:</i>	Number of pins in the PCB Decal assigned to each connector part is equal to the number of connector pins defined for the part type.
	<i>For normal parts:</i>	<p>Number of pins in the PCB Decal assigned to each normal part is equal to or greater than the highest electrical pin number used in the gates defined for the Library part type.</p> <p>Restriction: Pin counts cannot be updated in the schematic. Pin count differences are flagged as warnings in the update report.</p>
CAE Decals		
	<i>Gate Decal details:</i>	<p>Number of attribute labels, attribute label name, number of text, number of terminals.</p> <p>Restrictions:</p> <ul style="list-style-type: none"> • CAE Decal graphics are not compared. • Details of text items are not compared.
	<i>Terminal details:</i>	<p>Pin decal name assigned to each terminal.</p> <p>Restriction: If terminal pin counts differ, the decal cannot be updated in the schematic separately from the part types.</p>

Table 25. Compared and Reported Part Structures (continued)

Item	Item Component	What is Compared & Reported
Pin Decals		<p>Pin Decal name.</p> <p>Update notes:</p> <ul style="list-style-type: none"> Because default pin name and pin number label locations are “hard-wired” in the CAE decal when it is created, the locations of pin names and numbers in the schematic are unaffected when a Pin decal is updated. Pin decals do provide a default location for net names on the schematic, but the actual used net name locations are stored separately and therefore net name locations are not affected by an update.
Power & Ground Symbols		Symbol name, type, Signal Pin name
Off-Page Symbols		Symbol name, type

How to Read the Update Report

Every time you update a design from the library or generate a comparison report, Update from Library writes the results of the operation to a new UpdateReport.txt file in C:\SailWind Projects. You can use a text editor to open and review the content of this report and verify that the design objects are the intended versions.

The Update Report gives you information about the items you selected for comparison or update, such as:

- Whether the timestamp of an item in the design is older, newer, or the same as the item’s timestamp in the library.
- Whether the content of an item in the design is the same as or different from the item’s content in the library.

The report consists of 4 sections:

- Report header information** — General information about the report itself. Example:

```

1 / *PADS Logic 9.2* UPDATE FROM LIBRARY REPORT
2 / 12:10:48 05/14/10 DEMO.SCH
3 / C:\PADS Projects\UpdateFromLibrary_Logic_report.txt

```

- Report Options** — A list of options selected for this operation in the Update from Library dialog box. Example:

```
11 | =====
12 | REPORT OPTIONS
13 | =====
14 |
15 | This report was run with the following options:
16 | Update design from library
17 | Include items with identical time stamp
18 | Summary and Details
19 | Part types and attributes
20 |     Preserve design attributes not found in library
21 |     Add new attributes not found in design
22 |     Update common attributes
23 | CAE Decals
24 | Pin Decals:
25 |     PIN
26 |     PINB
27 |     PCLK
28 |     PCLKB
29 |     PINIEB
30 |     PINORB
31 |     PINSHORT
32 |     PINVRTS
33 | Off-Page symbols
34 | Ground symbols
35 | Power symbols
```

- **Report Summary** — A tabular listing of compared part types, CAE decals, off-page symbols, ground symbols, and power symbols, showing the comparison results and update status for the package as a whole. Example:

Schematic Parts

How to Read the Update Report

```
37 | -----
38 | REPORT SUMMARY
39 | -----
40 |
41 | Compare status:
42 |   Compared -      137
43 |   Differences -   72
44 |   Warnings -      3
45 |   Errors -         0
46 |   Skipped -        0
47 |
48 | Update status:
49 |   Updated -        137
50 |   Warnings -       3
51 |   Failed -         0
52 |   Skipped -        0
53 |
54 | -----
55 | Compared Part Types
56 | -----
57 |
58 |          Design
59 |  Part Type  Timestamp
60 |  -----  -----
61 |  7404     Older than library
62 |  7402     Older than library
63 |  7474     Older than library
64 |  7432     Older than library
65 |  CON\26P\ED Older than library
66 |  XTAL1    Older than library
                                         Content  Found in Library  Update Status  See Line
                                         Different  ti           UPDATED  307
                                         Different  ti           UPDATED  668
                                         Different  ti           UPDATED  861
                                         Different  ti           UPDATED  996
                                         Different  misc         UPDATED  1145
                                         Different  misc         UPDATED  1652
```

- **Report Details** — Complete comparison and update details for all compared packages.
Example:

```

299 | =====
300 | REPORT DETAILS
301 | =====
302 |
303 | -----
304 | -- Part Types --
305 | -----
306 |
307 | 7404
308 | -----
309 | Part Type Timestamp:          Older than library - CONTENT UPDATED
310 | Library:                   ti
311 | Details:
312 |   Logic Family: TTL          Same as library
313 |   Gate Count: 6              Same as library
314 |   Pin Count:                Same as library
315 |   Connector: 12             Same as library
316 |   Signal: 2                 Same as library
317 |   Gate 1 Details:
318 |     Logic Decal Details:
319 |       Count: 4               Same as library
320 |       1.Name: INV            Same as library
321 |       2.Name: INVL           Same as library
322 |       3.Name: INVDM          Same as library
323 |       4.Name: INVDM1         Same as library
324 |     Gate Swap Class: 1      Same as library
325 |     Gate Pin Count: 2       Same as library
326 |     Gate Pins Details:
327 |       #1.Pin:
328 |         Number: 1            Same as library
329 |         Name: Not assigned  Same as library
330 |         Pin Type: Load      Same as library
331 |         Swap Class: 0        Same as library
332 |       #2.Pin:
333 |         Number: 2            Same as library
334 |         Name: Not assigned  Same as library
335 |         Pin Type: Source    Same as library
336 |         Swap Class: 0        Same as library

```

Table 26 lists and describes the messages returned for compared/updated items in the Update Report.

Table 26. Update Report Messages

Message	Report Section	Refers to:	Means
Cannot get CAE DECAL from Library!!!	Details	CAE decal	CAE decal exists in design but not in library.

Table 26. Update Report Messages (continued)

Message	Report Section	Refers to:	Means
Cannot get GATE from Library!!!	Details	Gate	Gate exists in design but not in library.
Cannot get GATE from Library - Comparison Skipped	Details	Gate	Gate exists in design but not in library.
Cannot get PACKAGE from Library!!!	Details	Package	Package exists in design but not in library.
Cannot get PIN DECAL from Library!!!	Details	Pin decal	Pin decal exists in design but not in library.
CANCELLED	Summary	Item update	Operation was canceled by user.
COMPARE CANCELLED	Summary & Details	Compare operation	Operation was canceled by user.
COMPARE ERROR	Details	Compare operation	No items selected or item(s) unavailable.
COMPARE SKIPPED	Details	Compare operation	Cannot compare because CAE decal terminal count differs in design and library.
COMPARE STOPPED!	Details	Compare operation	Operation was canceled by user.
Compared items match	Summary	Item content	All compared item content is the same in the design and the library. The following items are not compared: <ul style="list-style-type: none">• CAE Decal graphics• Details of CAE Decal text items
CONTENT UPDATED	Details	Item update status	Content in the design updated with content from library.
Decal Count Mismatch	Details	CAE decal	CAE decal terminal count in design differs from count in library.
Different	Summary & Details	Item content	Item content in design differs from content in library.

Table 26. Update Report Messages (continued)

Message	Report Section	Refers to:	Means
Different timestamp	Summary	Pin Decal timestamp	Timestamps in design and library differ.
FAILED	Summary	Item update status	Update has failed due to an unspecified error.
In library: Not in design	Details	Item status	Item in library not found in design.
Newer than library	Summary & Details	Timestamp in design	Timestamp in design newer than in library.
No CAE Decal found!!!	Details	CAE decal	CAE Decals check box selected in dialog box but no CAE decal found in design.
No PIN Decal found or selected!!!	Details	Pin decal	Either: <ul style="list-style-type: none"> • Pin decal selected in dialog box not found in design, or • Pin decals check box selected but no pins selected.
Not assigned	Details	Gate pin	Gate pin assigned in design is not assigned in library.
Not assigned in library part	Details	Compared item	The PCB Decal assigned to the part in the design is not in the set of decals assigned to the part in the library.
Not in design	Summary	Off-page, PWR and GND symbols	The symbol exists in the library, but is not used in the design.
Not in library	Details	Compared item	Item in design not found in library.
Older than library	Summary & Details	Timestamp in design	Timestamp in design older than in library.
Operation canceled by USER!!!	Details	Update or Compare operation	Operation was canceled by user.
PERFORMED UNDO OPERATION	Details	Update or Compare operation	Operation canceled by user.

Table 26. Update Report Messages (continued)

Message	Report Section	Refers to:	Means
Same as library	Summary & Details	Timestamp in design Item content in design	Timestamps in design and library are equal. Item content is the same in design and library.
SKIPPED	Summary	Item update	User has replied "No" to the "<item>" in the schematic has a more recent time stamp than in the library. OK to overwrite the version in the schematic?" prompt.
STOPPED	Summary	Item update	Operation canceled by user.
UPDATE CANCELLED	Details	Item update	Operation was canceled by user.
UPDATE FAILED	Details	Item update	An error occurred in the update process and the update failed.
UPDATE MESSAGE	Details	Item update	Explains cause of an Update failure.
UPDATE SKIPPED	Details	Item update	User has replied "No" to the "<item>" in the schematic has a more recent time stamp than in the library. OK to overwrite the version in the schematic?" prompt.
UPDATE STOPPED!	Details	Item update	Operation canceled by user.
WARNING	Details	PCB Decal	PCB Decal in the library has fewer pins than in the design.

Related Topics

Related Topics

[The Update From Library Function](#)

[The Compare/Update Process](#)

Deleting a Part

When deleting a part that is selected in object mode, you can choose to delete the part only, or the part and its connections.

Procedure

1. Select a part.
2. If you want to keep connections attached to the part, right-click and click the **Preserve Connections** popup menu item to enable it. The **Preserve Connections** option is enabled by default.



Tip

Preserve Connections is enabled when a check mark appears next to it in the menu.

3. Right-click and click the **Delete** popup menu item.

The software leaves the connections at their original locations.

Deleting a Part and Its Connections

If a design no longer requires a part, and you are not going to replace the part with a similar one (and therefore you do not need to preserve its connections), you can delete the part and its connections.

Procedure

1. Select a part.
2. If you want to remove connections attached to the part, right-click and click the **Preserve Connections** popup menu item to disable it. The **Preserve Connections** option is enabled by default.



Tip

Preserve Connections is disabled when a check mark does not appear next to it in the menu.

3. Right-click and click the **Delete** popup menu item.

The software deletes the part and its connections.

Related Topics

[Object Select Mode \(Select Object First\)](#)

[Adding Off-Page References](#)

[Working With Floating Connections](#)

Cut, Copy, and Paste

In a manner similar to other standard applications, you can cut, copy and paste objects in your design.

- Use the **Edit** menu items or the Ctrl+X, Ctrl+C, or Ctrl+P shortcuts.
- Connections that extend outside a selection are assigned off-page symbols at the break point.
- SailWind Logic only maintains one cut or copy in the paste buffer.
- You can paste into a new design.

Copy as Bitmap

The Copy as Bitmap command lets you define a rectangular area to copy graphics information to the paste buffer as a bitmap image. Once copied, you can, for example switch to Microsoft Word and use the **Home > Paste** menu item to insert the bitmap into a document.

Procedure

1. Click the **Edit > Copy as Bitmap** menu item.
2. Drag the cursor over the area to copy.
All items in the rectangle including the background, dot grid, color, etc., are copied.
3. Open the application in which to place the bitmap and use the **Paste** command to place the image.

Creation of Groups

A group is a collection of objects on the schematic. By default, connections that extend outside the group are clipped so they stop at the boundary used to define the group. Once you select a group, you can move it, copy it to the paste buffer, delete it, duplicate it, or save it to a file for use in another design.



Tip

A group differs from a combination, which consists only of 2D lines or 2D lines and text.

See also [Combining 2D Lines and Text](#).

[Creating a Group](#)

[Setting the Origin of Groups](#)

Creating a Group

To perform an operation on multiple objects simultaneously, create a group and then move it, copy it to the paste buffer, delete it, duplicate it, or save it to a file for use in another design.

Restrictions and Limitations

- Certain group commands are available only if the group consists of mixed items. For example, the “Preserve by Boundary” popup menu item is unavailable if the group consists of components only.

Procedure

1. Set the selection filter to the objects you want to select.

See also [Using the Selection Filter](#).

2. Area select the objects in the group.

As an alternative, you can use Ctrl+click to select all objects to include in the group.

The software highlights the included objects.

3. To preserve connections that extend outside the group boundary, right-click and click the **Preserve by Boundary** popup menu item.

4. To preserve single pin nets that remain after you delete a part or bus, right-click and click the **Preserve Connections** popup menu item.

5. As needed, Ctrl+click objects in the group to remove them, or Ctrl+click objects outside the group to add them to the group.

The software highlights the included objects.

Setting the Origin of Groups

When you create a group, the software creates an origin automatically. The software uses this origin when moving or placing groups. You may want to change the origin when it is optimal to have the origin at a specific location.

Restrictions and Limitations

You must enable the **Preserve by Boundary** option and the boundary must be visible to set the origin. When the boundary is not visible, the group rotates around the center of the selection.

Procedure

1. Select the group.
2. Right-click and click the **Set Origin** popup menu item.
A bullseye attaches to the pointer.
3. Move the cursor to the new origin and click to set it.

Related Topics

[Management of Groups](#)

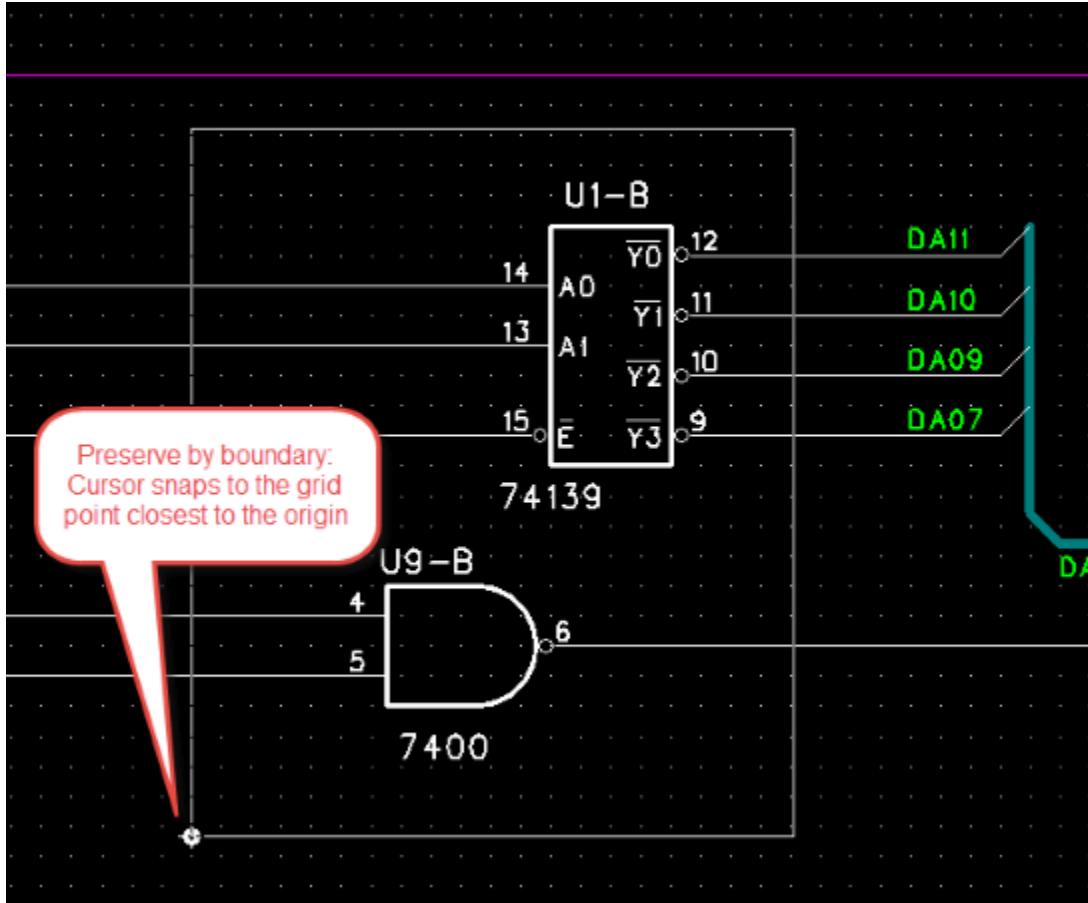
Group Anchor Points

When duplicating or moving a group, SailWind Logic places the anchor point of the group at a pre-defined location.

Keep the following anchor point criteria in mind:

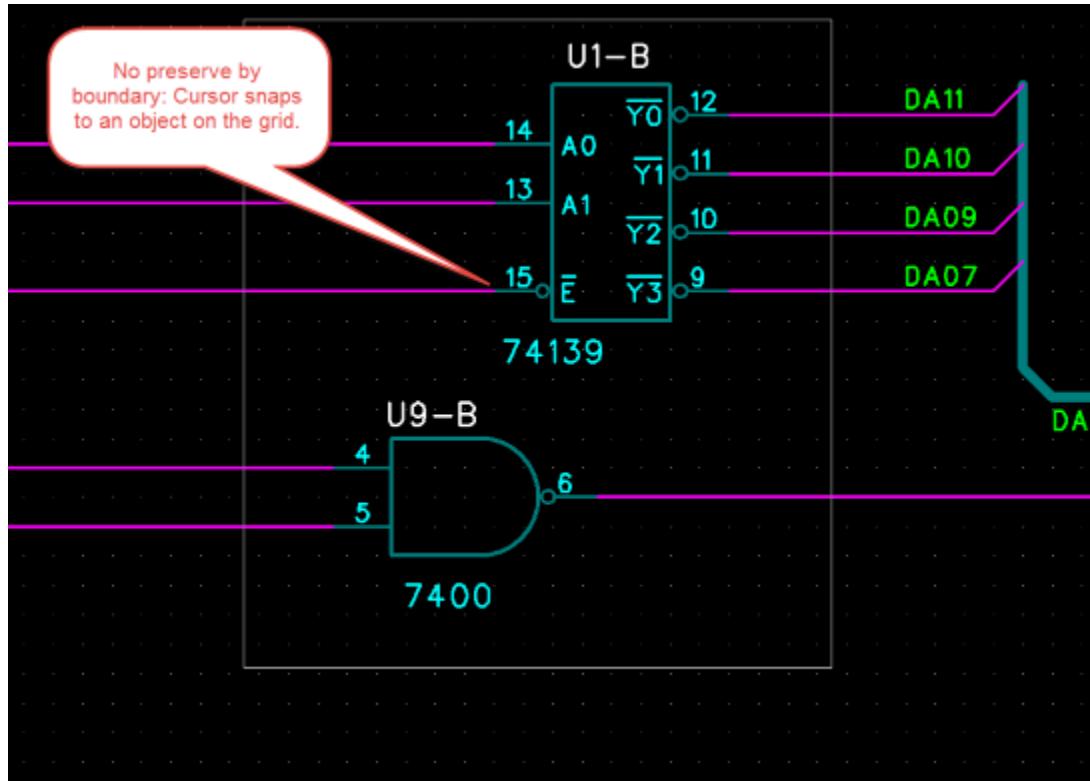
- If you have enabled the “Preserve by boundary” feature, SailWind Logic anchors to a point on the design grid closest to the group’s origin.

Figure 3. Anchor Point with “Preserve by Boundary”



- If you have not enabled the “Preserve by boundary” feature, SailWind Logic anchors to either a group object that lies on the design grid or—if no part of the group is on the design grid—it anchors to a group object that is closest to the origin.

Figure 4. Anchor Point with no “Preserve by Boundary”



Management of Groups

Using a group, you can perform an operation on its contents as a single entity. Once you create a group, you can save it, delete it, move it, or reuse it.

[Duplicating an Existing Group](#)

[Moving Groups](#)

[Deleting Groups](#)

[Saving Groups](#)

[Pasting Groups From a File](#)

[Automatic Connection When Pasting](#)

[Rotate or Mirror a Group](#)

Duplicating an Existing Group

You can duplicate all of the items in a group that already exist on a schematic. When you duplicate a group of objects, reference designator names are assigned automatically.

Procedure

1. Select a group, right-click and click the **Duplicate** popup menu item.
-



Note:

The anchor point of the duplicate group depends on certain conditions. For more information, see [Group Anchor Points](#).

2. Move the cursor to the location for the duplicate, and click to place it.
 3. Repeat the above step to create additional copies.
-



Tip

You can set SailWind Logic to display only the group's outline during copy operations.

See also [Creation of Groups](#).

Moving Groups

Using the Move command, you can move a group to any location on the sheet.

Procedure

1. Select a group, right-click and click the **Move** popup menu item.
-



Note:

As you move the group, its anchor point depends on certain conditions. For more information, see [Group Anchor Points](#).

2. Move the cursor to the new group location and click to place it.
-



Tip

You can set SailWind Logic to display only the group's outline during move operations.

See also [Creation of Groups](#).

Deleting Groups

When deleting a group, you can choose to delete or retain the connections that are outside the group boundary.

Procedure

1. Select a group, right-click and click the **Delete** popup menu item.
 2. To preserve connections that extend outside the group boundary, right-click and click the **Preserve by Boundary** popup menu item.
-



Tip

When preserving connections, note the following:

- **Preserve by Boundary** must be checked to use this capability, otherwise, the feature is disabled.
 - When you preserve connections outside the group boundary, the connections are automatically assigned off-page symbols upon deletion. Unnamed connections are uniquely named with a “\$\$\$” prefix.
-

3. To preserve single pin nets that remain after you delete a part or bus, right-click and click the **Preserve Connections** popup menu option.

See also [Detach](#), [Working With Floating Connections](#).

Saving Groups

You can save the contents of a group for use on other schematic sheets or a different design. Saved groups are stored in the `\SailWind Projects` folder with a `.grp` extension.

Restrictions and Limitations

- The group must consist of mixed items. The “Save to File” command is not available if you select only components or other objects.

Procedure

1. Select a group, right-click and click the **Save to File** popup menu item.
2. In the File Name area, type a name.
3. Click **Save**.

Pasting Groups From a File

You can save a group to a file so that you can use it again in the current or future designs. Use Paste from File to insert a group that was previously saved to a file.

When you paste a group into a sheet:

- Named nets retain their name, and unnamed nets are assigned default signal names.
- All parts are named with the next available reference designator names.



Tip

To control renaming of reference designators, use the “Preserve Reference Designators on Paste option” check box in the Options dialog box, Design category.

Procedure

1. Click the **Edit > Paste from File** menu item.
2. In the Load Group File dialog box, select the group to paste.
3. Click **Open**. The group attaches to the cursor.
4. Move the pointer to the location for the group, and click to place it.



Tip

If the design contains parts with the same reference designation as those in the group, a report generates.

See also [Preserving Reference Designators](#).

Automatic Connection When Pasting

When pasting a group from a file into your design, unconnected pins, unterminated wires, or bus segments in a group can connect automatically.

- Unconnected pins connect automatically with unterminated wire ends or unconnected pins.
- Unterminated wire ends connect automatically to bus segments only if the unterminated wire is a named subnet of the bus.

Rotate or Mirror a Group

You can rotate or mirror a group while the group is selected. You can also rotate or mirror a group during a move.

Procedure

1. Select a group and right-click.
2. Click the **Rotate 90**, **X Mirror**, or **Y Mirror** popup menu item.



Restriction:

The group must consist of mixed items. The commands are not available if only components are selected.

Related Topics

[Creation of Groups](#)

Attributes Overview

Attributes enable you to associate information with a schematic symbol. Attributes are made of two parts: an attribute name, and its corresponding value. For example, you can create an attribute named Part Description and assign its value as Hex Inverter.

You can create an attribute that is assigned to every part in the library, every part in the open design, a part type, or to only one part in the design.

- To add an attribute to all parts in a library, create the attribute in the Manage Library Attributes dialog box.

See also [Managing Library Attributes](#).

- To add an attribute to every part in the open design, create the attribute in the Managing Schematic Attributes dialog box.

See also [Manage Attributes in a Schematic](#).

- To add an attribute to a specific part, create the attribute using the **Attributes** tab while editing the part.

See also [Managing Attributes](#).

- To add an attribute to a part on the schematic without updating the part type, use the Part Attributes dialog box.

See also [Modifying Part Attributes](#) and [Resistor Values Defined on Parts](#).

- To display attributes in the schematic after creating them, use the visibility controls.

See also [Controlling Text Visibility for a Part](#).

- To create placeholders in CAE Decals so that attributes are placed in a predefined locations, use attribute labels.

See also [Attribute Labels](#).

Manage Attributes in a Schematic

Use the Manage Schematic Attributes dialog box to manage attributes at the schematic level. You can create a new attribute and automatically assign it to every part in your design. You can also rename an attribute in, or delete an attribute from, the schematic. All parts are automatically updated.

See also [Attributes Overview](#).

[Creating Attributes](#)
[Renaming Attributes](#)
[Deleting Attributes](#)

Creating Attributes

You can create a new attribute and automatically assign it to every part in a design.

Procedure

1. Click the **Edit > Attribute Manager** menu item.
2. In the Manage Schematic Attributes dialog box, click **Add Attr.**
3. In the Add New Attribute dialog box do one of the following:
 - Type a name for the attribute in the Attribute Name text box.
 - Click **Browse Lib Attr** to select an attribute already defined in the library. Select an attribute name from this dialog box and then click **OK**.
4. Type a value for the attribute in the Attribute Value text box.
5. Click **OK** to close the Add New Attribute dialog box.
6. Click **Close** to close the Manage Schematic Attributes dialog box.

Renaming Attributes

You can rename an existing attribute and apply the change to each part in the design.

Procedure

1. Click the **Edit > Attribute Manager** menu item.
2. In the Manage Schematic Attributes dialog box, select one or more attributes in the Attributes in Schematic list.
3. Click **Add**. The attributes are added to the Attributes Selected for Rename list.
4. Select the attribute in the New Name column and click **Edit New Name**.
5. In the Attribute Name text box, do one of the following:

- Type a name for the attribute
 - Click **Browse Lib Attr** to select an attribute already defined in the library. Select an attribute name from this dialog box, and then click **OK**.
6. Click **Rename Attrs** to rename the attribute and remove it from the Attributes Selected for Rename list box.
7. Click **Close**.



Tip

Click **Remove** or **Remove All** to place the selected attributes back into the Attributes in Schematic list and keep the current name.

Deleting Attributes

You can delete an attribute from all parts in the design:

Procedure

1. Click the **Edit > Attribute Manager** menu item.
2. Select one or more attributes in the Attributes in Schematic list.
3. Click **Delete Attrs** then click **Yes** at the prompt.
4. Click **Close**.

Resistor Values Defined on Parts

Discrete parts such as resistors and capacitors are created with Value and Tolerance attributes. The value for these attributes is not defined in the part symbol, but is set when the part is added to the drawing.

Use the [Part Attributes Dialog Box](#) on page 283 to modify these values.

Chapter 11

Sheets

You can use sheets to logically partition your design. You can also add custom borders and fields to your sheets.

[Editing Sheets](#)

[Creating a Custom Sheet Border](#)

[Adding a Field](#)

[Changing a Text String Into a Field](#)

[Managing Fields](#)

Editing Sheets

Use the Sheets dialog box to edit the sheet set of the current schematic in the work area. Using Sheets enables you to add and delete sheets from the set and to modify sheet names and the numeric order of the set. You can create up to 1024 sheets.

Procedure

1. Click the **Setup > Sheets** menu item.
 2. Click a sheet in the Numbered Sheets list box, and then click **View** to view it in the work area.
 3. Click **Add** to add a new sheet to the list.
The new sheet displays in the work area.
 4. Click the new sheet in the Numbered Sheets list box, and then click **Rename** to rename it.
-



Tip

Spaces are not valid characters in a sheet name.

5. If desired, perform any of the following additional operations from the Sheets dialog box:
 - Click **Delete** to remove a sheet from the list.
 - Click **Up** to change the numbering and position of the sheet. The sheet value changes as n-1.
 - Click **Down** to change the numbering and position of the sheet. The sheet value changes as n +1.

Creating a Custom Sheet Border

You can create a custom sheet border with a title block using specialized text strings called fields. When you add fields to the title block of a sheet border, these fields are automatically propagated to each new sheet.

Procedure

1. Draft the 2D line border. (See [Creating 2D Line Items.](#))
2. Add fields to the 2D line object. (See [Adding a Field.](#))
3. Save the object to the lines library. (See [Adding Drafting Items to a Library.](#))

Adding a Field

Fields are used as placeholders for data that might change in a document. You can add fields to a schematic or any 2D line object. If you selected a 2D line object, the added field is automatically combined with it upon placement.

There are two types of fields: system and user. You cannot modify system fields; the value is derived from the system. User fields are custom fields you create. There are no limitations to the name or value strings of user fields. User fields are available only in the schematic in which they are created.

Procedure

1. Select nothing or a 2D line object, right-click and click the **Add Field** popup menu item.
2. In the Name list, select a system field or type a custom field name.
3. In the Value box, type the value you want displayed.

Note that the Value box is unavailable for system fields since the value is derived from your system.
4. To place the field at a precise X,Y coordinate location, type the value in the X and Y boxes.



Tip

If this is blank when you click **OK**, the field attaches to the pointer until you click to indicate the location.

-
5. In the Rotation list, select the degree of rotation you want.
-



Tip

Rotation can be 0 or 90 degrees.

6. In the Size box, type the font size you want.

Stroke font sizes must be between 10 and 1000 mils; system font sizes must be between 1 and 72 points.

7. For stroke font, in the Line Width box, type the line width.

8. For system fonts, select the font you want to use in the Font list.

You can also click a system font style you want applied: **B** for Bold, **I** for Italic, or **U** for Underlined.

9. To set the Justification, click the Horizontal and Vertical options you want.

10. Click **OK**.

Related Topics

[Combining 2D Lines and Text](#)

Changing a Text String Into a Field

You can use the Make Field command to change an existing text string into a field.

Procedure

1. Select a text string, right-click and click **Make Field**.
 2. In the Name list, type a field name or select a system field.
 3. To place the field at a precise X,Y coordinate location, type the value in the X and Y boxes.
-



Tip

If this is blank when you click **OK**, the field attaches to the pointer until you click to indicate the location.

4. In the Rotation list, select the degree of rotation you want.
-



Tip

Rotation can be 0 or 90 degrees.

5. In the Size box, type the font size you want.

Stroke font sizes must be between 10 and 1000 mils; system font sizes must be between 1 and 72 points.

6. For stroke font, in the Line Width box, type the line width.

7. For system fonts, select the font you want to use in the Font list.

You can also click a system font style you want applied: **B** for Bold, **I** for Italic, or **U** for Underlined.

8. To set the Justification, click the Horizontal and Vertical options you want.

9. Click **OK**.

Managing Fields

Use the Fields dialog box to manage multiple fields. You can manage the fields in the entire schematic or in a 2D line object.

[Managing All Fields in the Schematic](#)

[Managing the Fields in a 2D Line Item](#)

Managing All Fields in the Schematic

If you selected nothing, the software lists all of the fields used in your schematic. The System fields display in the System area, and the fields you defined display in the User area. You can manage only the User fields.

Procedure

1. Select nothing or a 2D line object, right-click and click the **Fields** object.
2. In the User area, click **Add**.
3. Type a name in the Name column.
4. Type a value in the Value column.
5. To delete a field from the schematic, select the field name and click **Delete**.



Restriction:

You cannot delete a system field from the schematic.

6. To edit the name or value of a field, select the Name or Value you want to edit and click **Edit**.

Managing the Fields in a 2D Line Item

If you selected a 2D line object, the software lists all of the fields associated with the object. The System fields display in the System area, and the fields you defined display in the User area. You can delete System fields from the 2D line, but they are still available in the schematic.

Procedure

1. Select a 2D line item then right-click and click the **Fields** popup menu item. This opens the Fields dialog box.
2. To delete a system field from the 2D line item, in the System area, select the field name and click **Delete**.

The system field is still available in the schematic.
3. To delete a user field from the 2D line item, in the User area, select the field name and click **Delete**.

The field is still available in the schematic.

4. To edit the value of a user field, select the Value you want to edit and click **Edit**.
-



Restriction:

When editing a user field, note the following:

- You cannot edit the name of the user field.
 - You cannot add a field for later use in a 2D line.
-

Chapter 12

Non-Electrical Objects

A full complement of editing commands are available for working with non-electrical objects. Use the drafting commands to create lines, arcs, text, and all other items not generally associated with connectivity.

- [Creating 2D Line Items](#)
- [Adding Text](#)
- [Moving Text](#)
- [Adding Circles](#)
- [Adding Polygons or Paths](#)
- [Adding Rectangles](#)
- [Modifying 2D Line Items](#)
- [Pulling Arcs](#)
- [Combining 2D Lines and Text](#)
- [Uncombining 2D Lines and Text](#)
- [Exploding Combinations](#)
- [Adding Drafting Items to a Library](#)
- [Adding Drafting Items From a Library](#)
- [Modifying Objects in a 2D Lines Library](#)

Creating 2D Line Items

To add drafting objects such as polygons, circles, rectangles, paths, as well as, arcs to polygons or paths, click the **Create 2D Line** button on the Schematic Editing toolbar. This changes the default popup menu to a specific popup menu for creating 2D line items.

Procedure

1. On the Schematic Editing toolbar, click the **Create 2D Line** button.
2. Right-click and set one or more of the following values before you add the drafting object:

Table 27. Creating 2D Line Items - Options

Option	Description
(Shape)	Select to add a Polygon, Circle, Rectangle, or Path.
Width	Specify a width value to override the default.
Orthogonal	Adds segments in 90-degree increments.
Diagonal	Adds segments in 45-degree increments.
Any Angle	Adds segments at any angle.

Adding Text

Free text is text not belonging to another object. All alpha, numeric, and special characters on the keyboard are valid. The maximum text string length is 72 characters including spaces.

To add multiple lines of text to a design, see [Embedding a Text Document](#).

Procedure

1. On the Schematic Editing Toolbar, click the **Create Text** button.

2. Type the text string you want.

To add a text entry with a bar over the characters, precede the text with the \ character. To place a bar over only a portion of text, enclose that section with the \ character.

For example, typing \READ\WRITE places a bar over READ.

3. To place the field at a precise X,Y coordinate location, type the value in the X and Y boxes.
-



Tip

If this is blank when you click **OK**, the field attaches to the pointer until you click to indicate the location.

4. In the Rotation box, select the degree of rotation from the Rotation list.

Rotation can be 0 or 90 degrees.

5. For stroke font, in the Line Width box, type the line width.

6. In the Size box, type the size (in mils for stroke font, in points for system fonts).
-



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

7. For system fonts, select the font you want to use.

You can also click a font style: **B** for bold, **I** for Italic, or **U** for Underlined.

8. In the Justification area, set the horizontal and vertical justification of the text.

9. If you combined the selected text string with a drafting object, right-click the text string, click **Properties**, and then select **Parent** to display the Drafting Properties dialog box to modify the drafting object.

10. Click **OK**.

Related Topics

[Combining 2D Lines and Text](#)

Moving Text

When you add a part to the schematic, all of the text associated with the symbol (reference designator, part name, pin number, attributes, etc.) appears in a predefined location. You can move the text to make room for other parts or connections or to make the schematic more readable.

Restrictions and Limitations

- You can move a pin number a maximum of one inch from its pin terminal.
- If you select and move a visible net name, with snap to grid enabled, the new net name snaps to the Labels and Text grid.

To set the labels and text grid, use the General page in the Options dialog box. See “[Options Dialog Box, General Category](#)” on page 595.

Procedure

1. On the Schematic Editing toolbar, click the **Move** mode button.
2. Select the text string. It attaches to the cursor.
3. Move the cursor and indicate a new position for the text.

While moving text, you can modify the string using commands in the popup menu. The commands listed in the popup menu are based on the type of text being moved.

Adding Circles

You can use the 2D line drafting command to add circle drafting objects to your design.

Procedure

1. On the Schematic Editing toolbar, click **Create 2D Line** then right-click and click the **Circle** popup menu item.
2. Indicate the center point of the circle.
A circle outline follows the cursor movement.
3. Indicate the radius length to end the circle addition.

Adding Polygons or Paths

You can use the 2D drafting command to add polygons or paths (with or without arcs) to your design.

Procedure

1. On the Schematic Editing toolbar, click the **Create 2D Line** button then right-click and click the **Polygon or Path** popup menu item.
2. Indicate the start point. A line follows the cursor movement.

3. Move the cursor to the next corner and click again.
4. Continue adding corners in the same manner.
 - a. If you want to add an arc rather than a straight segment, right-click and click **Add Arc**.
 - b. To remove the last corner, press the backspace key or right-click and click the **Del Corner** popup menu item.
5. Double-click to end the addition.

Polygons automatically close; paths terminate at the last-defined corner.

Adding Rectangles

You can use the 2D drafting command to add rectangle drafting objects to your design.

Procedure

1. On the Schematic Editing toolbar, click the **Create 2D Line** button, right-click and click the **Rectangle** popup menu item.
2. Indicate one corner of the rectangle.
A box outline follows the cursor movement.
3. Indicate the diagonally opposite corner at which to end the rectangle.

Modifying 2D Line Items

You can use the **Modify 2D Line** button to change drafting objects. This changes the default popup menu to a specific popup menu for modifying 2D line items.

Procedure

1. On the Schematic Editing toolbar, click the **Modify 2D Line** button.
2. Select the line segment or corner to modify.

The object is placed in Move mode.

Use the popup menu to enable other 2D line modifications. If you select a segment, the following modification functions are available:

Table 28. Modify 2D Line Item Options

Option	Description
Pull Arc	Converts a segment or corner into an arc.
Split	Divides a segment into two segments.
Del Segment	Deletes the line segment at the pick point.
Width	Changes the line width of the item.

Option	Description
Filled	Fills a closed polygon.
Solid Style and Dotted Style	Changes the line style to a solid or a dashed line. You cannot specify dotted lines for circles or arcs.
Orthogonal	Moves are made in 90-degree increments.
Diagonal	Moves are made in 45-degree increments.
Any Angle	Moves are made at any angle.

If you choose a corner, the popup menu uses most of the options as selected segments, plus the following:

Table 29. Modify 2D Line Item - Corner Option

Option	Description
Del Corner	Deletes the corner. A line segment is created between the corner's original endpoints.

- Reposition the cursor and indicate a new location for the item.

Pulling Arcs

Pull Arc moves the selected line segment to create an arc, as the cursor is moved. Horizontal lines move in the vertical direction. Vertical lines move in the horizontal direction. Diagonal lines move in all directions, unless anchored by a line at the end of the diagonal. If an existing arc is selected, its endpoints remain fixed and the arc chord follows the cursor.

Use the Pull Arc command to convert a drafting segment or corner into an arc. The starting points of the drafting segment become the start and stop angle for the new arc.

Procedure

- On the Schematic Editing toolbar, click the **Modify 2D Line** button.
- Select the line for arc conversion.
- Right-click and click the **Pull Arc** popup menu item. An arc follows the cursor movement.
- Click to indicate the end of the arc conversion.

Combining 2D Lines and Text

In SailWind Logic, combining is the process of merging two or more 2D line or text items into a single complex 2D line item. After you have combined them, you can manipulate the group (move, rotate, duplicate, and so forth) as a single item. This is useful for creating title blocks or other similar features.

Restrictions and Limitations

- A combination can only include 2D lines or text. This differs from a [group](#) on page 215, which is a collection of objects (which may or may not include 2D lines or text) on the schematic.

Procedure

Use one of the following:

- Using Object Mode
 - a. Select a 2D line or block of text.
 - b. Using Ctrl-click, continue to select items to combine.
 - c. Right-click and click the **Combine** popup menu item.

The selected items are combined.



Tip

When you select a 2D line item from the combination for move, duplicate, etc., all items are selected for the operation. Selecting text first will not select the whole group.

- Using Verb Mode
 - a. On the Schematic Editing toolbar, click the **Combine/Uncombine** button.
 - b. Right-click and click the **Combine** popup menu item, then select a line item.



Tip

When combining 2D lines and text, always select the 2D line first, then select the text. If you try to select a text item before first selecting a 2D line, the message “No drawing item to add text to” displays.

- - c. Continue to select the items to combine. Selected line items change to the highlight color.
 - d. Right-click and click the **Complete** popup menu item.

The selected items are combined.

- - e. Click the **Select** button or press the Esc key to exit the Combine/Uncombine function.



Tip

Selecting a 2D line item in the combination for move, duplicate, and so on, selects all items for the operation. Selecting text first does not select the whole group.

Related Topics

- [Adding Text](#)
- [Uncombining 2D Lines and Text](#)
- [Exploding Combinations](#)

Uncombining 2D Lines and Text

You can use the Uncombine command to remove individual objects from a combination.

Procedure

1. On the Schematic Editing toolbar, click the **Combine/Uncombine** button, right-click and click the **Uncombine** popup menu item.
2. Select the line or text item to remove.
3. Continue selecting items to remove as necessary.
4. Right-click and click the **Complete** popup menu item or press the Esc key when you finish removing objects from the group.

The objects return to their original states and locations.

Related Topics

- [Exploding Combinations](#)

Exploding Combinations

You can use the Explode command to separate all objects from a combination.

Procedure

Use one of the following:

- Using object mode:
 - a. Select a 2D line item from the group.
 - b. Right-click and click **Explode**.

The combination is no longer combined, and objects are separated.



Tip

In object mode, you can only explode a combination if it contains text items. Use the verb mode method (below) to explode a combination that does not have text items.

- Using verb mode:

- a. On the Schematic Editing toolbar, click the **Combine/Uncombine** button.
- b. Right-click and click the **Explode** popup menu item and select the item to explode.
- c. Right-click and click the **Complete** popup menu item.

Adding Drafting Items to a Library

You can save a 2D line item or a complex 2D line item in the schematic to a lines library. This makes it available in the current or future design sessions.

Procedure

1. Select an item, right-click and click the **Save to Library** popup menu item.
2. Type the item name and specify the library location in the Save to Library dialog box.
3. Click **OK**.

Adding Drafting Items From a Library

Use the Get Drafting Item from Library dialog box to load a 2D line item from the available libraries to the current schematic.

Procedure

1. On the Schematic Editing toolbar, click the **Add 2D Line from Library** button.
2. In the Get Drafting Item from Library dialog box, select an item from the Drafting Items list.

If the list does not contain the item you need, use the filter to search the available libraries. Use the wildcard convention, with or without leading characters, to expand or narrow the filter. Click **Apply** to activate the filter. SailWind Logic searches only active libraries with the Lines parameter checked for the 2D element.
3. Click **OK**. The item follows the cursor movement.
4. (Optional) You can rotate or mirror the item before you indicate its location. Right-click and click **Rotate**, **Mirror X**, or **Mirror Y** as often as needed to set the correct orientation of the item.
5. Click to place the item.

Modifying Objects in a 2D Lines Library

To edit an object that is stored in a 2D Lines library, you must first bring that object into the design workspace, make the desired edits and then save it back into the library.

Procedure

1. In the SailWind Logic workspace, on the Schematic Editing Toolbar, click the **Add 2D Line from Library** button.
2. In the Get Drafting Item from Library dialog box, select the desired object from the Drafting Items list and click **OK**.
The item will attach to the cursor.
3. In the design workspace click a location to place the item.
4. Make the desired edits using the commands available on the Schematic Editing Toolbar and the right mouse button menus.
5. After you have completed the desired edits, select the objects that make up the drawing item, right click and choose **Save to Library**.
6. You can choose to either overwrite the item or save it as a new item.

Clicking **OK** saves the 2D line object back into the 2D Lines library.



Tip

To save multiple objects that include lines and text, you must first select all the items, choose **Combine** from the right mouse menu, and then choose **Save to Library** from the right mouse button menu.

Chapter 13

Connections

Connections are the heart of your design and enable you to capture and convey the interconnectivity of your design. SailWind Logic offers a robust selection of commands for working with adding, naming and connecting nets. You can fully connect your design objects, or use floating connections as placeholders for future design decisions. You can add, move split nets or segments. You can also swap pins on your parts as needed. There are also a powerful set of commands available for creating, editing, and managing buses.

[Adding Connections](#)
[Naming a Connection](#)
[Net Attributes Overview](#)
[Creating Net Attributes](#)
[Default Attributes for Nets](#)
[Adding Power and Ground Connections](#)
[Working With Floating Connections](#)
[Editing Off-Page Symbol Sheet Numbers Per Line](#)
[Changing a Connection](#)
[Moving Connections](#)
[Splitting a Connection](#)
[Splitting Segments](#)
[Swapping Pins](#)
[Managing Buses](#)
[Deleting Connections](#)
[Detach](#)

Adding Connections

When you add connections, they must have start and end points that are electrically meaningful. Start a connection on either a symbol pin or another connection. End a connection on a symbol pin, another connection, a bus, an off-page reference, or a ground or power symbol. SailWind Logic prevents you from starting or ending a connection just anywhere in the database unless you enable Floating Connections.

See also [Working With Floating Connections](#).

Procedure

1. Click the **Add Connection** button.
2. Indicate the starting point of the connection. The connection follows the cursor movement. Right-click and click **Angle** to specify diagonal segments in the connection.
3. Indicate corners as needed. Right-click and click **Del Corner** to back up to the previous corner if necessary.
4. A connection cannot end in open space. Locate the cursor and double-click to end the connection.

5. To end a connection with a power, ground, or off-page reference symbol, select the appropriate symbol from the popup menu during the last step of this procedure.

If you end the connection on a bus, see the remaining steps in the [Adding Connections to Buses](#) topic.



Tip

Special Symbols in SailWind Logic include power symbols, ground symbols, and off-page reference symbols. For more information, see [Special Schematic Symbols](#).



Note:

When you end a connection at another connection, SailWind Logic places a tie dot at the junction. The resulting net name is the name of the added connection, not the existing connection, unless you assign a name to the existing connection.

6. To assign a fixed net name to a connection, you must first add the connection and then select a segment that is directly attached to a part type pin. Then, you can select the net, right-click and click **Properties**, and enter the new net name in the [Net Properties](#) on page 289 dialog box.



Tip

If you select and move a visible net name, with snap to grid enabled, the new net name snaps to the Labels and Text grid. To set the labels and text grid, use the settings on the Tools > Options > General page. See “[Options Dialog Box, General Category](#)” on page 595.

Related Topics

[Adding Power and Ground Connections](#)

[Adding Off-Page References](#)

Naming a Connection

When you add a connection, SailWind Logic automatically assigns a name with the format \$\$\$nnnn, where nnnn is a random number. You can specify a different net name to replace the assigned name.

Procedure

1. Select a connection segment.
2. Right-click and click **Properties**.
3. Type a new net name in the Net Name box.



Tip

The Net Name Label check box exhibits the following behaviors:

- The Net Name Label check box is only available when you select a connection segment that enters or exits a part.
 - Clearing the Net Name Label check box removes the visible label. The net and all subnets retain the net name.
-

4. To place a bar over the net name, precede the text with the \ character. To place a bar over only a portion of text, enclose that section with the \ character.

For example, typing MAIN_CLK\ places a bar over CLK.



Tip

The \ character factors into the 47 character limit for the net name.

Related Topics

[Modifying Nets](#)

Net Attributes Overview

Net attributes enable you to associate information with a single net or set of nets on the schematic. Attributes are made of two parts, an attribute name and its corresponding value.

If connecting to SailWind Layout, these attributes are passed along with the rest of the schematic.



Tip

When working with attributes, note the following:

- SailWind Layout checks a limited set of net attributes. See the [Default Attributes for Nets](#) topic for more information.
 - To assign attributes to signal pins, see the [Signal Pin Nets](#) topic.
-

See also [Creating Net Attributes](#).

Creating Net Attributes

You can create net attributes and assign them to specific nets. Once created, the attributes can be edited or deleted.

Procedure

1. Select a net, right-click and click **Attributes**.
2. In the Net Attributes dialog box, click **Add** to add an attribute.
3. Click the Name column and type a new attribute name.

See also [Default Attributes for Nets](#).

4. Click the Value column and type an attribute value.
5. To remove the selected net attribute, click **Delete**.
6. To edit the selected name or value, click **Edit**.
7. Click **OK**.

Default Attributes for Nets

Refer to the Listing of Default Attributes for Nets table to determine which net attributes are checked by SailWind Layout for correct values and units.



Tip

Some values in the Description and use column are case-sensitive and must be written as such.

Table 30. Listing of Default Attributes for Nets

Attributes	Type	Description and Use
HyperLynx.Frequency	Frequency (Measure)	<p>Lists the working frequency for nets in BoardSim simulations.</p> <p>Range: 0 Hz-1000 GHz</p> <p>Units: Hz kHz MHz GHz (case-sensitive)</p>
HyperLynx.Duty Cycle	Percentage (Measure)	<p>Lists the percent of time the signal is in high state. Used for BoardSim simulations.</p> <p>Range: 0%-100%</p>
HyperLynx Signal Type	List	<p>Lists the signal type. Used in BoardSim simulations.</p> <p>Values (case-insensitive): One of, Clock, Strobe, Data, Address, Power Supply, Analog High Speed, Analog Low Speed, Do Not Analyze</p>
HyperLynx.Default IC.Model	Free Text	<p>Lists the part model in the file used for BoardSim simulations.</p> <p>Values: Text (model name)</p>
HyperLynx.Default IC.Model File	Free Text	<p>Lists the part model file in the file used for BoardSim simulations.</p> <p>Values: Text (filename)</p>
HyperLynx.Default IC.Model Pin	Free Text	<p>Lists the specific pin for the model in the above attribute, if applicable. Used in BoardSim simulations.</p> <p>Values: Text (pin name or number)</p>
PowerGround	Yes/No	Identifies nets as Ground and Power nets.

Table 30. Listing of Default Attributes for Nets (continued)

Attributes	Type	Description and Use
		Values: Yes or No (case-insensitive)
Voltage	Measure	Describes the voltage of the net. Range: -100kV-100kV (-100,000-100,000V) Units: nV uV mV V kV (case-sensitive)
DFT.Nail Count Per Net	Number	Indicates the ID of the probe in the test fixture. Values: (integer) 0-2000 (all exact limits are allowed; for example, both 0 and 2000 are acceptable)

Adding Power and Ground Connections

When you connect a part to ground or power nets, the normal convention is to use a special symbol to represent the net. The ground symbol ties the connection to ground. The power symbol adds voltage to connections.

Procedure

1. Specify the connection as described in the [Adding Connections](#) topic.
2. Right-click and click **Ground or Power**. The symbol attaches to the connection and follows the cursor movement.
(Optional) You can rotate or mirror the symbol by choosing the appropriate command from the popup menu. You can also modify the symbol representation by right-clicking and clicking **Alternate**. Right-click and click **Display PG Name** to toggle the visibility of the net name for the power or ground symbol.
3. Locate the cursor and double-click to end the connection or right-click and click **Complete**.
4. You can create and store a number of ground symbols in the library, each with its own net name. To select the appropriate ground symbol, right-click and click **Alternate**.
Three symbols are available with the standard library. You can modify these or add additional symbols.
SailWind Logic assigns the net name associated with the ground symbol to the connection, (for example, GND, Chassis ground, Analog-ground, etc.).
5. For each alternate power symbol, there is a different signal name: +5V, +24V, etc. To select the appropriate signal name, click **Alternate** while placing the symbol.



Tip

As you view the power or ground alternates in the working window, you can locate in the status bar the net name to which you want to tie the connection.

Related Topics

[Adding Connections](#)

[Using Alternate Symbols](#)

Working With Floating Connections

Floating Connections add another level of flexibility to designing your schematic. Enable Floating Connections and you can add a connection from a pin to a location where a part doesn't yet exist without requiring an off-page symbol. You can also begin a connection in empty space and connect it later. Unconnected nets are terminated with square markers.

[Enabling Floating Connections](#)

[Disabling Floating Connections and Running the Connectivity Report](#)

[Adding Floating Connections](#)

[Editing Floating Connections](#)

[Duplicating Floating Connections](#)

[Creating Floating Connections When Deleting Objects](#)

[Attaching Objects to Floating Connections](#)

[Adding Off-Page References](#)

Enabling Floating Connections

You can enable floating connections in your design. This enables you to create connections that are unterminated at one or both ends.

Procedure

1. Click the **Tools > Options** menu item, and then click the **Design** category.
2. In the Parameters area, select the Allow Floating Connections check box to enable Floating Connections in the design.

Disabling Floating Connections and Running the Connectivity Report

You can disable floating connections in your design. Once you have finished creating a design with floating connections enabled, you can disable the floating connections and run a connectivity check on the design to assure that all connections have been properly terminated.

Procedure

1. Click the **Tools > Options** menu item, and then click the **Design** category.
2. In the Parameters area, clear the Allow Floating Connections check box to disable Floating Connections in the design.

If Floating Connections exist in the design, a warning message appears.



Note:

Warning: Disabling Floating Connections does not remove them from the design. Disabling the preference only prevents additional Floating Connections from being created.

Floating Connections are undesirable on a finished schematic. A floating end should either be connected to a pin or terminated on an off-page symbol. Run the Connections Report to find the locations of Floating Connections in your design.

3. Click the **File > Report** menu item.
 4. In the Reports dialog box, select the Connectivity check box.
 5. Click **OK** to create the report.
-



Tip

Multiple Floating Connections are easily selected for deletion. In the Search and Select box of the Selection Toolbar, type "dangling" to select all the Floating Connections.

Adding Floating Connections

With the Floating Connections preference enabled, you can leave a connection dangling in space for a later connection. You can also start a connection in space in anticipation of a later connection. New parts and connection stubs added automatically through an ECO Import will receive Unconnected Markers instead of off-page symbols. Net stubs added through an ECO Import to existing parts will continue to receive off-page symbols.

[Creating a Dangling Connection](#)
[Creating a Floating Connection](#)

Creating a Dangling Connection

A dangling connection is tied to a design object at only one end in anticipation of it being connected at the other end at some future point in the design process.

Procedure

1. Start a connection on a pin.
2. Draw the connection.
3. To end the connection in space, double-click.

Alternatively, you can right-click and click **End**.

Creating a Floating Connection

A floating connection is not tied at either end. Use floating connections to add connections to your design as placeholders with the intent of completing the connections at some later time in the design process.

Procedure

1. To start a connection in space, double-click.
2. Draw the connection.
3. To end the connection in space, double-click.

Alternatively, you can right-click and click **End**.

Editing Floating Connections

You can add floating connections to your design and connect them at a later time. Once created, you can edit the floating connections in your design.

- Use Move mode or Add Connection mode to continue a connection from the Unconnected Marker.

Duplicating Floating Connections

With the Floating Connections preference enabled, you can duplicate segments of connections and paste them in space. You are not required to paste them where they will make a pin to pin connection.

Procedure

1. Select a segment(s) of a connection.
2. Right-click and click **Duplicate**.
3. Position the cursor in the paste location and click.

See also [Step and Repeat](#).

Creating Floating Connections When Deleting Objects

With the Floating Connections preference enabled, connections attached to deleted parts are terminated with Unconnected Markers instead of off-page symbols. When a bus is deleted, nets connected to the bus are also terminated with Unconnected Markers instead of off-page symbols.

Attaching Objects to Floating Connections

With the Floating Connections preference enabled, any component pins that touch a floating connection end will automatically connect. A pin that touches another component pin will automatically connect also. Move the components apart to make the connection visible.

Adding Off-Page References

To connect two pins on separate sheets of the schematic, or at opposite ends of the same sheet, without a graphical connection, use an off-page reference with identical net names.

Procedure

1. Specify the connection as described in the [Adding Connections](#) topic.
2. Right-click and click **Off-page**.

The symbol attaches to the connection and follows the cursor movement.

The default decal with the right pin is defined as a source pin; the alternate decal with the left pin is defined as a load pin. This pin type orientation plays a role when you create a hierarchical symbol of the sheet. All off-page sources are on the left of the hierarchical symbol outline, and all loads are on the right. You can place the default or right-click and click **Alternate**.

(Optional) You can rotate or mirror the symbol by using the appropriate command.

3. Locate the cursor and double-click to end the connection or right-click and click **Complete**.
4. Type the net name in the dialog box and click **OK**.

The sheet number for the connecting symbol appears next to the off-page symbol if the “Show Off-page Sheet Numbers” option is selected in the [Options Dialog Box, Design Category](#).

**Tip**

If you display the off-sheet numbers, you can line them up or stack them by specifying Numbers per Line in the Tools > Options > Design category.

Editing Off-Page Symbol Sheet Numbers Per Line

The displayed sheet number next to an off-page reference is set by default to enable 5 sheet numbers to be listed in a single text string, separated by commas. You can use Properties mode to edit the number of off-page symbol sheet numbers per line.

For example, if set to one, and there are two sheets containing the same net, the sheets will be listed in two lines.

Procedure

1. Select the off-page reference text string.
2. On the Schematic Editing toolbar, click the **Properties** button.
This displays an information dialog box.
3. Change the settings for sheet numbers per line.
4. Click **OK**.

See also [Options Dialog Box, Design Category](#).

Changing a Connection

Use Move to change a connection from one endpoint to another, without deleting the rest of the connection.

Procedure

1. Select the connection at any point on the segment entering or exiting the part, or where it enters another connection.
2. Right-click and click **Move**.
The connection follows the cursor movement.
3. Right-click and click **Angle** to specify diagonal segments in the connection.
4. Indicate corners as needed or right-click and click **Add Corner**. Right-click and click **Del Corner** to back up to the previous corner if necessary.

5. A connection cannot end in open space. Locate the cursor and double-click to end the connection or right-click and click **Complete**.
6. To end a connection with a power, ground, or off-page reference symbol, right-click and click the appropriate symbol during the last step of this procedure.

Related Topics

- [Adding Connections](#)
- [Adding Power and Ground Connections](#)
- [Adding Off-Page References](#)

Moving Connections

When you move a part, existing part connections follow the part movement. After part repositioning, you might need to move connections to improve the flow.

You can reposition a connection segment that is not a connection endpoint, or you can reconnect a segment endpoint. Refer to the [Changing a Connection](#) topic for information on disconnecting from a part or connection and reconnecting to another part or connection.

Related Topics

- [Working With Floating Connections](#)

Splitting a Connection

You can split a connection to create new corners or a 90-degree turn.

Restrictions and Limitations

You can not split segments which enter or exit a part.

Procedure

1. Select a segment, right-click and click **Move**.
2. Right-click and click **Split Segment**. Move the cursor to define the segment length.
3. Right-click and click **Swap Corner** to define the endpoint in the opposite direction.
4. Indicate the endpoint of the new segment to complete the split.

Splitting Segments

Split converts a line segment into two separate segments, joined by a third segment perpendicular to the original segment.

All line segments in a 2D line must have the same line width and style. If you need to create a 2D element with more than one line width or style, create two separate 2D lines and use the Combine command to join them.

Procedure

1. On the Schematic Editing toolbar, click the **Modify 2D Line** button.
2. Select the line to split.
3. Right-click and click **Split**. The new and divided segments follow the cursor movement.

You can right-click and click one of the following when splitting a segment:

Table 31. Splitting Segments Options

Option	Description
Swap Corner	Selects the other corner or line segment created by the split.
Del Corner	Deletes the current corner.
Width	Changes the line width of the item.
Solid Style and Dotted Style	Changes the line style to a solid or a dashed line. You cannot specify dotted lines for circles or arcs.
Orthogonal	Moves are made in 90-degree increments.
Diagonal	Moves are made in 45-degree increments.
Any Angle	Moves are made at any angle.

4. Indicate the endpoint of the new segment to complete the split.

Swapping Pins

To swap two part pins, you can use the **Swap Pins** button in the Schematic Editing toolbar or **Swap Pins** on the popup menu. SailWind Logic swaps the pin number, name, and terminal of the pins you select. You can also swap pins when you place a part using Mirror X and Mirror Y on the popup menu. This is often useful for improving the flow of the schematic.

[Swapping Pins Using the Swap Pins Button in Verb Mode](#)

[Swapping Pins by Selecting One Pin at a Time](#)

[Swapping Pins by Selecting Both Pins First](#)

Swapping Pins Using the Swap Pins Button in Verb Mode

You can swap pins using the **Swap Pins** button. Using this method enables you to initiate the command and then make your pin selections to swap. You can continue to select pin pairs to swap until you cancel the command.

Procedure

1. On the Schematic Editing toolbar, click the **Swap Pins** button.
2. Select the first pin number.
3. Select the pin number to swap with the first selected pin.
4. If the pin types you select do not match, the message “Warning, pin swap types don’t match...” message appears. Click **Yes** to continue. Click **No** to cancel the swap.



Tip

The message “Keep connections tied to same electrical pin?” may also appear. Click **Yes** to preserve the connections with the pin numbers. Click **No** to reconnect to the renumbered pins.

To use the popup menu to swap pins, you can either select one pin at a time or select both pins at once:

Swapping Pins by Selecting One Pin at a Time

You can swap pins by selecting one pin at a time. This method enables you to select the first pin, initiate the command and then select the second pin. To create another pin swap, you would repeat the process.

Procedure

1. Select the first pin number.
2. Right-click and click **Swap Pins**.
3. Select another pin number to complete the swap.

4. If the pin types you select do not match, the message “Warning, pin swap types don't match...” message appears. Click **Yes** to continue. Click **No** to cancel the swap.



Tip

The message “Keep connections tied to same electrical pin?” may also appear. Click **Yes** to preserve the connections with the pin numbers. Click **No** to reconnect to the renumbered pins.

Swapping Pins by Selecting Both Pins First

You can swap pins by selecting both pins first. After you have selected both pins, you can initiate the command. To create another pin swap, you would repeat the process.

Procedure

1. Using Ctrl+click, select the pins you want to swap.
2. Right-click and click **Swap Pins**. If the message “Swap connected pin?” appears, click **Yes** to continue. Click **No** to cancel the swap.
3. If the pin types you select do not match, the message “Warning, pin swap types don't match...” message appears. Click **Yes** to continue. Click **No** to cancel the swap.



Tip

The message “Keep connections tied to same electrical pin?” may also appear. Click **Yes** to preserve the connections with the pin numbers. Click **No** to reconnect to the renumbered pins.

Managing Buses

SailWind Logic enables you to create, name, edit, and delete buses in your design using a flexible array of available commands. Using the bus commands enables you to decrease the visual clutter in your design and accurately convey your design intent without having to graphically represent every net as a distinct line.

- [Choosing the Bus Type](#)
- [Naming Buses](#)
- [Adding Buses](#)
- [Adding Connections to Buses](#)
- [Adding Nets to a Mixed Net Bus](#)
- [Extending Buses](#)
- [Moving Bus Segments](#)
- [Splitting Buses](#)
- [Deleting Bus Segments](#)
- [Deleting Buses](#)

Choosing the Bus Type

There are two bus types: a bit format bus and a mixed bus. Choose the bus structure that can best visualize your design intent. The bit format bus will display sequential bus nets. The Mixed bus format will enable you to combine sequentially named bit format bus nets along with individual nets.

A bit format bus contains a series of nets with consecutive net names. For example, the bit format bus AD[0:4] contains nets AD0, AD1, AD2, AD3, and AD4.

A mixed net bus can contain one or more bit format buses in addition to individual nets that are not necessarily sequential. For example, a mixed net bus BUS1 can contain individual nets named RAS, CAS, and EN as well as the bit format bus AD[0:7].



Tip

The Bus Type you select determines which options appear in the Bus Name list. For example, if you select the Bit Format Bus Type, only bit format bus names appear in the Bus Name list.

Naming Buses

SailWind Logic defines a bus that appears on the same sheet, or any other sheet, with an identical name as the same bus.

Rules for naming bit format and mixed net buses differ as follows:

Naming a Bit Format Bus

The bit format bus name consists of two parts: the prefix and the bit range of the bus. The format is PREFIX[nn:mm] where nn is the lowest bit number and mm is the highest bit number. You can use leading zeros to determine the lowest bit number. For example DD[00:15] enables nets DD00, DD01,...DD15 to connect to the bus; whereas DD[0:15] enables nets DD0, DD1, DD2,...DD15.

The sheet number also displays with the bus name if the Show Off-Page Reference option is on the [Options Dialog Box, Design Category](#).

The total bit format bus name - including the prefix, brackets and colon - cannot exceed 47 characters. The bit numbers must be in the range of 0 to 32767 and cannot contain alphabetic characters.

Naming a Mixed Net Bus

The mixed net bus name can contain up to 47 alphanumeric characters. The mixed format bus name cannot include a bit range suffix or spaces. You cannot create a bus name that is the same as a net name.

Adding Buses

You can add a bus to your design so that you can display a collection of related nets as a single entity.

Procedure

1. On the Schematic Editing toolbar, click the **Add Bus** button.
2. Indicate the starting point. The bus follows the pointer movement.
3. Indicate bus corners as needed, or right-click and click **Add Corner**.
If you need to back up to the previous corner, right-click and click **Del Corner**.
4. Double-click to end the bus in open space or right-click and click **Complete**.
5. In the Add Bus dialog box, in the Bus Name box, type a new name for the bus, or select a bus name from the list.



Restriction:

The Rename area is available only in Query mode.

6. Select the Bus Type. The bus type determines which bus names appear in the Bus Name list.
7. Select the Add Bus Name Label check box to add the bus name as a label to the bus at the end of the bus closest to where you selected it.



Tip

When working with bus labels, observe the following:

- A bus can have two labels, one on each end.
- The check box is unavailable when the end of the selected bus has a label.
- A bus label is not required.
- To delete a bus label, select the label in the schematic and click the **Delete** button on the standard toolbar.

8. If the bus is a mixed net bus, add the bus names in the Bus Nets area.



Tip

To rotate the text before you place it, right-click and click **Rotate**, then indicate the name location

Adding Connections to Buses

When connecting to a bus, the connection must be perpendicular to the bus. SailWind Logic adds an angled bus-tap segment at the point of connection and gives the connection the next sequential net name in the bus.

Confirm or type the net name in the dialog box. The text follows the cursor movement. Position the text and indicate its location. To swap the bus tap angle, right-click and click **Swap Angle**.

Adding Nets to a Mixed Net Bus

Use the Bus Nets list in the Add Bus dialog box or the Bus Properties dialog box to add nets to a mixed net bus. The Bus Nets area is not available when the Bit Format bus type is selected.

Procedure

1. After you have added a bus, in the Add Bus dialog box, click **Add** to add a new row in the Bus Nets list.
 2. Type the bus prefix in the Name/Prefix text box.
-



Tip

Enter the names as follows:

- For a single net, type the net name.
 - For a range of sequential nets, type the prefix for the sequence of nets.
-

3. In the Start text box, type the starting bit number for a sequence of nets.
 4. In the End text box, type the ending bit number for a sequence of nets.
 5. To add an individual net to the bus, type the net name in the Prefix/Name text box.
Do not type Start and End values for a single net that is not sequenced by bit number.
 6. Click **Edit** to modify a selected cell in the Bus Nets list.
 7. Click **Delete** to remove a net from the Bus nets list.
 8. Click **Down** to moves the selected row down one position in the Bus Nets list.
-



Tip

The order in which nets appear in the Bus Nets list box determines the default order for naming nets as they are connected to a bus.

9. Click **Up** to move the selected row up one position in the Bus Nets list.
-

Extending Buses

If you need more room to fan out bus connections, or need to lengthen the path of an existing bus, you can extend a bus segment.

Procedure

1. On the Schematic Editing toolbar, click the **Extend Bus** button.
2. Select either endpoint of the bus.

The end segment follows the pointer movement.
3. Click to indicate corners as needed for the extension or right-click and click **Add Corner**.
4. Double-click to indicate the new endpoint or right-click and click **Complete**.
5. You cannot use the Extend Bus command to shorten a bus segment. To shorten a bus, delete the long segment, then use Extend Bus to redefine that segment, or alternately, you could use the Group command to shorten the bus segment.

See also [Creation of Groups](#).

Moving Bus Segments

If you need to reposition a bus segment to accommodate additional edits to your schematic, you can move a bus segment.

Procedure

1. Select a bus segment, right-click and click **Move**.
2. Indicate the new location with the pointer and then click to place.

Splitting Buses

If you need to edit a bus, you can create new corners or a 90-degree turn in the bus.

Procedure

1. Click the **Split Bus** button.
2. Select the bus segment. The new and divided segments follow the pointer movement.
3. Move the pointer to define the segment length. To define the endpoint in the opposite direction, right-click and click **Swap Corner**.
4. Indicate the endpoint of the new segment and click to complete the split.

Deleting Bus Segments

As your design progresses, if you determine that you no longer need a bus segment, you can remove a segment of the bus.

Procedure

1. Select the bus segment.
2. Right-click and click **Delete**.

If the deletion splits the bus into two separate buses, the bus name appears, identifying any unnamed bus segments. Each bus retains the same properties.

Deleting Buses

If you determine that you need to delete a bus that was selected in object mode, you can choose to delete the bus only or the bus and its connections.

See also [Object Select Mode \(Select Object First\)](#).

[Delete a Bus Only](#)

[Delete a Bus and Its Connections](#)

Delete a Bus Only

If you need to delete a bus, you have the option to delete a bus only and leave the connections intact.

Procedure

1. Select a bus.
2. If you want to keep connections attached to the bus, right-click and click **Preserve Connections** to enable it.

Preserve Connections is enabled by default.



Tip

Preserve Connections is enabled when a check mark appears next to it in the menu.

3. Right-click and click **Delete**.

Connections are left at their original locations.

Delete a Bus and Its Connections

If you need to delete a bus, you have the option to delete a bus and its connections.

Procedure

1. Select a bus.
2. If you want to remove connections attached to the bus, right-click and click **Preserve Connections** to disable it.

Preserve Connections is enabled by default.



Tip

Preserve Connections is disabled when a check mark does not appear next to it in the menu.

3. Right-click and click **Delete**.

The bus and connections are deleted.

Deleting Connections

You can delete connections, but be aware that when the deleted connections are subnets, the results can differ.

- Select a connection, right-click and click **Delete**.
- When a subnet with a system assigned name is split into two subnets by deleting a connection, one of the subnets is automatically assigned a new system name.
- When a subnet with a user assigned name is split into two subnets by deleting a connection, both new subnets retain the user assigned name, even if there is no visible net name label.
- When subnets are joined and both have user assigned names, the user is prompted to select one of the names for the combined net, even if there are no visible net name labels.
- When a system named subnet is joined with a user named subnet, the combined subnet automatically inherits the user assigned name.
- When two system named subnets are joined, the combined subnet inherits the name of one of the subnets.

Related Topics

[Splitting a Connection](#)

Detach

Use the detach command to free a gate, part, group of components, or group selection from any attached nets.

Procedure

1. Make a selection.
2. Right-click and click **Detach**.

All nets are detached. Nets are given unconnected markers or off-page symbols, depending on the Floating Connections preference.



Note:

Warning: A group selection with Preserve by Boundary enabled detaches nets at the boundary and not at net terminations. The Net is split into two instances. Named nets retain their original names on both sides of the split.

Related Topics

[Working With Floating Connections](#)

[Creation of Groups](#)

Chapter 14

Hierarchical Design

SailWind Logic supports hierarchical design where you can create symbols to represent entire sub-schematics and show system interactivity at a higher level. You can create hierarchy from either the top down or the bottom up.

- [Hierarchical Design Overview](#)
- [Creating a Top-Down Hierarchy](#)
- [Creating a Bottom-Up Hierarchy](#)
- [Pushing Into the Hierarchy](#)
- [Popping Up the Hierarchy](#)
- [Modifying a Hierarchical Symbol](#)
- [Copying a Hierarchical Symbol](#)
- [Deleting a Hierarchical Symbol](#)

Hierarchical Design Overview

Using hierarchical design, you can create high-level symbols to represent complex sub-schematics, or repetitive elements in your design. You can create a hierarchy by using either the top down or the bottom up design strategy.

- **Top down** — You define a hierarchical symbol and then the underlying logic.
- **Bottom up** — You create a schematic, then define the hierarchical symbol for the schematic.

You can also assign or unassign hierarchical connections in the [Hierarchical Component properties](#) on page 296 dialog box.

Level 0 is the top design level that contains the design contents. Numbered sheets from sheet 1 to 1024 represent the design contents. A SailWind Logic design cannot exceed a total number of 1024 hierarchical and numbered sheets.

You can navigate the hierarchy with the [Push Hierarchy](#) on page 269 and [Pop Hierarchy](#) on page 270 commands on the **View** menu.

When a hierarchical symbol is copied, any existing underlying schematic content is also copied. SailWind Logic updates the reference designator names of the underlying symbols. There is a one-to-one correspondence between hierarchy symbols and a schematic sheet. The underlying logic for duplicated hierarchical symbols is stored as a unique sheet, and can be edited without affecting the logic of other hierarchical symbols. Even though you copy a hierarchical symbol, its underlying schematics can be completely different from the copied symbol's underlying schematics.

Creating a Top-Down Hierarchy

In top-down design, you create a hierarchical symbol before the underlying schematic exists. This enables you to create the container object and then populate it with the lower level design elements. Pin names tie together signals between hierarchical levels.

SailWind Logic ties together all instances of a common signal name, regardless of where they are found in the design, into a single connection net. You establish connectivity across the hierarchy by using the same signal name in the underlying logic.

Prerequisites

- Ensure that the off-page signals in the underlying schematic are consistent with the pin names in the hierarchical symbol.

Procedure

1. On the Schematic Editing toolbar, click the **New Hierarchical Symbol** button.
2. In the **Hierarchical Symbol Wizard Dialog Box**, in the Hierarchical Sheet area, type the symbol name in the Sheet Name text box.
3. Select your preferred Pin Decal for both input and output pins from the appropriate dropdown list boxes.
4. Specify the Pin Count for both input and output pins.
The Preview area displays the symbol outline.
5. Click **OK**. The hierarchical symbol appears in the **Part Editor** on page 109 window.



Tip

Because the symbol is a hierarchical one, it has no pin numbers. The pin name associated with pins of the hierarchical symbol identify the net name of the connection that is tied to the pin. All pins on a hierarchical symbol must have a pin name. You cannot complete the hierarchical symbol until you assign a name to each pin.

6. In the Part Editor, double-click each pin to open the **Terminal Properties dialog box** on page 131 where you name the selected pin. You must name each pin of the symbol.
7. Click the **File > Complete** menu item.
This closes the Part Editor, and the new symbol attaches to the cursor.
8. Move the cursor to a desired location and click to place the symbol in the schematic.
9. You can navigate this hierarchy with the **View > Push Hierarchy**, and **View > Pop Hierarchy** menu commands.

Related Topics

[Creating a Bottom-Up Hierarchy](#)

Creating a Bottom-Up Hierarchy

In bottom-up design, you create a hierarchical symbol from an existing sheet that represents the underlying logic. This enables you to create the lower level design and then wrap it in the hierarchical symbol.

SailWind Logic adds an input or output pin in the symbol for each off-page reference on the schematic sheet. SailWind Logic places off-page references with a pin type of Source on the left side of the hierarchical symbol, and off-page references with a pin type of Load on the right side. (Pin types for off-page references are defined in the library under electrical.)

Procedure

1. Switch to the sheet to which you want to add the hierarchical symbol. If needed, add a new schematic sheet to the sheet set using the **Setup > Sheets** menu item.
2. On the Schematic Editing toolbar, click the **New Hierarchical Symbol** button.
3. On the Hierarchical Symbol Wizard dialog box, select an existing sheet from the Sheet Number dropdown list box in the Hierarchical Sheet section.
4. Type a name for the symbol in the Sheet Name text box.
5. Select your preferred Pin Decal for both input and output pins from the appropriate dropdown list box.

The Preview area displays the symbol outline. The number of input/output pins on the symbol is automatically set from the number and type of off-sheet reference symbols on the underlying schematic sheet and the pin count boxes are unavailable for editing.

6. Click **OK**.

The hierarchical symbol appears in the **Part Editor** on page 109 window. An input or output pin is located on the symbol to match each off-page reference on the underlying schematic sheet.

7. Click the **File > Complete** menu item.

This closes the Part Editor, and the new symbol attaches to the cursor.

8. Move the cursor to a desired location and click to place the symbol in the schematic.
9. You can navigate this hierarchy with the **View > Push Hierarchy** and **View > Pop Hierarchy** menu commands.

Related Topics

[Creating a Top-Down Hierarchy](#)

Pushing Into the Hierarchy

Use the Push Hierarchy command to look inside or push down into a hierarchical symbol to view the underlying logic.

Procedure

1. Click the **View > Push Hierarchy** menu item.
2. Select the hierarchical symbol.

Alternatively, you can select the hierarchical symbol, right-click and click **Push Hierarchy**.



Tip

If there is no underlying schematic representation of the selected symbol, a blank sheet appears.

Popping Up the Hierarchy

Use the Pop Hierarchy command to replace the current sheet with its corresponding hierarchical symbol.

Procedure

In a hierarchical sub-schematic, click the **View > Pop Hierarchy** menu item.



Tip

In the Sheets list on the main toolbar, a sub-schematic is displayed with an indented icon.

Modifying a Hierarchical Symbol

Once created, you can modify a hierarchical symbol to add or subtract design detail.

Procedure

1. Select a hierarchical symbol, right-click and click **Edit Hierarchical Symbol**.
 2. Modify the symbol in the [Part Editor](#) on page 109.
 3. Click the **File > Complete** menu item when you are finished with the changes.
-



Tip

If you add terminals to a hierarchical symbol, ensure that you give them pin names, and that the names also appear as off-page references in the sheet with the underlying schematic.

Related Topics

[Hierarchical Symbol Wizard Dialog Box](#)

[Deleting a Hierarchical Symbol](#)

Copying a Hierarchical Symbol

When you copy a hierarchical symbol, the sheets that the symbol references will also be copied and added as new sheets to the schematic. SailWind Logic assigns new reference designators to the part types in the copied sheets. If the referenced sheet also contains hierarchical symbols, then SailWind

Logic also copies the sheets referenced by those symbols, and so on, down the entire sub-tree of the hierarchy.

Procedure

1. Select the symbol in the design.

When you select a hierarchical symbol in copy mode, a warning prompt appears. If you click **Yes** in the prompt window, SailWind Logic copies the hierarchical symbol, and all directly or indirectly referenced sheets. If any numbered sheets are referenced, SailWind Logic assigns the copies new sheet numbers.

2. Ctrl+drag a copy.

3. In the warning prompt, click **Yes**.

The symbol copy attaches to the cursor.

4. Click the place the symbol.

Related Topics

[Hierarchical Symbol Wizard Dialog Box](#)

[Modifying a Hierarchical Symbol](#)

[Deleting a Hierarchical Symbol](#)

Deleting a Hierarchical Symbol

You can delete a hierarchical symbol, but use caution when performing the operation.

Procedure

1. On the Schematic Editing Toolbar, click the **Delete** button.

2. When you select a hierarchical symbol in delete mode, the message “OK to delete sheets associated with the hierarchy - Warning: there is no Undo for this operation” appears. Be sure of your intent before confirming the command.

- If you click **Yes**, SailWind Logic deletes the hierarchical symbol and any associated sheets throughout the hierarchy.
- If you click **No**, and it is a hierarchical symbol without a sheet number, SailWind Logic assigns it the next unused sheet number and it becomes a normal, numbered sheet.

Chapter 15

Schematic Object Modification

Changes and edits to your design objects are fully-supported in SailWind Logic. This includes the modification of drafting objects, fields, parts, reference designators, part attributes, labels, pins, and buses. You can modify individual instances of objects, or you can choose to update the library objects so that any modifications are available to your future design sessions.

- [Modifying Drafting Objects](#)
- [Modifying Fields](#)
- [Modifying Parts](#)
- [Reference Designator Renumbering](#)
- [Automatically Renumbering Reference Designators](#)
- [Setting Reference Designators by Sheet in a New Schematic](#)
- [Setting Reference Designators by Sheet in Completed Schematics](#)
- [Modifying Part Attributes](#)
- [Modifying Part Attribute Labels](#)
- [Modifying Part Type Labels](#)
- [Searching the Library for a Decal](#)
- [Rename Part](#)
- [Rename Gate](#)
- [Modifying Reference Designator Labels](#)
- [Modifying Pins](#)
- [Modifying Pin Label Fonts](#)
- [Modifying Nets](#)
- [Modifying Net Name Labels](#)
- [Modify Buses](#)
- [Modifying Bus Name Labels](#)
- [Modifying Off-Page Labels](#)
- [Modifying Label Font Sizes](#)
- [Modifying Text](#)
- [Modifying Hierarchical Components](#)

Modifying Drafting Objects

Use the Drafting Properties dialog box to modify the line width, style, and orientation of selected drafting objects.



Tip

Use the Schematic Editing toolbar to modify the shape of 2D lines.

Procedure

1. Select a drafting object, right-click and click **Properties**.

2. In the Width box, type a new line width for the drafting object.

The width box lists the current line width of the selected drafting object.

Use the **Line Widths** tab in the Options dialog box to change the default line width.

3. Select the Filled check box to create a filled shape from a selected polygon.



Restriction:

This option is unavailable for circles, paths, and if you used Pull Arc to modify the polygon.

4. In the Style area, select a line style option for the selected drafting object.

5. In the Rotation box, select the degree of rotation from the Rotation list.



Tip

When rotating objects, observe the following:

- Rotation can be 0 or 90 degrees.
- The point used when selecting the object is the also the point of rotation.

6. Select the Mirror by X or Mirror by Y check boxes to mirror the selected drafting object in the X (horizontal) or Y (vertical) direction.

Modifying Fields

Use the Field Properties dialog box to modify a field name or change its text size, orientation, or justification.

Procedure

1. Select a field, right-click and click **Properties**.

2. In the Field Properties dialog box, the Name box displays the selected Field string. Modify the existing string, select a new one, or type a new text string.

3. In the Value box, type the value you want displayed.

Note that the Value box is unavailable for system fields since the value is derived from your system.

4. To place the field at a precise X,Y coordinate location, type the value in the X and Y boxes.



Tip

If this is blank when you click **OK**, the field attaches to the pointer until you click to indicate the location.

5. In the Rotation list, select the degree of rotation you want.

Rotation can be 0 or 90 degrees.

6. In the Size box, type the font size you want.

Stroke font sizes must be between 10 and 1000 mils; system font sizes must be between 1 and 72 points.

7. For stroke font, in the Line Width box, type the line width.

8. For system fonts, select the font you want to use in the Font list.

You can also click a system font style you want applied: **B** for Bold, **I** for Italic, or **U** for Underlined.

9. To set the Justification, click the Horizontal and Vertical options you want.

10. Click **OK**.

Modifying Parts

Use the Part Properties dialog box to create and edit part attributes. You can also define signal pins and control the visibility of attributes assigned to the part.



Tip

This dialog box contains several sub-dialog boxes. Before using any of these sub-dialog boxes, changes made in open dialog box must be applied to the design.

- [Changing the Reference Designator](#)
- [Changing the Part Type](#)
- [Changing Part Information](#)
- [Changing the Visibility of Text](#)
- [Changing Part Attributes](#)
- [Changing PCB Decals](#)
- [Assigning Unused Pins as Signal Pins](#)

Changing the Reference Designator

If you need to reassign gates in your design, you can change the reference designator of a selected gate.

Procedure

1. Select a part, right-click and click **Properties**.
2. In the Reference Designator area, click **Rename Gate** to change the reference designator of the selected gate, type the new gate reference designator information and click **OK**.
3. Click **Rename Part** to change the reference designator of the selected part, type the new part reference designator information and click **OK**.

Changing the Part Type

Use the Change Part Type dialog box to change the selected part(s) to a new part type. The new part type can be one that already exists in the schematic or in the parts library.

Procedure

1. Click the **Tools > Options** menu item, then click the **Design** category.
2. Set the “Allow overwriting of attribute values in design with blank values from library” check box appropriately to allow or prevent overwriting of non-blank attribute values with blank (“placeholder”) values from the library.
3. Select a part or parts, right-click and click **Properties**.
4. In the Part Type area, click **Change Type** to change the selected part(s) to a new part type.

5. In the [Change Part Type Dialog Box](#), use the Filter to limit your search results to a chosen library (or libraries) for a specific part or item name, or names that match a [wildcard or expression](#) on page 105. Click **Apply** to search the libraries and display the search results.
 6. Use the Library dropdown list to select specific library directories or the All Libraries setting.
 7. Type * to view all parts or items in the chosen libraries.
 8. Scroll through the Parts list to find a part type.
-



Tip

When examining the parts list, note the following:

- The Parts list box displays the parts that matched the search filter settings.
 - The decal of a selected part displays in the preview area to the left of the list box.
-

9. Set how attributes are updated in the Attributes area.
 10. Set how parts are updated in the “Apply update to” area.
-



Tip

When updating parts, observe the following:

- You can update the part definition in the schematic with a modified version in the library. Select the same part name, then click All Parts This Type in the “Apply Update to” Area.
 - If you change a part type to one with fewer pins, the connections going to the missing pins are not deleted. They are attached to automatically generated off-page symbols. You are notified of all disconnected pins.
-

Changing Part Information

As your design needs evolve, you can change part properties as required.

Procedure

1. Select a part, right-click and click **Properties**.
2. Click **Statistics** to display gate and connection information for the selected part. This information is displayed in the default text editor so you can save the contents to a file.
3. Select a gate decal name from the Gate Decal list to change the gate decal of the selected gate or part to one of the predefined alternate decals.

Changing the Visibility of Text

You may decide to retain but turn off the visibility of certain text objects in your design. An example would be to turn off the display of the tolerance values for a family of components such as resistors where they all are assigned a default value.

Procedure

1. Select a part, right-click and click **Properties**.
2. Click **Visibility** to change the visibility of associated text.

See also [Controlling Text Visibility for a Part](#).

Changing Part Attributes

As your design progresses, you may find it necessary to change or modify attributes. You can change the attributes by modifying the part properties.

Procedure

1. Select a part, right-click and click **Properties**.
2. Click **Attributes** to assign or modify part attributes.

See also [Modifying Part Attributes](#).

Changing PCB Decals

Use the PCB Decal Assignment dialog box to assign alternate PCB decals to schematic parts. Decal names are included in the netlist file to display the proper decal, or footprint, when the file is imported into the PCB design file. You can select a decal assigned as an alternate during part creation, override the decal with one from the library, or enter a name for an undefined decal you plan to create later in the PCB design.

Procedure

1. Select a part, right-click and click **Properties**.
2. Click **PCB Decals** to assign alternate PCB decals.
3. In the PCB Decal Assignment dialog box, in the Assigned in Schematic box, type a new decal name for a decal you plan to create later in the PCB design.



Tip

The Assigned in Schematic box displays the name of the currently selected decal as it is assigned to the schematic from the current library.

4. Select the “No specific PCB Decal” check box to remove the assigned decal.
The default decal assigned to the part type is used when the netlist is imported to SailWind Layout; no decal assignment appears in the netlist.
5. In the Alternates in Library list, select a decal from the current part type definition in the library.
6. Click **Assign** to assign the decal to the part.



Tip

The preview window displays the currently selected decal.

7. Click the **Browse** button to search a library for a decal.

See also [Searching the Library for a Decal](#).

8. Set how parts are updated in the Apply update to area.

- **This Part** — Only updates the selected part.
- **All Parts This Type** — Updates all matching parts in the design with the new decal.

9. Click **OK**.

Assigning Unused Pins as Signal Pins

Use the Part Signal Pins dialog box to assign any unused pins as additional signal pins. When the part type is created and stored in the library, the standard power and ground pins for part types are defined.

Signal pins assigned during part creation cannot be modified through this dialog box. Instead, use the Assigning Signal Pin Names to Parts dialog box in the Library Manager.

Procedure

1. Select a part, right-click and click **Properties**.

2. Click **Sig Pins**.

The Part Signal Pins dialog box lists pins and their corresponding signal names. A signal pin is a pin that has a signal net (GND for example) assigned by a schematic capture program during part type creation.

See also [“Part Information - Pins” on page 147](#).

3. In the Part Signal Pins dialog box, select a pin in the Unused Pins list.

4. Click **Add** to add the unused pin to the Signal Pins list.

5. Assign a signal name in the Signal Pins area.

6. To modify the signal name of the pin, select the pin in the Signal Pins list and click **Edit**.

7. To move the pin to the Unused Pins list, select the pin in the Signal Pins list and then click **Remove**.

8. Set the update settings in the Apply update to area.

- **This Part** — Updates the selected part only.
- **All Parts This Type** — Updates all parts of the same type.

9. Click **OK**.

Reference Designator Renumbering

Manually renumber single or automatically renumber reference designators in the schematic. SailWind Layout can renumber the reference designators automatically in a pattern, to optimize finding your parts on a manufactured printed circuit board. Once the parts are renumbered in SailWind Layout, you can synchronize the designs again by back annotating an .eco file of the changed designators.

To manually renumber a reference designator in SailWind Logic, see [Changing the Reference Designator](#).

To automatically renumber reference designators in SailWind Logic, see [“Automatically Renumbering Reference Designators”](#) on page 280.

To automatically renumber the reference designators in SailWind Layout, see [Changing the Reference Designators of Multiple Components in ECO Mode \(Autorenumbering\)](#) in the *SailWind Layout Guide*.

Automatically Renumbering Reference Designators

Renumber reference designators on a sheet by sheet basis with control over the starting value, increment, and pattern.

Procedure

1. On the Schematic Editing toolbar, click the **Auto Rerumber Parts** button
2. In the “[Auto Rerumber Parts Dialog Box](#)” on page 473, select the sheets to renumber.
3. Choose which reference designator prefixes to renumber.
4. If needed, set the Cell size to smaller cells. See the example below.
5. Set the Rerumber settings for your situation.
6. Set the Precedence pattern.
7. Click **OK**.

An ECO file is automatically generated. If the Output window is open, a link to the file appears in the log. If the output window is closed, the ECO file opens in your default text editor.

8. If the board layout has already been started in SailWind Layout, you must import the ECO file to update the reference designators in SailWind Layout. This exact file must be used. Do not overwrite this file with a new one or generate an ECO file using the Compare/ECO Tools until SailWind Layout has been updated with the renumbering. In SailWind Layout, click the **File > Import** menu item. Browse for the .eco file and import the changes.

Examples

The following graphic displays the result of a smaller cell size on the parts in the schematic. Cell borders have been drawn to illustrate how it works. The parts within the cells are renumbered in the Precedence pattern and then the cells also are processed in the same pattern.

Figure 5. Cell Size Illustration



Setting Reference Designators by Sheet in a New Schematic

Set reference designator start values by sheet. If you will have multiple channels, each occupying one schematic sheet, and you want the reference designators to have predetermined ranges you can set a start value for designated sheets.

Procedure

1. Click the **Setup > Sheets** menu item.
2. In the “[Sheets Dialog Box](#)” on page 717. Click the **Add** button and add all the sheets for the multiple channels.
3. Name the schematic sheets.
4. In the RefDes Start Value column, for each channel sheet, type the starting value for reference designators of components added or copied and pasted to the schematic sheet.

Setting Reference Designators by Sheet in Completed Schematics

Renumber reference designators in schematics that already have assigned reference designators.. If you will have multiple channels, each occupying one schematic sheet, and you want the reference designators to have predetermined ranges you can renumber them.

Prerequisites

- Determine which schematic sheets need to be renumbered and which reference designators should be renumbered on those sheets.

Procedure

1. On the Schematic Editing toolbar, click the **Auto Renumber Parts** button.
2. In the “Auto Renumber Parts Dialog Box” on page 473, select the sheets to renumber.
3. Choose which reference designator prefixes to renumber.
4. If needed, set the Cell size to smaller cells. See the example below.
5. Set a Start Value and Increment value.
6. Select the “Increment Start Value by Sheet” check box.

This sets the starting number to match the sheet number and not the order of the sheet in the list.

7. Set the Precedence pattern.
8. Click **OK**.

Examples

The following graphic displays the result of a smaller cell size on the parts in the schematic. Cell borders have been drawn to illustrate how it works. The parts within the cells are renumbered in the Precedence pattern and then the cells also are processed in the same pattern.

Figure 6. Cell Size Illustration



Modifying Part Attributes

Use the Part Attributes dialog box to assign or modify part attributes, which is information about the part such as manufacturer and cost.

Procedure

1. Select a part, right-click and click **Attributes**.
2. To add an attribute to a part on the schematic, click **Add**.
3. In the Name column type a name or click **Browse Lib Attr** to select an existing attribute name.
4. In the Value column, type a value for the attribute.
5. To edit a name or value in the Attributes list, select a name or value and click **Edit** or double click a name or value.
6. To delete an attribute, select a name or value and click **Delete**.

7. Set how parts are updated in the “Apply update to” area.

- **This Part** — Only updates the selected part.
 - **All Parts This Type** — Updates all matching parts in the design.
-



Tip

Adding or changing attributes in a part at the schematic level does not update the part type (in the library). Edit the part in the Part Editor to update the library part.

Related Topics

[Part Editor Operations](#)

[Attributes Overview](#)

Modifying Part Attribute Labels

Use the Attribute Properties dialog box to provide text and font settings for one or more part attribute labels.

Procedure

1. Select nothing, right-click and click **Select Documentation**.
 2. Select a part attribute label, right-click and click **Properties**.
 3. In the Attribute Properties dialog box, type a new value for the attribute.
-



Tip

If you selected multiple attribute labels, you cannot change the value.

4. In the Rotation box, select the degree of rotation from the Rotation list.
Rotation can be 0 or 90 degrees.
 5. In the Size box, type the size in mils for the height of the stroke font (from the top of the tallest character to the bottom of the lowest character), or in points for system fonts.
-



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

6. For stroke font, in the Line Width box, type the width of the line used to create the characters.
7. For system fonts, select the font you want to use.
You can also click a system font style you want applied: **B** for bold, **I** for Italic, or **U** for Underlined.

8. To set the Justification, click the Horizontal and Vertical options you want.
9. Click **OK**.

Modifying Part Type Labels

Use the Part Type Label Properties dialog box to provide text and font settings for one or more part type labels.

Procedure

1. Select nothing, right-click and click **Select Documentation**.
2. Select a part type label, right-click and click **Properties**.
3. In the Part Type Label Properties dialog box, click **Change Type** if you want to change the part type.
See also [Changing the Part Type](#).
4. In the Rotation box, select the degree of rotation from the Rotation list.
Rotation can be 0 or 90 degrees.
5. In the Size box, type the size in mils for the height of the stroke font (from the top of the tallest character to the bottom of the lowest character), or in points for system fonts.



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

6. For stroke font, in the Line Width box, type the width of the line used to create the characters.
7. For system fonts, select the font you want to use.
You can also click a system font style you want applied: **B** for Bold, **I** for Italic, or **U** for Underlined.
8. To set the Justification, click the Horizontal and Vertical options you want.
9. Click **OK**.

Searching the Library for a Decal

Use the Get Decal from Library dialog box to search a library for a decal. You can use the filters to quickly narrow down your search parameters to find the exact part that you require.

Procedure

1. Select a part, right-click, click **Properties**, and then click the **PCB Decals** button.
2. In the PCB Decal Assignment dialog box, click the **Browse** button.

3. In the Get Decal from Library dialog box, use Filter to limit your search results to a chosen library (or libraries) for a specific part or item name, or names that match a [wildcard or expression](#) on page 105. Click **Apply** to search the libraries and display the search results.
-



Tip

When searching, observe the following:

- Use the Library dropdown list to select specific library directories or the All Libraries setting.
 - Type a prefix plus an asterisk in the Items text box. For example, type DIP* to view all decals that begin with DIP. Type an asterisk only to view all decals in the selected libraries. Type only* to view all parts or items in the chosen libraries.
 - Enter a value in the Pin Count text box to narrow the search further. Pin Count grays if a decal is already defined for the selected part. Pin Count is on if the No Specific PCB Decal is on in the PCB Decal Assignment dialog box.
-

4. Scroll through the Decals list box to find and select a decal.
-



Tip

Decals are displayed as follows:

- The Decals list box displays the decals that matched the search filter settings.
 - The selected decal displays in the preview area to the left of the list box.
-

5. Click **OK** to exit the dialog box.

You return to the PCB Decal Assignment dialog box.

Rename Part

Use Rename Part to change the reference designator for the selected part or gate. All gates are renamed if you change the reference designator of one gate of a multi-gated part. You are prevented from assigning an already used reference designator.

Procedure

1. Select a part, right-click and click **Properties**.
2. In the Part Properties dialog box, click **Rename Part**.
An information window displays.
3. Type the new part reference designator information.
4. Click **OK**.

Rename Gate

Use Rename Gate to change the reference designator of the selected gate. You are prevented from assigning an already used reference designator or an unused gate of a part with a different part type.

Procedure

1. Select a part, right-click and click **Properties**.
2. In the Part Properties dialog box, click **Rename Gate**.
An information window displays.
3. Type the new gate reference designator information.
4. Click **OK**.

Modifying Reference Designator Labels

Use the Reference Designator Properties dialog box to view and modify label size and justification as well as text and font settings for one or more reference designator labels.

Procedure

1. Select nothing, right-click and click **Select Documentation**.
 2. Select a reference designator label, right-click and click **Properties**.
 3. In the Reference Designator Properties dialog box, in the Rename area, select Gate to rename just the single gate. Select Part to rename the entire part.
-



Tip

If you select a gate with only one part, Gate is unavailable.

4. In the Rotation box, select the degree of rotation from the Rotation list.
Rotation can be 0 or 90 degrees.
 5. In the Size box, type the size in mils for the height of the stroke font (from the top of the tallest character to the bottom of the lowest character), or in points for system fonts.
-



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

6. For stroke font, in the Line Width box, type the width of the line used to create the characters.
7. For system fonts, select the font you want to use.
You can also click a font style you want applied: **B** for Bold, **I** for Italic, or **U** for Underlined.
8. To set the Justification, click the Horizontal and Vertical options you want.
9. Click **OK**.

Modifying Pins

Use the Pin Properties dialog box to view pin information, to modify parts and nets to which the selected pin is connected, and also to set font settings for pin number and pin name labels.

Procedure

1. Select a pin, right-click and click **Properties**.
2. In the Pin Properties dialog box, click the **Part/Gate** button to change part or gate attribute information.
See also [Modifying Parts](#).
3. Click the **Net** button to modify the net name and attributes of a named net.



Tip

To modify the net name or attributes of a \$\$\$ named net, use the **Query Mode** button to select a net near a pin to change it from a \$\$\$ named net.

See also [Modifying Nets](#).

4. Click the **Font** button to open the Pin Label Fonts dialog box where you set font preferences.
See also [Modifying Pin Label Fonts](#).
5. Click **OK**.

Modifying Pin Label Fonts

To improve the legibility of your schematic, you can use the Pin Label Fonts dialog box to change the fonts of a pin label.

Procedure

1. Select a pin label, right-click and click **Properties**.
2. In the Pin area of the Pin Label Fonts dialog box, type the line width in the Line Width box for stroke font.
3. For stroke or system fonts, in the Size box, type the size (in mils for stroke font, in points for system fonts).



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

4. For system fonts, select the font you want to use.
You can also click a font style if you want one: **B** for bold, **I** for italic, or **U** for underlined.
5. Click **OK**.

Modifying Nets

To rename a net or edit its attributes and values, you can use the Net Properties dialog box to access net specific properties.

Procedure

1. Select a net, right-click and click **Properties**.
-



Tip

Select the connection segment where it enters or exits a part. Selecting any other segment of the connection will not allow net name labels to be added.

2. Type a new name in the Net Name box or select an existing name from the list.
-



Tip

When working with net names, observe the following:

- If you select a name from the list, the message “Net <netname> already exists - OK to combine nets (Y/N)?” appears. Click **Yes** to combine the nets, or click **No** to enter a different net name.
 - If you delete all visible net name labels, the net and all subnets retain the net name. Net names do not revert to a generated \$\$\$nnnn system number. You must rename the net.
 - If you attempt to combine a named and an unnamed net, the resulting net will always take the name of the named net. Any similar attribute names will always take their values from the named net.
 - The Connectivity Report identifies subnets tied together without a visible net name by flagging them as missing an off-page symbol.
-

3. In the Rename area, click **This Instance** to apply the new net name to the selected connection or click **All Instances** to apply the new net name to all instances of this connection.
-



Tip

Selecting **This Instance** will cause the net to be split into two separate nets.

4. Select the Net Name Label check box to add a visible label to the selected net segment.

You must select the connection segment where it enters or exits a part, or the Net Name Label check box will be unavailable.



Tip

Clearing the Net Name Label check box removes the visible label. The net and all subnets retain the net name.

5. Click **Statistics** to display connection information in the default text editor.

6. Click **Attributes** to open the Net Attributes dialog box and modify the net attributes.

See also [Creating Net Attributes](#).

7. Click **Rules** to open the Net Rules dialog box and modify the net rules.

See also [Setting Up Net Rules](#).

8. Click **OK**.

The new net name is placed over the connection segment.



Tip

Use [Using Move Mode](#) to adjust the placement of the net name text.

Modifying Net Name Labels

Use the Net Name Properties dialog box to provide or change text and font settings for one or more net name labels.

Procedure

1. Select nothing, right-click and click **Select Documentation**.
 2. Select a net name label, right-click and click **Properties**.
 3. Type a new name in the Net Name box or select an existing name from the list.
-



Tip

When working with net names, observe the following:

- If you select a name from the list, the message “Net <netname> already exists - OK to combine nets (Y/N)?” appears. Click **Yes** to combine the nets, or click **No** to enter a different net name.
 - If you delete all visible net name labels, the net and all subnets retain the net name. Net names do not revert to a generated \$\$\$nnnn system number. You must rename the net.
 - If you attempt to combine a named and an unnamed net, the resulting net will always take the name of the named net. Any similar attribute names will always take their values from the named net.
 - The Connectivity Report identifies subnets tied together without a visible net name by flagging them as Subnets with Missing Net Name.
-

4. In the Rename area, click **This Instance** to apply the new net name to the selected connection or click **All Instances** to apply the new net name to all instances of this connection.
-



Tip

Selecting **This Instance** will cause the current segment of the net to take the new net name and be split from the other instances.

5. In the Rotation box, select the degree of rotation from the Rotation list.

Rotation can be 0 or 90 degrees.

6. In the Size box, type the size in mils for the height of the stroke font (from the top of the tallest character to the bottom of the lowest character), or in points for system fonts.
-



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

7. For stroke font, in the Line Width box, type the width of the line used to create the characters.

8. For system fonts, select the font you want to use.

You can also click a system font style you want applied: **B** for Bold, **I** for Italic, or **U** for Underlined.

9. To set the Justification, click the Horizontal and Vertical options you want.

10. Click **OK**.

Modify Buses

Use the Bus Properties dialog box to change the name of a bus or change the bus type.

[Changing the Name of a Bus](#)

[Changing the Bus Type](#)

[Managing Bus Nets](#)

Changing the Name of a Bus

As your design evolves, you may need to add or delete nets to or from a bus. Once you have made these modifications, you can change the name of a bus to reflect those changes.

Procedure

1. Select a bus, right-click and click **Properties**.
2. Type a new name for the bus in the Bus Name box or select an existing name from the list.

See also [Naming Buses](#).



Tip

If you change the bit range for a bit format bus, the new bit range nn:mm must be equal to or greater than the old bit range. The signals attached to the bus will keep their old bit number, but take the new name.

3. In the Rename area, click This instance to apply the new bus name to the selected bus, or click All instances to apply the new bus name to all instances of this bus.
4. Select the Bus Name Label check box to add a visible label to the selected bus.

Changing the Bus Type

As your design evolves, you may find it necessary to change the type of a bus from a bit format bus to a mixed net bus or the other way around. You can use Properties to change the bus type.

Procedure

1. Select a bus, right-click and click **Properties**.
2. In the Bus Type area, click the bus type you want.

The Bus Type determines which bus names appear in the Bus Name list.



Tip

If you change the bus type from bit format to mixed net bus, the bus name changes to just the prefix. The bus name prefix and bit range are added to the Bus Nets list box.



Restriction:

You can change a mixed net bus to a bit format bus only if all of the connected nets conform to the bit sequenced names, which you define in the Bus Name text box. Before you can change a mixed net bus to a bit format bus, you must delete the bus nets from the Bus Nets list box.

Managing Bus Nets

You may find it necessary to expand the range of a bus to include additional sequential nets. Use Properties to add sequential nets to a bus.

Restrictions and Limitations

The Bus Nets area is not available when the Bit Format bus type is selected.

Procedure

1. Select a bus, right-click and click **Properties**.
 2. Click **Add** to add a new row in the Bus Nets list.
 3. Type the bus prefix in the Name/Prefix column.
-



Tip

Use the following naming conventions:

- For a single net, type the net name.
 - For a range of sequential nets, type the prefix for the sequence of nets.
-

4. In the Start column, type the starting bit number for a sequence of nets.
 5. In the End column, type the ending bit number for a sequence of nets.
-



Tip

To add an individual net to the bus, type the net name in the Prefix/Name column. Do not type Start and End values for a single net that is not sequenced by bit number.

6. Click **Edit** to modify a selected cell in the Bus Nets list.
 7. Click **Delete** to remove a net from the Bus nets list.
 8. Click **Down** to move the selected row down one position in the Bus Nets list.
-



Tip

The order in which nets appear in the Bus Nets list determines the default order for naming nets as they are connected to a bus.

9. Click **Up** to move the selected row up one position in the Bus Nets list.

Related Topics

[Managing Buses](#)

[Modifying Bus Name Labels](#)

Modifying Bus Name Labels

Use the Bus Name Properties dialog box to provide or change text and font settings for one or more bus name labels.

Procedure

1. With nothing selected, right-click and click **Select Documentation**.
 2. Select a bus name label, right-click and click **Properties**.
 3. In the Bus Name Properties dialog box, click **Bus** to open the Bus Properties dialog Box.
See also [Modify Buses](#).
 4. In the Rotation box, select the degree of rotation from the Rotation list.
Rotation can be 0 or 90 degrees.
 5. In the Size box, type the size in mils for the height of the stroke font (from the top of the tallest character to the bottom of the lowest character), or in points for system fonts.
-



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

6. For stroke font, in the Line Width box, type the width of the line used to create the characters.
 7. For system fonts, select the font you want to use.
You can also click a system font style you want applied: **B** for Bold, **I** for Italic, or **U** for Underlined.
 8. To set the Justification, click the Horizontal and Vertical options you want.
 9. Click **OK**.
-



Tip

Font properties you specify in the Net Name Label Properties dialog box are also applied to the off-page reference label.

Modifying Off-Page Labels

Use the Off Page Properties dialog box to set the maximum sheet numbers per line and the rotation settings for off page labels.

Procedure

1. Select an off-page label, right-click and click **Properties**.
2. In the [Off-Page Properties Dialog Box](#), type the maximum sheet numbers allowed per line, from 0 to 99.
3. In the Rotation box, select the degree of rotation from the Rotation list.
Rotation can be 0 or 90 degrees.
4. Click **OK**.

Modifying Label Font Sizes

New labels are created using a system default font size. You can't change this default, but you can modify the font sizes of labels after they are created.

Procedure

1. Select the labels you want to change, right-click and click **Properties**.



Restriction:

All the labels selected must be of the same type; for example, all net name labels or all reference designator labels.

2. In the Properties dialog box, enter the new Size value.



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

3. Click **OK**.

Table 32. Related Topics

Attribute Properties Dialog Box	Part Type Label Properties Dialog Box
Bus Name Properties Dialog Box	Pin Label Fonts Dialog Box
Net Name Properties Dialog Box	Reference Designator Properties Dialog Box

Modifying Text

Use the Text Properties dialog box to modify free text or change its size, orientation or justification.

Procedure

1. Select text, right-click and click **Properties**.
2. In the Text Properties dialog box, the Text box displays the selected text string. Modify the existing text string, or type a new text string.
3. In the X,Y location boxes, type new values to move the text string to a specified location.
If you need to move text outside the sheet, you must move the text with the cursor instead.
4. In the Rotation box, select the degree of rotation from the Rotation list.
Rotation can be 0 or 90 degrees.
5. For stroke font, in the Line Width box, type the line width.
6. In the Size box, type the size (in mils for stroke font, in points for system fonts).



Tip

Type a stroke font size between 10 and 1000 mils or a system font size between 1 and 72 points.

-
7. For system fonts, select the font you want to use.
You can also click a font style: **B** for bold, **I** for Italic, or **U** for Underlined.
 8. In the Justification area, set the horizontal and vertical justification of the text.
 9. If you combined the selected text string with a drafting object, click **Parent** to display the Drafting Objects Properties dialog box to modify the drafting object.
 10. Click **OK**.

Related Topics

[Modifying Drafting Objects](#)

[Combining 2D Lines and Text](#)

Modifying Hierarchical Components

When created and added to an existing sheet, hierarchical components cannot be accessed through the Sheet list in the toolbar unless the parent sheet is displayed. They are also excluded from the Sheet command in the **Setup** menu. Use the Hierarchical Component Properties dialog box to assign a hierarchical component to the next available sheet number to make it accessible from the Sheet list when a sheet other than the parent sheet is displayed.

Use the Sheets command under the **Setup** menu to modify the sheet name or the numeric order.

Procedure

1. Select a hierarchical component, right-click and click **Properties**.
2. In the Query Hierarchical Component dialog box, type a new name in the Name box to rename the hierarchical component.
3. Select the Visibility check box to the right of the Name box to display the name on top of the hierarchical component in the schematic.
4. Click Numbered to assign the hierarchical component the next available sheet number.
The assigned sheet number is displayed in the field above this option.
5. Click Un-numbered to remove a sheet number assignment from a hierarchical component.
6. In the Associated Sheet area, select the Visibility check box to display the sheet number in the schematic.
7. Click **OK**.

Chapter 16

Rules

As your design complexity increases, you can use design rules to control the interaction of various design objects with one another. Use design rules to specify clearances between design objects such as pads, coppers and traces. You can also specify design rules for nets, groups of nets, classes, and differential pairs. Use the Rules Report to review your settings. You can also import and export rules between your schematics and your PCB designs.

[Rules Setup](#)
[Setting Up Clearance Rules](#)
[Same Net Matrix](#)
[Routing Rules](#)
[Setting Up High-Speed Rules](#)
[Setting Up Rules](#)
[Rules Hierarchy](#)
[Setting Up Default Rules](#)
[Setting Up Class Rules](#)
[Setting Up Net Rules](#)
[Setting Up Conditional Rules](#)
[Creating Differential Pairs](#)
[Differential Pair Layer Hierarchy](#)
[Creating a Rules Report](#)
[Import Rules from PCB](#)
[Export Rules to PCB](#)

Rules Setup

Design Rules enable you to assign general routing constraints for your design, such as trace width and spacing. An option in the netlist lets you pass established design rules along with the connectivity and parts information to SailWind Layout.

To access and examine the Design Rules setup, click the **Setup > Design Rules** menu item.

Rule Categories

Design rules are separated into three categories:

- [Setting Up Clearance Rules](#) — Specifies minimum allowable airgap between various object types in the design; trace to trace, via to trace, etc.
- [Routing Rules](#) — Assigns or prohibits via types, specifies length minimization types, or allows or prohibits routing.
- [Setting Up High-Speed Rules](#) — Specifies minimum and maximum parameters for advanced design rules such as parallelism, delay, capacitance, and others.

Information for each category is entered and edited in a separate dialog box that you can access through the Design Rules command from the **Setup** menu.

Setting Up Clearance Rules

Use the Clearance Rules dialog box to define the spacing permitted between objects. When objects are imported, the On-line DRC and Verify Design programs use these rules to check and report clearance violations.

**Tip**

You can use the [Setting Up Conditional Rules](#) dialog box to save a clearance configuration as a set, to apply to a selected item only when it comes in contact with different level of the [hierarchical order](#) on page 305.

Procedure

1. Click the **Setup > Design Rules** menu item, click a rule hierarchy, then click the **Clearance** button.
2. In the Same Net area, define edge-to-edge clearance values between items that are in the same net:
 - To define the minimum spacing between any two objects, type the value in the appropriate text box.
 - To define the same spacing value for all text boxes in one matrix column, press the button above the column and type a value.
 - To define the same spacing value for all text boxes in one matrix row, press the button in the left column and type a value.
 - To define the same spacing value for all text boxes in the matrix, press the **All** button and type a value.

Table 33. Clearance Rules - Same Net - Edge-to-Edge Clearance Options

Clearance	Description
SMD to Via	Minimum spacing between a surface mount pad and escape via.
SMD to Corner	Minimum spacing between a surface mount pad and the first trace bend point.
Via to Via	Minimum spacing between two vias in the same net.
Pad to Corner	Minimum spacing between a through hole pad and the first trace bend point.
Trace to Corner	Minimum spacing between a trace and the bend point of another trace; for example, when a trace splits at a T-junction and one of the two traces has a bend point.

3. In the Trace Width area, type values in the text boxes to restrict the trace width to a range of values:
 - In the Recommended box, type the width you want to assign to the trace when routing begins.
 - In the Minimum and Maximum boxes, values are respected by routing routines that must use trace width to achieve some high-speed routing functions, such as impedance matching.
4. In the Clearance area, use the Clearance matrix to define edge-to-edge clearances between two object types:
 - To define the minimum spacing between any two objects, type the value in the appropriate text box.
 - To define the same spacing value for all text boxes in one matrix column, press the button above the column and type a value.
 - To define the same spacing value for all text boxes in one matrix row, press the button in the left column and type a value.
 - To define the same spacing value for all text boxes in the matrix, press the **All** button and type a value.

5. In the Other area, optionally set other clearance values, which include:

Table 34. Clearance Rules - Other - Clearance Options

Clearance	Description
Drill to Drill	The minimum edge-to-edge spacing between two drill holes.
Body to Body	The minimum edge-to-edge spacing between two component bodies.

6. Click **OK**, or optionally click **Delete** to remove this set of Clearance rules from your rules hierarchy.



Tip

You cannot delete the Default Clearance rules.

Same Net Matrix

Use the Same Net Matrix to define edge-to-edge clearance values between items that are in the same net.

- To define the minimum spacing between any two objects, type the value in the appropriate text box.
- To define the same spacing value for all text boxes in one matrix column, press the button above the column. Type a value and click **OK**.

- To define the same spacing value for all text boxes in one matrix row, press the button in the left column. Type a value and click **OK**.
- To define the same spacing value for all text boxes in the matrix, press the **All** button. Type a value and click **OK**.

Table 35. Same Net Matrix - Clearance Options

Clearance	Description
SMD to Via	Minimum spacing between a surface mount pad and escape via.
SMD to Corner	Minimum spacing between a surface mount pad and the first trace bend point.
Via to Via	Minimum spacing between two vias in the same net.
Pad to Corner	Minimum spacing between a through hole pad and the first trace bend point.
Trace to Corner	Minimum spacing between a trace and the bend point of another trace; for example, when a trace splits at a T-junction and one of the two traces has a bend point.

Routing Rules

Use the Routing Rules dialog box to specify rules for interactive and automatic routing. You can specify the default set of routing rules and routing rules for specific nets.

Setting Up High-Speed Rules

Use the Hi Speed Rules dialog box to define rules for Parallelism, Tandem, Shielding, Routed Length, Stub Length, Delay, Capacitance, Impedance, and Matched Length.



Tip

When working with high-speed rules, observe the following:

- When imported into SailWind, the EDC (Electrodynamic Checking) routine checks to see if rules are met correctly after routing (except shielding and matched length).
- You can use the [Setting Up Conditional Rules](#) dialog box to save a high speed configuration as a set, or to apply for a selected item only when it comes in contact with different level of the [hierarchical order](#) on page 305.

Procedure

1. Click the **Setup > Design Rules** menu item, click a rule level, then click the **Hi Speed** button.
2. In the Parallelism area, type a value for length and gap in the parallelism boxes to restrict the distance that traces in different nets on the same layer can run together.

3. Type a value for length and gap in the tandem boxes to restrict the distance that traces in different nets on different layers can run together.

Table 36. High-Speed Rule Options

Option	Description
Parallelism	Restricts the distance that traces in different nets on the same layer can run together.
Tandem	Restricts the distance that traces in different nets on different layers can run together.

**Tip**

When configuring High-Speed Rule Options, observe these guidelines:

- These values determine the distance and standoff against the specific item and everything else in the design. To set a narrower check, use [Setting Up Conditional Rules](#) to define specific Source and Against items.
- Length defines the maximum allowable parallel/tandem distance.
- Gap defines the minimum gap between traces below which the parallel/tandem rules apply.

4. Click Aggressor to specify if a net is an aggressor, or source of interference, during parallel/tandem checks.

5. In the Rules area, type minimum and maximum values for:

Table 37. High-Speed Rules - Minimum/Maximum Values

Rule	Description
Length	Defines a minimum and maximum length.
Stub Length	Specifies a maximum stub length. The stub length is the distance from a T-point to the end of the route.
Delay	Defines a minimum and maximum delay time in nanoseconds.
Capacitance	Defines a minimum and maximum capacitance in picofarads.
Impedance	Defines a minimum and maximum impedance in ohms.

**Tip**

These text boxes restrict the trace width to a range of values. Recommended is the width you want to assign to the trace when routing begins. The Minimum and Maximum values are respected by routing routines which must use trace width to achieve some high-speed routing functions, such as impedance matching.

6. Some routers can arrange certain nets as shielding others if requested; the Net in the Use Net list box is routed up and down both sides of a selected net to provide protection from interference. In the Shielding area, click **Shield** to invoke the shielding rules.



Tip

When working with shielding, observe the following guidelines:

- You can only assign nets associated with plane layers in the Layer Definition dialog box to shield other nets. If there are no plane layers, the Shield area is grayed out.
 - If your router supports shielding, you can specify the shield gap value and net to use as the shield in the Gap and Use Net text boxes.
-

7. (Optional) In the Matching area, select the Match Length check box to invoke the matching rule, and type a tolerance value.
-



Tip

When working with Length Matching, observe the following:

- Length Matching is a same length requirement parameter you can pass on to autorouters that support it.
 - This rule specifies the maximum difference allowed between the shortest length and longest length in the matched length group.
-

8. Click **OK**, or click **Delete** to remove this set of High Speed rules from your rules hierarchy.
-



Tip

You cannot delete the Default High Speed rules.

Setting Up Rules

Use the Rules dialog box to enter item-to-item Clearance rules, routing guidelines, and values for the optional High Speed checking commands. You can also indicate the unit of measure for passing rules to SailWind Logic: mils, metric, or inches.

Procedure

1. Click the **Setup > Design Rules** menu item.
 2. In the Units list box, select Mils, Metric, or Inches.
 3. Select the rules hierarchy level for which you want to set or modify rules.
-



Tip

When you specify one of the hierarchical levels, you can access the Clearance, Routing, or High Speed forms.

4. Click the appropriate button to define rules for a particular level in the hierarchy.
 5. (Optional) Click the **Report** button to access the Rules Report dialog box where you can select a default report of a report for some or all of the rules you have defined.
-

Rules Hierarchy

In the rules hierarchy, certain rules have precedence over other rules. For example, a net rule overrides a class rule, and a class rule overrides a default rule.

Table 38. Rule Hierarchy and Order of Precedence

Rules	Precedence	Description
Setting Up Default Rules	Least	Rules that apply to an object if there are no other individually defined rules.
Setting Up Class Rules		Rules for a collection of nets, called a class, which needs identical rules.
Setting Up Net Rules	Highest	Rules for a specific net.

Rules Hierarchy Order of Precedence

The complete order of precedence for all rules follows, from least to most specific. Default, 1, represents the lowest level of the hierarchy with the least amount of precedence. At the opposite end of the order is Net against Net with Level, which is the highest level of the hierarchy and has the highest possible precedence. It represents the most specific rule you can assign to an object in SailWind Logic.

1. Default
2. Default with Level
3. Class
4. Class with Level
5. Net
6. Net with Level
7. Class against Class
8. Class against Class with Level
9. Net against Class
10. Net against Class with Level
11. Net against Net
12. Net against Net with Level

See also [Setting Up Default Rules](#), [Setting Up Class Rules](#), [Setting Up Net Rules](#), [Creating a Rules Report](#).

Setting Up Default Rules

Use the Default Rules dialog box to define rules which apply to all objects that are not included in any other rule definitions within the hierarchy.

Procedure

1. Click the **Setup > Design Rules** menu item, then click the **Default** button.
2. To define default [Clearance](#) on page 300, [Routing](#) on page 302, or [High Speed](#) on page 302 rules, click the appropriate button.
3. Click the [Report](#) on page 310 button to produce a rules report.

Setting Up Class Rules

Use the Class Rules dialog box to define rules that apply to a collection of nets known as a net class and to multiple net classes.

Procedure

1. Click the **Setup > Design Rules** menu item, then click the **Class** button.
2. In the Class Name list box, select a class name, or specify a new name for which you want to apply rules.



Tip

The Class list box defines net classes by name and parenthetically notes the rules that apply, if any, to the class.

3. In the Nets area, add or remove selected nets from the net class as follows:
 - Select a class in the list box to display nets in the Available and Selected list boxes, or click the **Add** button to create a new class.
 - To assign nets to a class, select the net(s) in the Available list box and click **Add**.



Tip

A net can only belong to one net class. The Available list box only contains nets that do not belong to a net class.

4. To delete nets from the class, select the net(s) in the Selected list box and click **Remove**.
5. (Optional) Select a class name and click **Rename** to change the class name for the selected class.
6. (Optional) Select the Show Classes with Rules check box to list only classes with at least one set of rule definitions.

-
7. For each class, click the appropriate button to define Clearance, Routing, or High Speed rules for a net class.

**Tip**

When you select a class name, an icon appears below each rule type to indicate the hierarchy level where the rule is defined for that class. For example, a class with only Clearance rules defined would have a **Class** icon below the **Clearance** icon and **Default** icons below the **Routing** and **High Speed** icons.

8. In the corresponding Rules dialog box that displays, supply values for the rules you want to apply, and click **OK** to return to the Class Rules dialog box.
9. When you have finished defining all class rules, click **OK**.

Setting Up Net Rules

Use the Net Rules dialog box to define rules that apply to a single net or multiple nets.

Procedure

1. Click the **Setup > Design Rules** menu item, the click the **Net** button.
2. Select the net(s) for which you want to define [Clearance](#) on page 300, [Routing](#) on page 302, and [High Speed](#) on page 302 rules.

Tip

When you select a net name, an icon appears below each rule type to indicate the hierarchy level where the rule is defined for that net. For example, a net with both Clearance and High Speed rules defined would have **Net** icons below the **Clearance** and **Hi Speed** icons and a **Default** icon below the Routing icon.

3. (Optional) Select the **Show Nets with Rules** check box to list only those nets with at least one set of rule definitions.
4. (Optional) Click the **Report** button to view a rules report.
5. Click the **Default** button to restore the net rules back to the default rules.

Setting Up Conditional Rules

Use the Conditional Rule Setup dialog box to apply a third overriding set of rules that apply only when the item meets other specific levels of the hierarchical order.

Once you set up Clearance rules for a group in the [hierarchical order](#) on page 305, the rules are applied to all other objects.



Tip

When working with conditional rules, use the following guidelines:

- You can use a layer as an against object, where rules you set for an object such as a net apply only when the net is routed on that layer.
 - You can further refine this to use another net as an against object and specify a layer to which the rules to apply. If these two nets meet on this layer, they must adhere to this clearance. You define these relationships by making conditional rule sets.
-

Procedure

1. Click the **Setup > Design Rules** menu item, then click the **Conditional Rules** button.
 2. In the Source Rule Object area, select Classes, Nets, or All to specify both classes and nets. Then specify the object(s) against which the rule is checked.
 3. In the Against Rule Object area, select Classes, Nets, or Layer, and then specify the object(s) against which the rule is checked.
-



Tip

Select Layer to use a layer as an against object or to apply an item-to-item rule on a specific layer.

4. (Optional) In the Define Conditional Objects area, in the Apply to Layer area, select a layer on which you want rules checked.
5. In the Existing Rule Sets list box, select the rule set to define the appropriate values for specific Clearance or High Speed definition of the selected rule set.
6. After you create a rule set, in the Current Rule Set area, define the Clearance or High Speed values.
7. Select Clearance to enter the minimum clearance gap you want the source and against items to maintain from each other, and then click **Matrix** to enter more item-specific standoffs.
8. Select High Speed to enter clearance values for [parallel and tandem checking](#) on page 302 for the condition. The source-against entries are used as the victim-aggressor designations for crosstalk conditions checking.
9. Click **Create** to compile the new rule set parameters and adds a description to the Existing Rule Sets list box, or click **Delete** to removes the selected rule set from the Existing Rule Sets list box.

Related Topics

[Setting Up Clearance Rules](#)

[Setting Up High-Speed Rules](#)

Creating Differential Pairs

Identify nets or pin pairs that behave electrically as differential pairs and define rules for them. You can set different properties for differential pairs, which affects how they are routed. Differential pair properties

determine the gap between the traces in the Controlled Gap Area, the minimum and maximum trace lengths, and how to respond to Obstacles in the controlled gap area.

Restrictions and Limitations

While you can define these rules in SailWind Logic, they are used only in SailWind Router.

Procedure

1. Click the **Setup > Design Rules** menu item, the click the **Differential Pairs** button.
2. In the Available list, double-click the first net, and then double-click the second net.



Tip

Nets cannot exist in more than one differential pair. The Available list displays only nets that have not been assigned to a differential pair.

3. Click **Add**.
4. Type minimum trace length value in the Minimum box.
5. Type the maximum trace length value in the Maximum box.
6. In the Properties of the pair area, type Width and Gap values in the <All layers> row.



Restriction:

You cannot delete the <All layers> row.

7. To set the width and gap per layer, click **Add**, click in the Layer cell in the newly added row, and select the layer for which to set width and gap values. Then type Width and Gap values in the appropriate cells.



Note:

When working with differential pairs, observe the following guidelines:

- Setting the differential pair width and gap per layer enables you to better control impedance.
- The gap rule cancels any other rule defining a clearance between the differential pairs. Therefore, the gap is the minimal clearance and must be provided when possible.
- If you select multiple differential pairs, and a layer setting doesn't belong to all of the selected pairs, the Layer column will be unavailable.
- If you select multiple differential pairs that have the same layer setting, but the Width and Gap values do not match, the Width and Gap cells will appear empty. You can, however, type a new value, and the new will be applied to all selected differential pairs when you click **OK** or **Apply**.

See also [Differential Pair Layer Hierarchy](#).

8. Click Restrict layer changes during autorouting to force the pair to be routed on a single layer.
This setting does not restrict layer changes when routing interactively.
9. Click Allow pair to split around obstacles to allow routing around an obstacle in the controlled gap area by temporarily exceeding the pair routing gap.

This setting applies to autorouting and does not restrict splitting around obstacles when routing interactively.

10. Type the maximum number of obstacles to route around in the Maximum number of obstacles box.
-



Tip

Obstacles in the [start zone](#) or [end zone](#) are not counted.

11. Type the maximum spacing allowed between traces around obstacles in the Maximum obstacle size box.

The size applies to the obstacle's longest horizontal or vertical dimension.



Tip

Obstacle size in the start zone or end zone is not checked.

12. Click **OK**.

Differential Pair Layer Hierarchy

You can assign differential pair width and gap values to layers and categories of layers; however, a layer may also fall into one or more categories. For example, Layer 2 may also be a plane layer, and an outer layer.

Therefore, the following hierarchy is followed to define which layer settings take priority:

1. All Layers
 2. Plane Layers
 3. Outer Layers
 4. Inner Layers
 5. Individual Layers
-



Tip

Individual Layers has highest priority.

Creating a Rules Report

Use the Rules Report dialog box to produce a report of some or all of the rules you have defined. By default, a complete report of all rules is reported.

Procedure

Click the **Setup > Design Rules** menu item, the click the **Report** button. You can produce reports for the following:

Table 39. Rules Report Types

Report Type	Description
Rule types	Displays the specified rules for the specified nets and classes. Click any combination of buttons, including Differential Pairs, to report net pairs.
Nets	Displays the specified rules for every net or selected nets. Click All Nets or select specific nets in the list box.
Classes	Displays the specified rules for every class or selected classes. Click All Classes or select specific net classes in the list box.
Output	Click Rule Sets to display all rules in the current hierarchy that are unique from the default rules. Click Rule Values to display all rules in the current hierarchy level, even if the values are the same as the default rules.
Default Rules	Displays the default rules for the specified nets and classes.

Import Rules from PCB

Use the **Design** Tab of SailWind Layout Link to read design rules from a specified SailWind ASCII file, including layer setups and layer counts, etc.

Procedure

1. Click the **Tools > SailWind Layout** menu item.
2. Click the **Design** tab.
3. On the **Design** tab, select the **Compare Design Rules** check box.
4. Click **ECO from PCB**.

The netlist in the current SailWind Logic schematic is compared to the part and netlist in the current SailWind Layout design and the schematic is updated.

Export Rules to PCB

Use the **Design** Tab of SailWind Layout Link to export design rules to a specified SailWind ASCII file, including layer setups and layer counts, etc.

Procedure

1. Click the **Tools > SailWind Layout** menu item.
2. Click the **Design** tab.

3. On the **Design** tab, select the Compare Design Rules check box.

4. Click **ECO to PCB**.

The netlist in the current SailWind Logic schematic is compared to the part and netlist in the current SailWind Layout design and the layout design is updated.

Chapter 17

Reports

You can generate reports to view and analyze data related to your design. Available reports include unused pins, parts statistics, net statistics, limits, connectivity, and a bill of materials. Use these reports to examine and interpret your design status at any point in the design cycle.

[Generating Reports](#)

[The Bill of Materials Report](#)

Generating Reports

Use the Reports dialog box to produce any of six different types of reports on the current schematic. You can save these reports as text files on your hard disk or output them to a printer.

Procedure

1. Click the **File > Reports** menu item.
2. In the [Reports Dialog Box](#), select each report you want to generate.
3. To specify or modify the report characteristics of the Bill of Materials report, click **Setup**.

See also [The Bill of Materials Report](#).

4. Click **OK**.

The reports are generated and the dialog box closes.

The Bill of Materials Report

The Bill of Materials report produces a user-configurable list of the parts contained in the current schematic. You can direct any part attribute in the schematic to a Bill of Materials report.

In the [Bill of Materials Setup dialog box](#) on page 479 are three different tabs for controlling report characteristics from which to choose:

- [Setting Up the Bill of Materials Attributes](#)
- [Setting Up the Bill of Materials Format](#)
- [Setting Up the Bill of Materials Configuration](#)

Setting Up the Bill of Materials Attributes

Use the **Attributes** tab to modify the Attribute content, the corresponding column headings, and column width of the report.

The attribute order in the content list determines the column arrangement in the BOM report. There is a limit of 12 attributes in the Bill of Materials report.

Restrictions and Limitations

Including non-ECO-registered parts and non-electrical parts in the bill of materials is constrained. See [Options Dialog Box, Design Category](#) for details.

Procedure

1. Click the **File > Reports** menu item.
2. In the [Reports Dialog Box](#) select the Bill of Materials check box and then click **Setup** to specify or modify the Bill of Materials report characteristics.
3. In the [Bill of Materials Setup dialog box](#) on page 479, click the **Attributes** tab.
4. To add a new attribute to the list, click **Add** and select an attribute from the list in the Part Attribute column,



Tip

You can list up to twelve attribute names. Each attribute defines a column in the BOM report.

5. In the Field Header column, specify the column heading for each attribute in the report. You can specify any character except the colon (:).
6. In the Width column, type an integer between 1 and 200 that represents the number of characters across the column for each attribute in the report.



Tip

SailWind Logic reserves a space to separate columns. Therefore, the actual column width is one character less than the specified number.

7. To move an attribute up a row, select the attribute to move and click **Up**.
 8. To move an attribute down a row, select the attribute to move and click **Down**.
 9. To edit an attribute, select the box to edit and click **Edit**.
-



Restriction:

You cannot edit the part attribute name, but you can select a new attribute to replace the one currently listed.

10. To remove an attribute from the report, select the row and click **Remove**.
11. To restore the default content from the *.ini* file, click **Reset**.
12. Click **OK**.

Setting Up the Bill of Materials Format

Use the **Format** tab to modify the output format of the Bill of Material report. This enables you to specify which fields to display and how the information will be displayed in the generated report.

The default settings originate from the *.ini* file.

Restrictions and Limitations

Including non-ECO-registered parts and non-electrical parts in the bill of materials is constrained. See [Options Dialog Box, Design Category](#) for details.

Procedure

1. Click the **File > Reports** menu item.
 2. In the [Reports Dialog Box](#) select the Bill of Materials check box and then click **Setup** to specify or modify the Bill of Materials report characteristics.
 3. In the [Bill of Materials Setup dialog box](#) on page 483, click the **Format** tab.
 4. Select the type of delimiter you want to distinguish the report columns.
 - **Separator** — places a vertical bar between report fields
 - **Tab** — separates columns with a tab spacing
 - **Comma** — places a comma character between report fields
 - **Custom** — specify any character as a delimiter
 5. Type a report title in the Report Title box.
-



Tip

The variable %j is replaced by the filename and %d is replaced by the date.

6. Select the Combine Value/Tolerance check box to combine the Value and Tolerance attributes of a part in the part name.

For example, the 1/4 watt resistor would have a part type name of R1/4W or R1/4W.4.7K,+/-5%. SailWind Logic evaluates parts that have either a different Value or Tolerance attribute as different part types.

7. In the Ref. Designator Separation Mode settings choose between giving each component a single line (Single Ref. Designator per line) or displaying the reference designators of identical components on one line (Multiple Ref. Designators per line).

If you choose multiple reference designators per line, you can also specify to compress ranges of reference designators and the delimiter between them.

8. Select an attribute in which to sort the report list in the Sort By list.
-

**Tip**

Select None to sort by Part Type.

9. Select the output file format from the File Format list.

10. To save report format settings for the current design to a specified file so you can create different format configurations for different designs, click **Save As** and type a filename in the Save Scheme dialog box.

SailWind Logic saves BOM format settings files in the \Settings folder with a .bom extension.

**Tip**

Click **Delete** to remove the selected setting file from the list.

11. To restore the default content from the .ini file, click **Reset**.

12. Click **OK**.

Setting Up the Bill of Materials Configuration

Use the **BOM Config** tab to preview the Bill of Materials report format and copy any selected lines of the file to a Windows clipboard. You can also export the BOM report to a TXT/CSV file.

The default view orders the attributes by the sort field you specified on the **Format** tab.

Restrictions and Limitations

Including non-ECO-registered parts and non-electrical parts in the bill of materials is constrained. See [Options Dialog Box, Design Category](#) for details.

Procedure

1. Click the **File > Reports** menu item.
 2. In the [Reports Dialog Box](#) select the Bill of Materials check box and then click **Setup** to specify or modify the Bill of Materials report characteristics.
 3. In the [Bill of Materials Setup dialog box](#), click the **BOM Config** tab.
-

4. To sort rows by an attribute other than the one you set on the **Format** tab, click the heading area of the attribute you want to sort.

To sort the same column in reverse order, click again.

5. Copy any selected lines of the file to a Windows clipboard:

- a. To include the column header information in the report, select the **Include table header** check box.
 - b. Select the rows you want to copy to the clipboard and click **Copy**.
-



Tip

Click **Select All** to select the entire table.

6. Export to a TXT/CSV file.

- a. To exclude any row in the report, select the **NC** checkbox.
- b. Click **Export**, and choose the file type in the popup dialog box.

7. Click **OK**.

Chapter 18

Working With SailWind Layout and SailWind Router

Using SailWind Logic, you can seamlessly exchange design data with SailWind Layout and SailWind Router. This is a bi-directional process that enables you to forward annotate and back annotate your data to keep your design changes synchronized. You can also generate a differences report so that you can compare one design with another and highlight any differences.

[Creation of a New PCB Layout from a SailWind Logic Design](#)

[Cross-Probe Between Sailwind Products](#)

[Forward Annotation From SailWind Logic to SailWind Layout](#)

[Backward Annotation From SailWind Layout to SailWind Logic](#)

[Backward Annotation Results](#)

[Contents of the Differences Report](#)

[ECO File Format](#)

Creation of a New PCB Layout from a SailWind Logic Design

There are two ways to create a new PCB design from a SailWind Logic design: you can use the SailWind Layout Link, or you can manually exchange a netlist between SailWind Logic and SailWind Layout.

- **Method 1** — Use if you have both SailWind Logic and SailWind Layout on your computer, use the “[Automatic Netlist Process Using the SailWind Layout Link](#)”. This automated method is the simplest.
- **Method 2** — Use if you don’t have both SailWind Logic and SailWind Layout on your computer, use the “[Manual Netlist Process Between SailWind Logic and SailWind Layout](#)”. This manual method requires you to manually export the netlist from SailWind Logic and then import it into SailWind Layout.

[Automatic Netlist Process Using the SailWind Layout Link](#)

[Manual Netlist Process Between SailWind Logic and SailWind Layout](#)

[Interpreting and Resolving the Netlist Process Error Report](#)

Automatic Netlist Process Using the SailWind Layout Link

If you have both SailWind Logic and SailWind Layout on your computer, you can automatically process a SailWind Logic netlist using the SailWind Layout Link. This automated method is the simplest.

Restrictions and Limitations

- Forward annotation of changes to an *existing* design requires a different process. See [Forward Annotation From SailWind Logic to SailWind Layout](#).
- Transferring non-ECO-registered parts and non-electrical parts is constrained by settings in the Options. See the [Design Options](#) on page 591 for details.

Prerequisites

You must have both SailWind Logic and SailWind Layout on the same computer.

Procedure

1. Click the **Tools > SailWind Layout** menu item.



Tip

If SailWind Layout is not already open, the [Connect to SailWind Layout Dialog Box](#) appears. Click **New** to start a new SailWind Layout session.

2. In the SailWind Layout Link dialog box, click the [Preferences Tab](#) on page 620, then set the appropriate options.

3. On the [Design Tab](#): on page 614

- a. If needed, select the Include Design Rules in Net list or the **AI layout reference data** check box.



Tip

The **AI layout reference data** item must be selected in order for the **AI Intelligent Layout** feature to work in SailWind Layout.

- b. Click the **Send Net List** button.

Results

The import process sources all the part types and decals from the library and stacks the decals at the origin. You can then use the SailWind Layout **Tools > Disperse Components** menu item to spread out the components.

If an errors report file (*ascii.err*) is generated and errors are found in the netlist, see [Interpreting and Resolving the Netlist Process Error Report](#) for more information.

Manual Netlist Process Between SailWind Logic and SailWind Layout

If you don't have both SailWind Logic and SailWind Layout on your computer, you can manually process the netlist. This manual method requires you to manually export the netlist from SailWind Logic and then import it into SailWind Layout.

Restrictions and Limitations

- Forward annotation of changes to an existing design requires a different process. See [Forward Annotation From SailWind Logic to SailWind Layout](#).
- Transferring non-ECO-registered parts and non-electrical parts is constrained by settings in the Options. See the [Design Options](#) on page 591 for details.

Procedure

1. Click the **Tools > Layout Netlist** menu item.
2. To select a different filename or location for the netlist, in the Netlist to PCB dialog box, click **Browse**.



Tip

The default is the design filename with an *.asc* extension, and is saved in the *\SailWind Projects* folder.

3. In the Select Sheets area, select the sheets you want to include in the netlist.
4. Select the **Include Subsheets** check box to include any connections to hierarchical symbols in the netlist.

5. Select the format you want from the Output Formats list.
6. Set the remaining dialog box options.
7. Click **OK** to generate the netlist.
8. Import the netlist into SailWind Layout. For instructions, see the Creating a New PCB Design by Manually Importing the SailWind Logic Netlist topic in the *SailWind Layout Guide*.

Results

The import process sources all the part types and decals from the library and stacks the decals at the origin. You can then use the **Tools > Disperse Components** menu item to spread out the components.

If an errors report file (*ascii.err*) is generated and errors are found in the netlist, see [Interpreting and Resolving the Netlist Process Error Report](#) for more information.

Interpreting and Resolving the Netlist Process Error Report

If errors are found in the netlist import process, an errors report file (*ascii.err*) is generated, and the error report file is displayed in a Notepad window. No errors file is generated if no errors are found.

The following are reported in the errors report file:

- Library issues
- Single or zero pin nets
- Totally floating connections or subnets
- Unnamed dangling connections (one end floating)
- Power and Ground symbols used on nets whose name is different from the default name on the symbol
- Multiple subnet nets where one or more subnets is missing an off-page symbol
- Single subnet nets with an off-page symbol (lonely subnet warning)
- User named subnets that have no visible net name label

Procedure

1. In SailWind Layout, click the **File > New** menu item.
2. Click **No** when you are prompted to save the design.
3. Add any missing components listed in the *ascii.err* file to your library, either by adding a library which contains the missing components to your library list, or by creating the missing part types and decals. (See Adding Libraries to the Library List, Creating and Modifying Part Types, and Creating and Editing PCB Decals in the *SailWind Layout Guide* for instructions.)

4. Resolve any other errors found in the `ascii.err` file.
5. When all the errors have been resolved, repeat the procedure you used to pass the netlist from SailWind Logic to SailWind Layout.

Related Topics

[Creation of a New PCB Layout from a SailWind Logic Design](#)

Cross-Probe Between Sailwind Products

You can cross-probe between SailWind Logic and SailWind Layout, or between SailWind Logic and SailWind Router, if the applications are on the same computer. You can cross-probe between only two applications at a time. In SailWind Logic, you can initiate cross-probing whether or not SailWind Layout or SailWind Router are open.

Through cross-probing, you can select items in both the schematic and design at the same time. For example, if you select objects in SailWind Layout (or SailWind Router), the object is automatically highlighted and centered in the SailWind Logic work area. For SailWind Logic and SailWind Layout only, cross-probing can also automate netlist comparisons and rules exports.

[Cross-Probing With SailWind Layout](#)

[Cross-Probing With SailWind Router](#)

Cross-Probing With SailWind Layout

Cross-probe with SailWind Layout to make simultaneous selections in both applications. Selecting an object in one of the programs will automatically select the same object in the other program.

Restrictions and Limitations

- The SailWind Logic [selection filter](#) on page 712 should be set to parts, gates, nets, or pins before you select items in SailWind Logic.
- Some parts may not exist in both of the databases. Non-ECO-registered parts and non-electrical parts may have been constrained. See [Options Dialog Box, Design Category](#) for details.

Procedure

1. Click the **Tools > SailWind Layout** menu item.

If SailWind Layout is not open, the [Connect to SailWind Layout Dialog Box](#) appears. Do one of the following:

- Click **New** to start a new SailWind Layout session with a new, untitled, design.
- Click **Open** to start a new SailWind Layout session with an existing design. In the File Open dialog box, select a design file and click **Open**.

2. In the [SailWind Layout Link Dialog Box](#), on the **Selection** tab, select the Receive Selections check box to enable selections in SailWind Layout to be received in SailWind Logic.

You can now proceed with cross-probing with SailWind Layout.

Cross-Probing With SailWind Router

Cross-probe with SailWind Router to actively make simultaneous selections in both applications. Selecting an object in one of the programs will automatically select the same object in the other program.

Restrictions and Limitations

- The SailWind Logic [selection filter](#) on page 712 should be set to parts, gates, nets, or pins before you select items in SailWind Logic.
- Some parts may not exist in both of the databases. Non-ECO-registered parts and non-electrical parts may have been constrained. See [Options Dialog Box, Design Category](#) for details.

Procedure

1. Click the **Tools > SailWind Router** menu item.

If SailWind Router is not open, the [Connect to SailWind Router Dialog Box](#) appears. Do one of the following:

- Click **New** to start a new SailWind Router session with a new, untitled, design.
- Click **Open** to start a new SailWind Router session with an existing design. In the File Open dialog box, select a design file and click **Open**.

2. On the [SailWind Router Link Dialog Box](#), on the **Selection** tab, select the Receive Selections check box to enable selections in SailWind Router to be received in SailWind Logic.

You can now proceed with cross-probing with SailWind Router.

Forward Annotation From SailWind Logic to SailWind Layout

You can export your schematic changes “forward” (known as forward annotation) to an existing PCB layout. You can choose between any of the three methods to forward annotate design changes.

- **Method 1** — If you have both SailWind Logic and SailWind Layout on your computer, use the “[Automated Forward Annotation Process](#)”. This automated method is the simplest and fastest.
- **Method 2** — If SailWind Layout is on another computer and you want the layout designer to compare the designs to generate the ECO file, and import it into SailWind Layout use the process in “[Generating the ECO File in SailWind Layout](#)”. This method is somewhat quicker than Method 3 since the Compare/ECO tool in SailWind Layout can automatically import the file after the designs are compared.
- **Method 3** — If SailWind Layout is on another computer and you want to compare the designs to generate the ECO file within SailWind Logic and send it to the layout designer to import into SailWind Layout, use the process in “[Generating the ECO File in SailWind Logic](#)”.

[Automated Forward Annotation Process](#)

[Generating the ECO File in SailWind Layout](#)

[Generating the ECO File in SailWind Logic](#)

[Forward Annotation Results](#)

Automated Forward Annotation Process

If SailWind Logic and SailWind Layout are on the same computer, you can use the SailWind Layout Link dialog box to compare a newer schematic with an older PCB design and update the original design in Layout from the new design in Logic. You can also create a differences report.

Restrictions and Limitations

- If you are creating a new PCB by sending a netlist for the first time, see [Creation of a New PCB Layout from a SailWind Logic Design](#).
- To avoid unexpected changes during forward annotation, consider comparing data before you forward-annotate.
- Transferring non-ECO-registered parts and non-electrical parts is constrained. See the override settings in the [Options dialog box, Design Category](#) on page 591 for details.
- You cannot forward annotate changes to SailWind Router. SailWind Router does not import ECO files.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.

Prerequisites

You must have both SailWind Logic and SailWind Layout on the same computer.

Procedure

1. Click the **Tools > SailWind Layout** menu item to open the SailWind Layout Link dialog box.
-



Tip

If SailWind Layout is not already open, the [Connect to SailWind Layout Dialog Box](#) appears. Click **Open** to start a new SailWind Layout session with the original design. In the File Open dialog box, select the original design file and click **Open**.

2. (Optional) If you want to check the design differences before updating, click the [Design Tab](#) on page 614, and then click the **Compare PCB** button.

The two versions are compared and differences written to *Logic.rep* in the *\SailWind Projects* folder. To see the report, click the *logic.rep* link in the Output Window.

3. On the [Preferences Tab](#) on page 620, then set the appropriate options.
 4. On the [ECO Names Tab](#) on page 618, set the appropriate options.
 5. On the [Design Tab](#): on page 614
 - a. If needed, check the Compare Design Rules and Show Net List errors report check boxes.
 - b. Click the **ECO To PCB** button to send the changes.
-



Tip

While the SailWind Layout Link dialog box is open, you can cross-probe. For more information see [Cross-Probe Between Sailwind Products](#).

See “Forward Annotation Results”.

Generating the ECO File in SailWind Layout

You can create a netlist (.asc file) and send it to the layout designer who can then generate an ECO file by comparing the designs, and then import the changes into SailWind Layout.

Restrictions and Limitations

- If you’re creating a new pcb by sending a netlist for the first time, see [Creation of a New PCB Layout from a SailWind Logic Design](#).
- To avoid unexpected changes during forward annotation, consider comparing data before you forward-annotate.
- Transferring non-ECO-registered parts and non-electrical parts is constrained. See the override settings in the [Options dialog box, Design Category](#) on page 591 for details.

- You cannot forward annotate changes to SailWind Router. SailWind Router does not import ECO files.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.

Procedure

1. Click the **Tools > Layout Netlist** menu item.
 2. To select a different filename or location for the netlist, in the Netlist to PCB dialog box, click **Browse**.
-
-  **Tip**
The default is the design filename with an *.asc* extension, and is saved in the *\SailWind Projects* folder.
-
3. In the Select Sheets area, select the sheets you want to include in the netlist.
 4. Select the Include Subsheets check box to include any connections to hierarchical symbols in the netlist.
 5. Select the format you want from the Output Formats list.
 6. Set the remaining dialog box options.
 7. Click **OK** to generate the netlist.
 8. Send the netlist to the layout designer.

For the continuation of this process, see Forward Annotating Using an ECO File Generated by SailWind Layout in the *SailWind Layout Guide*.

See “[Forward Annotation Results](#)”.

Generating the ECO File in SailWind Logic

Use the Compare/ECO dialog box to compare the netlists for a newer schematic and an older PCB design and create an ECO (engineering change order) file for import into the PCB design. You can also create a differences report file.

Restrictions and Limitations

- If you’re creating a new pcb by sending a netlist for the first time, see [Creation of a New PCB Layout from a SailWind Logic Design](#).
- To avoid unexpected changes during forward annotation, consider comparing data before you forward-annotate.
- Transferring non-ECO-registered parts and non-electrical parts is constrained. See the override settings in the [Options dialog box, Design Category](#) on page 591 for details.

- You cannot forward annotate changes to SailWind Router. SailWind Router does not import ECO files.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.

Prerequisites

You must have the .asc file exported from SailWind Layout, and have the schematic open in SailWind Logic.

Procedure

1. Click the **Tools > Compare/ECO** menu item.
2. In the Compare/ECO dialog box, click the [Documents Tab](#) on page 505.
3. In the Original Design to Compare and Update area, browse for the previous netlist .asc file sent to SailWind Layout.
(Optional) You can acquire a new Layout .asc file from SailWind Layout to generate the .eco file of design differences.



Tip

As long as the PCB design hasn't undergone engineering changes, the last .asc file from SailWind Logic can be compared to the current design in SailWind Logic to generate the .eco file of the engineering changes in the design. If the last exported .asc file is lost, you can export an .asc file from SailWind Layout to compare to the current schematic to generate the .eco file that gets imported into SailWind Layout to update the PCB layout. The same effect is created by [Generating the ECO File in SailWind Layout](#) and the process is semi-automated by automatically importing the .eco file.

4. Click the [Comparison Tab](#) on page 507, and select the options you want to use for design comparison.
5. (Optional) If you want to check the design differences before creating the ECO file:
 - a. Select the Generate Differences Report check box in the **Documents** tab.
 - b. Clear the Generate ECO File check box.
 - c. Click **Run**.

The netlist and PCB files are compared and differences written to *Logic.rep* in the \SailWind Projects folder. To see the differences, click **Show Report** in the Process Status dialog box.
6. Select the Generate ECO File check box, and verify the ECO Filename.



Tip

Give the file a unique name to avoid overwriting any existing ECO files.

7. Click **Run**.
Output files are written to the \SailWind Projects folder.



Tip

In addition to the files listed above, messages or errors that occur during comparison are also written to *Logic_Session.log* and *Logic.err* in the *\SailWind Projects* folder.

8. Send the ECO file to the layout designer for import into SailWind Layout.

For the continuation of this process, see Forward Annotating by Importing an ECO File from SailWind Logic in the *SailWind Layout Guide*.

See “[Forward Annotation Results](#)”.

Forward Annotation Results

The forward annotation process generates a group of files (even in the background during the automated forward annotation process). These include an ECO file, a differences report, ASCII netlist files, and (optionally) an error report.

Table 40. Forward Annotation Generated Files

<schematic_name>.eco	The ECO file. Contains ECO commands that describe the changes needed to update the original design to match the new design. Generated when you select Generate ECO File from the Compare/ECO Tools Documents tab. See “ ECO File Format ” on page 342 for a description of this file.
<i>Logic.rep</i>	The Differences Report file. Describes the differences between the “old” and the “new” compared files. Generated when you select Generate Differences Report from the Compare/ECO Tools Documents tab. See “ Contents of the Differences Report ” on page 340 for a description of this file.
<i>ecogtmp0.asc</i> <i>ecogtmp1.asc</i>	Temporary copy of the “old” netlist and a temporary copy of the “new” netlist.
<i>ecogtmp[0 1].err</i>	Generated only if errors are found in the netlist. A link to this file is displayed in the Output Window.

In addition to the files listed above, messages or errors that occur during comparison are also written to *logic_session.log* and *logic.err* in the *\SailWind Projects* folder. The following are reported in the errors report file:

- Library issues
- Single or zero pin nets
- Totally floating connections or subnets
- Unnamed dangling connections (one end floating)
- Power and Ground symbols used on nets whose name is different from the default name on the symbol

- Multiple subnet nets where one or more subnets is missing an off-page symbol
- Single subnet nets with an off-page symbol (lonely subnet warning)
- User named subnets that have no visible net name label

Related Topics

[ECO File Format](#)

[Contents of the Differences Report](#)

Backward Annotation From SailWind Layout to SailWind Logic

You can export your PCB layout changes “back” (known as *backward annotation*) to the schematic. You can choose between the three methods to backward annotate design changes.

You can backward annotate part, gate, pin, net, and attribute changes. For details, see [Backward Annotation Results](#).

- **Method 1** — If you have both SailWind Logic and SailWind Layout on your computer, you can use the “[Automated Backward Annotation Process](#)”. While this automated method is the simplest and fastest, this method is not recommended because it generates its own .eco file by design comparison. You should use Method 2 to record the .eco file in SailWind Layout and manually import it into SailWind Layout for the best results. For more information, see Recorded Versus Generated ECO Files in the *SailWind Layout Guide*.
- **Method 2** — If SailWind Layout is on another computer and you want the layout designer to generate the ECO file, follow the process in “[Creating the ECO File in SailWind Layout](#)”. This is the most accurate method. You must manually import the .eco file into the SailWind Logic design.
- **Method 3** — If SailWind Layout is on another computer and you must compare the designs and generate the ECO file within SailWind Logic, follow the process in “[Creating the ECO File in SailWind Logic](#)”. This method is not recommended since you are generating the .eco file by design comparison. You should use Method 2 to record the .eco file in SailWind Layout and manually import it into SailWind Layout for the best results. For more information, see Recorded Versus Generated ECO Files in the *SailWind Layout Guide*.

[Automated Backward Annotation Process](#)

[Creating the ECO File in SailWind Layout](#)

[Creating the ECO File in SailWind Logic](#)

Automated Backward Annotation Process

If SailWind Logic and SailWind Layout are on the same computer, you can use the SailWind Layout Link dialog box to compare a newer PCB design with an older schematic and update the older schematic from the newer PCB design. You can also create a differences report.

This method generates a new .eco file and does not use a recorded .eco file. Recording the exact changes in an .eco file gives the best back annotation results. Use Method 2 for the best results. For more information, see Recorded Versus Generated ECO Files in the *SailWind Layout Guide*.

Restrictions and Limitations

- Transferring non-ECO-registered parts and non-electrical parts is constrained. See [Options dialog box, Design Category](#) on page 591 for details.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.
- You cannot perform backward annotation between SailWind Logic and SailWind Router because SailWind Router does not export ECO files.

Prerequisites

You must have the older schematic open in SailWind Logic, and the newer PCB design open in SailWind Layout.

Procedure

1. In SailWind Logic, click the **Tools > SailWind Layout** menu item to open the SailWind Layout Link dialog box.



Tip

If SailWind Layout is not open, the [Connect to SailWind Layout Dialog Box](#) appears. Click **Open** to open the PCB design you want to annotate from in SailWind Layout. In the File Open dialog box, select the .pcb file and click **Open**.

2. (Optional) If you want to check the design differences before updating, click the [Design Tab](#) on page 614, and then click the **Compare PCB** button.

The two versions are compared and differences written to *Logic.rep* in the \SailWind Projects folder. To see the report, click the *logic.rep* link in the Output Window.

3. On the [Preferences Tab](#) on page 620, set the appropriate options.
4. On the [ECO Names Tab](#) on page 618, set the appropriate options.
5. On the [Design Tab](#): on page 614
 - a. If needed, check the Compare Design Rules and Show Net List errors report check boxes.
 - b. Click the **ECO From PCB** button.

Creating the ECO File in SailWind Layout

Create the .eco file of design changes using SailWind Layout and then import it into the SailWind Logic design. This enables you to synchronize changes done in SailWind Layout back to the schematic.

This procedure only documents recording the .eco file as you made engineering changes to your design, which creates a before and after record of changes. Generating an .eco file by comparing designs will be electrically correct, but it does not create a perfect before and after record of components that are identical in their part types and connections. For more information, see Recorded Versus Generated ECO Files in the *SailWind Layout Guide*.

If you neglected to record the .eco changes and you must generate the .eco file by comparing two designs, see Comparing Two Versions of a Design in the *SailWind Layout Guide*.

Restrictions and Limitations

- Transferring non-ECO-registered parts and non-electrical parts is constrained. See [Options dialog box, Design Category](#) on page 591 for details.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.
- You cannot perform backward annotation between SailWind Logic and SailWind Router because SailWind Router does not export ECO files.

Procedure

1. Record the engineering/netlist changes in an .eco file. For more details, see Recording ECO Changes in the *SailWind Layout Guide*.
2. In SailWind Logic, click the **Tools > Options** menu item, then click the **Design** category.
3. Set the “Allow overwriting of attribute values in design with blank values from library attributes” check box appropriately to allow or prevent overwriting of non-blank attribute values with blank (“placeholder”) values from the library.
4. With your design open in SailWind Logic, click the SailWind Logic **File > Import** menu item.
5. In the File Import dialog box, in the Files of Type: list, select ECO Files (*.eco).
6. Browse for and select the ECO file to import.
7. Click **Open**.

If no errors occur, the schematic is updated. If errors occur, the schematic is not updated, and the errors, along with a link to the ECO Import errors file, are written to the Output window.

Creating the ECO File in SailWind Logic

Acquire the layout design in an .asc file format, compare it to the schematic design with the SailWind Logic Compare/ECO tools, and then import the .eco file of design changes into the SailWind Logic schematic design.

This method generates a new .eco file and does not use a recorded .eco file. Recording the exact changes in an .eco file gives the best back annotation results. Use Method 2 for the best results. For more information, see Recorded Versus Generated ECO Files in the *SailWind Layout Guide*.

Restrictions and Limitations

- Transferring non-ECO-registered parts and non-electrical parts is constrained. See [Options dialog box, Design Category](#) on page 591 for details.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.
- You cannot perform backward annotation between SailWind Logic and SailWind Router because SailWind Router does not export ECO files.

Prerequisites

You must have an .asc file exported from SailWind Layout. For more details, see [Exporting an ASCII File](#) in the *SailWind Layout Guide*.

Procedure

1. In SailWind Logic, click the **Tools > Options** menu item, then click the **Design** category.
2. Set the “Allow overwriting of attribute values in design with blank values from library attributes” check box appropriately to allow or prevent overwriting of non-blank attribute values with blank (“placeholder”) values from the library.
3. In SailWind Logic, click the **Tools > Compare/ECO** menu item.
4. In the Compare/ECO dialog box, click the [Documents Tab](#) on page 505.
5. In the Original Design to Compare and Update area, select the Use Current Schematic Design check box.
6. In the New Schematic Design with Changes area, browse for the new .asc file exported from SailWind Layout.
7. Click the [Comparison Tab](#) on page 507, and select the options you want to use for design comparison.
8. (Optional) If you want to check the design differences before creating the ECO file:
 - a. Select the Generate Differences Report check box in the **Documents** tab.
 - b. Clear the Generate ECO File check box.
 - c. Click **Run**.
- The netlist and PCB files are compared and differences written to *Logic.rep* in the *\SailWind Projects* folder. To see the differences, click **Show Report** in the Process Status dialog box.
9. Select the Generate ECO File check box, and verify the ECO Filename.



Tip

Give the file a unique name to avoid overwriting any existing ECO files.

10. Click **Run**.

Output files are written to the *\SailWind Projects* folder.



Tip

In addition to the files listed above, messages or errors that occur during comparison are also written to *Logic_Session.log* and *Logic.err* in the *\SailWind Projects* folder.

11. Click the SailWind Logic **File > Import** menu item.
12. In the File Import dialog box, in the Files of Type: list, select ECO Files (*.eco).
13. Browse for and select the ECO file to import.
14. Click **Open**.

If no errors occur, the schematic is updated. If errors occur, the schematic is not updated, and the errors, along with a link to the ECO Import errors file, are written to the Output window.

Related Topics

[Backward Annotation Results](#)

Backward Annotation Results

Backward annotation is supported at multiple levels of design activity. You can use backward annotation to exchange design updates and changes to attributes, parts, gates, nets, and pins. Results are dependent upon the type of design object data that you are updating.

[Attribute Level Backward Annotation](#)

[Part Level Backward Annotation](#)

[Gate Level Backward Annotation](#)

[Net Level Backward Annotation](#)

[Pin Level Backward Annotation](#)

Attribute Level Backward Annotation

If you add, delete or modify attributes in SailWind Layout, you can backward annotate new attributes and deleted attributes.

New Attributes

- A new attribute in a part updates all parts of the same type. If the attribute name does not exist, it is added with the assigned value.
- An error is created if the part does not exist.



Tip

Unsupported attribute types, such as net or net class, are ignored.

Deleted Attributes

- Deleting an attribute for a part-type deletes the attribute on all parts of that type in the design.
- An error message is generated if the part or attribute name does not exist.
- If the attribute command specifies an object type not supported for general attributes, such as net or net class, the attribute command is ignored.

Related Topics

[Creating the ECO File in SailWind Layout](#)

Part Level Backward Annotation

You can backward annotate added parts, changed parts, deleted parts, and the reference designator name.

Added Parts

- A new sheet is created and all new parts are added to the sheet. Parts are placed on a grid so that parts of a medium size do not overlap. No attempt is made to avoid overlapping of larger parts.
 - An error message is generated if the reference designator of the newly added part already exists or if the part does not exist in the Library.
-



Tip

The part is not added to the schematic if the reference designator already exists.

- If the part contains Signal Pins, these pins are included in the add pin function. Backward annotation does not currently support signal pins so an error message is created.

Changed Parts

- If the changed part is a multigate part, all gates are updated to the new part type.
- An error message is generated if the new part does not exist in the design or in the Library, or if the gate or pin count is incompatible.

Deleted Parts

- If the deleted part is a multigate part, all gates are deleted.
- An error message is generated if the part is still connected to a net or the part does not exist.

Reference Designator Name

- If the part being renamed is a multigate part, all gates are updated.
- An error message is generated if the old reference designator does not exist.

Related Topics

[Creating the ECO File in SailWind Layout](#)

Gate Level Backward Annotation

If you swap gates in SailWind Layout to improve your routing, you can backward annotate the swapped gates to the schematic.

SailWind Logic creates an off-page symbol at each swapped gate. An error message is created if the gate does not exist.

Related Topics

[Creating the ECO File in SailWind Layout](#)

Net Level Backward Annotation

You can backward annotate joined nets, nets created by splitting an existing net, and renamed nets.

Joined Nets

- The first net is renamed to be the same name as the second net.

Nets Created by Splitting an Existing Net

- A Delete Pin from Net operation is performed to remove them from the existing net, followed by an Add Pin to Net operation to add the pin to the new net.

Renaming Nets

- All subnets of the old net on all sheets are renamed. If any of the subnets contain Power or Ground symbols without netnames, netnames are added to these symbols.
- An error message is created if the new net already exists.

Related Topics

[Creating the ECO File in SailWind Layout](#)

Pin Level Backward Annotation

You can backward annotate swapped pins, pins added to a net, and pins disconnected from a net.

Swapped Pins

- SailWind Logic creates an off-page symbol at each swapped pin.

Pins Added to a Net

- A pin can be added only if it is not already connected to another net. If the pin is a gate pin (a visible terminal pin on the gate symbol), an off-page symbol is created.
- An error is created if pin is already connected or the pin is a signal pin already assigned to a net.

Pins Disconnected From a Net

- If the pin is a gate pin, the connection is deleted if it connects to a tie-dot or off-page symbol. If the connection goes to another gate pin, the connection is broken and, an off-page symbol is added.
- This command generates an error message if the pin is not connected to the net in question.

Related Topics

[Creating the ECO File in SailWind Layout](#)

Contents of the Differences Report

When you compare two versions of a design (**Tools > Compare/ECO**), you can create an output file that lists the differences between the two versions. The report file is named *Logic.rep* and is written to the *\SailWind Projects* folder.

Table 41. Sections of the Difference Report

Option	Description
Part differences	This section of the report includes the reference designator and the part type for both the old and new designs. Parts that exist only in the old design are listed under the New Design column as <none>. Parts that exist only in the new design are listed under the Old Design column as <none>. Parts that have identical reference designators and part types in both designs are not listed.
Net differences	This section lists names of the nets that do not exist under two columns: New Design and Old Design. It lists the names of the nets that do not exist in one design or the other. It also lists the nets that match, but that have different names, including nets in the old design that have been split into multiple nets in the new design. A net split operation appears as pin differences. Nets are listed alphabetically under the Old Design column, except where multiple nets are combined, where they are listed in succession. Nets that do not exist in the old design are listed at the end of this section.
Unmatched net pins in old design	This section lists any connected pins in the old design that are missing or connected to other nets in the old design. These pins are deleted from nets during the ECO process. This list provides net names in the old design followed by unmatched pins in the net. If the net does not exist in the new design, all pins in the net are listed.
Unmatched net pins in new design	This section lists any connected pins in the new design that are missing or connected to other nets in the old design. These pins are added to nets during the ECO process. This list provides net names in the new design followed by unmatched pins in the net. If the net does not exist in the old design, all pins in the net are listed.
Attribute differences	This section of the report lists each object under the headings: Attribute Name, Old Value, and New Value. Subheadings, including object type, object name in old design, and object name in new design appear for each object in the list if different.

Table 41. Sections of the Difference Report (continued)

Option	Description									
	Attribute differences are included only for objects that exist in both the old and new design. If an attribute is missing in either design, the value is listed as <no attr>. If the attribute exists, but has no value, it is listed as <no value>.									
Rules differences	<p>This section reports each object or object pair that has rules differences. Each object or object pair has three columns of information (Object Type, Object Name, and Rule Type) and a subheading that lists Rule Name, Old Value, and New Value.</p> <p>If a rule set is missing on an object in either the old or new design, then the old or new values of all missing rule entries are listed as <no rule>.</p> <p>The following example shows a high-speed rule change for net \$\$\$1963 that changes the maximum length and minimum impedance rules.</p> <table style="margin-left: 40px;"> <tr> <td>NET</td> <td>\$\$\$1963</td> <td>HIGH_SPEED</td> </tr> <tr> <td>MAX_LENGTH</td> <td>50000</td> <td>20000</td> </tr> <tr> <td>MIN_IMPEDANCE</td> <td>50.0</td> <td>70.0</td> </tr> </table>	NET	\$\$\$1963	HIGH_SPEED	MAX_LENGTH	50000	20000	MIN_IMPEDANCE	50.0	70.0
NET	\$\$\$1963	HIGH_SPEED								
MAX_LENGTH	50000	20000								
MIN_IMPEDANCE	50.0	70.0								
Net class differences	<p>This section reports the names of net classes that:</p> <ul style="list-style-type: none"> • Do not exist in one design or the other. (Classes that do not exist in the Original Design appear at the end of the section.) • Match but have different names 									
Added class nets	<p>This section reports nets that are not in the Original Design and are added in the New Design. (The net class in the New Design has nets either not included in the Original Design or are included in different net classes in the Original Design.)</p> <p>This section lists:</p> <ul style="list-style-type: none"> • Each net class that has added nets in the New Design and the names of the added nets • All of the nets in the net class if the net class is new (does not exist in the Original Design) 									
Removed class nets	<p>This section reports nets that are in the Original Design but have been removed from the New Design. (The net class in the Original Design has nets either not included in the New Design or included in different net classes in the New Design.)</p> <p>This section lists:</p> <ul style="list-style-type: none"> • Each net class in the old design from which nets were removed and the names of those removed nets • All of the nets in the net class if the net class does not exist in the New Design 									
Pin-pair group differences	<p>This section lists pin-pair groups that:</p> <ul style="list-style-type: none"> • Do not exist in one design or the other. (Pin-pair groups that do not exist in the Original Design appear at the end of the section.) • Match but have different names. 									
Removed group pin-pairs	<p>This section reports pin-pairs that are in the Original Design but removed from the New Design. (The pin-pair group in the Original Design has pin-pairs either not included in the New Design or included in different groups in the New Design.)</p>									

Table 41. Sections of the Difference Report (continued)

Option	Description
	This section lists: <ul style="list-style-type: none">• Each group in the old design from which pin-pairs were removed and the names of its removed pin-pairs• All of the pin-pairs in the group if the group does not exist in the New Design
Added group pin-pairs	This section reports pin-pairs in the New Design that are not in the Original Design. (The pin-pair group in the New Design has pin-pairs either not included in the Original Design or included in different groups in the Original Design.) An ECO operation adds these pin-pairs. This section lists: <ul style="list-style-type: none">• Each group with added pin-pairs in the New Design and the names of added pin-pairs• All of the pin-pairs in the group if the group is new (does not exist in the Original Design)

Related Topics

[Compare/ECO Tools Dialog Box, Documents Tab](#)

[Compare/ECO Tools Dialog Box, Comparison Tab](#)

ECO File Format

The format used is similar to PADS-format ASCII. Each type of data begins with a header line with a key word surrounded by asterisks (*).

The first line of the file is formatted:

```
*PADS-ECO*
```

The end of the file (EOF) entry is formatted:

```
*END*
```

Add remark lines with the entry:

```
*REMARK* remark information etc.
```

The following are the available ECO commands.

Add a Pin to the Net

The command starts with the format:

```
*NET*
```

A line indicating the net to which the pin is added follows this line:

```
*SIGNAL* netname 10
```

where *netname* is the net to add the pins to and 10 is the trace width to associate with the connections. If the net name does not currently exist in the design, it will be added. This is followed by the pin(s) to add to the net as shown below:

```
ref1.pin1 ref2.pin2
```

Add a Part

The command is formatted:

```
*PART*
```

The part entry line is formatted:

```
refdes parttype
```

where *refdes* is the part reference name and *parttype* is the part type name. When parts are added in the PCB, they will be placed at system origin of 0,0. If a board outline is present, the parts are instead placed at the lower left corner of its box.

Join Two Nets Together

The command starts with the format:

```
*JOINNET*
```

This is followed with the line indicating the nets to join:

```
OLDNET0 OLDNET1
```

where *OLDNET0* and *OLDNET1* are the names of the nets to combine. The new combined net uses the name of *OLDNET1*. A connection is added between the nets using two random pins in the selected nets. The trace width of the added connection is the same as that of a connection in the first net (*OLDNET0*).

Delete a Part

The command is formatted:

```
*DELPART*
```

The line is formatted:

```
refdes parttype
```

where *refdes* is the part reference name to delete and *parttype* is the part type name. If all pins of the part to be deleted have not already been disconnected from the connection nets in the design, an error is reported.

Delete a Pin from a Net

The command starts with the format:

```
*DELPIN*
```

This is followed by a list of pins to delete from each net, in the form:

```
refdes.pinnumber signature
```

where *refdes* is the part reference name, *pinnumber* is the pin number to disconnect, and *signature* is the net of which the pin is currently part.

Change a Component's Part Type

The command is formatted:

```
*CHGPART*
```

The change part line is formatted:

```
refdes oldparttype newparttype
```

where *refdes* is the reference name of the part to change, *oldparttype* is the old part type, and *newparttype* is the new part type.

Split a Net into Two Nets

The command starts with the format:

```
*SPLITNET* SPLIT NET INTO TWO NEW NETS
```

Two lines follow, listing the new signal names and the pins:

```
*SIGNAL* oldsignature  
ref1.pin1 ref2.pin2  
*SIGNAL* newsignature1  
ref3.pin3 ref4.pin4
```

where the pins following the *SIGNAL* statement are in the first net, and those following the second line are in the second net.

Rename a Part

The command is formatted:

```
*RENPART*
```

The rename line is formatted:

```
oldrefdes newrefdes
```

where *oldrefdes* is the old name and *newrefdes* is the new name. To facilitate the renaming operation, duplicate name checking is not run until after all rename information has been read. This allows the above rename list to run without a conflict. If any error is encountered, no parts in the list are renamed.

Rename a Net

The command is formatted:

```
*RENNET* RENAME NET
```

The rename net entry line is formatted:

```
oldname newname
```

where *oldname* is the old net name and *newname* is the new net name.

Swap a Gate

The command is formatted:

```
*SWPGATE* GATE1 GATE2
```

Swap Pins

The command is formatted:

```
*SWPPINS* REFDES PIN1.PIN2
```


Chapter 19

Plotting and Printing

Various options are available for plotting and printing your design output. You can choose to print your design on a printer connected to your computer or network for a quick visual design review. You can generate output to a pen plotter or a photo plotter. You can also generate an intelligent PDF file for sharing your design output electronically.

[Setting Printing and Plotting Output Options](#)

[Previewing Your Output](#)

[Creating a PDF](#)

[Plotting Output](#)

[Setting Up a Pen Plotter](#)

[Adding a New Pen Plotter](#)

[Setting Up a Photo Plotter](#)

[Printing Output](#)

[Printing to PDF](#)

[Setting the Printer to Print to a File](#)

Setting Printing and Plotting Output Options

Use the Options dialog box to set the output options for printing or plotting. Setting options enables you to set the position, orientation, and color of selected sheets and objects.

Procedure

1. In the Print or Plot dialog box, click **Options**.
2. In the Sheet Selections area in the Options dialog box, select the name of the available sheets to add to the Sheets to Print list, and click **Add**.
3. Select an Orientation angle and a Justification position for the output.
4. Type an offset value in the X Offset and Y Offset boxes.



Tip

When setting an offset, observe the following:

- You can only set offsets if you clicked something other than Scale to Fit and Centered in the Justification list.
- The print or plot location is calculated after the plot is rotated and scaled.

5. In the scaling boxes, type the plot size to actual size ratio.

Example: A 2 to 1 scaling results in a printout or plot that is twice the actual size.



Tip

The Preview area graphically shows the position, orientation, justification, and scaling of the output.

6. In the Items area, select the check boxes for the items you want to include in your output.
7. In the Selected Color area, click a color tile, then in the Items area, click the tile to the right of the item to change its color.
8. Select the Plot Jobname check box to output the schematic name, time, and date.
9. Select the Print Window check box to print the displayed window.
10. Click **OK** to return to the Print or Plot dialog box.

Previewing Your Output

You can use the Selections Preview dialog box to preview your options and output. This enables you to examine the output and determine if you need to make any changes before generating your final output files.

Procedure

1. Click the **File > Plot** or the **File > Print** menu item, then in the Plot or Print dialog box, click the **Preview** button to open the Selections Preview dialog box and verify the settings for the selected sheets you want to print or plot.
 2. In the Selections Preview dialog box, click **Print** or **Plot** to send the output to the printer or plotter.
-



Tip

You can navigate within the Selections Preview dialog box using the following methods:

- Zoom into the Sheet or Extents of the selected sheet by clicking the appropriate buttons.
 - Click the left mouse button in the preview window to zoom in at the cursor location. Click the right mouse button to zoom out.
-

Creating a PDF

You can create an intelligent PDF of your schematic, choosing which sheets you want to share and show to others in your organization. You can create the PDF in full-color or black and white, with hyperlinks to part attributes, and with search capabilities, making it easy to locate parts and nets. Once you locate a net, you can find other instances of it through the entire schematic, even when the net is on a different page. You can also create a black and white, non-searchable PDF of your schematic.

See also [Printing to PDF](#).



Tip

Adobe® Acrobat® Distiller™ is not required on your system to create a PDF.

Restrictions and Limitations

To search a PDF, you must substitute the Stroke font with a System font.

Procedure

1. Click the **File > Create PDF** menu item.
 2. In the File Create PDF file dialog box, type the name of the PDF and then click **Save**.
-



Tip

The default name of the file is the name of the schematic with a PDF extension.

3. In the Create PDF dialog box, in the Sheets to PDF list, select the schematic sheets you do not want to include in the PDF, and then click **Remove**.
-



Tip

By default, all schematic sheets are selected for PDF creation.

4. To automatically open the resulting PDF, select the “View PDF after creation” check box.
5. To replace a stroke font in your schematic with a system font in the PDF, select the “Replace stroke font with” check box, and then select a font from the list.

Use the Font style buttons to add Bold, Italic, or Underline styles.



Restriction:

When replacing stroke fonts, the following restrictions apply:

- You can only search a PDF if you replace the Stroke font with a System font.
 - There is no font size control. Text heights are already set for each text item in the design. The height will be converted to the nearest point size in the PDF.
-

6. To be able to view part attributes such as the reference designator and the part type in the PDF, select the “Create hyperlinks that will display part attributes” check box.

In the resulting PDF, you will see yellow boxes around each part. If you click inside the box, you will get a listing of the part attributes.

7. To enable finding the next instance of a net or bus in the PDF, select the “Create hyperlinks that will pan through nets” check box.
-



Restriction:

If the net name is not visible in the schematic, you will not be able to pan through the nets.

In the resulting PDF, you will see blue boxes around each net name and bus name. If you click inside the box, you will pan to the next instance of that net or bus.

8. To create hyperlinks on parts, net names, and buses without their yellow and blue boxes, clear the “Visible rectangle around objects with hyperlinks” check box.
-

In the resulting PDF, the hyperlinks will be invisible.

9. In the Color Scheme area, click the scheme of your choice.
-



Tip

The colors used in the Color on Black or Color on White schemes are the same colors currently used in your schematic. The Black on White scheme shows all currently visible items in the schematic in black.

10. Click **OK**.

Plotting Output

After you have completed your design you can generate output for design review. You can output your designs to pen plotters or photo plotters.

Procedure

1. Click the **File > Plot** menu item.
 2. Choose Pen Plot or Photo Plot.
 3. [Set up output options](#) on page 347.
 4. Set up the [pen plotter](#) on page 350 or [photo plotter](#) on page 352.
 5. [Preview your output](#) on page 348.
 6. Click **Run**.
-



Tip

The Plot dialog box displays the following information related to the plot configuration:

- The Summary area lists the numbered available sheets you can plot, and the items contained in the sheets.
- The File Prefix box lists the prefix name of the file you want to plot.

Related Topics

[Printing to PDF](#)

[Setting the Printer to Print to a File](#)

[Printing Output](#)

Setting Up a Pen Plotter

Before you can output your design to a pen plotter, specific options need to be configured so that the output is correctly reproduced by the plotting device. Use the Pen Plotter Setup dialog box to set various options for the plotter.

Procedure

1. Click the **File > Plot** menu item.
2. In the Plot dialog box, click **Setup**.
3. Type the number of pens (1-16) in your device, and type the pen line width in mils.
4. Select the Rotate Axis check box to reverse the X and Y axes of the design.
5. To assign colors, click a Selected Color tile, and then click a pen number in the Pen Colors area. Repeat this step for every pen listed.
6. Click a **Plotting Size** button, or click **Other** to define a custom size.
If you click **Other**, type the X and Y dimensions to use.
7. In the Device list, select the plotter device to use.

Adding a New Pen Plotter

Use the Pen Plotter Advanced Setup dialog box to add a new pen plotter to the list of available plotters.

Restrictions and Limitations

- You can also edit the data for an existing plotter, but only if it is a custom setup.
- You cannot modify the SailWind-supplied advanced plotter settings.

Procedure

1. Click the **File > Plot** menu item.
2. In the Plot dialog box, click the **Setup** button.
3. In the Pen Plotter Setup dialog box, click the **Advanced** button.
4. Type the name of a different pen plotter you want to use.
Exception: Do not reuse one of the existing, supplied device names.
5. In the device type list, select the interface language the plotter uses: HPGL or HGML.
6. Set the plotter resolution by providing a scaling ratio. Type a number in the Multiplier and Divisor boxes. The ratio defined is the scale factor to convert from mils (0.001 in) to plotter units.
Example: Most Hewlett-Packard plotters have a resolution of 0.025 mm or 1/40 mm. This means that a distance of one inch (1000 mils) is 1016 plotter units (25.4 X 40). So a ratio of 1016 to 1000 would be defined. The ratio actually used is 254 to 250 which is the same as 1016 to 1000.
7. Select the Origin at Center check box if the origin of the plotter is at the center of the paper. Clear this check box if the origin is in the lower left corner or other location.
8. Click **OK** to return to the Pen Plot Setup dialog box.
9. Click **OK** to return to the Plot dialog box.

Related Topics

[Setting Up a Photo Plotter](#)

Setting Up a Photo Plotter

Before you can output your design to a photo plotter, specific options need to be configured so that the output is correctly reproduced by the plotting device. Use the Photo Plotter Setup dialog box to define the aperture and other photo plotter options.

Procedure

1. Click the **File > Plot** menu item.
2. In the Plot dialog box, select Photo Plot, and then click the **Setup** button.
3. To add a new D-code to the D-Code list, click **Add**, type a code, and click **OK**.
4. To delete a D-code, select a code in the D-Code list, and click **Delete**.

Alternatives:

- Click the **Augment** button to automatically generate D-codes for items in the schematic file that are not in the current list.
- Click the **Regenerate** button to clear the current D-code list and automatically add D-codes for all items in the schematic.

5. Set the shape for a selected D-code.
 - Select the **Same Aperture for Flashes/Lines** check box to draw lines and flashed items with the same aperture. Round and square shapes for lines will be gray.
 - If you clear this check box, then you can click either **Flash** (to set a flash aperture) or **Line** (to set a draw aperture).
 - In the **Width** box, type a diameter for round shapes. This box is unavailable if a width is not appropriate for the specified shape.

6. In the **Aperture Count** box, type the maximum aperture count.
7. Select the **Augment on-the-fly** check box to add apertures to the D-code list when photo plots are run if any newly created lines were added to the schematic.

When you open the Photo Plotter Setup dialog box, the necessary apertures will be present.

8. In the Photo Plotter Setup dialog box, click the **Advanced** button.
9. Click English for mils or Metric for millimeters.
10. To specify the precision of the output file coordinates, type the number of digits that should lead and trail the decimal point. Type the digits in the **Leading** and **Trailing** boxes.

11. To set the coordinates for the output file, click Absolute for absolute coordinates, or click Incremental for relative coordinates.
12. Define how to handle zero suppression in the output file:
 - Click None to retain leading and trailing zeros.
 - Click Leading to suppress zeros before the decimal point.
 - Click Trailing to suppress zeros after the decimal point.
13. Define how to draw arcs and circles:
 - Click None if your photo plotter does not support circular interpolation. Arcs and circles are drawn as small straight-line segments.
 - Click Quadrant if your photo plotter does not support full, 360-degree circular interpolation.
 - Click Full if your photo plotter supports full, 360-degree circular interpolation.
14. To set the plotting size, click a Plotting size button, or click **Other** to define a custom size.
If you click **Other**, type the X and Y dimensions to use.
15. Select the Suppress Repeated Coords check box to eliminate repeated coordinates from the output file.
16. Click **OK** to return to the Photo Plotter Setup dialog box.
17. Click **OK** to return to the Plot dialog box.

Related Topics

[Setting Up a Pen Plotter](#)

Printing Output

You can output your design to any standard printer that is connected to your system. Set up your print options as required by your installed printer driver.

Procedure

1. Click the **File > Print** menu item.
-



Note:

If you have previously set up your printer configuration options, then on the toolbar, click the **Print** button to immediately print your design using the last saved print configuration.

2. Select the name of the printer to which you want to print.
 3. [Set up print options](#) on page 347.
-

4. Click **Network** to display the Connect to Printer dialog box.
5. [Preview your output](#) on page 348.
6. Click **OK** to send the output to the printer.

Related Topics

[Printing to PDF](#)

[Setting the Printer to Print to a File](#)

Printing to PDF

You can print your schematic sheets to a single PDF. The resulting PDF is a non-searchable, black and white image of your schematic.

To create a color PDF that is searchable, see the [Creating a PDF](#) topic.

Restrictions and Limitations

You must have a PDF generator (such as Adobe PDF, Cute PDF, or Microsoft Print to PDF) installed before you can print to PDF.

Procedure

1. On the toolbar, click the **Print** button.
2. In the Print dialog box, select your preferred PDF generator in the Printer name list.
3. Click **OK**.

Related Topics

[Creating a PDF](#)

[Plotting Output](#)

[Printing Output](#)

Setting the Printer to Print to a File

SailWind Logic supports PostScript printing to a file through the use of the Windows printer properties. Before you create a PostScript file, you must first set up the printer to print to a file.

Procedure

1. Locate printer information based on your platform:
 - For Windows 7 Professional, click **Start**, click **Control Panel**, then double-click **Devices and Printers**.
 - For Windows 10, click **Start**, click the **Settings** icon, click **Devices**, then click **Printers and scanners**.
 2. Select the PostScript printer you want to use, then right-click and click **Properties**.
 3. Click the **Ports** tab.
 4. In the Port column, select the Print to File check box.
-



Tip

This procedure works with local printers. If you are using network printers, you may not have access rights to select or change the port information.

The options on the **Ports** tab may differ depending upon the specific version of your operating system.

5. Click **OK**.

Related Topics

[Printing to PDF](#)

[Plotting Output](#)

[Printing Output](#)

Chapter 20

Object Linking and Embedding

SailWind Logic object embedding capabilities enable design engineers to embed an object in a SailWind Logic design file and hold it within its framework. You can also embed a link to an outside object so that the linked object automatically updates each time you open the SailWind Logic database.



Restriction:

The insertion of SailWind Logic, Layout or Router files as OLE objects in other files (including other SailWind files) is not supported. Any SailWind Logic, Layout or Router file inserted in another file will not behave properly and cannot be edited within the “container” application (Visual Editing).

You can insert other files or other applications as linked or embedded objects within a SailWind Logic schematic. You can insert a Microsoft Word document, a Microsoft Excel spreadsheet containing a Bill of Materials, video or audio clips, and so forth. SailWind Logic does not need to understand the format of the inserted object; SailWind Logic communicates with the application that created the file and that source application tells SailWind Logic what information to display and how to display it.



Tip

You cannot insert or modify OLE linked or embedded objects in the Part Editor.

[Insertion of OLE Objects in SailWind Logic](#)

[Embedding a Text Document](#)

[Turning Off the Display of OLE Objects](#)

[OLE Object Selection](#)

[Moving and Sizing OLE Objects](#)

[Changing an OLE Object's Icon or Label](#)

[Converting an OLE Object to Another Type](#)

[Edits of OLE Objects](#)

[OLE and Print/Plot](#)

[Deleting OLE Objects](#)

[Redraws of a Screen Containing OLE Objects](#)

[OLE and View Menu Commands](#)

[Changing the OLE Object Background Color](#)

[Saving OLE Objects](#)

Insertion of OLE Objects in SailWind Logic

When you insert an OLE object in a SailWind Logic schematic, you can choose to either embed it or link to it.

- An embedded object resides in the schematic, and can be accessed only from within SailWind Logic. An embedded Word object, for instance, can be opened and edited only from the schematic it resides in.
- A linked object resides on disk, and can be opened in its appropriate application either via SailWind Logic or directly from the disk. It is not copied permanently into the schematic, but is read in from disk each time the schematic is opened.



Restriction:

Insertion of SailWind Logic, Layout or Router files as OLE objects in other files (including other SailWind files) is not supported. Any SailWind Logic, Layout or Router file inserted in another file will not behave properly and cannot be edited within the “container” application (Visual Editing).

There are three ways of inserting an OLE object in a SailWind Logic schematic:

- Insert a new (empty) embedded object, and then create the object's content. For instance, insert a new Word document, then edit it with Word from within the schematic.
- Insert a copy of an existing application or file as an embedded object. Embedded objects are not updated when the original file is updated.
- Insert a link to an existing application or file as a linked object. Linked objects are updated whenever the file they link to is updated.

The following sections of this topic describe how to insert OLE objects in a SailWind schematic:

[Inserting a New Embedded OLE Object](#)

[Inserting an Existing File as an Embedded Object](#)

[Inserting an Existing File as a Linked Object](#)

Inserting a New Embedded OLE Object

You can insert a new embedded OLE object in a SailWind Logic schematic. By embedding the object, it is not affected by changes made to an external source file.

Procedure

1. Click the **Edit > Insert New Object** menu item.
2. From the Insert Object dialog box, select **Create New**.
3. From the Object Type list box, select the type of OLE object you want to create.



Tip

You can only create new OLE objects using applications that are installed and registered on your system and are OLE servers. Most OLE servers are Microsoft products, such as Word or Excel.

4. To display the new object in the schematic as an icon, check the **Display As Icon** check box.
The icon that will be displayed appears beneath the check box.
Uncheck the check box to display the entire object; for example, to display the actual Word document instead of a **Word** icon.
5. Click **OK**. The application associated with the object type you selected opens, and you can [edit](#) on page 364 the new object's content:
6. If the application is an OLE Linking and Embedding Server, it opens inside SailWind Logic, but runs in the background. The application's toolbar takes over SailWind Logic's toolbar. You can then work with the source application as you would if it was started outside SailWind Logic. This is called Visual Editing.
Click outside the object, and the SailWind Logic toolbar takes over again. You can continue to design in SailWind Logic. The application continues to run in the background; therefore, you can click on the object and work in the source application at any time.
If the application associated with the object is not an OLE Linking and Embedding Server, the application opens in a new window.
7. When you are finished editing, close the object. You can reopen the object at any time by double-clicking on it.

Inserting an Existing File as an Embedded Object

An existing file inserted in a SailWind Logic schematic as an embedded object is a copy of the source file it is created from. It is not linked to the original source file, is not updated when the source file is changed, and can be accessed only from SailWind Logic.

Procedure

1. Click the **Edit > Insert New Object** menu item.
2. From the Insert Object dialog box, select **Create from File**.
3. In the File edit box, type the pathname of the file to insert, or click the **Browse** button to search for the file.
4. Uncheck the **Link** checkbox to insert the OLE object as an embedded object.
5. To display the object as an icon in the schematic, check the **Display As Icon** check box.
The icon that will be displayed appears beneath the check box.
Uncheck the check box to display the entire object; for example, to display the actual Word document instead of a **Word** icon.
6. Click **OK**. The object is inserted in the schematic.

Inserting an Existing File as a Linked Object

An existing file inserted in a SailWind Logic schematic as a linked object is merely a link to the source file on the disk. The file is not copied permanently into the schematic, but is read in from disk each time the schematic is opened. It can be opened from within SailWind Logic, or directly from the disk.

Procedure

1. Click the **Edit > Insert New Object** menu item.
2. From the Insert Object dialog box, select **Create from File**.
3. In the File edit box, type the pathname of the file to insert, or click the **Browse** button to search for the file.
4. Check the **Link** checkbox to insert the OLE object as a linked object.
5. To display the object as an icon in the schematic, check the **Display As Icon** check box.
The icon that will be displayed appears beneath the check box.
Uncheck the check box to display the entire object; for example, to display the actual Word document instead of a **Word** icon.
6. Click **OK**. The object is inserted in the schematic.

Embedding a Text Document

Using OLE, you can embed a text document in your schematic to more quickly add multiple lines of text since the Create Text tool on the Schematic Editing Toolbar only allows single lines of text.

Embedding is recommended over linking since the embedded document will reside inside the .sch file and can't get lost or accidentally deleted as an external file.

You can see a sample of this in the preview.sch sample design. See the Notes section on the lower left corner of sheet 1. Double-click the text to activate the Microsoft Word document.

Restrictions and Limitations

- OLE objects can only be printed. They cannot be plotted by a pen or photo plotter.
- OLE objects must be enabled in the print [Options dialog box](#) on page 606 to appear in the printout, but they will never be visible when viewing the Print Preview.
- OLE objects can only be printed using a zero plot orientation.

Procedure

Follow the appropriate instructions in [Insertion of OLE Objects in SailWind Logic](#).



Tip

You can resize the object within your design when the object is active for editing.

Turning Off the Display of OLE Objects

SailWind Logic does not understand the format of an inserted object; it only communicates with the application that created the OLE linked or embedded object to display the information. If the application that created the linked or embedded OLE object is installed and registered on your system, then SailWind Logic calls upon that application to display the OLE object in SailWind Logic as it would appear in the source application. For example, a Word document can appear within SailWind Logic and the Word toolbars appear within SailWind Logic.

If the source application is not installed and registered on your system, then SailWind Logic can only display the inserted OLE object as an icon. It cannot open or display the OLE object as it would appear in the source application. SailWind Logic also displays the OLE object as an icon if the object is an application.

You may want to turn off the display of OLE objects when SailWind Logic contains many linked or embedded objects because the redraw speed can decrease. Redraw speed noticeably decreases if the OLE Linking and Embedding servers, or source applications, which actually display the OLE items are not optimized for remote display.

Procedure

1. Click the **Tools > Options** menu item, then in the Options dialog box, click the **General** category.
2. Clear the Display OLE Objects check box.

See also [Options Dialog Box, Design Category](#).

OLE Object Selection

Selection of OLE objects in SailWind Logic operates differently than selection of other objects such as pads, nets, components and so forth.

The differences are:

- You cannot select more than one OLE object at a time.
- You cannot use area select to select OLE objects.
- Commands apply to selected OLE objects only, even if you also select SailWind Logic objects. OLE objects have selection priority over SailWind Logic components.
- OLE objects are always on top; to select SailWind Logic items under an OLE object, you must move the OLE object.

When you click on an OLE object in SailWind Logic, it behaves like a non-text item in a Word file. that is, it becomes a rectangular area with sizing handles to indicate that it is selected. (Sizing handles are small, black squares that appear at the corners and along the sides of a rectangular area surrounding a selected object.)

Right-click a selected OLE object to access a shortcut menu that lists all commands that you can apply to the OLE object.

Moving and Sizing OLE Objects

Move and resize OLE linked or embedded objects just as you resize non-text objects in Word.

Procedure

Select the object.

- To move the OLE object, click and hold the left mouse button. Move the cursor to move the object. Release the mouse button once the object is in the correct location.
- To size an OLE object, click and hold the left mouse button on one of the sizing handles. Move the cursor; the object changes size according to the cursor movements. Release the mouse button when the object is sized correctly.

Changing an OLE Object's Icon or Label

You can change the icon that represents an embedded or linked object in your schematic. You can also change the descriptive label that is displayed below the icon.

Procedure

1. Click on the object to select it.
2. Right click and select <object_type> > **Convert** to display the Convert Dialog Box.
3. Click the **Change Icon** button to display the Change Icon dialog box.
4. In the Change Icon dialog box, click an Icon radio button to select the icon for the object:
5. Leave the Current radio button set if you want to keep the current icon but change the object's label.
6. Click the Default radio button to change to the default icon.
7. Click the From File radio button to select the new icon from a file. If no file is shown in the From File edit box, click the **Browse** button to search for the file containing the icon you want to represent the object. Then click on the desired icon in the list box.
8. To change the object's label, edit it in the Label edit box.
9. Click **OK** to return to the Convert dialog box.
10. In the Convert dialog box, click **OK** to save your changes.

Converting an OLE Object to Another Type

You can change the file format of an inserted OLE object so that you can open it in the applications you use. You can change the format temporarily--for a single editing session, after which the object is saved in its original format--or permanently, so that the object is converted to--and saved in--the new format.

Procedure

1. Click on the object to select it.
2. Right click and select <object_type> > **Convert** to display the Convert Dialog Box.
3. In the Object Type list box, click the object type you want to convert the object to.
4. Click Convert to or Activate as to specify the conversion mode:
 - a. Click Convert to convert the object to the type selected in the Object Type box.
The object will be converted to the new object type, and saved.
 - b. Click Activate as to open the object as the type selected in the Object Type box.
After you activate and edit the object, it returns to the current type.
5. To display the inserted object in the schematic as an icon, check the Display As Icon check box.
The icon that will be displayed appears beneath the check box.
Uncheck the check box to display the entire object; for example, to display the actual Word document instead of a **Word** icon.



Tip

If an object's source application is not registered on your system, then the object can be displayed only as an icon.

6. Click **OK** to convert or activate the object.

Edits of OLE Objects

You can perform a limited set of edits on an OLE linked or embedded object. These include copy, paste, delete, open, and convert OLE Objects.

- [Cut, Copy, and Paste SailWind Logic OLE Objects](#)
- **Edit > Delete All OLE Objects** — Enables you to remove all of the OLE objects in a design. The prompt, “All current OLE Linked and Embedded objects will be removed from this Design. Do you want to continue?”, appears.
- [Edit OLE Links](#)
- [Open, Edit, Convert OLE Objects](#)
- [Editing an OLE Object's Content in SailWind Logic](#)

[Cut, Copy, and Paste SailWind Logic OLE Objects](#)

[Edit OLE Links](#)

[Open, Edit, Convert OLE Objects](#)

[Editing an OLE Object's Content in SailWind Logic](#)

Cut, Copy, and Paste SailWind Logic OLE Objects

Editing an OLE linked or embedded object is similar to editing SailWind Logic objects. You can cut, copy, and paste OLE objects using the Cut, Copy, and Paste commands from the **Edit** menu. You must select the OLE object before you can edit the object. You can copy objects from one sheet to another.



Tip

You cannot cut, copy, or paste OLE objects when in the Part Editor.

Procedure

1. Copy or cut the OLE object as follows:
 - a. Select the OLE object.
 - b. Click **Edit > Copy** menu item (to copy the object) or the **Edit > Cut** menu item (to cut the object).
2. Click the **Edit > Paste** menu item to paste the object.
3. Relocate the pasted object as necessary.

Related Topics

[Open, Edit, Convert OLE Objects](#)

[Edit OLE Links](#)

[Edits of OLE Objects](#)

[Editing an OLE Object's Content in SailWind Logic](#)

Edit OLE Links

You can edit a linked OLE object's link to change the source file, break the link to the source file, specify update options or manually update the linked object.

- Change the object's source file (that is, link the object to a different file).
- Break the link with the source file, making the linked object an embedded object.
- Select the object's update mode, that is, specify whether the object will be updated automatically (whenever the source file changes), or only when you execute an Update command.
- Manually update the linked object.

The following sections of this topic describe how to edit an object's link:

[Changing a Linked OLE Object's Source File](#)

[Breaking the Link to a Linked OLE Object's Source File](#)

[Setting the Update Mode For a Linked OLE Object](#)

[Manually Updating a Linked OLE Object](#)

Changing a Linked OLE Object's Source File

If your design objectives change, you can change the source file that an object links to.

Procedure

1. Click the **Edit > Links** menu item.
2. From the list in the Links dialog box, select the object whose source file you want to change.
3. Click the **Change Source** button.
4. In the Change Source dialog box, browse for and select the new source file for the object, and click **Open**.

The object is linked to the new source file.

Breaking the Link to a Linked OLE Object's Source File

If you no longer want to be linked to a source object, you can break the link to an object's source file so that it will not automatically update when the source file changes.



Tip

Once an object's link has been broken, it cannot be reconnected.

Procedure

1. Click the **Edit > Links** menu item.
 2. From the list in the Links dialog box, select the object whose link you want to break.
 3. Click the **Break Link** button, and click **Yes** in the popup that appears. The link is broken and the object becomes an embedded OLE object.
-



Tip

If the object whose link you broke is iconized, to view the object you must first [convert](#) on page 362 it to a Picture object.

Setting the Update Mode For a Linked OLE Object

You can set the update mode for a linked OLE object and choose between automatic or manual updating.

Procedure

1. Click the **Edit > Links** menu item.
2. From the list in the Links dialog box, select the object whose update mode you want to set.
3. Click the Automatic or Manual radio button to set the update mode:
 - a. Click Automatic to have the object automatically updated when you open the SailWind Logic file, and whenever the source file changes.
 - b. Click Manual to have the object updated only when you execute an Update command for the object.

Manually Updating a Linked OLE Object

You can manually update a linked object whose update mode is set to Manual. (Objects set to Automatic update are automatically updated, so manually updating these objects has no effect.)

Procedure

1. Click the **Edit > Links** menu item.
2. From the list in the Links dialog box, select the object you want to update.
3. Click the **Update Now** button; the object is updated.

Related Topics

[Open, Edit, Convert OLE Objects](#)

[Cut, Copy, and Paste SailWind Logic OLE Objects](#)

[Editing an OLE Object's Content in SailWind Logic](#)

Open, Edit, Convert OLE Objects

Once you insert an OLE object, the name of the object appears at the bottom of the **Edit** menu. For example, if you insert a video clip, Video Clip Object appears at the bottom of the **Edit** menu. If you highlight the Video Clip Object command, another menu appears listing all commands you can perform on the linked or embedded OLE object. For a Video Clip Object, you can click **Play**, **Edit**, or **Open**.



Tip

The commands that appear for each object depend on the object type; therefore, a Word object will not have the same options that a video clip has.

Some of the more common commands you will see relative to OLE objects are:

- **Edit** — Use Edit to edit the linked or embedded OLE object in SailWind Logic. You can edit the object using all of the source application's commands and tools.
 - **Open** — Use Open to open the linked or embedded OLE object in the source application. You can then edit the object within the source application.
 - **Convert** — You can convert an OLE object to another object. You can also convert the OLE object from displaying as an icon to displaying the actual object; for example, the Word document instead of an icon. Use Convert to convert a linked or embedded OLE object to another type of object; for example, convert a Word document to a Word picture.
-



Tip

The object's source application determines what the object can be converted to.

Related Topics

[Edit OLE Links](#)

[Edits of OLE Objects](#)

[Cut, Copy, and Paste SailWind Logic OLE Objects](#)

[Editing an OLE Object's Content in SailWind Logic](#)

Editing an OLE Object's Content in SailWind Logic

You can edit an object's content within SailWind Logic (known as in-place visual editing), or in a separate window. In either case, you edit its contents as you normally would using all of the source application's commands and tools.

In-place Visual Editing

Visual Editing occurs when the source application for a linked or embedded OLE object opens within SailWind Logic. You can also edit an OLE object by opening the source application and editing the object in the environment in which it was created.

To edit an object within SailWind Logic, double-click on the object.

Click outside of the object to deactivate visual editing. Updates are automatically reflected in the object.



Restriction:

You cannot edit embedded SailWind programs within SailWind Logic with visual editing.



Tip

When choosing objects for in-place editing, the following exceptions may apply:

- Linked objects cannot be edited in-place: they open in a separate window for editing.
 - If the container application does not support in-place visual editing, the object will open in a separate window.
-

Separate Window Editing

You can edit objects outside SailWind Logic, in a separate window. To edit an object outside SailWind Logic:

- Select the SailWind Logic object and click the **Edit > (Linked) Document Object > Edit** menu item. You can also select the **(Linked) Document Object > Edit** menu item from the popup menu.
- Ctrl+double-click the OLE object to edit in the source application. The source application opens and you edit the object.

To update the object in SailWind Logic:

- Click the **File > Update Document** menu item. This forces a redraw of the object.
- Set a preference for SailWind Logic OLE objects in the “[Options Dialog Box, General Category](#)” on page 595. When you select the Update on Redraw check box, the object in the container application will update whenever you perform a redraw in the separate editing window.

For best performance clear this option.

To return to the container application, select the **File > Exit and Return to <host>** menu item.



Tip

If you want to save the object you edit in the separate window, you can click the **File > Save Copy As** menu item. The object is really a copy of the original, and this command lets you save this copy. You cannot open other files, create new files, or save original designs in the separate window.

Related Topics

[Open, Edit, Convert OLE Objects](#)

[Edit OLE Links](#)

[Edits of OLE Objects](#)

[Cut, Copy, and Paste SailWind Logic OLE Objects](#)

OLE and Print/Plot

You can print OLE linked or embedded objects to any Windows-supported printer or plotter. You cannot photo plot or pen plot OLE objects. Also, OLE objects do not appear when previewing prints.

See also [Plotting Output](#).

Related Topics

[Setting Up a Pen Plotter](#)

[Setting Up a Photo Plotter](#)

[Printing Output](#)

Deleting OLE Objects

If they are no longer needed in your design, you can delete OLE linked or embedded objects.

Procedure

1. Select the OLE object to delete.
2. Click the **Edit > Delete** menu item, or click the Delete key.



Tip

When you delete a sheet with OLE objects on it, all of the OLE objects are also deleted.

3. To delete all OLE objects in the design, click the **Edit > Delete All OLE Objects** menu item.
This enables you to remove all of the OLE objects in a design.
The prompt, “All current OLE Linked and Embedded objects will be removed from this Design. Do you want to continue?” appears.

Redraws of a Screen Containing OLE Objects

When SailWind Logic redraws, SailWind Logic components are redrawn first, then OLE linked or embedded objects are redrawn. OLE objects always redraw in the same order and always redraw after SailWind Logic objects; therefore, they always appear on top of SailWind Logic components.

You can also choose to update linked and embedded objects when you redraw the work area using the Redraw OLE Objects check box on the Options > **Global** category. This option is grayed when no OLE objects exist.

OLE and View Menu Commands

You can use all of the **View** menu commands with OLE linked or embedded objects; you can zoom into and zoom out of the objects.

Changing the OLE Object Background Color

OLE linked or embedded objects are displayed with a solid white background. You may, in some cases, prefer to display the OLE object with a transparent background; for example, since a bitmap already contains a background, you may prefer to use a transparent background. If your object is a Word document, then you may prefer a white background because black text on a transparent background results in black on black, or an invisible object.

Procedure

1. Select the OLE object.
2. Right-click and click **White Background** to change the background color.

A check next to the command indicates that the object will use a white background.

Saving OLE Objects

Linked and embedded objects are automatically saved as part of the schematic when you save a SailWind Logic schematic. If you want to save OLE objects separately, use **File > Export** to save the objects in an .ole file. You can then use **File > Import** to import the objects into other designs.

SailWind Logic .ole files can be opened in other applications that understand the .ole file format. For example, if you insert a Word document into SailWind Logic and then save the Word object, you can later open Word and open the Word documents stored in the SailWind Logic .ole file.

Procedure

1. Click the **File > Export** menu item.
2. In the File Export dialog box, select the location for the file from the Save in dropdown list.
3. In the File name edit box, type a name for the OLE file you are saving.
4. From the Save as type dropdown list box, select OLE Files (*.ole).
5. Type a name for the OLE file you are saving.
6. Click **Save**.

Chapter 21

Using Advanced Procedures

SailWind Logic supports interaction with advanced procedures such as SPICE Simulation and Basic scripting. You can set up advanced SPICE design simulations to analyze your design behaviors, and use the Basic scripting to automate many repetitive design tasks. You can also view and manage your license file information.

[Spice Simulation](#)

[Basic Scripting](#)

[Managing Licensed Options](#)

Spice Simulation

The Spice simulation in SailWind Logic enables you to set up and analyze your design using advanced procedures including setting up Spice Netlists, AC Analysis, DC Source Sweep Analysis, Transient Analysis, and applying attributes to your analog designs..

- [Analog Schematics for Simulation](#)
- [Creating a SPICE Netlist](#)
- [Setting Up AC Analysis](#)
- [Setting Up DC Source Sweep Analysis](#)
- [Setting Up the SPICE Netlister](#)
- [Setting Up Transient Analysis](#)
- [Apply Attributes to Your Analog Design](#)

Analog Schematics for Simulation

You can create analog schematics and generate netlists for SPICE Simulators. Add analog attributes with simulation values to parts and nets for SPICE simulation.

Several analog parts are supplied in the *misc* library as examples. In addition, several examples of component simulation models are supplied in the Analog Models subfolder of the library.



Note:

Each SPICE model must have a separate *.mod* file. A model file can have subcircuit sections, but multiple models cannot be stored in one file since there is no mechanism for the software to extract the different models from within the file.

[Add SPICE Attributes to Library Parts Versus Schematic Parts](#)

[Adding SPICE Attributes to Schematic Parts or Nets](#)

Add SPICE Attributes to Library Parts Versus Schematic Parts

You can add SPICE attributes to library parts instead of parts in the schematic. When you add attributes to schematic parts, you must add them each time you use the part. When you add attributes to parts in the library; however, you don't have to add them again.

See also [Attributes Overview](#).

Adding SPICE Attributes to Schematic Parts or Nets

When you add attributes to schematic parts, you must add them each time you use the part. Consider adding the attributes to your library parts.

Procedure

1. Select a part or a net, right-click and click **Attributes**.
2. Add SPICE attributes to the part. You can also click **Browse Lib. Attr.** to browse the Attribute Dictionary for SPICE attributes.

See also SPICE Netlist Attribute Glossary in the *SailWind Logic Command Reference*.



Note:

Use the MODEL attribute to refer to component simulation models. Models take the device name with a *.mod* extension. Models are searched in the SailWind installation folder, SailWind libraries folder, SailWind project folder, and finally the library list using the search order. Each SPICE model must have a separate *.mod* file. A model file can have subcircuit sections, but multiple models cannot be stored in one file since there is no mechanism for the software to extract the different models from within the file.

Related Topics

[Creating a SPICE Netlist](#)

Creating a SPICE Netlist

After you add parts (with SPICE attributes) to your schematic, or add SPICE attributes to existing parts, you can create a SPICE netlist in preparation for simulation.

Procedure

1. Click the **Tools > SPICE Netlist** menu item.
 2. In the Output File Name box, use the default or type the path and name of the SPICE netlist file.
You can also use the **Browse** button to search for a path.
 3. In the Select Sheets area, select the sheets to include in the SPICE netlist. You can also use:
 - **Select All** — Selects all sheets in the Select Sheets box.
 - **Unselect All** — Clears all the selected sheets in the Select Sheets box.
-



Tip

Only complete schematic sheets can be simulated.

4. If the design is hierarchical, select the **Include Subsheets** check box to include any underlying hierarchy.
 5. In the Output Formats box, select the target SPICE software.
 6. Click **Simulation Setup** to change or enable the default simulation values.
-



Tip

Your simulation values are saved for future SPICE netlisting.

See also [Setting Up the SPICE Netlister](#).

7. Click **OK** to create the SPICE netlist.

An output window displays the resulting netlist. All warnings, errors, and comments during netlisting are embedded in the final netlist. You can edit the netlist before starting simulation.

Related Topics

[Analog Schematics for Simulation](#)

Setting Up AC Analysis

As part of setting up your SPICE netlister, you can set options specifically for an AC analysis.

Procedure

1. Click the **Tools > SPICE Netlist** menu item.
 2. In the SPICEnet dialog box, click **Simulation Setup**.
 3. In the Simulation Setup dialog box, click **AC Analysis**.
 4. In the Interval area, type the number of points and then select your choice of variation: Decade, Octave, or Linear.
 5. In the Frequency area, type a Starting and Ending frequency.
-



Tip

Your simulation setup values are saved for future SPICE netlisting.

For more information on AC Analysis, see the Help in your SPICE simulator.

Related Topics

[Setting Up the SPICE Netlister](#)

Setting Up DC Source Sweep Analysis

As part of setting up your SPICE netlister, you can set options specifically for a DC Sweep analysis.

Procedure

1. Click the **Tools > SPICE Netlist** menu item.
 2. In the SPICEnet dialog box, click **Simulation Setup**.
 3. On the Simulation Setup dialog box, click **DC Sweep**.
 4. In the Source box, type the name of the voltage or current source.
 5. In the Start box, type the starting voltage for the sweep.
 6. In the Stop box, type the stopping voltage for the sweep.
 7. In the Step box, type the incrementing values for the sweep.
-



Tip

Your simulation setup values are saved for future SPICE netlisting.

For more information on DC Source Sweep Analysis, see the Help in your SPICE simulator.

Related Topics

[Setting Up the SPICE Netlister](#)

Setting Up the SPICE Netlister

After you add parts (with SPICE attributes) to your schematic, or add SPICE attributes to existing parts, you can create a SPICE netlist in preparation for simulation.

Procedure

1. Click the **Tools > SPICE Netlist** menu item.
2. In the SPICEnet dialog box, click **Simulation Setup**.
3. Select the check boxes beside any analyses you want to enable.
4. Click an analysis button to change default simulation values.
 - **AC Analysis** — Directs the SPICE simulator to perform frequency analyses
 - **DC Source Sweep Analysis** — Directs the SPICE simulator to perform operating point analyses at specified values
 - **Transient Analysis** — Directs the SPICE simulator to perform time analyses
 - **Operating Point** — Directs the SPICE simulator to determine the DC operating point of the circuit



Tip

Your simulation setup values are saved for future SPICE netlisting.

For more information on Simulation Setup, see the Help in your SPICE simulator.

Related Topics

- [Creating a SPICE Netlist](#)
- [Setting Up AC Analysis](#)
- [Setting Up DC Source Sweep Analysis](#)
- [Setting Up Transient Analysis](#)

Setting Up Transient Analysis

As part of setting up your SPICE netlister, you can set options specifically for a Transient analysis.

Procedure

1. Click the **Tools > SPICE Netlist** menu item.
2. On the Simulation Setup dialog box, click **Transient**. The Transient Analysis dialog box displays.
3. In the Data Step Time box, type the increment for the analysis.

4. In the Total Analysis Time box, type the time to end the analysis.
5. In the Time to Start Recording Data box, type the time to start recording data from the analysis.
This is helpful if your simulation files become too large and you are not interested in data from the beginning of the analysis.
6. In the Maximum Time Step box, type a maximum time step value.
7. If you do not want SPICE to solve for the quiescent operating point before beginning the transient analysis, select the Use Initial Conditions check box.

If this option is enabled, SPICE uses the values specified using IC=... on the various elements as the initial transient condition and proceeds with the analysis.



Tip

Your simulation setup values are saved for future SPICE netlisting.

For more information on Transient Analysis, see the Help in your SPICE simulator.

Related Topics

[Setting Up the SPICE Netlister](#)

Apply Attributes to Your Analog Design

There is a list of attributes that you can apply to your analog design. Review the list and select those attributes that are relevant to your design intent.

Please see the “SPICE Netlist Attribute Glossary” in the *SailWind Logic Command Reference*.

Basic Scripting

Basic is a simple scripting language. Like many Windows applications, such as Microsoft Word and Excel, SailWind applications include Basic capabilities to enable users to customize their applications using a standard scripting language.

- [Managing Scripts](#)
- [Creation of Scripts](#)
- [Run Scripts](#)
- [Debug Scripts](#)
- [Accessing Help on the Basic Language](#)
- [Using the Basic Scripts Dialog Box](#)
- [Manage the Sax Basic Engine](#)
- [Basic Sample Scripts](#)

Managing Scripts

SailWind Logic offers a flexible number of options for managing scripts. You can open existing scripts, manage open scripts, edit scripts, edit user dialog boxes to enable user interaction, find automation statements, and watch variables. This gives you a high level of control over setting up and managing your scripts.

- [Opening an Existing Script](#)
- [Manage Open Scripts](#)
- [Editing a Script](#)
- [Editing a User Dialog Box](#)
- [Finding an Automation Statement](#)
- [Watching a Variable](#)

Opening an Existing Script

Scripts are created in and stored in script files that have a *.bas* extension. The default location for *.bas* files is *C:\SailWind Projects*.

Procedure

1. Click the **Tools > Basic Scripts > Basic Script Editor** menu item, then click the **Open** button.
2. Select the script and then click **Open**.

You can have up to nine scripts open at the same time.



Note:

A selection of sample scripts is available in the *C:\SailWind Projects\Samples\Scripts\Logic* folder.

Manage Open Scripts

The commands on the **Sheet** submenu provide script management methods. Since you can have up to nine scripts open at the same time, you can open #uses, close sheets, close multiple sheets, and choose scripts to view and edit.

Opening #uses Modules

#Uses modules are Basic scripts that are called from within other scripts. To open these secondary scripts:

- In the Basic Script Editor, right-click and select **Sheet > Open Uses**.

The #uses modules called in the script appear as script sheets in the Basic Script Editor. They are assigned a numbered tab and you can edit or run them.

Closing an Open Script

- In the Basic Script Editor, right-click and select **Sheet > Close**.

Alternatively, you can double-click the script's numbered tab in the gutter.

Closing all Open Scripts

- In the Basic Script Editor, right-click and select **Sheet > Close All**.

Viewing a Particular Script

If you have multiple scripts open, you can view a particular open script. You can have up to nine scripts open at the same time.

To view a particular script:

- Right-click and select **Sheet**. Then click the script you want to view from the list of open scripts on the submenu. Alternatively, you can click the script's numbered tab in the gutter.

Editing a Script

You can copy or cut selected text from the Basic Script Editor to the Clipboard. You can also paste a selection from the Clipboard into the text window. You can also paste text from the Clipboard into other applications.

Procedure

1. In the Basic Script Editor, select the text you want to copy or cut.
2. Right-click and click **Edit > Copy** or **Cut**.
3. Right-click and click **Edit > Paste** to paste the script text.

Your selection is pasted in the Output window at the insertion point.

As an alternative, you can click the **Copy**, **Cut**, and **Paste** buttons on the Basic Script Editor toolbar.

Editing a User Dialog Box

A UserDialog is defined by a Begin Dialog...End Dialog block.

Procedure

1. In the Basic Script Editor, put your cursor in a UserDialog block of the script.
2. Click the **Edit UserDialog** button.

See also *Sax Basic Editor On Line Help (C:\<install_folder>\<version>\Programs\sbe5_000.hlp)*

Finding an Automation Statement

If you are working with a long script, you can search for particular statements.

Procedure

1. In the Basic Script Editor, click the Object list and select an object type. The Object list shows all the objects for the current module. The (General) object groups all of the procedures that are not part of any specific object.
2. Click the Procedure list and select a bold procedure. The Procedure list shows all the procedures for the current object. Selecting a procedure that is bold locates the procedure in the script.

The statement appears in the Basic Script Editor.

Watching a Variable

Quick Watch shows the value of the expression under the cursor in the immediate window.

Procedure

Right-click and click **Quick Watch**.

As an alternative, in the Basic Script Editor, click the **Quick Watch** button.

See also *Sax Basic Editor Online Help*.

Creation of Scripts

You can create scripts to simplify redundant activities.

[Creating a Script](#)

[Inserting an Automation Statement Using the Object and Procedure Lists](#)

[Inserting an Automation Statement Using the ActiveX Automation Members Dialog](#)

[Setting the Next Statement](#)

[Showing the Next Statement](#)

Creating a Script

You can create a script using the Basic Script Editor.

Procedure

1. Click the **Tools > Basic Scripts > Basic Script Editor** menu item.

In SailWind Layout and SailWind Logic, the SAX Basic Engine dialog box appears.

2. Click the **New** button.

Inserting an Automation Statement Using the Object and Procedure Lists

Use the Object and Procedure lists to select and insert a statement. These lists contain the most commonly used statements.

Procedure

1. Click the Object list and click an object type. The Object list shows all the objects for the current module. The (General) object groups all of the procedures that are not part of any specific object.
2. Click the Procedure list and click a non-bold procedure to insert. The Procedure list shows all the procedures for the current object. Selecting a procedure that is not bold inserts the proper procedure definition for that procedure.

The statement appears at the bottom of the script.

Inserting an Automation Statement Using the ActiveX Automation Members Dialog

Use the ActiveX Automation Members dialog box to insert a statement from the extensive list provided.

Procedure

1. In the Basic Script Editor, right-click and click the **Debug > Browse** menu item.
2. Use the ActiveX Automation Members dialog box to select and insert a statement.

This dialog box contains an extensive list of statements.



Tip

If the pointer is on any line in the script other than the bottom line, the line is overwritten.

Setting the Next Statement

You can force a particular line in a script to run next. You can only select statements in the current subroutine or function.

Procedure

1. In the Basic Script Editor, put your cursor on the line you want to run next.
2. Right-click and click the **Debug > Set Next Statement** menu item.

An instruction pointer appears next to the selected line. This line, and only this line, will run next. If you go to other parts of the script, you can return to this line by clicking **Show Next Statement**.

Showing the Next Statement

In the Basic Script Editor, you can navigate the right mouse button menu to initiate the Show Next Statement command.

Procedure

In the Basic Script Editor, right-click and click **Debug > Show Next Statement** menu item.

An instruction pointer indicates the next statement to run.



Tip

Pausing a running script or setting a statement to run next sets the next statement. You can locate the set statement from anywhere in the script.

Run Scripts

You can run an existing script using Run. Run also resumes the playback of a paused script. When you run a script, you cannot use the mouse in the workspace.

- [Running a Script](#)
- [Pausing a Running Script](#)
- [Stopping a Running Script](#)

Running a Script

You can run a script using the Macro command in the Basic Script Editor.

Procedure

1. In the Basic Script Editor, open a script file.
2. Right-click and click the **Macro > Run** menu item.

As an alternative, on the Basic Script Editor toolbar, click the **Start/Resume** button.

Pausing a Running Script

When running a long script, you may need to pause it to perform some other design activity.

Procedure

1. In the Basic Script Editor, right-click and click the **Macro > Pause** menu item.
2. As an alternative, on the Basic Script Editor toolbar, click the **Pause** button.



Tip

If you paused the script, you can also use **Run**, **Step Over**, or **Step to Cursor** to resume running the script. Right-click and select **Run** to resume running the script.

Stopping a Running Script

You can stop a running script at any time. However, you cannot resume running a script once you have stopped it. When you click **Run**, the script starts from the beginning.

Procedure

In the Basic Script Editor, right-click and click the **Macro > End** menu item.

As an alternative, on the Basic Script Editor toolbar, click the **Stop** button.

Debug Scripts

When running a script, you can run it step-by-step or to a certain location in the script. To perform these debugging tasks, insert breakpoints in the script at the points at which you want the script to stop.

- [Setting or Removing the Breakpoints](#)
- [Debug the Scripts](#)
- [Removing All Breakpoints in the Script](#)
- [Correction of Run-Time Errors](#)

Setting or Removing the Breakpoints

The ability to set or remove breakpoints is useful when you debug a script. If the Basic engine encounters a breakpoint when running a script, it pauses the script.

Procedure

1. Place the cursor on the line to which to add a breakpoint.
2. On the Basic Script Editor toolbar, click the **Toggle Breakpoint** button.

As an alternative, in the Basic Editor, right-click and click the **Debug > Toggle Break** menu item.

This action inserts a breakpoint at the current cursor location. A breakpoint marker appears in the gutter area.



Note:

When the Basic engine encounters a breakpoint while running a script, it pauses the script. The next line in the script is marked with the instruction pointer.

Debug the Scripts

Once breakpoints are inserted, you can debug scripts using the Debug commands.

To run a single line of the script:

- On the Basic Script Editor toolbar, click the **Step over** button.

To perform a subroutine call on the current line:

- On the Basic Script Editor toolbar, click the **Step into** button.

As an alternative, in the Basic Script Editor, right-click and click the **Debug > Step Into** menu item.

To return from the subroutine to the point from which it was called:

- On the Basic Script Editor toolbar, click the **Step out** button.

To run a script to a point:

- In the Basic Script Editor, right-click and click the **Debug > Step to cursor** menu item.

To continue the execution from the current point:

- On the Basic Script Editor toolbar, click the **Run** button.

As an alternative, in the Basic Script Editor, right-click and click the **Macro > Run** menu item.

Removing All Breakpoints in the Script

You have the option to remove all breakpoints instead of single breakpoints.

Procedure

In the Basic Script Editor, right-click and click the **Debug > Clear All Breaks** menu item.

This removes all breakpoints in the script.

Correction of Run-Time Errors

If run-time errors occur, the script debugger switches to step-by-step mode and displays a detailed message on the status bar. The instruction pointer is set on the line that produced the error. After fixing the error, you can resume running the script.

Accessing Help on the Basic Language

While writing or running scripts, you can access Help that provides information and a sample script using the Basic language statements.

Procedure

Select or click in an item in color in the edit area of the Basic Script Editor and then press F1.

Help appears for the current statement.

Using the Basic Scripts Dialog Box

The Basic Scripts dialog box provides easy access to your Basic scripts. From there you can edit and debug your scripts.

Procedure

1. Click the **Tools > Basic Scripts** menu item, then click **Basic Scripts**.
 2. Select the script you want to manage.
 3. To run the script, click **Run**.
-



Restriction:

You cannot run multiple scripts at the same time.



Tip

If the selected script has an error during compilation, it automatically opens in the Basic Script editor for correction.

4. To edit the script, click **Edit**. The Sax Basic Engine dialog box appears.

See also [Manage the Sax Basic Engine](#).

5. To add the script to the **Basic Scripts** menu, click the **In Menu** check box.
 6. To remove the script from the list, click **Unload File**.
 7. To add a new script to the list, click **Load File**.
-



Tip

When loading scripts, note the following:

- You can load up to 32,767 scripts. Scripts are not compiled when they are loaded; they are compiled when you run them.
 - The list of scripts you load into this dialog box is saved in the **VBScripts.ini** file, so they load every time you open the Basic Scripts dialog box.
-

Related Topics

[Basic Sample Scripts](#)

Manage the Sax Basic Engine

The Sax Basic Engine dialog box provides access to the Sax Basic Engine Script editor. You can design, develop and edit scripts that add to, replace, enhance, or customize existing SailWind Logic features.

Scripts written in the Basic Script editor comply with all Microsoft requirements in terms of Visual Basic syntax; therefore, you can play these scripts in any other Visual Basic interpreter, such as Word or Excel. However, you cannot run all Basic scripts created outside of the Sax Basic Engine within the Sax Basic Engine because the Sax Basic Engine is a subset of Visual Basic. For example, you cannot run the Automation samples within the Sax Basic Engine.

- Click the **Tools > Basic Scripts > Basic Script Editor** menu item, then click **New** button.

See also [Sax Basic Editor On-line Help](#).

There are four types of scripts: scripts, code modules, object modules, and class modules.

You can create a script that calls another script. For example, ScriptA can call ScriptB.

You can also create a script that runs a series of scripts, or a "master" script. For example:

```
'$Include: "scriptA.bas"  
'$Include: "scriptB.bas"  
'$Include: "scriptC.bas"  
Sub Main
```

```
Call scriptA  
Call scriptB  
Call scriptC  
End Sub
```

Editor Colors

The Basic Script Editor displays source code using different colors. The color is context-sensitive: when you place the cursor on the text and press F1, the correct help file opens to the correct help topic. For example, if the cursor is on a SailWind Logic Automation Object (purple) when you press F1, the Automation Server On-line Help appears.

Table 42. Basic Script Editor Color Representations

Color	Represents
Blue	Basic Keywords
Black	User Variables
Cyan	Basic Functions
Purple	SailWind Logic Automation Objects or Members
Red	Errors
Green	Comments

Basic Sample Scripts

To address common design scenarios, and to provide a starting point for the development of custom user scripts, SailWind Logic includes a collection of sample Basic scripts. All sample scripts are commented.

[Basic Sample Scripts 00 Through 11](#)

[Basic Sample Scripts — RGL Reports](#)

[Basic Sample Scripts — Advanced](#)

Basic Sample Scripts 00 Through 11

Samples 00 through 09 provide an overview of Basic if you don't have experience with Basic scripts. Samples 10 and 11 provide small SailWind Logic features that add, enhance, or customize a SailWind Logic feature.

These files are located in *C:\<install_folder>\<version>\Samples\Scripts\Logic\tutorial*.

Table 43. Basic Sample Script Listing

Script Filename	Description
00 What is a Script.BAS	Empty script demonstrating what a Basic script is and how to define it.
01 Using a Message Box.BAS	Demonstrates how to display an OK dialog box.
02 Using a Variable.BAS	Demonstrates a Basic variable: how to assign a value and how to get its value.
03 Using a Basic Function.BAS	Demonstrates how to use a standard Basic function and display its result in a message box.
04 Using a SailWind Logic Function.BAS	Demonstrates Basic interaction with a SailWind Logic Automation function.
05 Using If and Then Statements.BAS	Demonstrates the If, Then statements.
06 Using a Custom Dialog1.BAS	Demonstrates a simple dialog box using the Basic dialog editor.
07 Using a Custom Dialog2.BAS	Demonstrates a standard dialog box using the Basic dialog box editor.
08 Using a Custom Dialog3.BAS	Demonstrates a complex dialog box using the Basic dialog box editor.

Table 43. Basic Sample Script Listing (continued)

Script Filename	Description
09 Using it All Together.BAS	Provides a "real life" example. Lists all design files in the default files directory. Selecting a file from the list will open that file in SailWind Logic.
10 List Of Comps and Nets.BAS	Lists all components and nets.
11 Select by Pin Count.BAS	Enables you to enter a number of pins. All parts with that number of pins are selected.

Basic Sample Scripts — RGL Reports

Basic scripts are equivalent to existing RGL reports.

These files are located in *C:\<install_folder>\<version>\Samples\Scripts\Logic\rgl samples*.

Table 44. Basic Sample Scripts/RGL Reports Listing

Script Filename	Description
RGL.BAS	Contains a library of functions, which is used by the other scripts in this group; the scripts in this group must contain RGL.BAS to function.
Net Statistics.BAS	Lists all nets in the schematic and identifies any questionable nets.
Part Statistics.BAS	Lists all parts in the schematic.
Unused Gates.BAS	Lists all unused gates in the schematic.
Unused Pins.BAS	Lists all unused pins in the schematic.
Unused.BAS	Lists all unused pins and gates in the schematic.

Basic Sample Scripts — Advanced

Advanced Basic script files are located in *C:\<install_folder>\<version>\Samples\Scripts\Logic*.

Table 45. Basic Sample Scripts/Advanced Listing

Script Filename	Description
Add Part.BAS	Adds a new gate to the schematic.
Alive Net List.BAS	Creates a netlist in Excel, which enables you to cross-probe between SailWind Logic objects and the Excel cells containing the object names.
Bill of Materials.BAS	Produces a bill of materials in a user-customizable format.

Table 45. Basic Sample Scripts/Advanced Listing (continued)

Script Filename	Description
Modeless Attributes.BAS	Generates a modeless dialog box, which you use to manage part attributes.
Modeless QM Part.BAS	Generates a modeless dialog box, which you use to manage part and gate properties.
Modeless Visibility.BAS	Generates a modeless dialog box, which you use to manage gate and attribute visibility.
SailWind Layout Net List Without Rules.BAS	Generates a netlist report in ASCII format.
SailWind Logic Script Wizard.BAS	Generates a Wizard dialog box, which you use to create a Basic report.
Sheet Hierarchy to Excel.BAS	Creates a sheet hierarchy report in Excel, which enables you to cross-probe between SailWind Logic objects and the Excel cells containing the object names.

Managing Licensed Options

You can view and manage your licensed SailWind Logic options. You can view the available options for node-locked or floating licenses.

[Viewing a License File or License Status](#)

[License File Definition](#)

Viewing a License File or License Status

If you are using node-locked licensing, you can view the contents of a license file. If you are using floating licenses, you cannot view the actual license file, but you can view the status of the features associated with a server license.

Procedure

1. Click the **Help > Installed Options** menu item, then click the **License File** tab.
2. Select one of these methods to view the license file:
 - **For Node-locked Licenses** — To view a license file:
 - i. Select the license file that you want to view from your list of license files.
 - ii. Click **View**. The bottom portion of the screen displays the contents of the selected file.
 - **For Floating Licenses** — To view the status of the features associated with a server license:
 - i. Select the server license file for which you want to display feature status.
 - ii. Click **Status**. The feature usage information appears in the bottom portion of the screen.

Related Topics

[Installed Options Dialog Box](#)

[License File Definition](#)

License File Definition

This section describes the license file and the possible options enabled in it.

All installed options appear in the [Installed Options Dialog Box](#) on page 394.

Options

```
FEATURE SailWindLogic paizieda \
```

Controls entry into the program and provides access to the basic SailWind Logic schematic functions. This feature is not listed on the Installed Options dialog.

Chapter 22

Custom Interface

You can customize the interface of SailWind tools by customizing elements of the interface such as toolbars, menus, and shortcut keys or by customizing the way windows of the interface are displayed.

[Customizing the SailWind Interface](#)

[Organizing Windows](#)

Customizing the SailWind Interface

You can customize the SailWind interface to suit your work style and design work. You can determine which toolbars are displayed, add items to toolbars and menus, and create custom toolbars, menus and shortcut keys.

To make customizations, use the Customize dialog box. You can invoke the dialog box in two ways:

- From the SailWind interface, click the **Tools > Customize** menu item. All customizations you make are applied to the main view of the SailWind tool.
- In a window of the interface (for example, the Output Window), right-click and select **Customize**. Your customizations apply only to that window.

Your customizations are saved with your current workspace so that all of the changes you make to toolbars, menus, and shortcut keys are present when you work in that workspace again.

[Customizing Toolbars](#)
[Creation of Custom Commands](#)
[Creating a Custom Menu](#)
[Adding Items to Toolbars and Menus](#)
[Moving Buttons on Toolbars](#)
[Moving Items on Menus](#)
[Removing Items From Toolbars and Menus](#)
[Customizing Shortcut Keys](#)
[Assigning Shortcut Keys to Macros](#)
[Creating a Command From a Macro and Adding it to a Menu](#)
[Customized Appearance of the Screen](#)
[Resizing the Sheets List](#)

Customizing Toolbars

Use the **Toolbars and Menus** tab on the Customize dialog box (**Tools > Customize > Toolbars and Menus** tab) to create custom toolbars and shortcut menus.



Tip

To create a custom main menu, use the **Commands** tab on the Customize dialog box. See [Creating a Custom Menu](#).

See also [Moving Buttons on Toolbars](#).

- [Creating a Custom Toolbar](#)
- [Showing or Hiding a Toolbar](#)
- [Deleting a Custom Toolbar](#)
- [Renaming a Custom Toolbar](#)
- [Resetting Toolbars to Defaults](#)

Creating a Custom Toolbar

To create a custom toolbar, you create a new empty toolbar and add items (commands) to it.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Toolbars and Menus** tab.
2. In the Toolbars box, click the **New** button.
3. Type the name for the toolbar and click **OK**.

This results in the following:

- The new (empty) toolbar appears on the SailWind interface.
 - The **Toolbars and Menus** tab lists the new toolbar, showing it as selected and enabled for display (the check box to the left of its name is selected).
4. Drag the toolbar to the place on the SailWind interface where you want it.
 5. To add items (commands) to your new toolbar, click the **Commands** tab.
 6. In the Categories list, select a menu or toolbar name to display commands specific to that menu or toolbar. Or select All Commands.



Restriction:

If you are working in a special mode in SailWind Layout or SailWind Logic (for example, the Decal Editor in SailWind Layout), some categories of commands are not available for customization.

7. In the Commands list, select the command you want and drag it to the toolbar.
8. When you have finished adding commands, click **Close**.

Showing or Hiding a Toolbar

To increase space in the software interface, you can show the toolbars you need to use and hide others.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Toolbars and Menus** tab.
2. In the Toolbars list, select the toolbar.
3. To display the toolbar in the interface, select the check box to the left of its name. To hide the toolbar, clear the check box.
4. Click **Close**.



Tip

For information on other ways you can customize the appearance of toolbars and menus, see [Customized Appearance of the Screen](#).

Deleting a Custom Toolbar

If you decide it is no longer needed, you can delete a custom toolbar (a toolbar you created).

Restrictions and Limitations

You cannot delete a system toolbar.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Toolbars and Menus** tab.
2. In the Toolbars list, select a custom toolbar. Then click the **Delete** button.

Renaming a Custom Toolbar

After it has been created, you can rename a custom toolbar (a toolbar you created).

Restrictions and Limitations

You cannot rename a system toolbar.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Toolbars and Menus** tab.
2. In the Toolbars list, select a custom toolbar and click the **Edit** button.
3. In the Toolbar Name dialog box, type the new name and click **OK**.

Resetting Toolbars to Defaults

If you decide changes you have made are no longer needed, you can reset one or all system toolbars to their default buttons.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Toolbars and Menus** tab.
2. In the Toolbars list, select the toolbar.
3. Click **Reset**.



Tip

To reset all system toolbars to defaults, click **Reset All**.

Creation of Custom Commands

Use the **Commands** tab to create commands that you can then use as selections on menus or as buttons on toolbars and to create custom menus.

- [Creating a Custom Command](#)
- [Defining Properties for a New Command](#)
- [Editing a Custom Command](#)
- [Deleting a Custom Command](#)

Creating a Custom Command

You can create a custom command from a command that already exists as a menu item or toolbar button. To create this kind of command, you select an existing command on which to base your new command. Then you define the properties of your new command.

You can also create a custom command from a macro command file. See [Creating a Command From a Macro and Adding it to a Menu](#).

Procedure

1. Click the **Tools > Customize** menu item, then click the **Commands** tab.
2. In the Categories list, click a menu or toolbar name to display items (commands) specific to that menu or toolbar, or click All Commands.



Tip

If you made macro commands (on the **Macro Files** tab) available as commands, the Categories list includes the Macro category and the Commands list includes the macros. For more information, see [Creating a Command From a Macro and Adding it to a Menu](#).

3. In the Commands list, select the command on which you want to base your custom command, and then click the **New** button.
4. In the Add Command dialog box, specify the properties of your new command:
 - a. In the Command name box, type the name of the command.
 - b. In the Description box, type a description of the custom command.
 - c. If an image was associated with the original command, select “Use Default Image” to use that same image with your custom command. Select “Select User-Defined Image” to use a different image, edit an image, or create a new image.
 - d. Click **OK** to close the Add commands dialog box and return to the Customize dialog box.
5. If you are finished with all customizations, click **Close**.



Tip

To add the command to a toolbar or menu, click the command and drag it from the Commands list to the toolbar or menu.

Defining Properties for a New Command

Once you create a new command, you can use the **Commands** tab to set the properties for a new command.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Commands** tab.
2. In the Commands list, click New, then click the **New** button.
The Add command dialog box opens.
3. In the Command name box, type the name of the new command.
4. (Optional) In the Based on box, type the name of the command on which the new command is based.
5. In the Description box, edit the command description, for example, to represent the argument values you added.
6. If an image was associated with the original command, do one of the following:
 - Click “Use Default Image”.
 - Click “Select User-defined Image”, then select a image, edit an image, or create a new one.
7. Click **OK** to close the Add Command dialog box and return to the Customize dialog box.

Editing a Custom Command

You can edit only custom commands (commands you created). You cannot edit system commands.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Commands** tab.
2. In the Categories list, click a menu or toolbar name to display items (commands) specific to that menu or toolbar, or click All Commands.
3. In the Commands list, select a command and click the **Edit** button.
4. In the Edit commands dialog box, change the properties of your custom command:
 - a. In the Command name box, type the name of the command.
 - b. In the Description box, type a description of the custom command.
 - c. If an image was associated with the original command, select “Use Default Image” to use that same image with your custom command. Choose “Select a User-Defined Image” to use a different image, edit an image, or create a new image.
 - d. Click **OK** to close the Edit commands dialog box and return to the Customize dialog box.

5. When you are finished with all customizations, click **Close**.
6. Click **OK** to close the Edit commands dialog box and return to the Customize dialog.

Deleting a Custom Command

You can delete only custom commands (commands you created). You cannot delete system commands.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Commands** tab.
2. In the Categories list, click a menu or toolbar name to display items (commands) specific to that menu or toolbar, or click All Commands.
3. In the Commands list, select a command and click the **Delete** button.
4. Click **Close**.

Related Topics

[Adding Items to Toolbars and Menus](#)

[Resetting Toolbars to Defaults](#)

Creating a Custom Menu

Create a custom menu, you first create a new empty menu and then add items (commands) to it:

Procedure

1. Click the **Tools > Customize** menu item, then click the **Commands** tab.
2. In the Categories list, select New Menu.
3. In the Commands list, select New Menu and drag it to the location you want.
 - To create a top-level menu, drag the new menu to the Menu Bar.
 - To create a submenu, drag it over an existing menu name.
4. Click your new menu to select it. Then right-click and select **Button Appearance**.
5. In the Button text field, type the name for the menu and click **OK**.
Leave the Customize dialog box open.
6. To add items (commands) to your new menu, click the **Commands** tab.
7. In the Categories list, select a menu or toolbar name to display commands specific to that menu or toolbar. Or select All Commands.



Restriction:

If you are working in a special mode in SailWind Layout or SailWind Logic (for example, the Decal Editor in SailWind Layout), some categories of commands are not available for customization.

8. In the Commands list, select the command you want and drag it to the menu.
9. When you have finished adding commands, click **Close**.

Adding Items to Toolbars and Menus

If there are commands that you frequently use that are not on the standard toolbars and menus, or you want to include them on frequently-used toolbars and menus, you can add these items.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Commands** tab.
 2. In the Categories list, select a toolbar or menu name to display commands specific to that menu or toolbar, or select All Commands.
-



Restriction:

If you are working in a special mode in SailWind Layout or SailWind Logic (for example, the Decal Editor in SailWind Layout), some categories of commands are not available for customization.

3. In the Commands list, select the command you want and drag it to the toolbar or menu.
-



Tip

To remove an item from a toolbar or menu (while the Customize dialog box is open), click the item and drag it outside the toolbar or menu.

4. When you have finished adding commands, click **Close**.

Related Topics

[Moving Items on Menus](#)

Moving Buttons on Toolbars

You can rearrange buttons on a toolbar. You can also move or copy a button from one toolbar to another.

Procedure

1. Click the **Tools > Customize** menu item.
 2. With the Customize dialog box is open, click the toolbar button and drag it to a new place on the same toolbar or to a different toolbar.
-



Tip

Instead of moving a button, you can copy it and move the copy. Press and hold the Ctrl key while dragging the button.

Moving Items on Menus

You can rearrange items on a menu. You can also move or copy an item from one menu to another.

Restrictions and Limitations

To move menu items, the Customize dialog box must be open.

Procedure

1. Click the **Tools > Customize** menu item.
 2. In the main window of the SailWind application, display the menu containing the item you want to move.
 3. Click the menu item and drag it to its new location on the same menu or to a different menu.
-



Tip

Instead of moving a menu item, you can copy it and move the copy. Press and hold the Ctrl key while dragging the item.

4. Click **Close**.

Related Topics

[Adding Items to Toolbars and Menus](#)

[Resetting Toolbars to Defaults](#)

Removing Items From Toolbars and Menus

If there are items that you do not use or want to appear, you can remove a menu item or toolbar button.

Procedure

1. Click the **Tools > Customize** menu item.
2. With the Customize dialog box is open, drag the item outside the toolbar or menu.

3. Close the Customize dialog box.

Note that you can reset a toolbar or shortcut menu back to its default list of items. See [Resetting Toolbars to Defaults](#).

Related Topics

[Adding Items to Toolbars and Menus](#)

Customizing Shortcut Keys

You can create and customize shortcut keys by using the **Keyboard and Mouse** tab of the Customize dialog box (**Tools** menu > **Customize > Keyboard and Mouse tab**).

[Creating a New Shortcut Key](#)

[Listing Shortcut Keys](#)

[Expressions in Shortcut Keys](#)

[Deleting a Shortcut Key](#)

[Resetting Default Shortcut Keys](#)

Creating a New Shortcut Key

You create shortcuts that apply in any mode. Thus, the same shortcut key may have different functionality depending on the mode in which you are working.

Restrictions and Limitations

The first character may consist of the following, plus Alt, Ctrl, or Shift modifiers:

- All printable characters including Space and Tab
- All function keys
- Extended keys: Up, Down, Left, Right, Insert, Delete, Home, PageUp, PageDown, End
- Numerical keypad keys (when Num Lock is off): Up, Down, Left, Right, Insert, Home, PageUp, PageDown, Del, End, /, *, +, -
- Mouse pointer events: Click, Double-click, RotateForward, RotateBackward



Restriction:

Mouse pointer events cannot be combined with key sequences, although the Ctrl, Alt, and Shift modifiers are allowed.

Subsequent characters may consist of the following:

- Alphanumeric (a-z0-9)



Note:

There are some exceptions: Some combinations, like Alt+Tab, are intercepted by Windows and thus are not available.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Keyboard and Mouse** tab.

2. In the Mode box, select the mode to which you want to apply the shortcut.

The available commands for that mode appear in the Commands box.

3. In the Commands box, select the command for which you want to create a new shortcut.

If a shortcut already exists, it appears in the Current shortcuts box.



Tip

To replace an existing shortcut, click **Delete** to remove the existing shortcut, and create a new shortcut for the command.

4. Above the Current shortcuts box, click the **New** button to open the Assign shortcut dialog box.

5. Select one of the following types of shortcut:

- To assign shortcut keys, select “Press new shortcut key(s)”, and then press the keys that you want to use.
-



Tip

As you enter the new shortcut, similar shortcuts appear in the Similar shortcuts assigned to other commands box. This helps you to avoid creating a new shortcut that conflicts with an existing shortcut.

- To create a mouse action, select “or select a pointer event”, and then select a combination of list box options, mouse button events, and modifier keys.

6. Click **OK** to close the Assign shortcut dialog box.

The new shortcut appears in the Current shortcuts box on the Customize dialog box.

Listing Shortcut Keys

You can create a table of commands and the shortcuts assigned to them in an HTML file, letting you share the information over the Web with other members of the design team.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Keyboard and Mouse** tab.

2. Click **Report** and then select or type the HTML filename, then click **Save**.

A hyperlink to the file appears in the Output window, under the **Status** tab.

Expressions in Shortcut Keys

You can substitute a regular expression for characters in shortcut key command arguments.

Table 46. Expressions in Shortcut Keys

Expression	Use to
*	Match any number of characters.
?	Match any one character.
[set]	Match any character in the specified set. i Tip A set is composed of characters or ranges. A range has the form: Character Hyphen Character, such as A-Z or 0-9. The minimum set of characters supported in a set consists of [0-9a-zA-Z_].
[!set] or [^set]	Match any character not in the specified set.
\	To suppress the special syntactic significance of the characters ` [] * ? ! ^ - \ ' within a set, and to match the character exactly.

The following table shows examples of regular expressions used in command arguments using the preview.pcb design, see [Table 47](#).

Table 47. Shortcut Key Expression Examples

Shortcut key	Result
H A*	Highlights all nets starting with A, such as A00, A01, A02.
H +??	Highlights all nets starting with +, having two digits or characters after 0, such as +5V.
H A?0	Highlights all nets starting with A, ending with 0, and with any character in between, such as A00 and A10.
H [C-D]*	Highlights all nets starting with C or D, such as CLKIN, D00.
H [!C-D]*	Highlights all nets not starting with C or D, such as A00, GND.

Deleting a Shortcut Key

Delete shortcuts you no longer want to use, or as the first step to changing an existing shortcut.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Keyboard and Mouse** tab.
2. In the Mode box, select the mode for the shortcut you want to delete.
The available commands for that mode appear in the Commands box.
3. In the Commands list, select the command whose shortcut you want to delete.

4. In the Current shortcuts list, select the shortcut you want to delete.
5. Click the **Delete** button.

Resetting Default Shortcut Keys

If you have made modifications that you no longer require, you can restore all shortcut keys to the default settings.

Procedure

1. Click the **Tools > Customize** menu item, then click the **Keyboard and Mouse** tab.
2. Click **Reset All**.
3. On the confirmation dialog box, click **Yes**.

Assigning Shortcut Keys to Macros

To simplify your design tasks, you can create a shortcut key that executes a macro.



Tip

To assign a macro to a shortcut key, the macro command file (*.mcr*) must already exist. You can create a macro by recording it in a SailWind tool or scripting it in Macro language. For more information, see [Macros](#).

Procedure

1. Click the **Tools > Customize** menu item, then click the **Macro Files** tab.
2. In the Macro Command Files area, click the **New** button.
3. In the Open macro file dialog box, select the macro file you want, and then click **Open**.

The SailWind tool loads the macro and makes it available for use as a command (the check box to the left of the macro name is selected).



Tip

To close the macro file or make it unavailable in the Customize dialog box, clear the check box next to the macro name.

4. To assign the macro to a shortcut key, click the **Keyboard and Mouse** tab.
5. In the Mode list, select All modes.
6. In the Commands area, double-click Macros to display a list of available macros, and then select the macro you want.
7. In the Current Shortcuts area, click the **New** button.

The SailWind tool displays the Assign shortcut dialog box.

8. Select one of the following types of shortcut:

- To assign shortcut keys, select “Press new shortcut key(s)”, and then press the keys that you want to use. For detailed information about rules and restrictions for creating shortcut keys, see the restrictions and limitations section of [Creating a New Shortcut Key](#).

**Tip**

As you enter the new shortcut, similar shortcuts appear in the “Similar shortcuts assigned to other commands” box. This helps you to avoid creating a new shortcut that conflicts with an existing shortcut.

- To create a mouse action, select “or select a pointer event”, and then select a combination of list box options, mouse button events, and modifier keys.

9. Click **OK** to close the Assign shortcut dialog box.

The new shortcut appears in the Current shortcuts box on the Customize dialog box.

Creating a Command From a Macro and Adding it to a Menu

You can create commands from macro files and add them to toolbars and menus.

**Tip**

To create a command from a macro command file or add it to a menu, the macro command file (.mcr) must already exist. You can create a macro by recording it in a SailWind tool or scripting it in Macro language. For more information, see [Macros](#).

Procedure

1. Click the **Tools > Customize** menu item, then click the **Macro Files** tab.
2. Click the **New** button.
3. In the Open macro file dialog box, select the macro file you want to use as a command. Then click **Open**.

The SailWind tool loads the macro and makes it available for use as a command (the check box to the left of the macro name is selected).

**Tip**

To close the macro file, or make it unavailable in the Customize dialog box, clear the check box next to the macro name.

4. To add the macro to the menu, click the **Commands** tab.
5. From the Categories list, select Macros.
6. In the Commands list, select the macro and drag it to the menu.

7. When you finish adding macros, click **Close**.
-



Tip

You can use the same steps to add macros to shortcut menus and toolbars.

Related Topics

[Adding Items to Toolbars and Menus](#)

Customized Appearance of the Screen

You can customize the SailWind interface by toggling the appearance of tooltips, changing the button icon size, enabling personalized menus (shortened menus based on usage), changing the menu display, changing the Microsoft visual style of the windows and dialog boxes, and changing the interface language.

For more information, see “[Customize Dialog Box, Options Tab](#)” on page 523.

Resizing the Sheets List

You can change the width of the Sheets list on the Standard toolbar of the Schematic Editor.

Procedure

1. In the Schematic Editor, click the **Tools > Customize** menu item.
 2. On the Standard toolbar, select the Sheets list box.
 3. Resize as needed.
 4. Click **Close**.
-



Restriction:

You cannot use the Alt key to resize the Sheets list.

Organizing Windows

You can customize the way windows appear in your workspace. Use the available options to show windows, hide windows, detach a window from the current view, attach a window to the current view, embed windows within other windows, and manage your windows tabs.

[Showing Windows](#)

[Hidden Windows](#)

[Detaching Windows From the Current View](#)

[Attach Windows to the Current View](#)

[Embed Windows Within Other Windows](#)

[Managing Window Tabs](#)

Showing Windows

When you first start the application, several windows display. You can show, hide, and automatically hide any of the windows in the application.

Procedure

On the **View** menu, click the name of the window you want to show.

Your choices may include Navigation Window, Output Window, Project Explorer, Help Window, Spreadsheet, and Shortcut Dialog.

Hidden Windows

When the application opens, several windows are open in addition to your workspace. You can close some of these windows or hide them automatically to maximize your design space.

[Closing Windows](#)

[Hiding Windows Automatically](#)

Closing Windows

If you do not want a particular window to appear in the workspace, you can hide windows by closing them manually.

Procedure

1. Move your pointer to the title bar of the window you want to hide.
2. Click the small downward pointing arrow  on the right side of the window's title bar.
3. In the resulting menu, click **Hide**.

The window closes.

Hiding Windows Automatically

You can also set a window to hide automatically so that it appears when you hover the pointer near it, and automatically minimizes when you move the pointer away from it.

Procedure

1. Move your pointer to the right side of the title bar in the window you want to hide.
2. Click the thumbtack  in the window's title bar.

The thumbtack picture changes to point sideways . A new bar appears on the side of the interface. The side on which the bar appears depends on the location of the window. For example, if the Project Explorer is located on the left side of the user interface, when you click the Auto Hide setting from the menu, the new bar appears on the left side of the interface.

The new bar contains a tab that has the same name as the window.

3. Hover over the tab in the new bar.
4. The window reappears, covering the application.

4. Move the pointer away from the window. The window minimizes to a tab.



Tip

To turn off the Auto Hide feature, hover over the tab in the new bar so the window reappears. Then repeat steps 1-2 in reverse.

Detaching Windows From the Current View

You can detach a window from the current view. This is called floating. A floating window is not attached to the current view; instead, it hovers, blocking the view to anything below it.

Restrictions and Limitations

You cannot float a window that is currently set to hide automatically. Turn off the Auto Hide feature before floating a window.

Procedure

Double-click the window's title bar. The window detaches and you can move it to any part of the screen.



Tip

To undo the floating, see [Attach Windows to the Current View](#).

Attach Windows to the Current View

You can attach a window to the current view. This is called docking. A docked window is attached to the current view, and therefore does not block the view to anything below it. You can dock a window in its last docked location, or dock a window to a different location.

[Docking to the Last Location](#)

[Docking to a New Location](#)

Docking to the Last Location

If you have moved a window, you can use a title bar keystroke command to dock it to its last docked location.

Procedure

Double-click the window's title bar. The window reattaches to the interface.

Docking to a New Location

You can use a drag operation to dock a window to a new location.

Procedure

1. Using the title bar, drag the window.

When you start dragging the window, additional graphics appear in the user interface. At the edges of the user interface, arrows containing graphics appear, as shown in [Figure 7](#):

Figure 7. Window Dragging Graphic



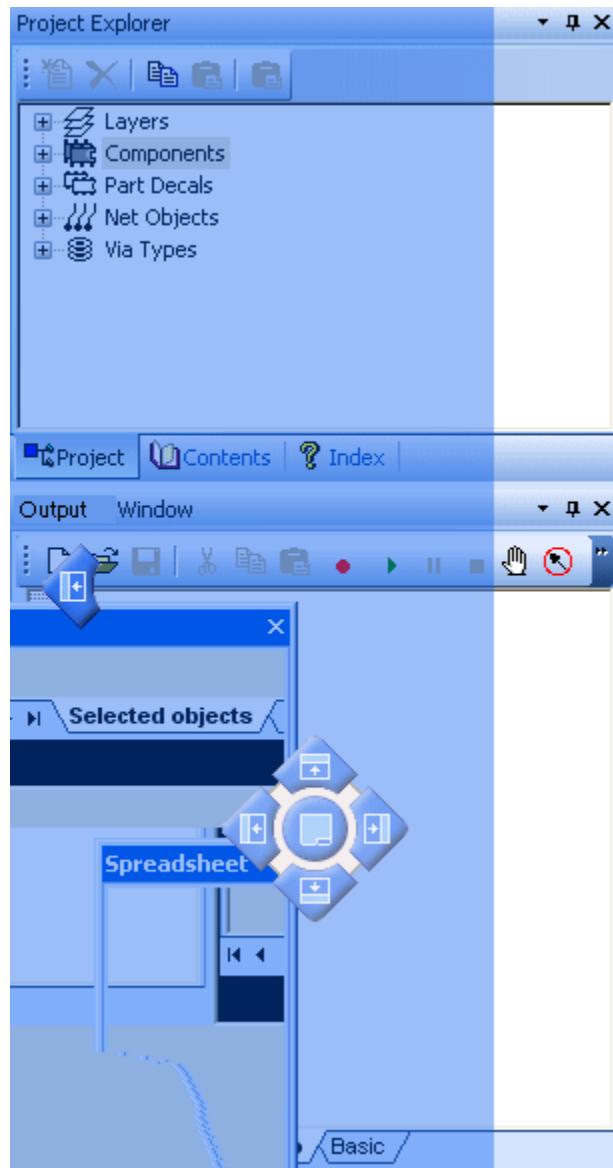
Tip

A similar group of arrows appears in a group near the center of the screen. Ignore that group of arrows for this procedure.

2. While dragging the window, hover over one of the arrows on the edge of the user interface. For example, hover over the arrow on the left side of the user interface.

A transparent colored block appears along the side of the user interface to which you are pointing. This block indicates where the window will be docked when you release the mouse button. For example, if you hover over the arrow on the left side of the user interface, a block appears along the left side of the screen, as shown in [Figure 8](#).

Figure 8. Docking a Window



3. Release the mouse button while hovering over the arrow that indicates where you want to dock the window.

The window docks to the user interface, and the other windows in the user interface resize.

Embed Windows Within Other Windows

In addition to attaching a window to a side of the user interface, you can embed a window within another window, so that it shares the window space with the original window, or becomes a tab within the original window.

[Two Windows Sharing One Window Space](#)
[Creating Tabs Within Windows](#)

Two Windows Sharing One Window Space

Use a drag operation to embed a window and share another window's space.

Procedure

1. Using the title bar, drag a window into another window.

When you start dragging a window, additional graphics appear in the user interface. A group of arrows containing graphics appears in the center of the window you are dragging, as shown in [Figure 9](#). Depending on the window you are dragging, the group of arrows may also have a tab graphic in the center.

Figure 9. Dragging a Window—Arrow Group



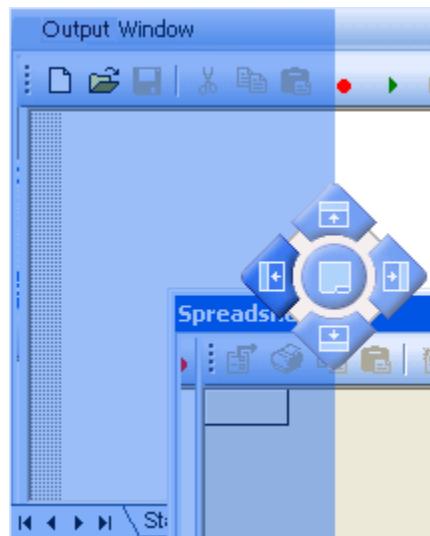
Tip

A similar group of arrows appears at the sides of the user interface. Ignore those arrows for this procedure.

-
2. While dragging the window, hover over one of the arrows. For example, hover over the left arrow.

A transparent colored block appears along the side of the window you are dragging, as shown in [Figure 10](#). This block indicates where the window will be docked when you release the mouse button. For example, if you hover over the left arrow, a block appears along the left side of the Project Explorer.

Figure 10. Dragging and Docking a Window



3. Release the mouse button while hovering over the arrow that indicates where you want to dock the window.

The window is embedded within another window, both sharing the space the original window occupied.



Tip

To maximize your workspace, try setting these embedded windows to hide automatically. Ctrl+click the thumbtack in one of the window's title bars, and all of the windows within the original window frame hide automatically.

Creating Tabs Within Windows

Embed a window and use the whole space by creating tabs.

Procedure

1. Using the title bar, drag a window into another window.

When you start dragging a window, additional graphics appear in the user interface. A group of arrows containing graphics appears in the center of the window you are dragging, as shown in [Figure 11](#). Depending on the window you are dragging, the group of arrows may also have a tab graphic in the center.

Figure 11. Dragging and Docking a Window—Arrow Commands





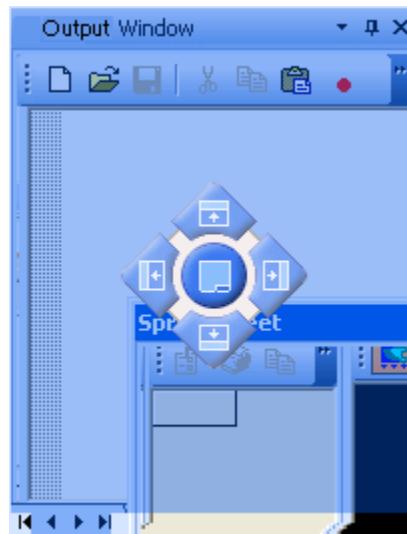
Tip

A similar group of arrows appears at the sides of the user interface. Ignore those arrows for this procedure.

2. While dragging the window, hover over the tab graphic.

A transparent colored block appears over the window you are dragging, as shown in [Figure 12](#). This block indicates where the window will be docked when you release the mouse button. For example, if you hover over the tab in the Project Explorer window, a block appears over the Project Explorer.

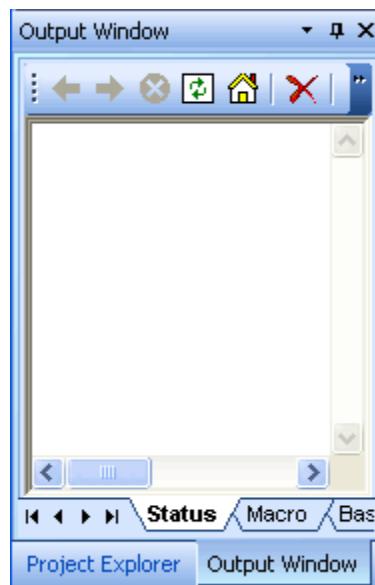
Figure 12. Dragging a Window—Transparent Block



3. Release the mouse button while hovering over the tab.

The window is embedded as a tab within a window, as shown in [Figure 13](#). You can click each tab to access each window.

Figure 13. Window Embedded as a Tab



Tip

To maximize your workspace, try setting these embedded windows to hide automatically. Ctrl+click the thumbtack in one of the window's title bars, and all of the windows within the original window frame hide automatically.

Managing Window Tabs

Some of the windows in the user interface contain tabs. However, you may decide you do not like the organization or grouping of the tabs. You can create additional tabs by embedding windows within other windows.

For more information, see “[Embed Windows Within Other Windows](#)” on page 419

[Rearranging Tabs in a Window](#)

[Moving Tabs Between Windows](#)

[Converting Tabs to Windows](#)

Rearranging Tabs in a Window

You can only rearrange tabs that you created by embedding a window within one of the docking windows. You cannot rearrange regular tabs, such as those in the Output window.

Procedure

Drag the tab to a new position within the row of tabs.

Moving Tabs Between Windows

You can only move tabs that you have embedded in other windows. In windows that have tabs by default (such as the Output Window in SailWind Router), you cannot move the tabs. You can only rearrange them. For information on rearranging tabs, see the previous section.

See also “[Embed Windows Within Other Windows](#)” on page 419.

Procedure

1. Drag the tab to a new window.

When you start dragging, the tab automatically behaves like a window.

2. Place the tab as you would a window.

See also “[Organizing Windows](#)” on page 414.

Converting Tabs to Windows

You can use a drag operation to create a new window from a tab.

Procedure

1. Drag the tab.

When you start dragging, the tab automatically behaves like a window.

2. Release the mouse button. Make sure the pointer is not over any arrow graphics.

You now have a floating window.

3. Place the tab as you would any floating window.

See also “[Organizing Windows](#)” on page 414.

Chapter 23

Crash Detection, BMW and BLT

If unexpected behaviors occur with the application, you can use crash detection to capture the crash event. You can then use the Basic Media Wizard and Basic Log Test tools to analyze the event and better understand the circumstances that may have caused the crash. You can capture all of this data into a log file that can be forwarded to SailWind Technical Support for further analysis and support.

[Crash Detection](#)

[BMW and BLT](#)

[Session Log Files](#)

[Session Media Files](#)

[Replaying Session Playback Media With BLT](#)

[The /BMW Command Line Switch](#)

[Scripting and Macros](#)

Crash Detection

The Error Detected dialog box opens at a crash and enables you to save a report of the SailWind environment as well as pertinent files into a compressed Dump File for troubleshooting.

This Error Detected dialog box is inaccessible unless the software crashes and crash detection is enabled in the software *.ini* file.

Crash detection is controlled by the *CrashDetection* switch in the *.ini* file; it is turned off by default.

- If no *CrashDetection* switch exists in the *.ini* file or if, in the [General] section of the *.ini* file, the switch exists with a value of 0 (zero), then crash detection is turned off. No report is created of the environment at the time of the crash.
- If the *CrashDetection* switch exists in the [General] section of the *.ini* file, or if the switch exists with a value of 1, then crashes are detected and the Error Detection dialog box appears.

BMW and BLT

BMW (Basic Media Wizard) and BLT (Basic Log Test) are tools that you can use to record and play back SailWind Logic, SailWind Layout and SailWind Router sessions. They are particularly useful as a means of supplying information to SailWind Software Support engineers trying to identify and resolve any problematical behavior you may encounter.

If you report problematical behavior for one of the SailWind tools to Technical Support, Tech Support engineers may ask you to use BMW to record session playback media documenting the actions that caused the problem. Tech support engineers can then replay the session with BLT to help them identify and resolve the problem.

[Creation of Session Playback Media With BMW](#)

[Creating Session Playback Media for a Normal Session](#)

[Automatically Creating Session Playback Media for a Crashed Session](#)

[Manually Creating Session Playback Media for a Crashed Session](#)

Creation of Session Playback Media With BMW

To create session playback media, BMW session logging must be enabled when the problem occurs. If session logging was not enabled when you encountered the problem, you must recreate the actions that caused the problem in a new session with session logging enabled.

Also, depending on whether the problem you want to document caused the SailWind tool to crash, you can create session playback media based on either the current or the immediately previous SailWind tool session.

The following table specifies which of the procedures described below you must use to create session playback media.

Was logging enabled?	Did the SailWind tool crash?	Then use this procedure.
Yes	No	Creating Session Playback Media for a Normal Session
No	No	Creating Session Playback Media for a Normal Session
Yes	Yes	Automatically Creating Session Playback Media for a Crashed Session
No	Yes	Manually Creating Session Playback Media for a Crashed Session

Creating Session Playback Media for a Normal Session

You can use the Basic Media Wizard (BMW) to create session playback media when the session you are recreating did not cause a SailWind tool crash.

Procedure

1. Start the SailWind tool, but do not open the file which you encountered the problematical behavior. (You must enable session logging before you open the file.)

2. Type the modeless command **BMW ON** and press Enter to enable session logging.

Logging remains enabled for this and all future sessions until you disable it with the **BMW OFF** command.

3. Open the file in which you encountered the problematical behavior.

4. Perform the series of actions that produced the problematical behavior.

The series of actions, as well as changes to the board or to the configuration, are stored in the [Session Log Files](#) for the current session.

5. Type **BMW** (the modeless command) and press the Enter key.

6. In the Media Wizard dialog box, click “Create Media from Current Session”.

7. Type your initials in the User Initials box. (They are included in the playback media filenames to identify the files as yours.)

8. To delete all entries in the session log file between the first Open and the last Save command, click “Delete Actions Before Last Save”.

You can do this to eliminate any actions you may have performed before beginning the series of actions that produced the problematical behavior. This makes it easier for the Tech Support engineer to identify the problem.

9. Click **OK** to create the [Session Media Files](#).

Automatically Creating Session Playback Media for a Crashed Session

You can use the Basic Media Wizard (BMW) to create session playback media when the session you are recreating caused the SailWind tool to crash, and none of the listed restrictions applies.

Restrictions and Limitations

- This procedure works only if the previous (crashed) session started with BMW logging already enabled and logging remained enabled throughout the session.
- This procedure does not give useful results if any additional instance of the SailWind tool ran concurrently (for any period) with the previous (crashed) session.

Procedure

1. Start the SailWind tool, but *do not* open the file which you encountered the problematic behavior. (You *must* enable session logging *before* you open the file.).

2. Type the modeless command **BMW ON** and press the Enter key to enable session logging.

Logging remains enabled for this and all future sessions until you disable it with the **BMW OFF** command.

3. Open the file in which you encountered the problematical behavior.
4. Perform the series of actions that produced the problematical behavior.

The series of actions, as well as changes to the board or to the configuration, are stored in the [Session Log Files](#) for the current session.

5. After the crash, restart the SailWind tool.

A dialog box is displayed asking if you want to save media files for the crashed session. Click **Yes** to create the [Session Media Files](#).

Manually Creating Session Playback Media for a Crashed Session

You can use the Basic Media Wizard (BMW) to manually create session playback media when the session you are recreating caused the SailWind tool to crash, and the automatic procedure cannot be used due to one of the restrictions listed in that section.

The automatic procedure is described in [Automatically Creating Session Playback Media for a Crashed Session](#).

Procedure

1. Start the SailWind tool, but *do not* open the file which you encountered the problematic behavior. (You *must* enable session logging *before* you open the file.).

2. Type the modeless command **BMW ON** and press the Enter key to enable session logging.

Logging remains enabled for this and all future sessions until you disable it with the **BMW OFF** command.

3. Open the file in which you encountered the problematical behavior.

4. Perform the series of actions that produced the problematical behavior.

The series of actions, as well as changes to the board or to the configuration, are stored in the [Session Log Files](#) for the current session.

5. After the crash, restart the SailWind tool.

6. Type **BMW** (the Modeless command) and press Enter.

7. In the Media Wizard dialog box, click “Create Media from Previous Session”.

8. Type your initials in the User Initials box to identify your session playback media files.

9. Click **OK** to create the [Session Media Files](#).

Session Log Files

Whenever BMW session logging is enabled, two sets of session log files are maintained in the `\SailWind\Projects` folder. These logs record actions performed in the current session, and in the immediately previous session.

BMW names these files as follows:

Current Session Log Files	Previous Session Log Files
<pads_tool>_Next.log	<pads_tool>_NextBak.log
<pads_tool>_Next.reg	<pads_tool>_NextBak.reg
<pads_tool>_Next.ini	<pads_tool>_NextBak.ini

These files are dynamic; each time you start a session, the current session log files are renamed as the previous session log files, and new current session log files are created. The contents of the old previous session log files are lost.

Whenever you elect to create session media files for a session, the appropriate set of these log files is saved in a permanent location, as described in [Session Media Files](#).



Tip

You may see a log file named <pads_tool>_Session.log listed in the \SailWind Projects folder. This file is unrelated to the session playback media created by BMW.

Session Media Files

Each time you create session playback media, BMW creates a new session media folder in the \SailWind Projects folder, and copies into it:

- The .pcb or .sch file for which you are recording the session
- The [Session Log Files](#) for the session. BMW then renames these files based on the session media folder name.

The session media folder is named <month><day><initials><sequential letter>, where:

- <month><day> is the date.
- <initials> are letters you type in the Media Wizard dialog box to personalize the media files.
- <sequential letter> is a letter automatically assigned to sequence the directories created on a specific date.

Example: \SailWind Projects\0530jsb represents a session media folder created on May 30, using the initials js, and that was the second session media folder created on that day.

When creating the session playback media, the following files are written to the session media folder:

Replaying Session Playback Media With BLT

You can use BLT to replay session playback media created by BMW.

Procedure

1. Type **BLT** (the Modeless command) and press Enter.
2. Select the session playback media from the Media Directories list and click **OK**.

The session is replayed.



Tip

To personalize the media folder and session playback media filenames, select a media session from the Media Directories list, type the new name into the New name box, and then click **Rename**.

The /BMW Command Line Switch

You can use a command line switch in the startup options of the software if you want to automatically record every SailWind Layout session.

If you want BMW to automatically prompt you to create media from the previous session each time you start a SailWind tool, open the SailWind tool using the /BMW command line switch. Or use /BMW-xx (where xx represents your initials, which are used in folder and filenames to identify them as yours).

When you use BMW as a command line option, it creates media of the previous session; use the *BMW* modeless command to create media of your current session.

Scripting and Macros

All scripting and macro documentation has been moved into a separate manual.

It can be found in the SailWind Logic Command Reference Manual.

Chapter 24

SailWind Logic GUI Reference

The section contains information on all of the GUI elements in SailWind Logic.

- [AC Analysis Dialog Box](#)
- [Add Attribute Label Dialog Box](#)
- [Add Bus Dialog Box](#)
- [Add/Edit Command Dialog Box](#)
- [Add Field Dialog Box](#)
- [Add Free Text Dialog Box](#)
- [Add Net to Class Dialog Box](#)
- [Add New Attribute Dialog Box](#)
- [Add New Attribute to Library Dialog Box](#)
- [Add Part From Library Dialog Box](#)
- [Add Pins Dialog Box](#)
- [Archiver Dialog Box](#)
- [Archiver Additional Files Dialog Box](#)
- [Archiver Libraries Dialog Box](#)
- [ASCII Output Dialog Box](#)
- [Assign Alternatives for Ground Part Dialog Box](#)
- [Assign Alternatives for Off-Page Part Dialog Box](#)
- [Assign Alternatives for Power Part Dialog Box](#)
- [Assign Decal to Gate Dialog Box](#)
- [Assign New Gate Decal Dialog Box](#)
- [Assign New PCB Decal Dialog Box](#)
- [Assign Shortcut Dialog Box](#)
- [Attribute Properties Dialog Box](#)
- [Auto Renumber Parts Dialog Box](#)
- [Basic Script Editor](#)
- [Basic Scripts Dialog Box](#)
- [Bill of Materials Setup Dialog Box](#)
- [Browse for Connectors Dialog Box](#)
- [Browse for Special Symbols Dialog Box](#)
- [Browse Library Attributes Dialog Box](#)
- [Bus Name Properties Dialog Box](#)
- [Bus Properties Dialog Box](#)
- [CAE Decal Wizard Dialog Box](#)
- [Capture a New View Dialog Box](#)
- [Change Part Type Dialog Box](#)
- [Check for Updates Dialog Box](#)
- [Class Rules Dialog Box](#)
- [Clearance Rules Dialog Box](#)

[Compare/ECO Tools Dialog Box](#)
[Conditional Rule Setup Dialog Box](#)
[Connect to SailWind Layout Dialog Box](#)
[Connect to SailWind Router Dialog Box](#)
[Crash Detected Dialog Box](#)
[Create PDF Dialog Box](#)
[Customize Dialog Box](#)
[DC Source Sweep Analysis Dialog Box](#)
[Default Rules Dialog Box](#)
[Differential Pairs Dialog Box](#)
[Display Colors Dialog Box](#)
[Display Colors Dialog Box - Part Editor](#)
[Drafting Properties Dialog Box](#)
[Edit Button Image Dialog Box](#)
[Fields Dialog Box](#)
[Field Properties Dialog Box](#)
[Font Replacement Dialog Box](#)
[Fonts Dialog Box](#)
[Get Drafting Items From Library Dialog Box](#)
[Get Gate Decal From Library Dialog Box](#)
[Get PCB Decal From Library Dialog Box](#)
[Hierarchical Symbol Wizard Dialog Box](#)
[HiSpeed Rules Dialog Box](#)
[Installed Options Dialog Box](#)
[Library List Dialog Box](#)
[Library Manager Dialog Box](#)
[Log Test Dialog Box](#)
[Logic Families Dialog Box](#)
[Make Field Dialog Box](#)
[Manage Library Attributes Dialog Box](#)
[Manage Schematic Attributes Dialog Box](#)
[Media Wizard Dialog Box](#)
[Modeless Commands and Keyboard Shortcuts](#)
[Net Attributes Dialog Box](#)
[Net Name Properties Dialog Box](#)
[Net Properties Dialog Box](#)
[Net Rules Dialog Box](#)
[Netlist to PCB Dialog Box](#)
[Off-Page Properties Dialog Box](#)
[Options Dialog Box](#)
[Options Dialog Box - Part Editor, General Category](#)
[Options Dialog Box - Print/Plot](#)
[Output Window](#)
[SailWind Layout Link Dialog Box](#)

SailWind Router Link Dialog Box
SailWind Suite Configuration Dialog Box
Part Attributes Dialog Box
Part Information Dialog Box
Part Properties Dialog Box
Part Signal Pins Dialog Box
Part Text Visibility Dialog Box
Part Type Label Properties Dialog Box
Pen Plotter Advanced Setup Dialog Box
Pen Plotter Setup Dialog Box
PCB Decal Assignment Dialog Box
Photo Plotter Advanced Setup Dialog Box
Photo Plotter Setup Dialog Box
Pin Decal Browse Dialog Box
Pin Decal List Management Dialog Box
Pin Label Fonts Dialog Box
Pin Properties Dialog Box
Plot Dialog Box
Print Dialog Box
Project Explorer
Query Hierarchical Component Dialog Box
Reference Designator Properties Dialog Box
Remap Special Symbols Dialog Box
Rename Gate Dialog Box
Rename Part Dialog Box
Renumber Pins Dialog Box
Report Manager Dialog Box
Reports Dialog Box
Routing Rules Dialog Box
Rules Dialog Box
Rules Report Dialog Box
Save CAE Decal to Library Dialog Box
Save Configuration Dialog Box
Save Drafting Item to Library Dialog Box
Save Off-Page to Library Dialog Box
Save Part and Gate Decals As Dialog Box
Save Part Types to Library Dialog Box
Save Part Type to Library Dialog Box
Save PCB Decal to Library Dialog Box
Save View Dialog Box
Select Gate Decal Dialog Box
Select Pin Decal Dialog Box
Select Type of Editing Item Dialog Box
Selection Filter Dialog Box

[Selections Preview Dialog Box](#)
[Server Busy Dialog Box](#)
[Sheets Dialog Box](#)
[Signal Pin Nets Dialog Box](#)
[Simulation Setup Dialog Box](#)
[SPICEnet Dialog Box](#)
[Step and Repeat Dialog Box](#)
[Text Properties Dialog Box](#)
[Transient Analysis Dialog Box](#)
[Update From Library Dialog Box](#)
[Update Selected CAE Decals From Library Dialog Box](#)
[Update Selected Part Type From Library Dialog Box](#)
[Update Selected Pin Decals From Library Dialog Box](#)

AC Analysis Dialog Box

To access: Tools > SPICE Netlist menu item > Simulation Setup button > AC Analysis button

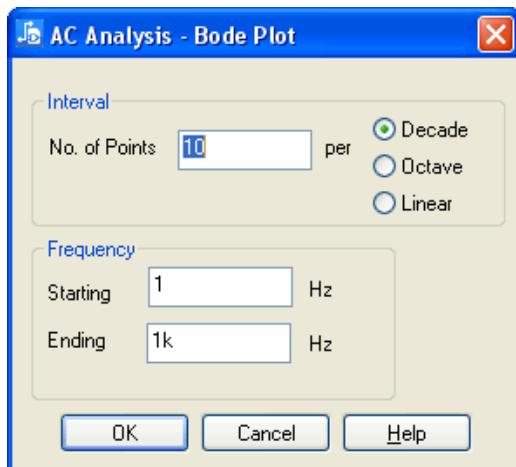
Set options specifically for an AC analysis.



Note:

Pictures in this document are for reference only, to help users better understand the software operation. In the case of interface difference due to version changes, the interface of SailWind Logic in practice shall prevail.

Figure 14. AC Analysis Dialog Box



Objects

Table 48. AC Analysis Fields

Name	Description
Interval area	Specifies the number of points and the variation: Decade, Octave, or Linear.
Frequency	Specifies the Starting and Ending frequencies.

Related Topics

[Creating a SPICE Netlist](#)

[Setting Up AC Analysis](#)

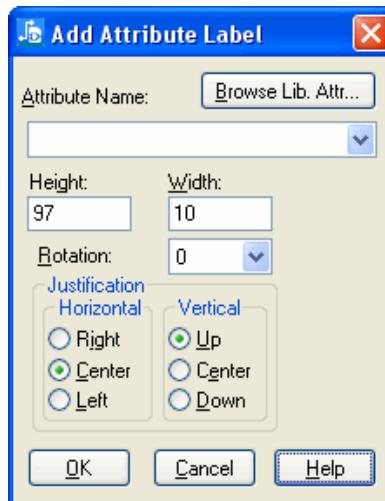
Add Attribute Label Dialog Box

To access:

- Select a part > right-click > **Edit Part** menu item > **Edit Graphics** button > select a gate > **Decal Editing Toolbar** button > **Add Attribute Label** button
- With nothing selected > **Tools** > **Part Editor** menu item > **Edit Graphics** button > **Decal Editing Toolbar** button > **Add Attribute Label** button

Use the Add Attribute Label dialog box to create attribute labels while editing or creating a CAE Decal.

Figure 15. Add Attribute Label Dialog Box



Objects

Table 49. Edit Attribute Label Fields

Name	Description
Attribute Name list	Lists all of the attributes available to you. Tip You can type the attribute name or browse the Library Attributes.
Browse Lib. Attr. button	Opens the Browse Library Attributes Dialog Box .
Height	Specifies the height of the text.
Width	Specifies the width of the text.
Rotation	Specifies the rotation of the text: 0 or 90 degrees.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

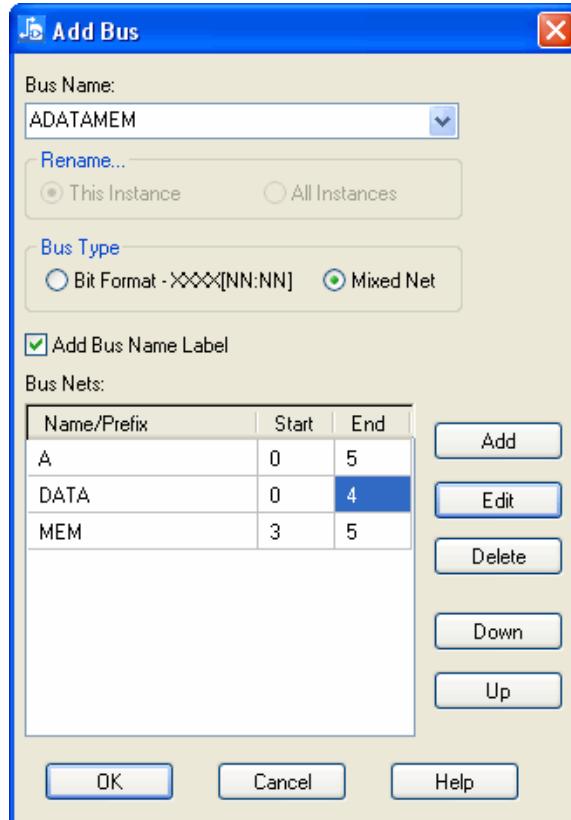
[Creating Attribute Labels](#)

Add Bus Dialog Box

To access: Schematic Editing toolbar > **Add Bus** button> click to indicate starting point > double-click to indicate ending point

Use the Add Bus dialog box to create attribute labels while editing or creating a CAE Decal.

Figure 16. Add Bus Dialog Box



Objects

Table 50. Add Bus Dialog Box Fields

Name	Description
Bus Name	Specifies the name of the bus. Select or type the name you want.
Rename area	Specifies to rename this instance or all instances of the bus.  Restriction: Available only in query mode.
Bus Type area	Specifies which bus names appear in the Bus Name list.

Table 50. Add Bus Dialog Box Fields (continued)

Name	Description
Add Bus Name Label	<p>Select the Add Bus Name Label check box to add the bus name as a label to the bus at the end of the bus closest to where you selected it.</p> <p> Restriction: The check box is unavailable when the end of the selected bus has a label.</p> <p> Tip</p> <ul style="list-style-type: none"> • A bus can have two labels, one on each end. • A bus label is not required. • To delete a bus label, select the label in the schematic and click the Delete button on the standard toolbar.
Bus Nets table	<p>Lists the name or prefix of the bus net, the starting bit number for a sequence of nets, and the ending bit number for a sequence of nets.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p> <p> Tip</p> <ul style="list-style-type: none"> • For a single net, type the net name. • For a range of sequential nets, type the prefix for the sequence of nets.
Add button	<p>Adds a row to the Bus Nets table.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>
Edit button	<p>Makes the selected row available for editing.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>
Delete button	<p>Removes the selected row from the Bus Nets table.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>
Down/Up buttons	<p>Moves the selected row up or down in the Bus Nets table.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>

Related Topics

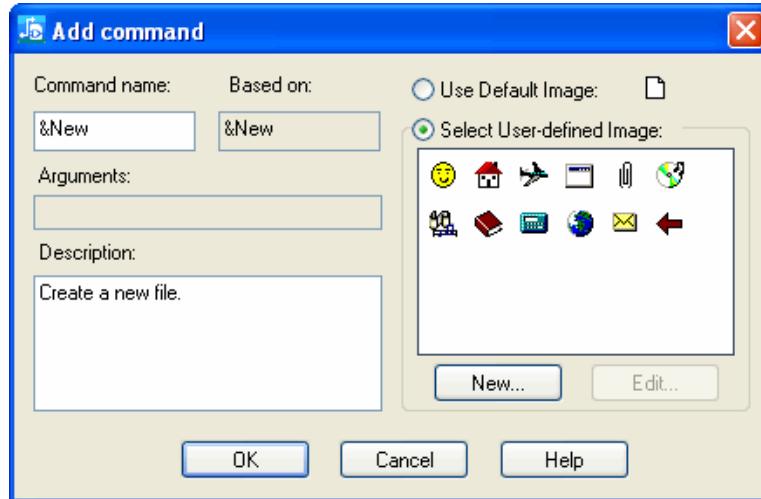
[Managing Buses](#)

Add/Edit Command Dialog Box

To access: Tools > Customize menu item > Commands tab > select a command > New or Edit button

Use the Add Command dialog box to create commands that you can then use as selections on menus or as buttons on toolbars.

Figure 17. Add Command Dialog Box



Objects

Table 51. Add/Edit Dialog Box Fields

Name	Description
Command name	The name of the new command. Tip Type an ampersand before the letter you want to use as the Alt keyboard shortcut.
Based on	The command on which you want to base the new command.
Arguments	Any arguments for the new command. Tip Use a space to separate arguments. If an argument contains a space, enclose the argument in quotation marks (""). Restriction: SailWind Router only.
Description	Lists what the new command does.
Use Default Image	Use the recommended image.

Table 51. Add/Edit Dialog Box Fields (continued)

Name	Description
Select User-defined Image	Select or create your own image to associate with the new command.
New button	Open the Edit Button Image Dialog Box .
Edit button	Open a button in the Edit Button Image Dialog Box .

Related Topics

[Creating a Custom Command](#)

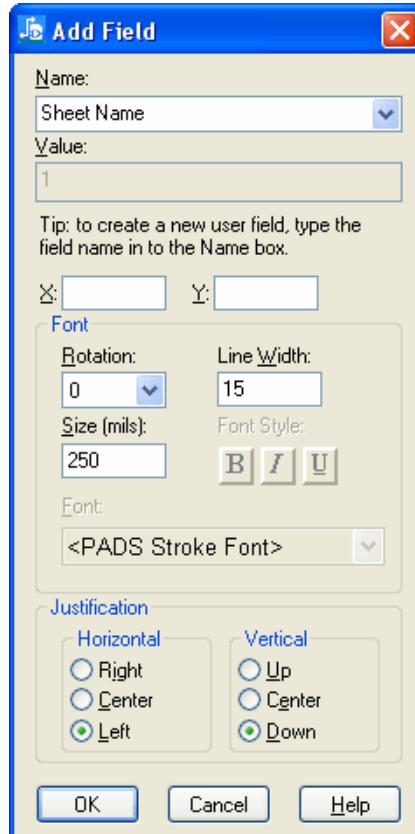
[Creating a Custom Menu](#)

Add Field Dialog Box

To access: Select nothing or a 2D line object > right-click > **Add Field** menu item

Use the Add Field dialog box to add a field to your schematic.

Figure 18. Add Field Dialog Box



Objects

Table 52. Add Field Dialog Box Fields

Name	Description
Name list	Type the name of a new field or select from a list of the fields available to you.
Value	The value of the field.  Restriction: The Value box is unavailable for system fields since the value is derived from your system.
X/Y	Type coordinates to place the field label in a specified location.

Table 52. Add Field Dialog Box Fields (continued)

Name	Description
	 Tip Leave these blank to attach the field to your pointer and click to indicate the location.
Rotation	Specifies the rotation of the text: 0 or 90 degrees.
Line Width	Specifies the line width for stroke fonts only.  Stroke Line Width
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.  Stroke Font - Size
Font Style	Enables you to change the font style to bold, italic, and underlined.  Restriction: System fonts only.
Font list	The fonts available to you. This lists either stroke fonts or system fonts. You choose which type of font to use in the Fonts Dialog Box .  Tip <ul style="list-style-type: none"> Select stroke font or a system font. For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

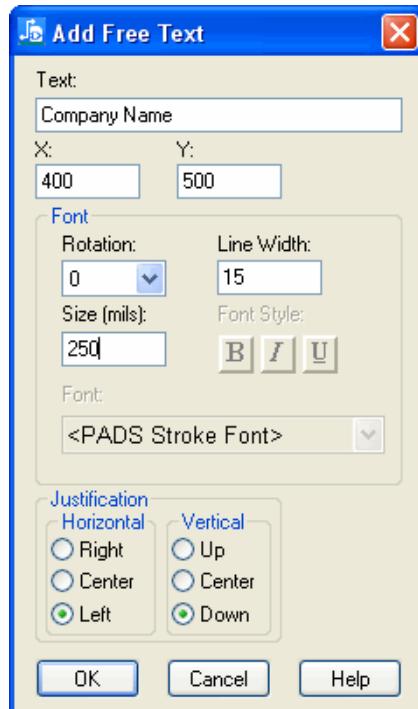
[Adding a Field](#)

Add Free Text Dialog Box

To access: Schematic Editing toolbar > **Create Text** button

Use the Add Free Text dialog box to add free text (not belonging to another object).

Figure 19. Add Free Text Dialog Box



Objects

Table 53. Add Free Text Dialog Box Fields

Name	Description
Text	Type the text you want in the schematic.
X/Y	Places the text in a specified location. Negative coordinates are not permitted. If you want to place text outside the sheet, you must move it there with the cursor. Tip Leave these blank to attach the text to your pointer and click to indicate the location.
Rotation	Specifies the rotation of the text: 0 or 90 degrees.
Line Width	Specifies the line width for stroke fonts only.

Table 53. Add Free Text Dialog Box Fields (continued)

Name	Description
	 Stroke Line Width
Size	<p>Specifies the size of the font.</p> <p>Size (pts): This is font size in points and appears for system fonts</p> <p>Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.</p>  Stroke Font - Size
Font Style	<p>Enables you to change the font style to bold, italic, and underlined.</p> <p> Restriction: System fonts only.</p>
Font list	<p>The fonts available to you. This lists either stroke fonts or system fonts. You choose which type of font to use in the Fonts Dialog Box.</p> <p> Tip</p> <ul style="list-style-type: none"> • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

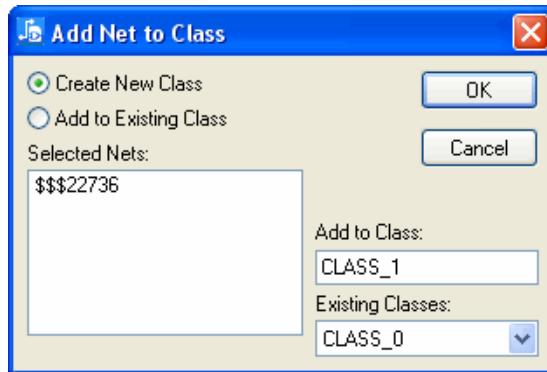
[Adding Text](#)

Add Net to Class Dialog Box

To access: With nothing selected, right-click > **Select Nets** > select one or more nets > right-click > **Make Class** menu item

Use the Add Net to Class dialog box to create a new class or to add nets to an existing class.

Figure 20. Add Net to Class Dialog Box



Objects

Table 54. Add Net Class Dialog Box Fields

Name	Description
Create New Class	Specifies to create a new class using the selected net(s).
Add to Existing Class	Specifies to add the selected net(s) to an existing class.  Restriction: This is unavailable if there are no existing classes.
Selected Nets	Lists all of the nets you selected in the schematic.
Add to Class	Specifies the name of the class. Type a new name or accept the default.  Restriction: This is unavailable if you select Add to Existing Class and only one class exists.
Existing Classes	Lists all of the existing classes in the schematic.

Add New Attribute Dialog Box

To access: **Edit > Attribute Manager** menu item > **Add Attr** button

Use the Add New Attribute dialog box to set name and value properties when adding new attributes to the schematic.

Figure 21. Add New Attribute Dialog Box



Objects

Table 55. Add New Attribute Dialog Box Fields

Name	Description
Browse Lib. Attr button	Opens the Browse Library Attributes Dialog Box .
Attribute Name	The name of the new attribute.
Attribute Value	The value of the new attribute.

Related Topics

[Manage Attributes in a Schematic](#)

Add New Attribute to Library Dialog Box

To access: **File > Library** menu item > Library Manager dialog box > **Attr Manager** button > **Add Attr** button

Use the Add New Attribute to Library dialog box to set name and value properties when adding new attributes to libraries.

Figure 22. Add New Attribute to Library Dialog Box



Objects

Table 56. Add New Attribute to Dialog Box Fields

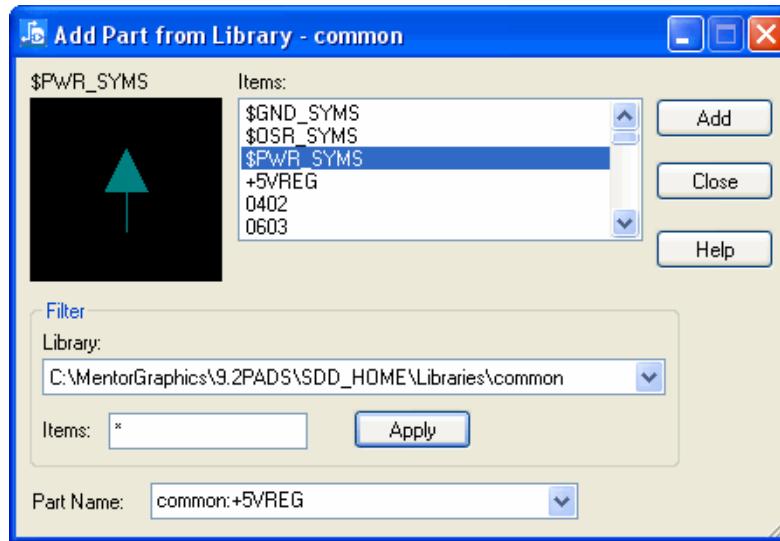
Name	Description
Browse Lib. Attr button	Opens the Browse Library Attributes Dialog Box .
Attribute Name	The name of the new attribute.
Attribute Value	The value of the new attribute.

Add Part From Library Dialog Box

To access: Schematic Editing toolbar > **Add Part** button

Use the Add Part from Library dialog box to load a part from a library into the current schematic drawing. SailWind Logic automatically assigns a reference designator when you add the part.

Figure 23. Add Part From Library Dialog Box



Objects

Table 57. Add Part From Library Dialog Box Fields

Name	Description
Preview area	Shows the selected item.
Items	Lists the items in the selected library. The number of objects that appear depends on the filter settings.
Library list	Specifies the library you want to use.
Items	Narrows the search. You can use wildcards or expressions on page 105. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.
Part Name list	Select a recently used part.
	<p>Tip</p> <p>The sixteen most recently used parts are available in the Part Name dropdown list box. You can clear this buffer by removing the entries in the <i>SailWindlogic.ini</i> file, under the [Last Added Parts] heading.</p>

Related Topics

[Adding Parts](#)

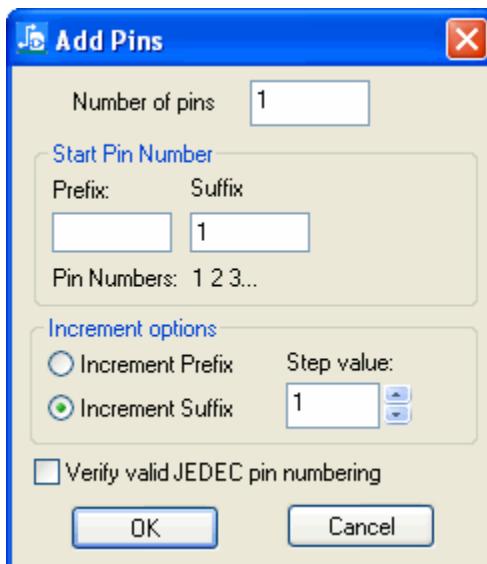
Add Pins Dialog Box

To access:

- **File > Library** menu item > select a Library > **Parts** button > **New** button > on the Part Editor Toolbar click **Edit Electrical > Pins** tab > **Add Pins** button
- **File > Library** menu item> select a Library > **Parts** button > select a part > **Edit** button > on the Part Editor Toolbar click **Edit Electrical > Pins** tab > **Add Pins** button

Use the Add Pins dialog box to add pins to a part type.

Figure 24. Add Pins Dialog Box



Objects

Table 58. Add Pins Dialog Box Fields

Name	Description
Number of pins	Specifies the number of pins to add using the Add Pins dialog box.
Prefix	The prefix you want for your pins. Tip <ul style="list-style-type: none"> • Alphabetic and numeric values can be used. For example, A1 or 1A. • For a single numeric, use either Prefix or Suffix box, and void the other box.
Suffix	The suffix you want for your pins.

Table 58. Add Pins Dialog Box Fields (continued)

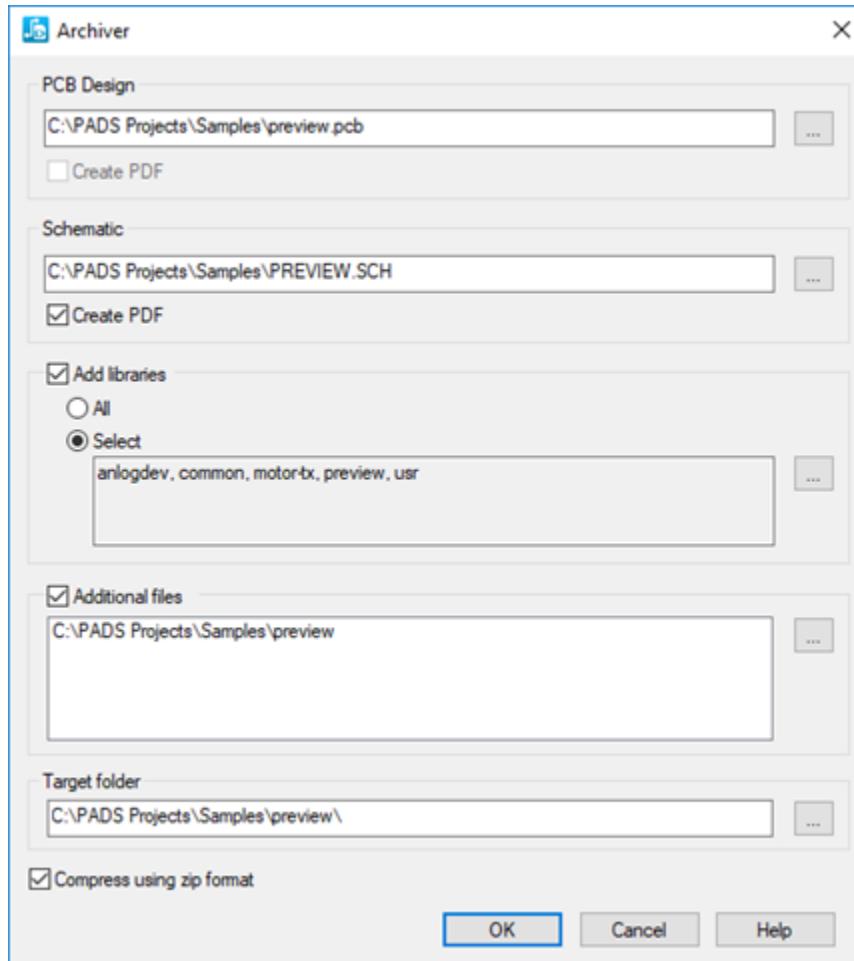
Name	Description
	 Tip <ul style="list-style-type: none">Alphabetic and numeric values can be used. For example, A1 or 1A.For a single numeric, use either Prefix or Suffix box, and void the other box.
Pin numbers	A preview of pin numbers based on your input in the Prefix and Suffix boxes.
Increment prefix/Increment suffix	Indicates whether you want the prefix or the suffix to increment.
Step value	A positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
Verify valid JEDEC pin numbering	Ensures that legal alphanumeric values are used.

Archiver Dialog Box

To access: **File > Archive** menu item

Use the Archiver dialog box to create archives of your schematics, designs, files and folders, and libraries.

Figure 25. Archiver Dialog Box



Objects

Table 59. Archiver Dialog Box Fields

Name	Description
PCB Design	Specifies the location and name of the PCB design you want to archive. To choose the file you want, type the location or click the Browse button. Select Create PDF to create a PDF file of the PCB design.

Table 59. Archiver Dialog Box Fields (continued)

Name	Description
Schematic	<p>Specifies the location and name of the schematic file you want to archive. This is automatically populated with the information from the current design. To change the design, or if no design was opened, type the location or click the Browse button.</p> <p>Specifies to create a PDF file of the schematic file.</p> <p> Restriction: This is unavailable if the file you chose is different from the current design.</p>
Add libraries	<p>Specifies that you want to include libraries in the archive.</p> <ul style="list-style-type: none">• All — Add all of your libraries to the archive.• Select — Add only the libraries you specify. <p>Click the Browse button to open the Archiver Libraries Dialog Box.</p>
Additional files	<p>Specifies that you want to include other files and folders in your archive. Click the Browse button to open the Archiver Additional Files Dialog Box.</p>
Target folder	<p>Specifies where you want the archive to be located. Type the path or click the Browse button.</p> <p>Requirement: The target folder must be empty.</p>
Compress using zip format	<p>Specifies to create a zip file. The file will be in the following format:</p> <div style="background-color: #f0f0f0; padding: 5px; margin-top: 10px;"><code><project_name>YYYYMMDDHHMMSS.zip</code></div> <p>Where YYYY is the year, MM is the month, DD is the day, HH is the hour - in military time, MM is the minute, and SS is the second of the exact time you created the file.</p>

Related Topics

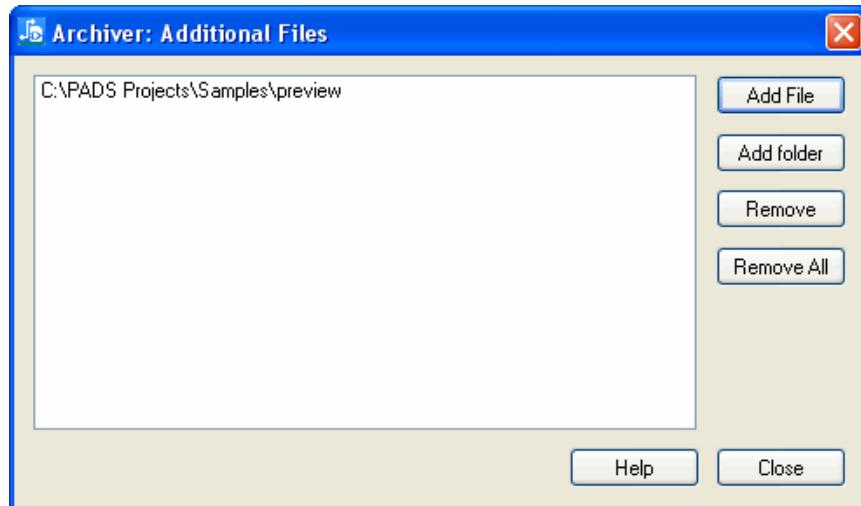
[Archiving Your Schematic](#)

Archiver Additional Files Dialog Box

To access: **File > Archive** menu item > Additional Files check box > **Browse** button

Use the Archiver: Additional dialog box to add files and folders to the schematic you want to archive.

Figure 26. Archiver Additional Files Dialog Box



Objects

Table 60. Archiver Additional Fields Dialog Box

Name	Description
Additional files list	Lists the files and folders you want to include in your archive.
Add File button	Opens the Additional File dialog box where you can select individual files you want to add to the Additional files list.
Add folder button	Opens the Browse for Folder dialog box where you can select an entire folder to add to the Additional files list.
Remove button	Removes the selected file or folder from the Additional files list.
Remove All button	Removes all of the files and folders from the Additional files list.

Related Topics

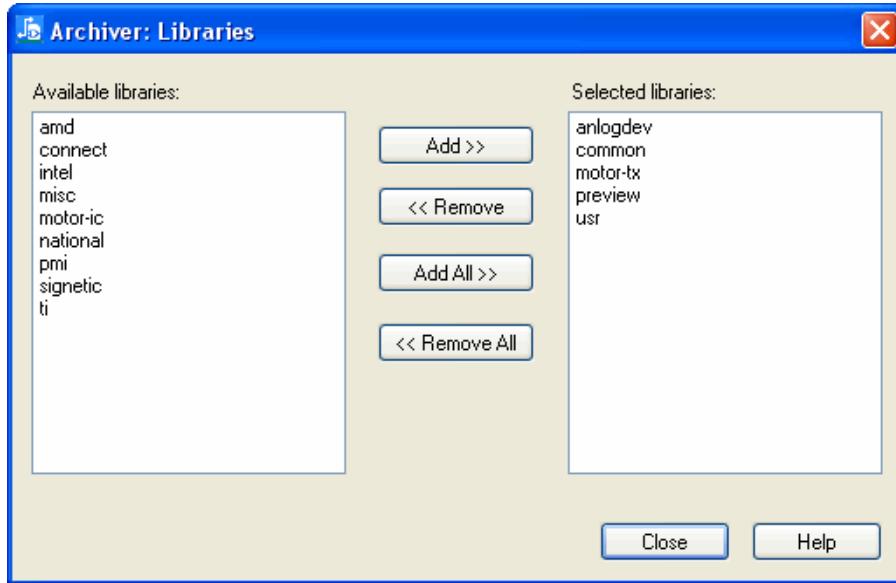
[Archiving Your Schematic](#)

Archiver Libraries Dialog Box

To access: **File > Archive** menu item > Add libraries check box > Select > **Browse** button

Use the Archiver: Libraries dialog box to add libraries to the schematic you want to archive.

Figure 27. Archiver Libraries Dialog Box



Objects

Table 61. Archiver Libraries Dialog Box Fields

Name	Description
Available libraries	Lists all of the libraries available for you to add to the archive. Restriction: If your library is not listed in the Library Manager, it will not appear in this list.
Add >> button	Moves the selected library from the Available libraries list to the Selected libraries list.
<< Remove button	Moves the selected library from the Selected libraries list to the Available libraries list.
Add all >> button	Moves all of the libraries from the Available libraries list to the Selected libraries list.
<< Remove all button	Moves all of the libraries from the Selected libraries list to the Available libraries list.

Related Topics

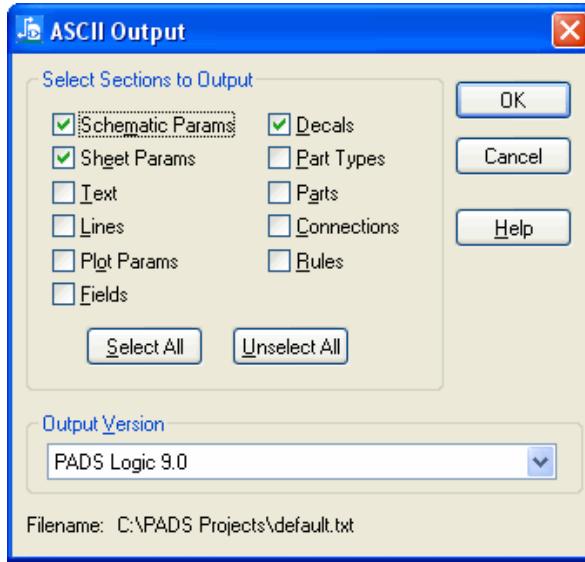
[Archiving Your Schematic](#)

ASCII Output Dialog Box

To access: **File > Export** menu item > type a filename > **Save** button

Use the ASCII Output dialog box to select the sections you want to export to the ASCII file.

Figure 28. ASCII Output Dialog Box



Objects

Table 62. ASCII Output Dialog Box Fields

Name	Description
Schematic Params	Specifies to export the default system settings from the Options tabs.
Sheet Params	Specifies to export the specific sheet information such as window scale, centering, etc.
Text	Specifies to export the free text together with location, level, and size.
Lines	Specifies to export the 2D-line items.
Plot Params	Specifies to export the information related to the CAM output settings and configurations generated using the Plot command.
Fields	Specifies to export the fields used in the schematic and their values.
Decals	Specifies to export the part decals and their contents.
Part Types	Specifies to export the library part attributes such as manufacturer, cost, and notes.

Table 62. ASCII Output Dialog Box Fields (continued)

Name	Description
Parts	Specifies to export the parts used in the schematic and their reference designators.
Connections	Specifies to export all connections on the schematic, including paths, tie-dots, and off-page flags.
Rules	Specifies to export the clearance, routing, and others specified in Design Rules.
Select All button	Selects all items.
Unselect All button	Clears all items.
Output Version	Specifies the version of the software you are using.
Filename	Displays the location and name of the file.  Tip This was specified in the File Export dialog box.

Related Topics

[Exporting to ASCII Output](#)

Assign Alternatives for Ground Part Dialog Box

To access: **Tools > Part Editor** menu item > Part Editor > **Open** button > Ground > **OK** button > **Edit Electrical** button

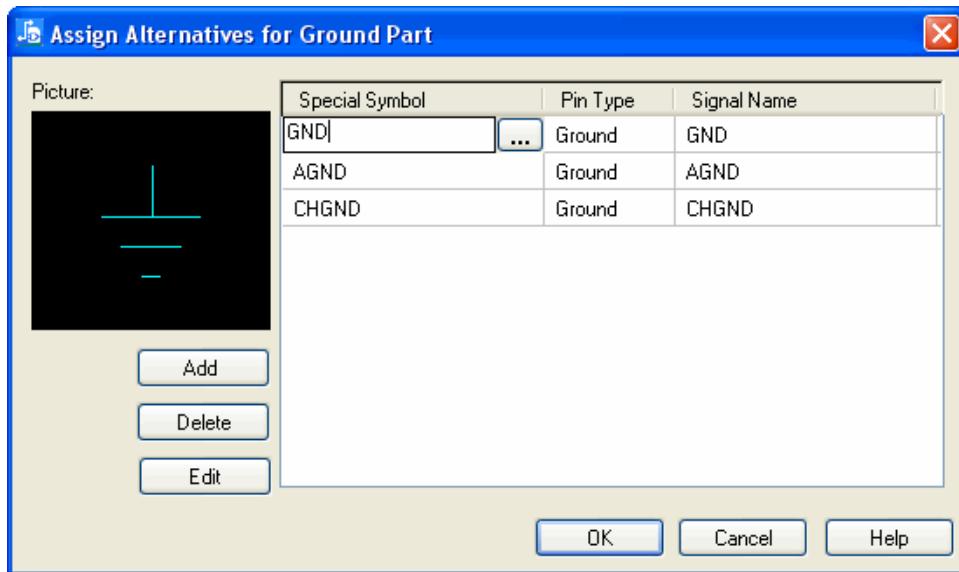
Use the Assign Alternatives for Ground Part dialog box to assign or create additional ground symbols.



Note:

For additional information on the creation and use of special schematic symbols, see “[Special Schematic Symbols](#)” on page 167.

Figure 29. Assign Alternatives for Ground Part Dialog Box



Objects

Table 63. Assign Alternatives For Ground Part Dialog Box Fields

Name	Description
Picture	Displays a picture of the selected Special Symbol.
Attribute table	<ul style="list-style-type: none">• Special Symbol — The name of a connector pin decal for use in the schematic.• Pin Type — The function of the special symbol.• Signal Name — The name of the signal.
[...]	Opens the Browse for Special Symbols Dialog Box .

Table 63. Assign Alternatives For Ground Part Dialog Box Fields (continued)

Name	Description
	 Tip This button is available only in the Special Symbols columns, and only when the cell is available for editing.
Edit button	Makes the selected cell available for editing.  Tip You can also double-click the cell to edit the contents.
Add button	Adds a new row at the bottom of the table.
Delete button	Removes the selected row.

Related Topics

[Assigning Alternate Logic Decals for Connector Symbols](#)

[Special Schematic Symbols](#)

Assign Alternatives for Off-Page Part Dialog Box

To access: **Tools > Part Editor** menu item > **Open** button > Off-page > **OK** button > **Edit Electrical** button

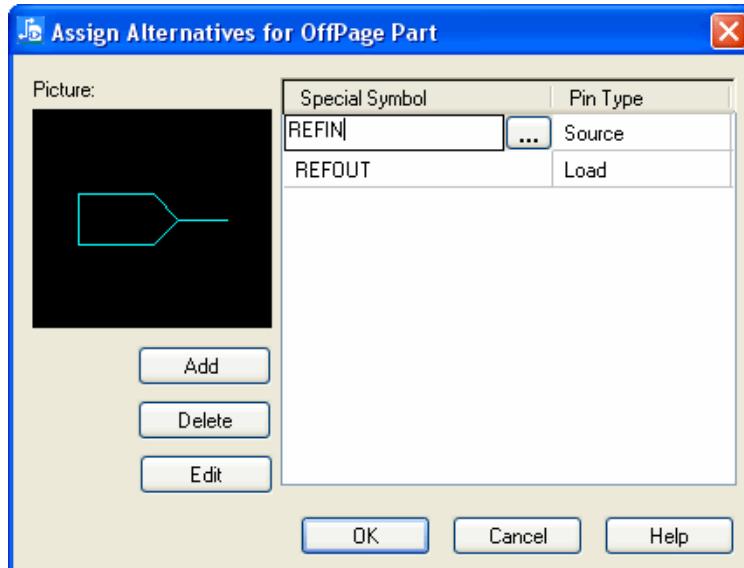
Use the Assign Alternatives for Off-Page Part dialog box to assign or create additional off-page reference symbols.



Note:

For additional information on the creation and use of special schematic symbols, see “[Special Schematic Symbols](#)” on page 167.

Figure 30. Assign Alternatives for Off-Page Part Dialog Box



Objects

Table 64. Assign Alternatives for Off-Page Part Dialog Box Fields

Name	Description
Picture	Displays a picture of the selected Special Symbol.
Attribute table	<ul style="list-style-type: none">• Special Symbol — The name of a connector pin decal for use in the schematic.• Pin Type — The function of the special symbol.
	Opens the Browse for Special Symbols Dialog Box .

Table 64. Assign Alternatives for Off-Page Part Dialog Box Fields (continued)

Name	Description
	 Tip This button is available only in the Special Symbols columns, and only when the cell is available for editing.
Edit button	Makes the selected cell available for editing.  Tip You can also double-click the cell to edit the contents.
Add button	Adds a new row at the bottom of the table.
Delete button	Removes the selected row.

Related Topics

[Assigning Alternative Symbols for the Off-Page Part](#)

[Special Schematic Symbols](#)

Assign Alternatives for Power Part Dialog Box

To access: Tools > Part Editor menu item > Open button > Power > OK button > Edit Electrical button

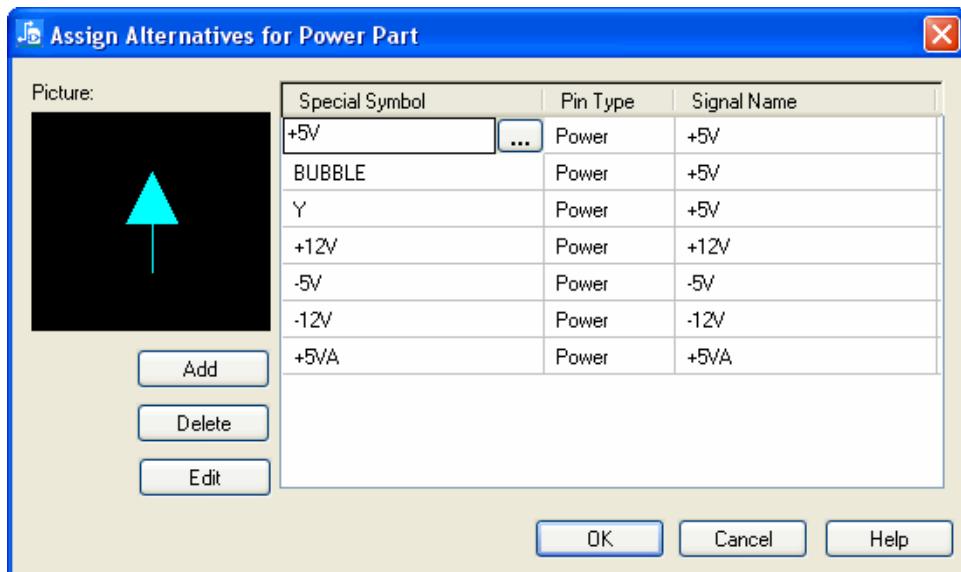
Use the Assign Alternatives for Power Part dialog box to assign or create additional power symbols.



Note:

For additional information on the creation and use of special schematic symbols, see “[Special Schematic Symbols](#)” on page 167.

Figure 31. Assign Alternatives for Power Part Dialog Box



Objects

Table 65. Assign Alternatives for Power Part Dialog Box Fields

Name	Description
Picture	Displays a picture of the selected Special Symbol.
Attribute table	<ul style="list-style-type: none">Special Symbol — The name of a connector pin decal for use in the schematic.Pin Type — The function of the special symbol.Signal Name — The name of the signal.
	Opens the Browse for Special Symbols Dialog Box . Tip This button is available only in the Special Symbols columns, and only when the cell is available for editing.

Table 65. Assign Alternatives for Power Part Dialog Box Fields (continued)

Name	Description
Edit button	Makes the selected cell available for editing.  Tip You can also double-click the cell to edit the contents.
Add button	Adds a new row at the bottom of the table.
Delete button	Removes the selected row.

Related Topics

[Assigning Alternative Symbols for the Power Part](#)

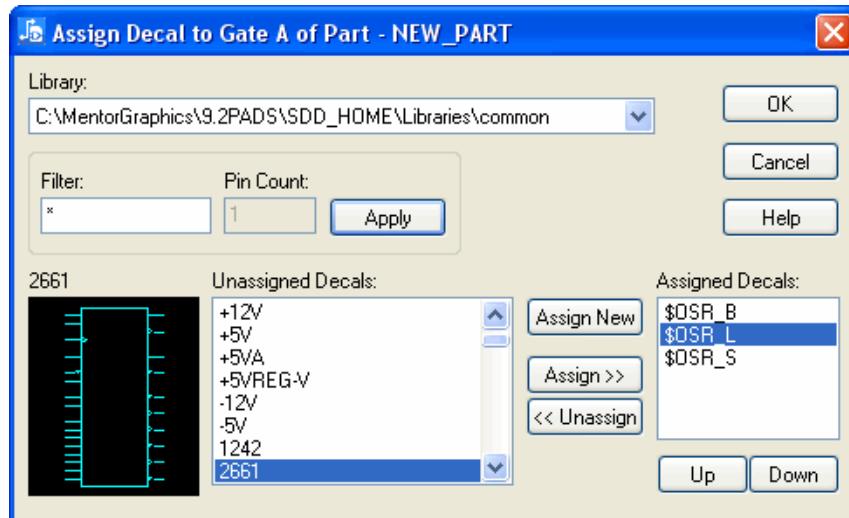
[Special Schematic Symbols](#)

Assign Decal to Gate Dialog Box

To access: **Tools > Part Editor** menu item > **Edit Electrical** button > **Gates** tab > (if a new part, click **Add**) > double-click a CAE Decal cell > **Browse** button

Use the Assign Alternatives for Power Part dialog box to assign or create additional power symbols.

Figure 32. Assign Decal to Gate Dialog Box



Objects

Table 66. Assign Decal to Gate Dialog Box Fields

Name	Description
Library list	Lists all your available libraries. Filters the Unassigned Decals list to only the selected library.
Filter	Searches the chosen library (or libraries) for a specific part or item name, or names that match a wildcard or expression on page 105. Type * to view all parts or items in the chosen libraries. Click Apply to search the libraries and display the search results.
Preview area	Displays the item selected in the Assigned Decals list.
Unassigned Decals list	Lists all of the decals that are available to assign.
Assign New button	Opens the Assign New Gate Decal Dialog Box .
Assign >> button	Moves the decal from the Unassigned Decals list to the Assigned Decals list.
<< Unassign button	Moves the decal from the Assigned Decals list to the Unassigned Decals list.
Assigned Decals list	Lists all of the decals that have been assigned.

Table 66. Assign Decal to Gate Dialog Box Fields (continued)

Name	Description
Up/Down buttons	Moves the selected decal up or down in the Assigned Decals list.

Related Topics

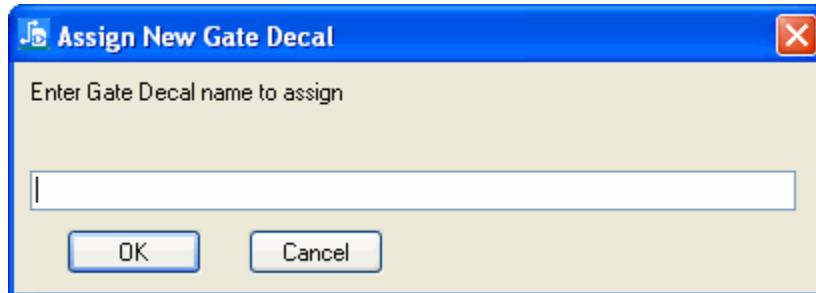
[Assigning CAE Decals to Gates](#)

Assign New Gate Decal Dialog Box

To access: **File > Library** menu item > select a Library > **Parts** button > select part > **New** or **Edit** button > on Part Editor Toolbar click **Edit Electrical > Gates** tab > (if new part click **Add**) > double-click CAE Decal cell > **Browse** button > **Assign New**

Use the Assign New Gate dialog box to assign a new gate decal when it doesn't yet exist in the Library.

Figure 33. Assign New Gate Decal Dialog Box



Objects

Table 67. Assign New Gate Decal Dialog Box Fields

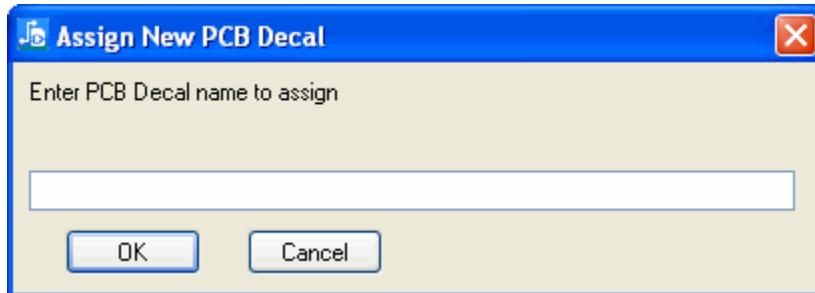
Name	Description
Text box	Enter the name of the new gate decal you intend to add to the library.

Assign New PCB Decal Dialog Box

To access: **File > Library** menu item > select a Library > **Parts** button > select part > **New** or **Edit** button > on Part Editor Toolbar click **Edit Electrical > PCB Decals tab > Assign New**

Use the Assign New PCB Decal dialog box to assign a new PCB Decal when it doesn't yet exist in the Library.

Figure 34. Assign New PCB Decal Dialog Box



Objects

Table 68. Assign New PCB Decal Dialog Box Fields

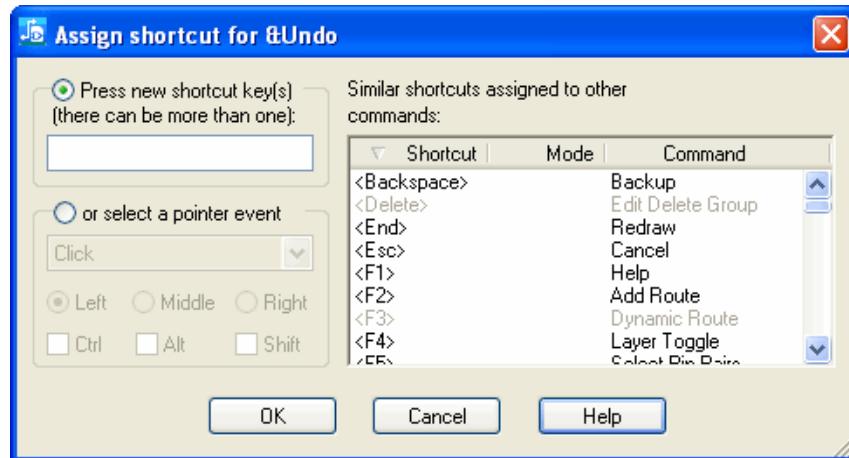
Name	Description
Text box	Enter the name of the new PCB decal you intend to add to the library.

Assign Shortcut Dialog Box

To access: **Tools > Customize** menu item > **Keyboard and Mouse** tab > select a mode > select a command folder > select a command > **New** button

Create a new shortcut key using the Assign Shortcut dialog box.

Figure 35. Assign Shortcut Dialog Box



Objects

Table 69. Assign Shortcut Dialog Box Fields

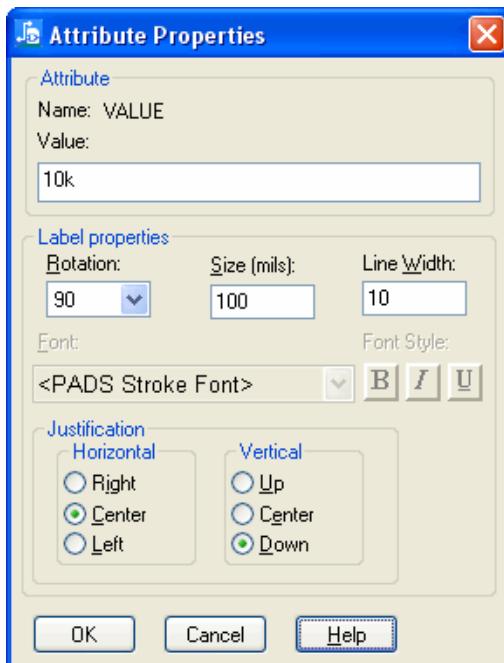
Name	Description
Press new shortcut key	Type the shortcut you want to use.
Select a pointer event	Set a pointer event shortcut
Similar shortcuts list	Lists the shortcut keys already assigned to other commands.

Attribute Properties Dialog Box

To access: Select a part attribute label > right-click > **Properties** menu item

Use the Attribute Properties dialog box to provide text and font settings for one or more part attribute labels.

Figure 36. Attribute Properties Dialog Box



Objects

Table 70. Attribute Properties Dialog Box Fields

Name	Description
Name	The name of the selected attribute.
Value	Specifies the label you want in the schematic.
Rotation	Specifies the rotation of the label: 0 or 90 degrees.
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.

Table 70. Attribute Properties Dialog Box Fields (continued)

Name	Description
	 Stroke Font - Size
Line Width	Specifies the line width for stroke fonts only.  Stroke Line Width
Font list	The fonts available to you. This lists either stroke fonts or system fonts. You choose which type of font to use in the Fonts Dialog Box .  Tip <ul style="list-style-type: none">• Select stroke font or a system font.• For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

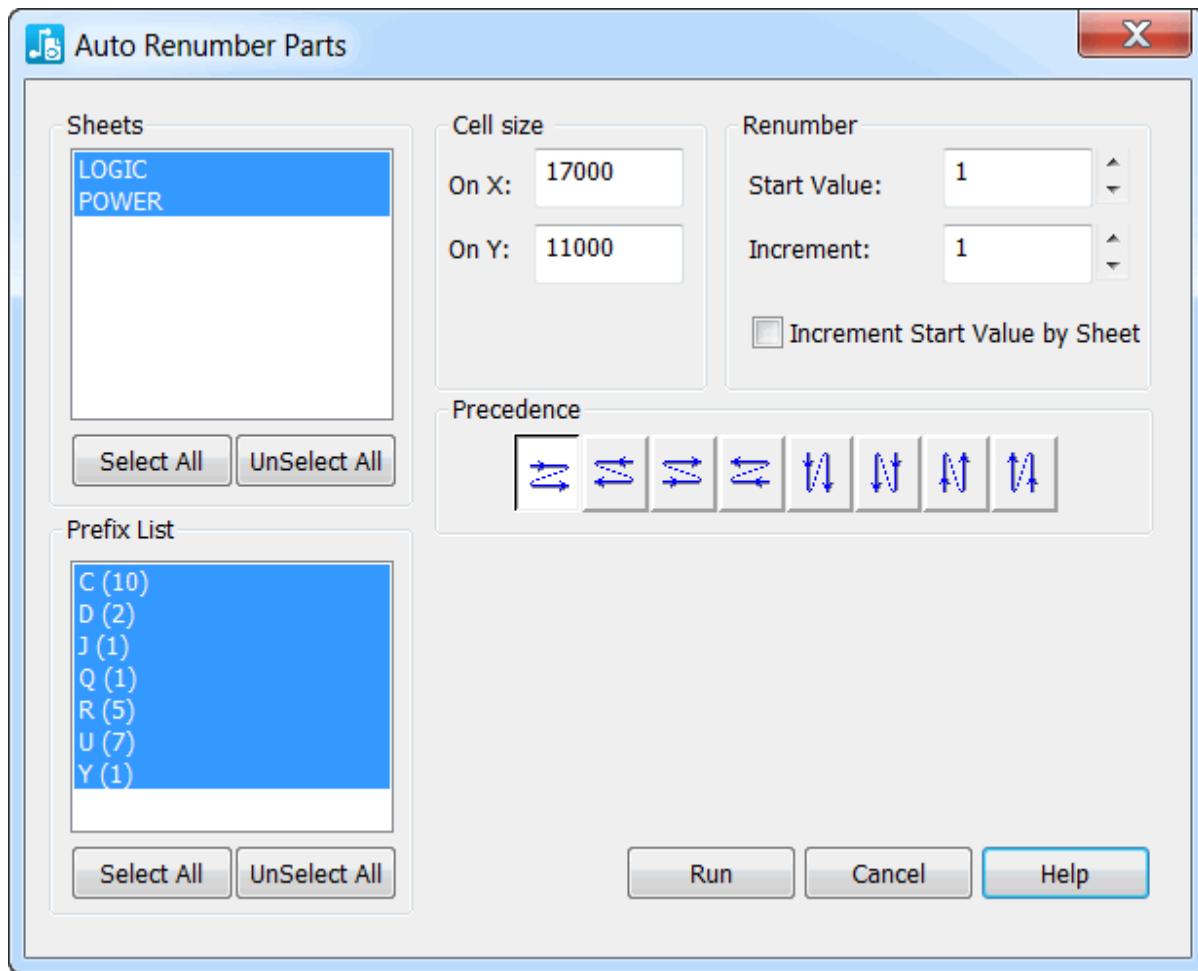
Related Topics

[Modifying Part Attribute Labels](#)

Auto Renumber Parts Dialog Box

To access: Schematic Editing toolbar > **Auto Renumber Parts** button

Automatically renumber the reference designators on schematic sheets in a pattern. Choose the sheets, reference designator prefixes, pattern and numbering range.



Objects

Name	Description
Sheets	Choose which sheets to renumber. Hierarchical sheets appear differently depending on whether they have been created from the top down or the bottom up.

Name	Description
	<ul style="list-style-type: none"> • Bottom-up — Sheets display at the primary level instead of being indented in the list. When using Increment Start Value by Sheet, the reference designators are renumbered on each sheet. • Top-down — Sheets are indented to show they are sub-sheets. When using Increment Start Value by Sheet, the sheets inherit the same base level number as the parent sheet.
Prefix List	Choose which reference designator families to renumber. The list displays the total number of reference designators in brackets. The prefix types and totals update based on the sheets selected in the Sheets list.
Cell size	Renumbering is applied according to the Precedence pattern within the cell area. Specify the size of cells in which to apply the renumbering pattern. Cells are arranged across the schematic area in selected Precedence pattern. For an illustration explaining cell size, see “ Automatically Renumbering Reference Designators ” on page 280.
Renumber	<ul style="list-style-type: none"> • Start Value — The number to apply to the first of each type of renumbered reference designator. • Increment — The gap in numbers to apply to the reference designator numbers. • Increment Start Value by Sheet — Use the sheet number as the start number - the sheet number as shown in the Sheets Dialog Box and not the order of the sheet in this dialog. For example, if the Start Value is 101, then sheet number 1 will start with number 101, but sheet 2 will start with number 201. If used in combination with large start values, the reference designator numbers may quickly run into the software limit. For example, if the start value is set to 1000, and there are 40 schematic sheets, then reference designators on sheet 40 would start at 40,000 - well over the limit of 32,766. <p>Start value and increment are applied to each reference designator family separately. For example, the schematic has the following components: C1, C2, C3, R1, R2. Start Value is 10 and Increment is 5. The result of renumbering is: C10, C15, C20, R10, R15. Parts on sheets are renumbered in the order of the sheets: first, parts from sheet number 1, then from sheet number 2 and so on. If a part belongs to more than one sheet, it gets renumbered on the selected sheet with the lowest number.</p>
Precedence	Specifies the pattern used to renumber parts within a cell area. Also specifies the pattern used for the application of cells to process over the schematic page. Renumbering is performed based upon the first pin of the symbol encountered in the given precedence direction.

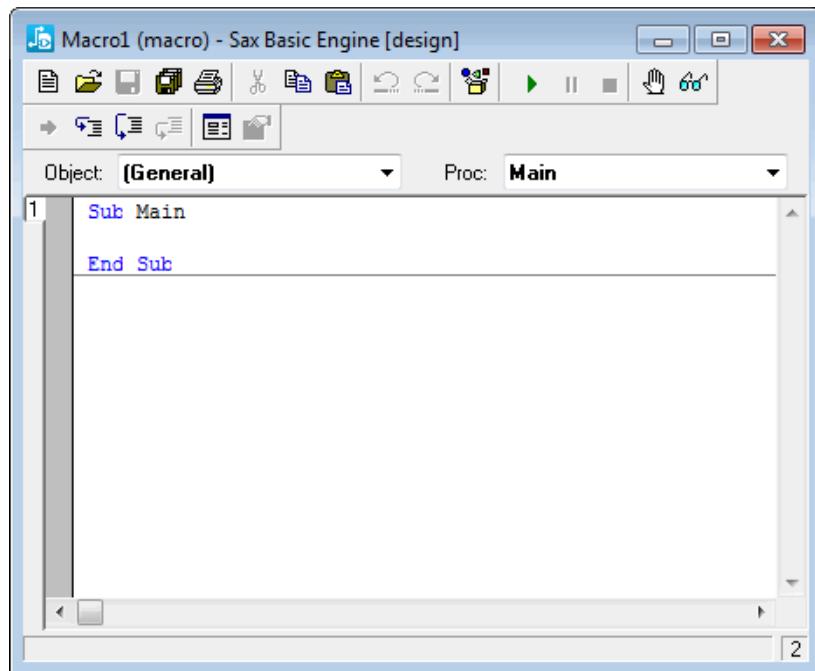
Basic Script Editor

Basic is a simple scripting language. Like many Windows applications, such as Microsoft Word and Excel, SailWind applications include Basic capabilities to enable users to customize their applications using a standard scripting language.

You can use the Basic Script Editor to create, edit, run, and troubleshoot Basic scripts from SailWind applications. To open the editor:

- Tools > Basic Scripts > Basic Script Editor menu item

Figure 37. Basic Script Editor

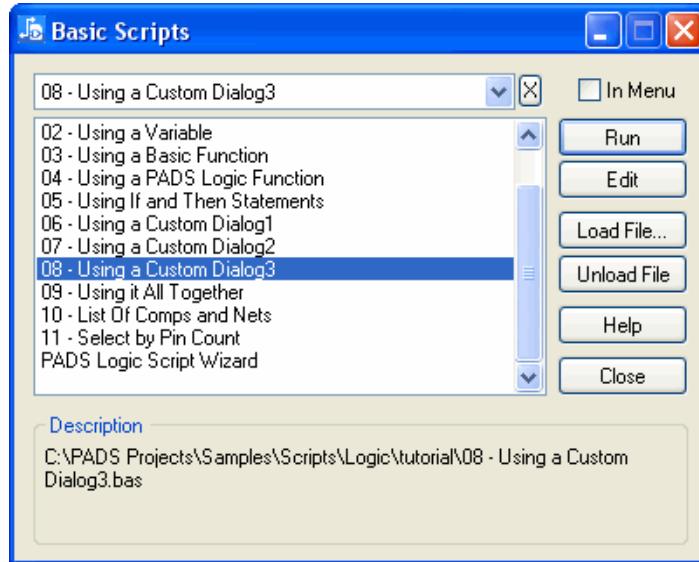


Basic Scripts Dialog Box

To access: Tools > Basic Scripts > Basic Scripts menu item

The Basic Scripts dialog box provides easy access to your Basic scripts.

Figure 38. Basic Scripts Dialog Box



Objects

Table 71. Basic Scripts Dialog Box Fields

Name	Description
Basic Script List	Lists the scripts available to you.
X button	Specifies to use the smaller dialog box vs. the entire dialog box. Tip To use the smaller dialog box, select the script you want to run and press Enter.
In Menu	Specifies to add the selected script to the Basic Scripts menu.
Run button	Runs the selected script. Restriction: You can not run multiple scripts at the same time. Tip If the selected script has an error during compilation, it automatically opens in the Basic Script editor for correction.
Edit button	Opens the Sax Basic Engine dialog box with the selected script loaded. See also Managing the Sax Basic Engine on page 389

Table 71. Basic Scripts Dialog Box Fields (continued)

Name	Description
Load File button	Adds a new script to the list.  Tip <ul style="list-style-type: none">• You can load up to 32,767 scripts. Scripts are not compiled when they are loaded; they are compiled when you run them• The list of scripts you load into this dialog box is saved in the <i>VBScripts.ini</i> file, so they load every time you open the Basic Scripts dialog box.
Unload button	Removes the selected script from the list.
Description area	Displays the location of the selected script.

Related Topics

[Using the Basic Scripts Dialog Box](#)

Bill of Materials Setup Dialog Box

The Bill of Materials report produces a user-configurable list of the parts contained in the current schematic. You can direct any part attribute in the schematic to a Bill of Materials report.



Restriction:

Including non-ECO-registered parts and non-electrical parts in the bill of materials is constrained. See [Options Dialog Box, Design Category](#) for details.

To access: **File > Reports** menu item > **Setup** button

[Bill of Materials Setup Dialog Box, Attributes Tab](#)

[Bill of Materials Setup Dialog Box, BOM Config Tab](#)

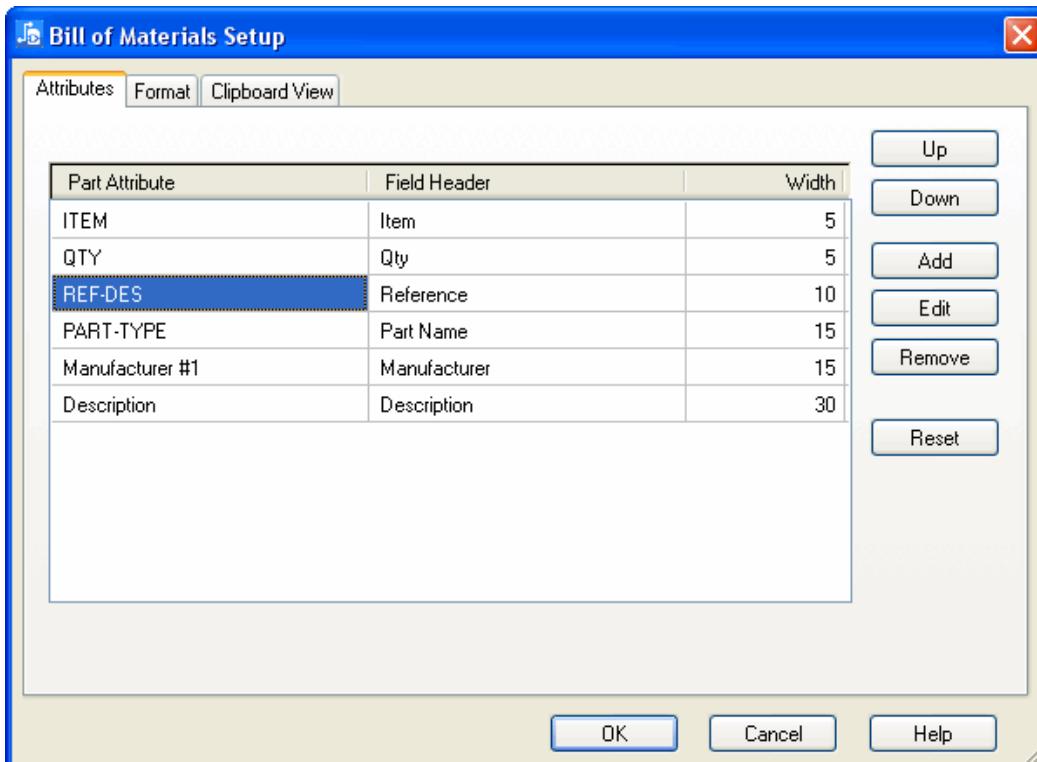
[Bill of Materials Setup Dialog Box, Format Tab](#)

Bill of Materials Setup Dialog Box, Attributes Tab

To access: **File > Reports** menu item > **Setup** button > **Attributes** tab

Use the **Attributes** tab to modify the Attribute content, the corresponding column headings, and column width of the report. The attribute order in the content list determines the column arrangement in the BOM report. There is a limit of 12 attributes in the Bill of Materials report.

Figure 39. Attributes tab



Objects

Table 72. Bill of Materials Setup Dialog Box Fields

Name	Description
Attribute table	<p>Specifies the attributes in, headings for, and width of the BOM report.</p> <ul style="list-style-type: none"> Part Attribute column — Specifies the attributes you want in the report. You can list up to twelve attribute names. Each attribute defines a column in the report. Field Header column — Specifies the column heading for each attribute in the report. You can specify any character except the colon (:). Width column — Specifies the number of characters across the column for each attribute in the report: 1 to 200.

Table 72. Bill of Materials Setup Dialog Box Fields (continued)

Name	Description
Up button	Moves the selected row up by one.  Tip The attribute order in the content list determines the column arrangement in the BOM report.
Down button	Moves the selected row down by one.  Tip The attribute order in the content list determines the column arrangement in the BOM report.
Add button	Adds a row to the bottom of the table where you can select a new part attribute.  Tip You can list up to twelve attribute names. Each attribute defines a column in the BOM report.
Edit button	Makes the selected box editable.  Restriction: You cannot edit the part attribute name, but you can select a new attribute to replace the one currently listed.
Remove button	Removes the selected row from the table and therefore the attribute from the report.
Reset button	Restores the default content from the .ini file.

Related Topics

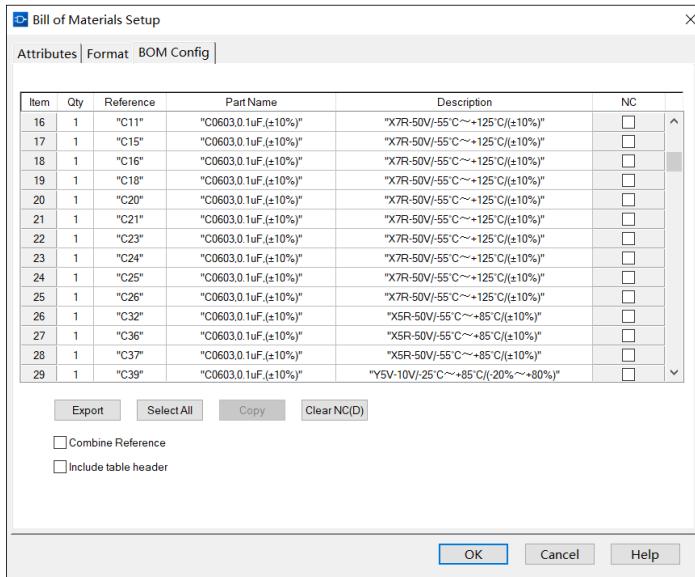
[The Bill of Materials Report](#)

Bill of Materials Setup Dialog Box, BOM Config Tab

To access: **File > Reports** menu item > **Setup** button > **BOM Config** tab

Use the **BOM Config** tab to preview the Bill of Materials report format and copy any selected lines of the file to a Windows clipboard. You can also specify rows to export to a TXT/CSV file. The default view orders the attributes by the sort field you specified on the **Format** tab.

Figure 40. BOM Config Tab



Objects

Table 73. Bill of Materials Setup Dialog Box-BOM Config Fields

Name	Description	Remark
Report table	Displays the BOM report in table format for you to select and copy any row or export to a CSV/TXT file. This table shows the data from the attributes you selected on the Attributes tab .	
Select All button	Selects all rows in the table.	Use these functions to select all rows in the table.
Copy button	Specifies to copy the row(s) you've selected to the Windows clipboard.	
Include table header	Specifies to copy the table header in addition to the row(s) you've selected.	
Export button	Exports the BOM report to a CSV/TXT file. You can exclude the row(s) you don't want by checking the NC checkbox.	Use these functions to export the BOM report to a CSV/TXT file. You can exclude specific rows by checking the NC checkbox.
Combine Reference button	Specifies to include reference designators sharing identical Part Name in the same cell.	

Table 73. Bill of Materials Setup Dialog Box-BOM Config Fields (continued)

Name	Description	Remark
Clear NC button	Specifies to clear all the NC checkboxes selected in the table.	

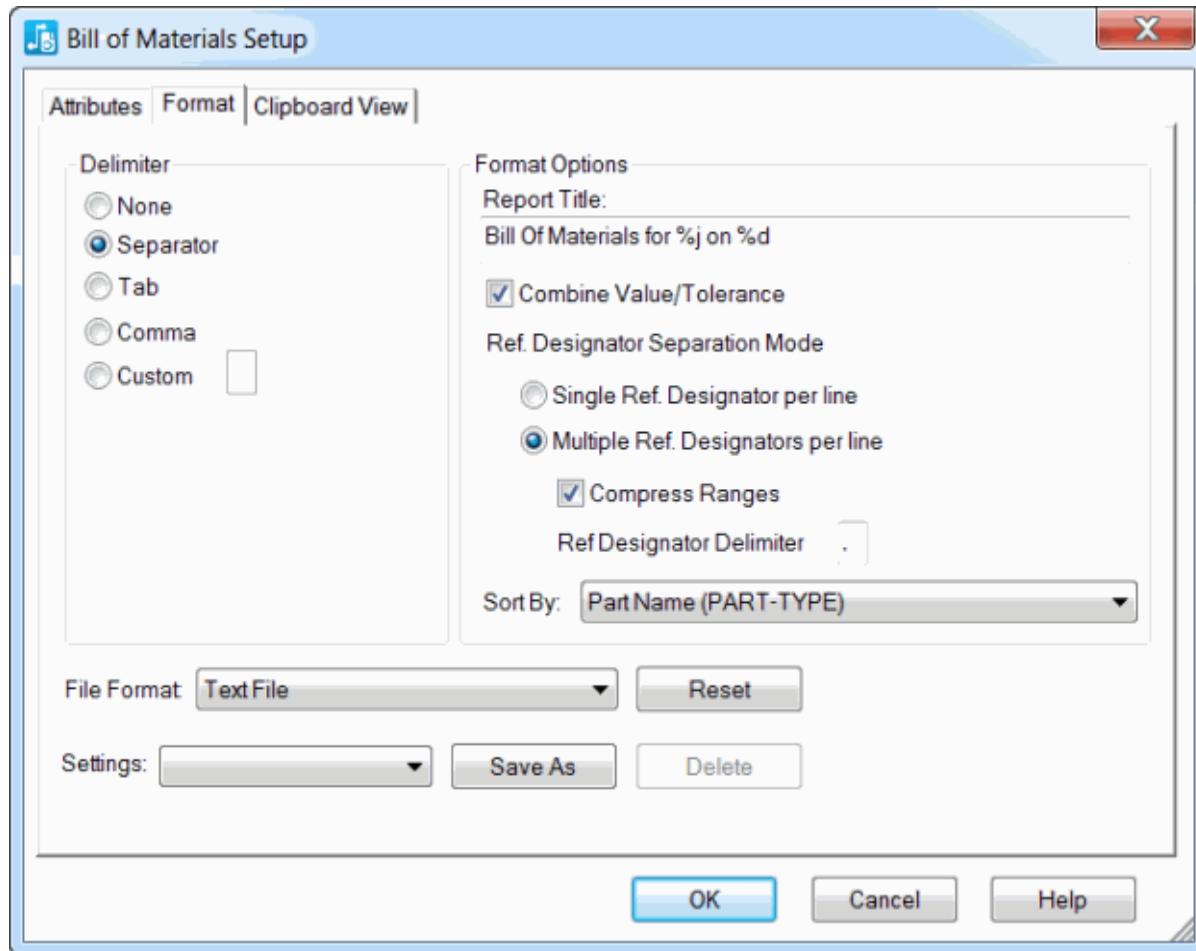
Related Topics

[The Bill of Materials Report](#)

Bill of Materials Setup Dialog Box, Format Tab

To access: File > Reports menu item > Setup button > Format tab

Use the **Format** tab to modify the output format of the Bill of Material report. The default settings originate from the *.ini* file.



Objects

Table 74. Bill of Materials Setup Dialog Box-Format Tab Fields

Name	Description
Delimiter area	Specifies the type of delimiter you want to distinguish the report columns. <ul style="list-style-type: none">• None — no delimiter used.• Separator — places a vertical bar between report fields.• Tab — separates columns with a tab spacing.

Table 74. Bill of Materials Setup Dialog Box-Format Tab Fields (continued)

Name	Description
	<ul style="list-style-type: none"> Comma — places a comma character between report fields. Custom — specify any character as a delimiter.
File Format list	Specifies the output file format.
Reset button	Restores the default content from the <i>.ini</i> file.
Settings list	Specifies to use a previously saved report format setting.
Save As button	Saves report format settings for the current design to a specified file so you can create different format configurations for different designs
Delete button	Removes the selected setting in the Settings list.
Format Options area	
Report Title	Specifies the title of the report.
Combine Value/Tolerance	<p>Specifies to combine the Value and Tolerance attributes of a part in the part name.</p> <p>Example: The 1/4 watt resistor would have a part type name of R1/4W or R1/4W.4.7K,+/-5%. SailWind Logic evaluates parts that have either a different Value or Tolerance attribute as different part types.</p>
Ref. Designator Separation Mode	<ul style="list-style-type: none"> Single Ref. Designator per line — Although some components are identical, display each component instance on a new line. This increases the BOM report size. Multiple Ref. Designator per line — Combines identical component instances on one line and lists all reference designators based on the settings below: <ul style="list-style-type: none"> Compress Ranges — Displays ranges of components with a dash between min and max value. For example, if components C1, C2, C3, and C4 are identical, they are displayed as C1-4. This can shorten the list of reference designators listed for a component type. <p>If you clear this check box, ranges are not compressed and individual reference designators are listed for the line entry. For example, C1,C2,C3,C4.</p> <ul style="list-style-type: none"> Ref designator Delimiter — Specify a single character to place between multiple reference designators. For example, use a comma for “C1,C2,C3”, or use an asterisk for “C1*C2*C3”. Although you can type multiple characters in this box, only the first character applies.
Sort By list	<p>Specifies the attribute in which to sort the report list.</p> <p>Tip Select None to sort by Part Type.</p>

Related Topics

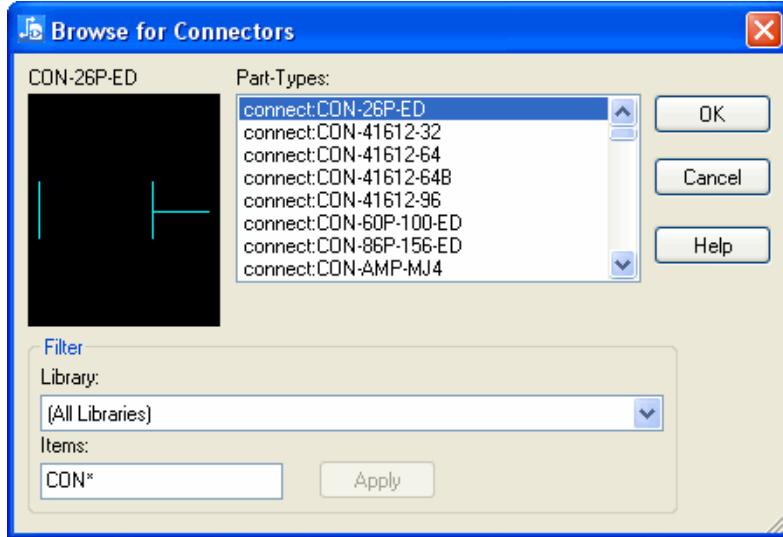
[The Bill of Materials Report](#)

Browse for Connectors Dialog Box

To access: **Tools > Part Editor** menu item > **Open** button > select Connector > **OK** button

Use the Browse for Connectors dialog box to browse a library and select a connector for editing from the library (or libraries) specified in the Library list.

Figure 41. Browse for Connectors Dialog Box



Objects

Table 75. Browse for Connectors Dialog Box Fields

Name	Description
Preview area	Shows the selected part.
Part Types	Lists the items in the selected library. The number of objects that appear depends on the filter settings.
Library list	Specifies the library you want to use.
Items	Narrows the search. You can use wildcards or expressions on page 105. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.

Related Topics

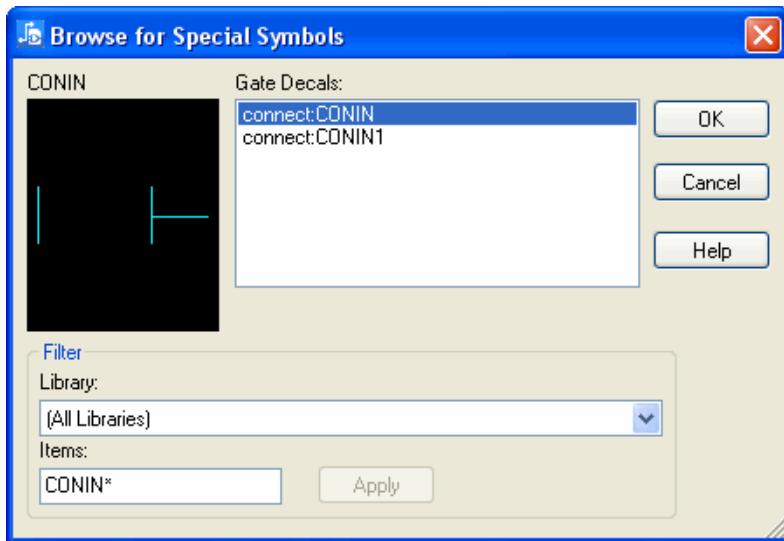
[Browsing for Connectors](#)

Browse for Special Symbols Dialog Box

To access: **Tools > Part Editor** menu item > open an off-page part > **Edit Electrical** button > double-click in a field in the Special Symbols column > **Browse** button

Use the Browse For Special Symbols dialog box to access the library for decals you want to specify as Special Symbols. Special Symbols are those used to create off-page reference, power and ground symbols, and connectors.

Figure 42. Browse for Special Symbols Dialog Box



Objects

Table 76. Browse for Special Symbols Dialog Box Fields

Name	Description
Preview area	Shows the selected item.
Gate Decals	Lists the items in the selected library. The number of objects that appear depends on the filter settings.
Library list	Specifies the library you want to use.
Items	Narrows the search. You can use wildcards or expressions on page 105. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.

Related Topics

[Assigning Alternative Symbols for the Ground Part](#)

[Assigning Alternative Symbols for the Off-Page Part](#)

[Assigning Alternative Symbols for the Power Part](#)

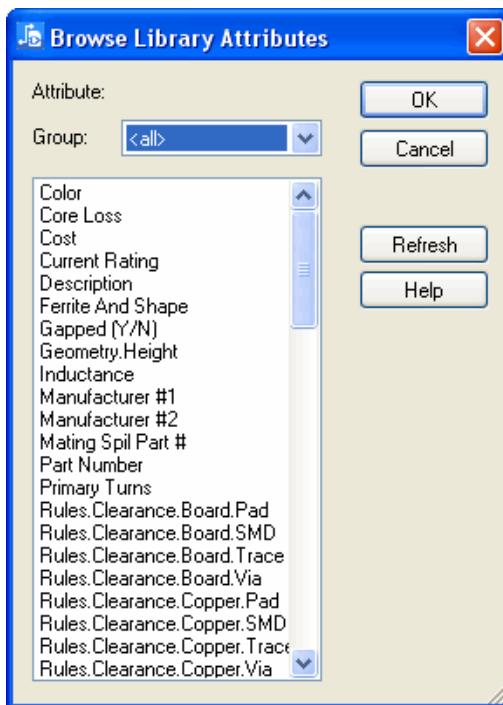
[Assigning Alternate Logic Decals for Connector Symbols](#)

Browse Library Attributes Dialog Box

To access: Tools > Part Editor menu item > Edit Electrical button > Attributes tab > Browse Lib. Attr button

Use the Browse Library Attributes dialog box to browse and list all attribute names from libraries specified in the Library List dialog box.

Figure 43. Browse Library Attributes Dialog Box



Objects

Table 77. Browse Library Attributes Dialog Box Fields

Name	Description
Attribute	Displays the selected attribute.
Group	Filters the attribute list. (Includes structured attributes.)
Refresh button	Manually updates the attribute list if you change the list of libraries in the Library Manager.

Related Topics

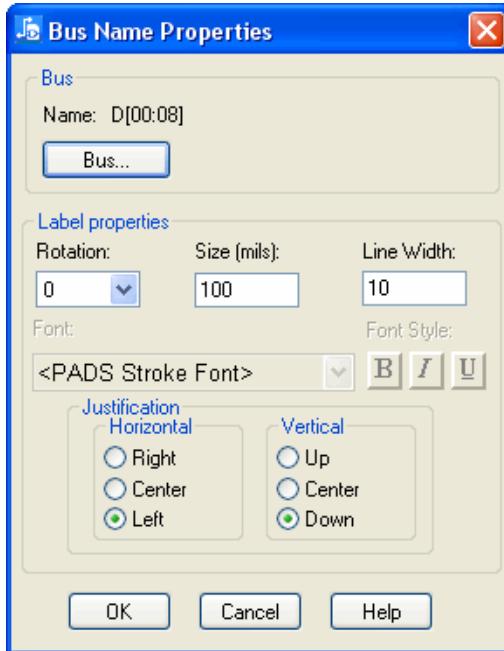
[Browsing Library Attributes](#)

Bus Name Properties Dialog Box

To access: Select a bus name label > right-click > **Properties** menu item

Use the Bus Name Properties dialog box to provide or change text and font settings for one or more bus name labels.

Figure 44. Bus Name Properties Dialog Box



Objects

Table 78. Bus Name Properties Dialog Box Fields

Name	Description
Name	The name of the selected bus.
Bus button	Opens the Bus Properties Dialog Box .
Rotation	Specifies the rotation of the label: 0 or 90 degrees.
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.

Table 78. Bus Name Properties Dialog Box Fields (continued)

Name	Description
	 <p>Stroke Font - Size</p>
Line Width	Specifies the line width for stroke fonts only.  <p>Stroke Line Width</p>
Font list	The fonts available to you. <p> Tip</p> <ul style="list-style-type: none"> • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Font Style	Enables you to change the font style to bold, italic, and underlined. <p> Restriction: System fonts only.</p>
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

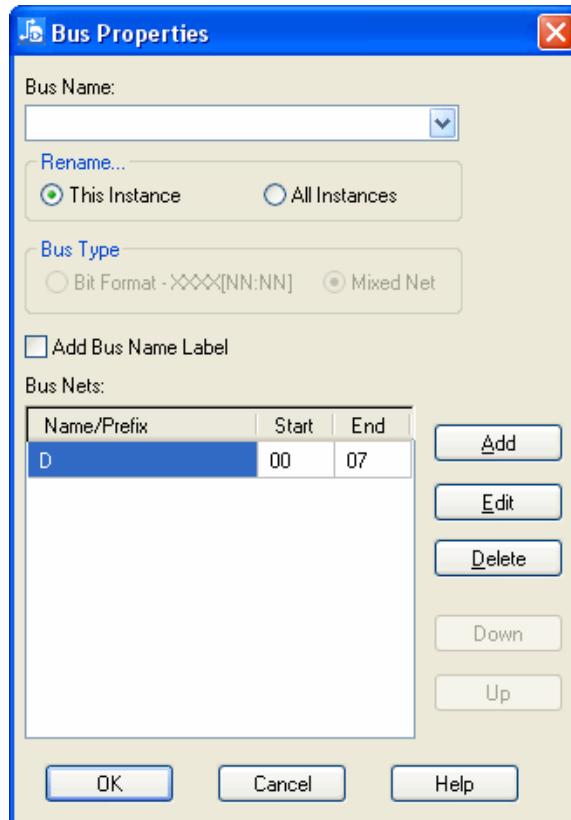
[Modifying Bus Name Labels](#)

Bus Properties Dialog Box

To access: Select a bus > right-click > **Properties** menu item

Use the Bus Properties dialog box to change the name of the bus, change the bus type, and manage bus nets.

Figure 45. Bus Properties Dialog Box



Objects

Table 79. Bus Properties Dialog Box Fields

Name	Description
Bus Name	Specifies the name of the bus. Select or type the name you want.
Rename area	Specifies to rename this instance or all instances of the bus.
Bus Type area	Specifies which bus names appear in the Bus Name list.
Add Bus Name Label	Select the Add Bus Name Label check box to add the bus name as a label to the bus at the end of the bus closest to where you selected it.

Table 79. Bus Properties Dialog Box Fields (continued)

Name	Description
	 Tip <ul style="list-style-type: none"> A bus can have two labels, one on each end. The check box is unavailable when the end of the selected bus has a label. A bus label is not required. To delete a bus label, select the label in the schematic and click the Delete button on the standard toolbar.
Bus Nets table	<p>Lists the name or prefix of the bus net, the starting bit number for a sequence of nets, and the ending bit number for a sequence of nets.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>  Tip <ul style="list-style-type: none"> For a single net, type the net name. For a range of sequential nets, type the prefix for the sequence of nets.
Add button	<p>Adds a row to the Bus Nets table.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>
Edit button	<p>Makes the selected row available for editing.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>
Delete button	<p>Removes the selected row from the Bus Nets table.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>
Down/Up buttons	<p>Moves the selected row up or down in the Bus Nets table.</p> <p> Restriction: Available only if the bus is a mixed net bus.</p>

Related Topics

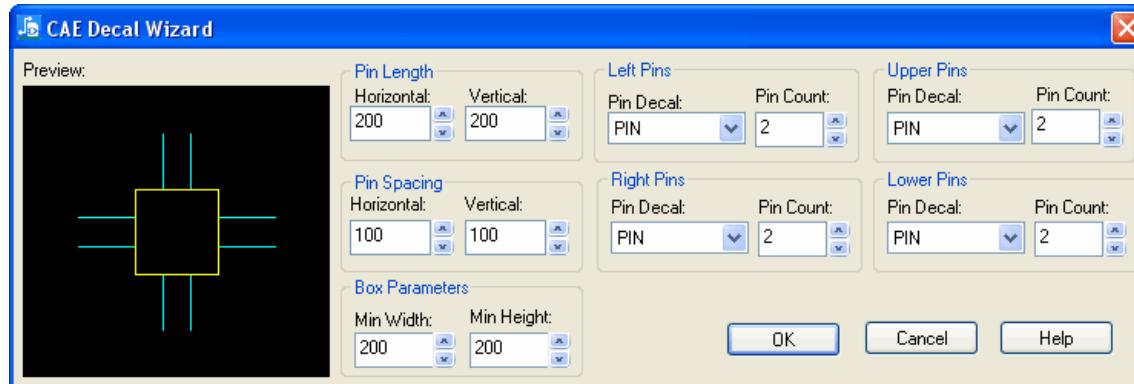
[Managing Buses](#)

CAE Decal Wizard Dialog Box

To access: Tools > Part Editor menu item > New button > CAE Decal > OK button > Decal Editing Toolbar button > CAE Decal Wizard button

Use the Decal Wizard dialog box to automatically create a new CAE decal. You must be in the Decal Editor mode of the Part Editor, and creating gate information, to use this feature.

Figure 46. CAE Decal Wizard Dialog Box



Objects

Table 80. CAE Decal Wizard Dialog Box contents

Name	Description
Preview area	Displays what the CAE decal looks like based on your settings.
Pin Length area	Sets the horizontal and vertical distance from the terminal connection point to the decal outline.
Pin Spacing area	Sets the horizontal and vertical distance between pins.
Box Parameters area	Sets the width and height of the decal outline. Tip <ul style="list-style-type: none">• Pin decals are moved left or right to accommodate the box width.• If you enter a value larger than needed to accommodate the number of input or output pins, space is added to the bottom of the decal.
Left Pins area	Specifies the pin decal and pin count for the left, or input, side of the part. <ul style="list-style-type: none">• Pin Decal — Specifies the pin decal to use for this side.• Pin Count — Specifies the number of pins to use on this side.
Right Pins area	Specifies the pin decal and pin count for the right, or output, side of the part.

Table 80. CAE Decal Wizard Dialog Box contents (continued)

Name	Description
	<ul style="list-style-type: none">• Pin Decal — Specifies the pin decal to use for this side.• Pin Count — Specifies the number of pins to use on this side.
Upper Pins area	Specifies the pin decal and pin count for the upper side of the part. <ul style="list-style-type: none">• Pin Decal — Specifies the pin decal to use for this side.• Pin Count — Specifies the number of pins to use on this side.
Lower Pins area	Specifies the pin decal and pin count for the lower side of the part. <ul style="list-style-type: none">• Pin Decal — Specifies the pin decal to use for this side.• Pin Count — Specifies the number of pins to use on this side.

Related Topics

[Using the Decal Wizard](#)

Capture a New View Dialog Box

To access: **View > Save View** menu item > **Capture** button

Use the Capture a New View dialog box to name the view you want to save.

Figure 47. Capture a New View Dialog Box



Objects

Table 81. Capture a New View Dialog Box contents

Name	Description
Name	Specifies the name you want for the view. View <i>n</i> is the default name where <i>n</i> is the next number available.  Tip You can create up to nine views. The view names appear at the bottom of the View menu.

Related Topics

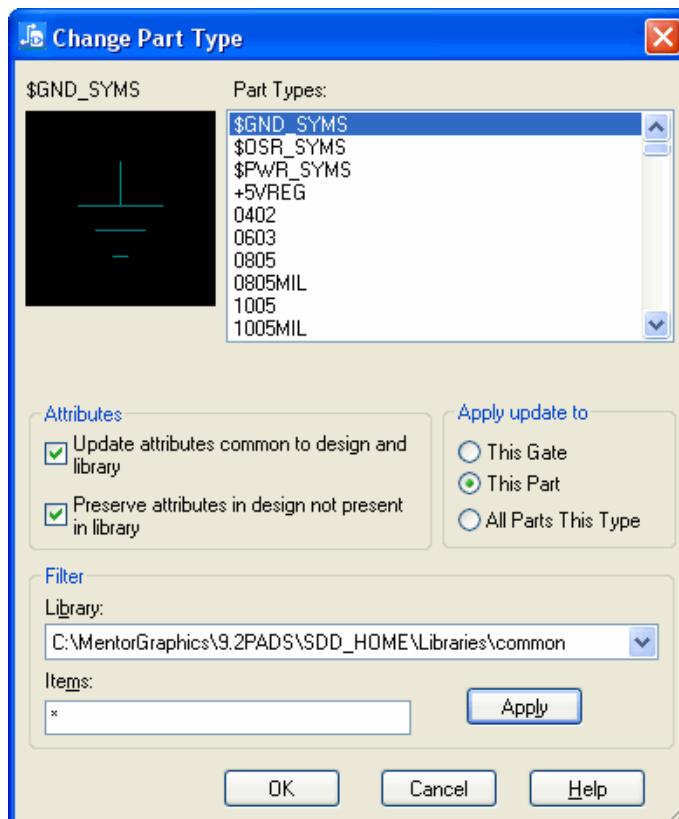
[Saving and Restoring Views](#)

Change Part Type Dialog Box

To access: Select a part > right-click > **Properties** > **Change Type** button

Use the Change Part Type dialog box to change the selected part to a new part type. The new part type can be one that already exists in the schematic or in the parts library. If multiple parts are selected, all of the selected parts are changed.

Figure 48. Change Part Type Dialog Box



Objects

Table 82. Change Part Type Dialog Box Contents

Name	Description
Picture area	Displays the part selected in the Part Types list.
Part Types list	Lists all of the part types according to the filter settings.
Update attributes common to design and library	Updates parts having common attributes with the attribute values contained in the new part.
Preserve attributes in design not present in library	Retains attributes that exist in the current part even though they do not exist in the new part.

Table 82. Change Part Type Dialog Box Contents (continued)

Name	Description
Apply update to area	<p>Sets how parts are updated in the Apply update to area:</p> <ul style="list-style-type: none"> • This Gate — Only updates the selected gate. • Selected Gates — Only updates the selected gates. <p> Restriction: Available only when multiple parts are selected.</p> <ul style="list-style-type: none"> • This Part — Only updates a part or all associated gates of a part. • Selected Parts — Only updates a part or all associated gates of the selected parts. <p> Restriction: Available only when multiple parts are selected.</p> <ul style="list-style-type: none"> • All Parts This Type — Updates all matching gates and/or parts in the design. <p> Restriction: Unavailable when multiple parts are selected.</p> <p> Tip</p> <ul style="list-style-type: none"> • You can update the part definition in the schematic with a modified version in the library. Select the same part name, then click All Parts This Type in the Apply Update to Area. • If you change a part type to one with fewer pins, the connections going to the missing pins are not deleted. They are attached to automatically generated off-page symbols. You are notified of all disconnected pins.
Library list	Specifies the library you want to use.
Items	Narrows the search. You can use wildcards or expressions on page 105. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.

Related Topics

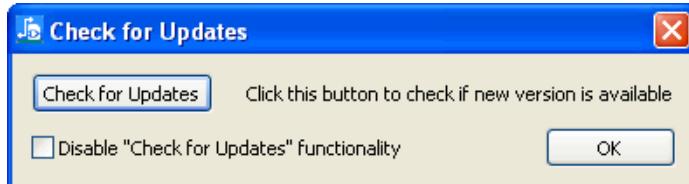
[Modifying Parts](#)

Check for Updates Dialog Box

To access: **Help > Check for Updates** menu item

Use the Check for Updates dialog box to manually check for a new version of SailWind, and to disable or enable automatic checks.

Figure 49. Check for Updates Dialog Box



Objects

Table 83. Check for Updates Dialog Box contents

Name	Description
Check for Updates button	Manually checks for a new version of the SailWind software.
Disable "Check for Updates" functionality	Determines if SailWind automatically checks for a new version of the software. Click to stop SailWind from automatically checking for a new version of the SailWind products; click to clear to have SailWind automatically check for a new version.

Related Topics

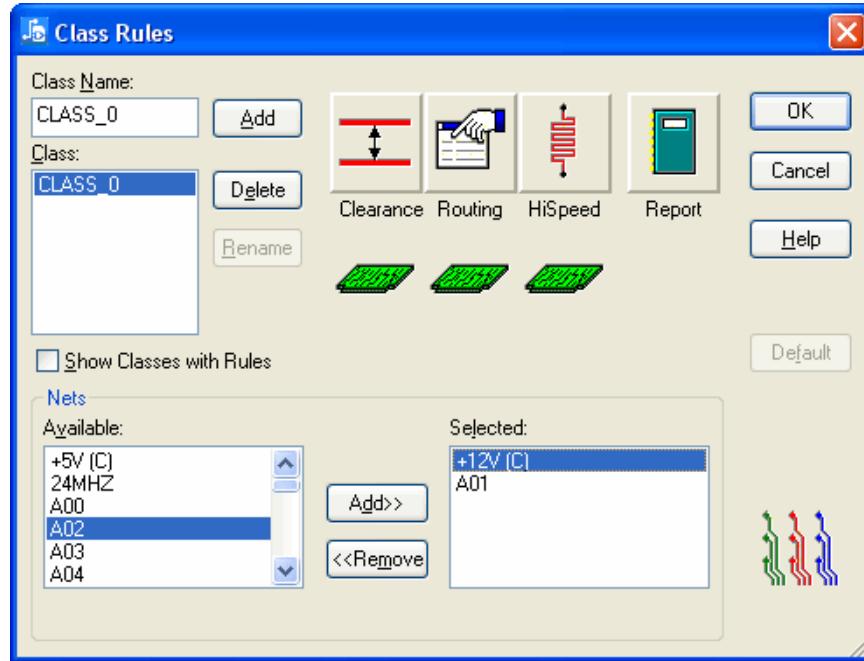
[SailWind Updates](#)

Class Rules Dialog Box

To access: **Setup > Design Rules** menu item > **Class** button

Use the Class Rules dialog box to define rules that apply to a collection of nets known as a net class and to multiple net classes.

Figure 50. Class Rules Dialog Box



Objects

Table 84. Class Rules Dialog Box Contents

Name	Description
Class Name	Specifies the class name for which you want to apply rules.
Class list	Defines net classes by name and parenthetically notes the rules that apply, if any, to the class.
Show Classes with Rules	Specifies to list only classes with at least one set of rule definitions.
Add button	Adds a new Class Name to the Class list.
Delete button	Removes the selected Class Name from the Class list.
Rename button	Renames the selected Class Name in the Class list. Tip Select the name in the list, type the new name in the Class Name box and then click Rename.

Table 84. Class Rules Dialog Box Contents (continued)

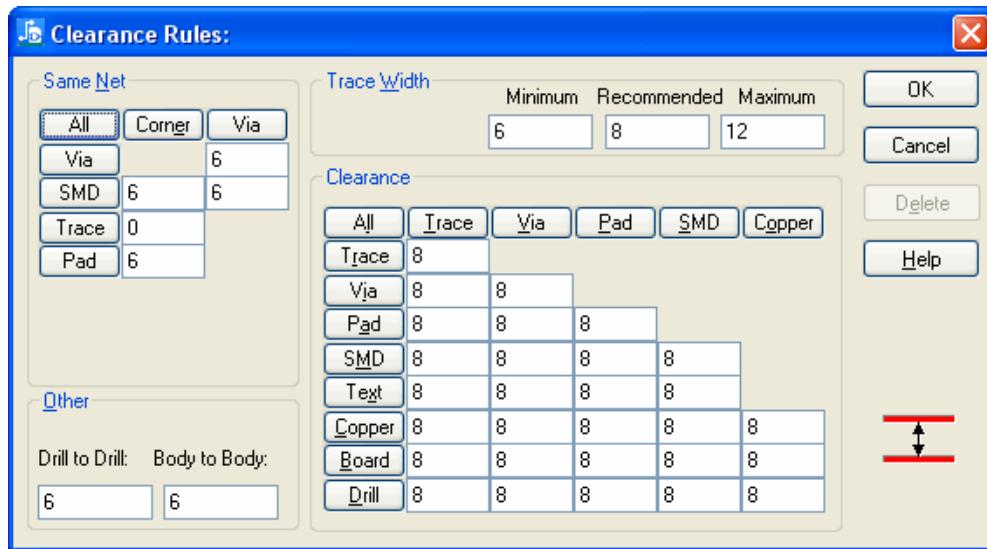
Name	Description
Clearance button	Opens the Clearance Rules Dialog Box .
Routing button	Opens the Routing Rules Dialog Box .
Hi Speed button	Opens the HiSpeed Rules Dialog Box .
Report button	Opens the Rules Report Dialog Box .
Available list	Lists all of the available nets for this Class.
Add>> button	Moves the net from the Available list to the Selected list.
<<Remove button	Moves the net from the Selected list to the Available list.
Selected list	Lists all of the selected nets for this Class.
Default button	Removes non-default rules from the selected nets, so that only default rules apply.

Clearance Rules Dialog Box

To access: **Setup > Design Rules** menu item > a rule hierarchy button > **Clearance** button

Use the Clearance Rules dialog box to define the spacing permitted between objects. When objects are imported, the On-line DRC and Verify Design programs use these rules to check and report clearance violations.

Figure 51. Clearance Rules Dialog Box



Objects

Table 85. Clearance Rules Dialog Box contents

Name	Description
Same Net area	<p>Defines edge-to-edge clearance values between items that are in the same net.</p> <p>i Tip</p> <ul style="list-style-type: none"> To define the minimum spacing between any two objects, type the value in the corresponding text box. To define the same spacing value for all text boxes in one matrix column, press the button above the column. Type a value and click OK. To define the same spacing value for all text boxes in one matrix row, press the button in the left column. Type a value and click OK. To define the same spacing value for all text boxes in the matrix, press the All button. Type a value and click OK. <p>See also Same Net Matrix</p>
Trace Width area	Specifies to restrict the trace width to a range of values.

Table 85. Clearance Rules Dialog Box contents (continued)

Name	Description
	 Tip <ul style="list-style-type: none"> In the Recommended box, type the width you want to assign to the trace when routing begins. In the Minimum and Maximum boxes, values are respected by routing routines that must use trace width to achieve some high-speed routing functions, such as impedance matching.
Clearance area	<p>Defines edge-to-edge clearances between two object types:</p> <ul style="list-style-type: none"> To define the minimum spacing between any two objects, type the value in the appropriate text box. To define the same spacing value for all text boxes in one matrix column, press the button above the column and type a value. To define the same spacing value for all text boxes in one matrix row, press the button in the left column and type a value. To define the same spacing value for all text boxes in the matrix, press the All button and type a value.
Other area	<p>Specifies other optional clearance values.</p> <ul style="list-style-type: none"> Drill to Drill — The minimum edge-to-edge spacing between two drill holes. Body to Body — The minimum edge-to-edge spacing between two component bodies.
Delete button	<p>Removes this set of Clearance rules from your rules hierarchy.</p>  Tip <p>You cannot delete the Default Clearance rules.</p>

Related Topics

[Setting Up Clearance Rules](#)

Compare/ECO Tools Dialog Box

When you compare two versions of a design, you can create an output file that lists the differences between the two versions. The report file is named *Logic.rep* and is written to the *\SailWind Projects* folder.



Restriction:

Transferring non-ECO-registered parts and non-electrical parts is constrained. See [Options Dialog Box, Design Category](#) for details.

To access: **Tools > Compare/ECO** menu item

[Compare/ECO Tools Dialog Box, Documents Tab](#)

[Compare/ECO Tools Dialog Box, Comparison Tab](#)

Compare/ECO Tools Dialog Box, Documents Tab

To access: Tools > Compare/ECO menu item > Documents tab

Use the **Documents** tab to specify the design netlists to compare and the files to create.



Tip

You can compare designs in any of the following forms:

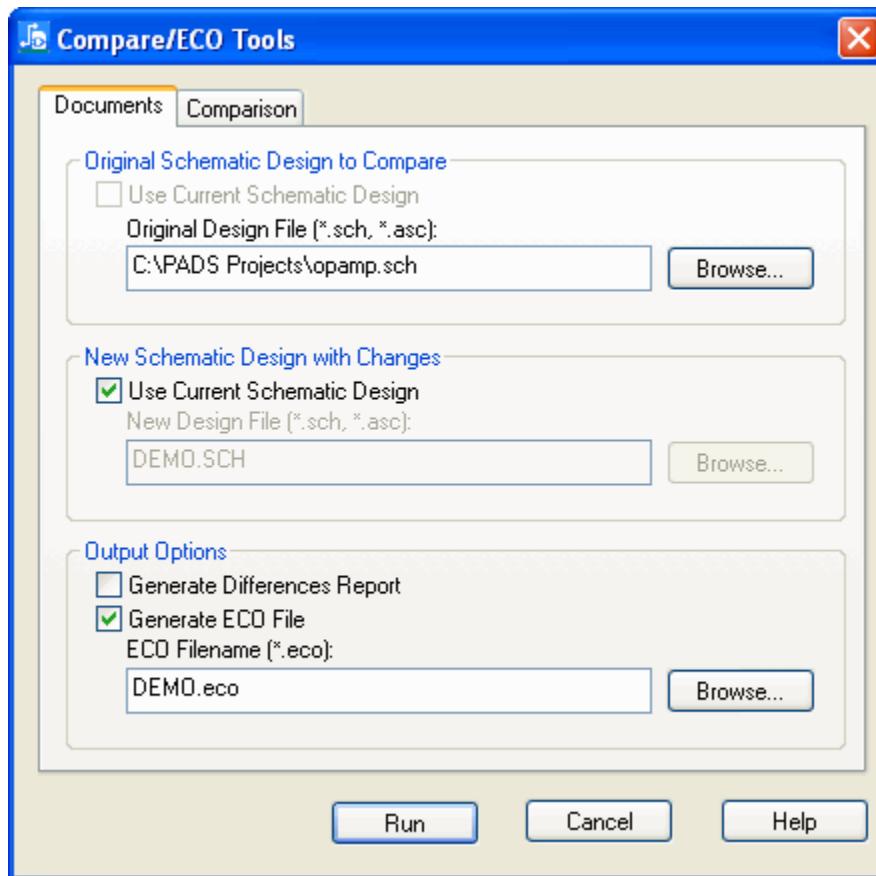
- Schematic in memory
- Binary schematic file (.sch)
- PADS-format ASCII file (.asc)



Restriction:

Transferring non-ECO-registered parts and non-electrical parts is constrained. See [Options Dialog Box, Design Category](#) for details.

Figure 52. Compare/ECO Tools Dialog Box, Documents Tab



Objects

Table 86. Compare/ECO Tools Dialog Box, Documents Tab Contents

Name	Description
Original Schematic Design to Compare	<p>Specify the design you want to update. Do one of the following:</p> <ul style="list-style-type: none">Select Use Current Schematic Design to use the schematic in memory as the original design.Clear Use Current Schematic Design, and then type or browse to the original design file. The original design file format can be binary (.sch) or PADS-format ASCII (.asc) output from the schematic or layout. <p> Restriction: The original design cannot have the same filename as the new design.</p>
New Current Schematic Design	<p>Specify the design containing the changes you want to place into the original design. Do one of the following:</p> <ul style="list-style-type: none">Select Use Current Schematic Design to use the schematic in memory as the new design.Clear Use Current Schematic Design, and then type or browse to the new design file. The new design file format can be binary (.sch) or PADS-format ASCII (.asc) output from the schematic or layout. <p> Restriction: The new design cannot have the same filename as the original design.</p>
Output Options	Specifies to create a report file containing a description of the differences between the two design versions. This file is named <i>Logic.rep</i> and is stored in the \SailWind Projects folder.
Generate ECO File	Specifies to create an ECO file. Type or browse to the ECO file. The ECO file contains ECO commands that describe the changes needed to update the original design to match the new design.

Related Topics

[Forward Annotation From SailWind Logic to SailWind Layout](#)

[Backward Annotation From SailWind Layout to SailWind Logic](#)

[Contents of the Differences Report](#)

Compare/ECO Tools Dialog Box, Comparison Tab

To access: Tools > Compare/ECO menu item> Comparison tab

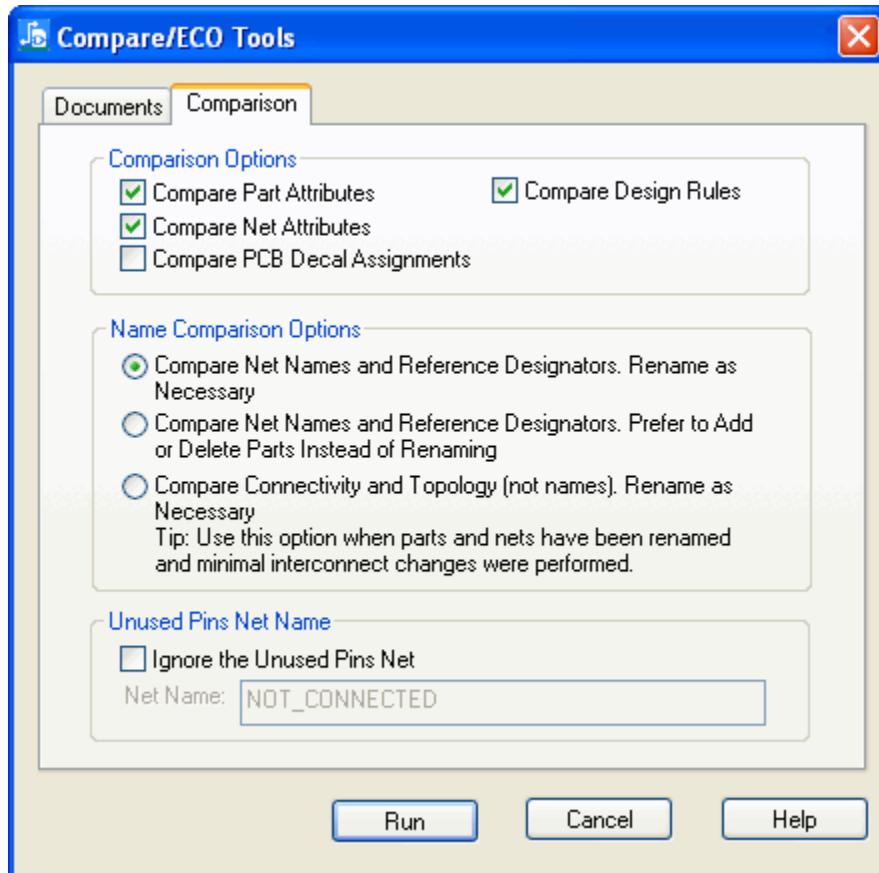
Use the **Comparison** tab to specify the design elements to include in the design netlists comparison.



Restriction:

Transferring non-ECO-registered parts and non-electrical parts is constrained. See [Options Dialog Box, Design Category](#) for details.

Figure 53. Compare/ECO Tools Dialog Box, Comparison Tab



Objects

Table 87. Compare/ECO Tools Dialog Box, Comparison Tab Contents

Name	Description
Comparison Options area	
Compare Part Attributes	Comparison includes part attributes. Only part attributes are compared and updated or reported. Each part receives attributes

Table 87. Compare/ECO Tools Dialog Box, Comparison Tab Contents (continued)

Name	Description
	<p>from its corresponding Decal and Part Type, but modification is performed only at the part level.</p> <p>SailWind Logic categorizes parts that have different attributes as different part types. Therefore, if you select Include Part Attributes when you generate the netlist, you must also select Compare Part Attributes when you perform an ECO comparison of that netlist. Otherwise the comparison considers the part types to be different and reports errors.</p>
Compare Net Attributes	<p>Comparison includes net attributes.</p> <p>SailWind Logic categorizes nets that have different attributes as different nets. Therefore, if you select Include Net Attributes when you generate the netlist, you must also select Compare Net Attributes when you perform an ECO comparison of that netlist. Otherwise the comparison considers the net types to be different and reports errors.</p>
Compare PCB Decal Assignments	Comparison includes PCB decal assignments.
Compare Design Rules	Comparison includes design rules. Only Default, Net, and Net Class rules are compared. Rules on other objects in the original design are preserved where possible.
Name Comparison Options area	
Compare Net Names and Reference Designators. Rename as Necessary	<p>Compare differences using reference designators and net names. Best used to minimize changes to routed traces. Selecting this option may result in the positional swapping of parts.</p>
Compare Net Names and Reference Designators. Prefer to Add or Delete Parts Instead of Renaming	<p>Compare differences using reference designators and net names on the basis that few reference designators have been renamed and nets have not been renamed.</p> <p>Best used to minimize the positional swapping of parts, and the design disruption that may result.</p>
Compare Connectivity and Topology (not names). Rename as necessary.	<p>Compare differences without using reference designators or net names. Compare differences using pin names, part type names, and so on.</p> <p>Best used to compare designs when parts and nets have been renamed and minimal interconnect changes have been performed.</p>
Unused Pins Net Name area	
Ignore the Unused Pins Net	<p>Select the check box and type the name of the unused pins net if you want the design comparison to exclude the unused pins net in the original design. The unused pins net contains pins that have no logical net association. An unused pins net may be created in the PCB design process.</p> <p> Restriction: Specify a net name of 47 characters or less. Use any alphanumeric characters except curly braces {}, asterisks *, or spaces.</p>

Table 87. Compare/ECO Tools Dialog Box, Comparison Tab Contents (continued)

Name	Description
	 Tip If you clear this option and you update the PCB layout from SailWind Logic, the unused pins net may be deleted.

Related Topics

[Forward Annotation From SailWind Logic to SailWind Layout](#)

[Backward Annotation From SailWind Layout to SailWind Logic](#)

[Contents of the Differences Report](#)

Conditional Rule Setup Dialog Box

To access: **Setup > Design Rules** menu item > **Conditional Rules** button

Once you set up Clearance rules for a group in the hierarchical order, the rules are applied to all other objects. Use the Conditional Rule Setup dialog box to apply a third overriding set of rules that apply only when the item meets other specific levels of the hierarchical order.

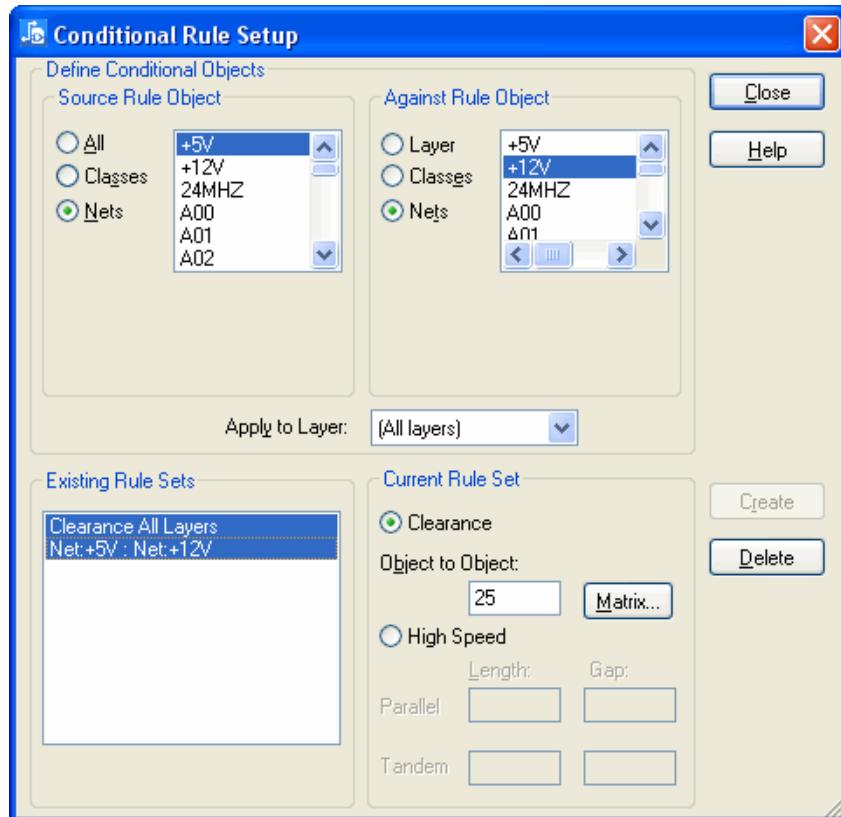


Tip

When setting up conditional rules, observe the following:

- You can use a layer as an against object, where rules you set for an object such as a net apply only when the net is routed on that layer.
- You can further refine this to use another net as an against object and specify a layer to which the rules to apply. If these two nets meet on this layer, they must adhere to this clearance. You define these relationships by making conditional rule sets.

Figure 54. Conditional Rule Setup Dialog Box



Objects

Table 88. Conditional Rule Setup Dialog Box Contents

Name	Description
Source Rule Object area	Specifies the object you want to use to check rules. Choose Classes, Nets, or All.
Against Rule Object area	Specifies which object to use to check the rules against. Choose Layer, Classes, or Nets.
Apply to Layer list	Specifies the layer on which you want the rules to be checked.
Existing Rule Sets list	Lists rule sets that have already been created.
Current Rule Set area	
Clearance Object to Object	Specifies the minimum clearance gap you want the source and against items to maintain from each other.
Matrix button	Opens the Clearance Rules Dialog Box .
High Speed	Specifies the clearance values for parallel and tandem checking for the condition. The source-against entries are used as the victim-aggressor designations for crosstalk conditions checking. For more information, see Setting Up High-Speed Rules .
Create button	Compile the new rule set parameters and adds a description to the Existing Rule Sets list box.
Delete button	Removes the selected rule set from the Existing Rule Sets list box

Connect to SailWind Layout Dialog Box

To access: Tools > SailWind Layout menu item

Use the Connect to SailWind Layout dialog box to choose your connection to SailWind Layout. You can then gain access to the SailWind Layout Link dialog box.

Figure 55. Connect to SailWind Layout Dialog Box



Objects

Table 89. Connect to SailWind Layout Dialog Box Contents

Name	Description
New button	<p>Opens and connects SailWind Logic to a new session of SailWind Layout then opens the SailWind Layout Link Dialog Box.</p> <p>i Tip If there is a delay in launching and connecting to SailWind Layout, the Server Busy Dialog Box may appear. Wait a short time or till SailWind Layout appears on your Windows Taskbar then click Retry.</p>
Open button	<p>Opens and connects to an existing design file in a new SailWind Layout session then opens the SailWind Layout Link Dialog Box.</p> <p>i Tip If a prompt window is open in SailWind Layout, the Server Busy Dialog Box may appear. Click the Switch To button and take care of the prompt to enable SailWind Logic to connect to SailWind Layout.</p>

Connect to SailWind Router Dialog Box

To access: Tools > SailWind Router menu item

Use the Connect to SailWind Router dialog box to choose your connection to SailWind Router. You can then gain access to the SailWind Router Link dialog box.

Figure 56. Connect to SailWind Router Dialog Box



Objects

Table 90. Connect to SailWind Router Dialog Box Contents

Name	Description
New button	Opens and connects SailWind Logic to a new session of SailWind Router then opens the SailWind Router Link Dialog Box . Tip If there is a delay in launching and connecting to SailWind Router, the Server Busy Dialog Box may appear. Wait a short time or till SailWind Router appears on your Windows Taskbar then click Retry .
Open button	Opens and connects to an existing design file in a new SailWind Router session then opens the SailWind Router Link Dialog Box . Tip If a prompt window is open in SailWind Router, the Server Busy Dialog Box may appear. Click the Switch To button and take care of the prompt to enable SailWind Logic to connect to SailWind Router.

Crash Detected Dialog Box

To access: This dialog box is inaccessible unless the software crashes and crash detection is enabled in the software *.ini* file.

The Crash Detected dialog box opens at a crash and enables you to save a report of the SailWind environment as well as pertinent files into a compressed SailWind Dump File for troubleshooting.

Objects

Table 91. Crash Detected Dialog Box Contents

Name	Description
Comments box	You can describe what you were doing when the error occurred or anything else you can think of that might help when investigating the crash.
Attach BMW data check box	You can include BMW data and your project files. This will enable customer support to play back what you were doing in your design that led up to the crash. This check box is unavailable if the BMW feature is not enabled. See also BMW and BLT .
Save button	You must click the Save button if you want to create a report file. When you click the Save button, you are prompted with a Save As dialog box. The file that is created is called a Dump File and is compressed in the <i>.zip</i> format. This is the file that you must send to customer support. It will include the report, the BMW data and the project files.

Create PDF Dialog Box

To access: **File > Create PDF** menu item > enter filename > **Save**

You can create an intelligent PDF of your schematic, choosing which sheets you want to share and show to others in your organization. You can create the PDF in full-color or black and white, with hyperlinks to part attributes, and with search capabilities, making it easy to locate parts and nets. Once you locate a net, you can find other instances of it through the entire schematic, even when the net is on a different page. You can also create a black and white, non-searchable PDF of your schematic.



Restriction:

To search a PDF, you must substitute the Stroke font with a System font.

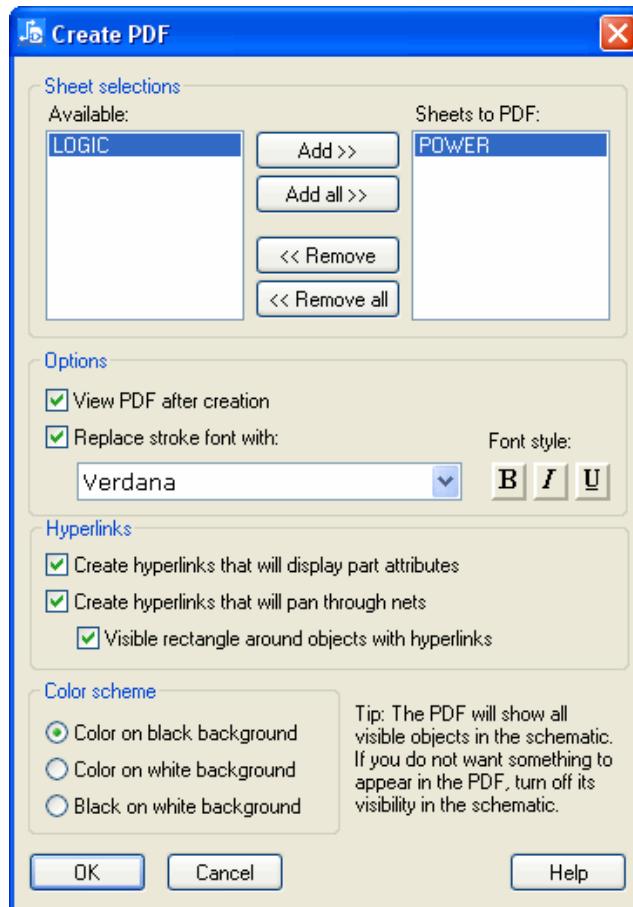


Tip

Adobe® Acrobat® Distiller™ is not required on your system to create a PDF.

See also [Printing to PDF](#).

Figure 57. Create PDF Dialog Box



Objects

Table 92. Create PDF Dialog Box Contents

Name	Description
Available list	List all of the sheets available to you.
Add >> Button	Moves the selected sheet to the Sheets to PDF list.
Add All >> Button	Moves all sheets to the Sheets to PDF list.
<< Remove button	Moves the selected sheet to the Available list.
<< Remove All Button	Moves all sheets to the Available list.
Sheets to PDF list	Lists the sheets you want to PDF.
View PDF after creation	Specifies to open the PDF after it has been created.
Replace Stroke font with	Specifies to replace the Stroke font with a system font. Select the font you want from the list. Use the Font style buttons to add Bold, Italic, or Underline styles.  Restriction: <ul style="list-style-type: none"> You can only search a PDF if you replace the Stroke font with a System font. There is no font size control. Text heights are already set for each text item in the design. The height will be converted to the nearest point size in the PDF.
Create hyperlinks that will display part attributes	Specifies to create a listing of the part attributes as a hyperlink.
Create hyperlinks that will pan through nets	Specifies to create a link to pan to the next instance of that net or bus.  Restriction: <p>If the net name is not visible in the schematic, you will not be able to pan through the nets.</p>
Visible rectangle around objects with hyperlinks	Specifies to create all hyperlinks with visible boxes: yellow around the part attributes, blue around the net and bus names.  Restriction: <p>This check box is only available when one of the two check boxes above it is clicked.</p>
Color scheme area	Specifies the color scheme you want to use.  Tip <p>The colors used in the Color on Black or Color on White schemes are the same colors currently used in your schematic. The Black on White scheme shows all currently visible items in the schematic in black.</p>

Customize Dialog Box

Use the **Commands** tab to add commands to menus or toolbars, or to create custom menus.

To access: **Tools > Customize** menu item

[Customize Dialog Box, Commands Tab](#)

[Customize Dialog Box, Keyboard and Mouse Tab](#)

[Customize Dialog Box, Macro Files Tab](#)

[Customize Dialog Box, Options Tab](#)

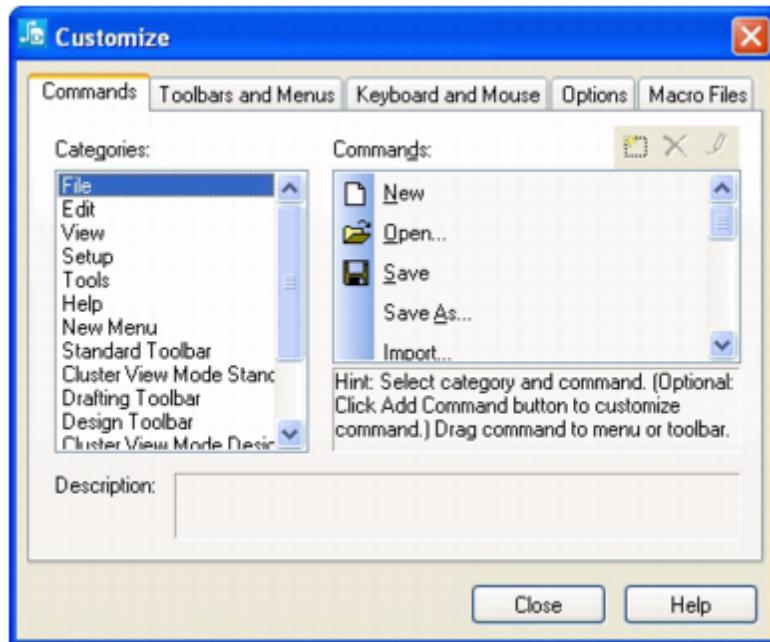
[Customize Dialog Box, Toolbars and Menus Tab](#)

Customize Dialog Box, Commands Tab

To access: Tools > Customize menu item > Commands tab

Use the **Commands** tab to add commands to menus or toolbars, or to create custom menus.

Figure 58. Commands Tab



Objects

Table 93. Command Tab Contents

Name	Description
Categories list	Narrows down the list of commands.
Commands list	List of commands available to add to a menu or toolbar.
	Add a new command, delete a command you've added, or rename a command you've added.

Related Topics

[Creating a Custom Command](#)

[Creating a Custom Menu](#)

[Defining Properties for a New Command](#)

[Editing a Custom Command](#)

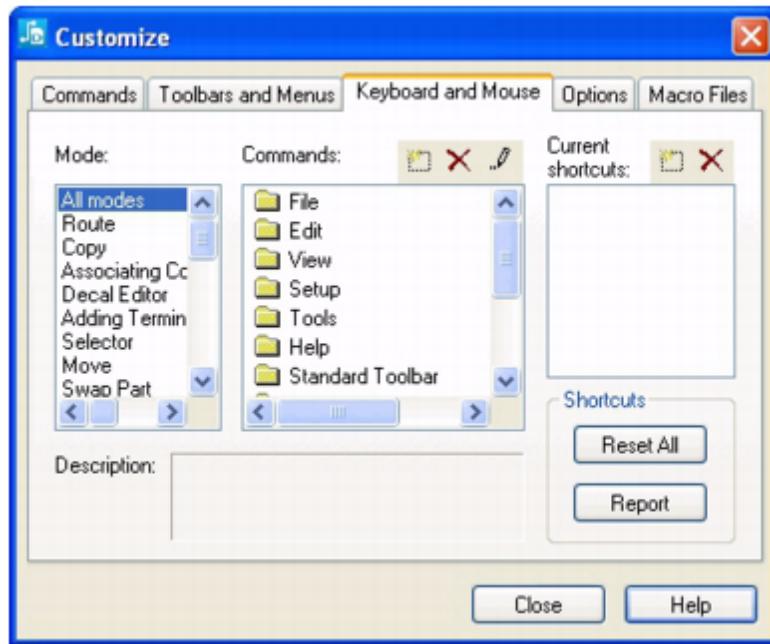
[Adding Items to Toolbars and Menus](#)

Customize Dialog Box, Keyboard and Mouse Tab

To access: Tools > Customize menu item > **Keyboard and Mouse** tab

Create and customize shortcut keys using the **Keyboard and Mouse** tab of the Customize dialog box.

Figure 59. Keyboard and Mouse Tab



Objects

Table 94. Keyboard and Mouse Tab Contents

Name	Description
Mode list	Narrows down the list of commands.
Commands list	The list of commands available for which to assign a shortcut.
	Add a new command (opens the Add Command Dialog Box on page 440), delete a command you've added, or rename a command you've added (opens the Edit Command dialog box on page 440).
Current shortcuts list	The list of shortcuts assigned to the selected command.
	Add a new shortcut (open the Assign Shortcut Dialog Box), or delete a shortcut you've added.
Description	Lists what the selected command does.

Table 94. Keyboard and Mouse Tab Contents (continued)

Name	Description
Reset All button	Sets the selected toolbar or shortcut menu to the default settings.
Report button	Saves a report of all current shortcut commands.

Customize Dialog Box, Macro Files Tab

To access: Tools > Customize menu item > **Macro Files** tab

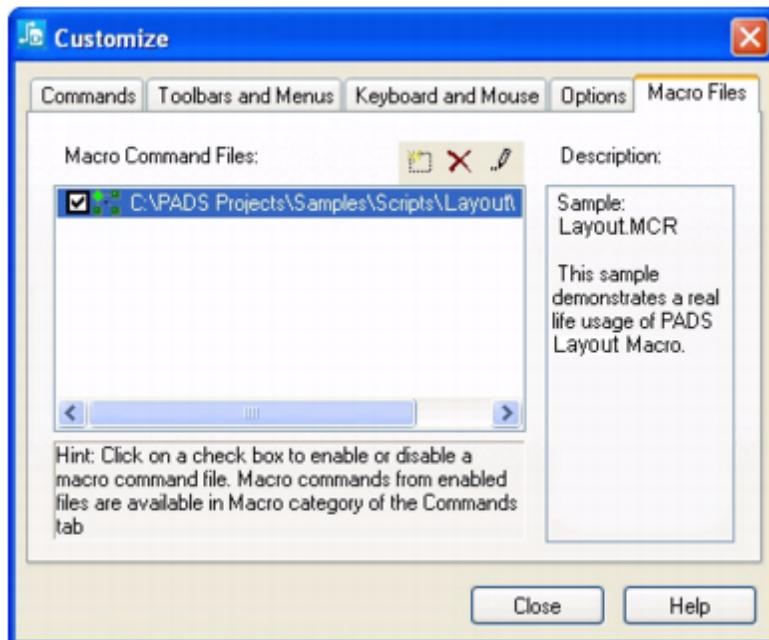
Create commands from macro files and add them to toolbars and menus using the **Macro Files** tab.



Tip

To create a command from a macro command file, the macro command file (.mcr) must already exist. You can create a macro by recording it in a SailWind tool or scripting it in Macro language. For more information, see “[Macros](#)” on page 47.

Figure 60. Macro Files Tab



Objects

Table 95. Macro Files Tab Contents

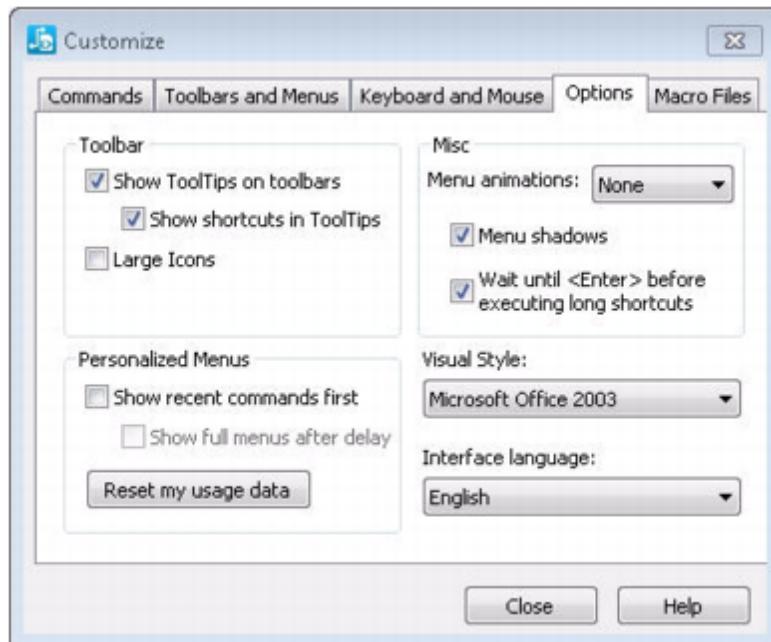
Name	Description
Macro Command Files list	The list of macro files you have opened.
	Add a macro to the list (opens the Open Macro dialog box), delete a macro from the list, or edit the location of a macro you've added.
Description	Lists what the selected macro does.

Customize Dialog Box, Options Tab

To access: Tools > Customize menu item > Options tab

Customize the SailWind interface by changing the appearance of menus and toolbars using the **Options** tab of the Customize dialog box.

Figure 61. Options Tab



Objects

Table 96. Options Tab Contents

Name	Description
Show ToolTips on toolbars	Displays the button name over the toolbar button when you hover over it with your pointer.
Show shortcuts in ToolTips	In addition to the name in the ToolTip, displays the shortcut for the button.
Large Icons	Displays icons on the toolbar larger than the default size.
Menu animations list	The type of animation for your menus: None, Unfold, Slide, or Fade.
Menu shadows	Displays a shadow behind the menu.

Table 96. Options Tab Contents (continued)

Name	Description
Wait until <Enter> before executing long shortcuts	Delays the execution of shortcut keys until you press Enter.
Show recent commands first	Displays your recent menu command selections at the top of the list.
Show full menus after delay	Displays the full menu after a slight pause.
Reset my usage data button	Restores the default set of commands to the menus and toolbars.  Tip This option does not undo any explicit customizations you made.
Visual Style	Sets the look and feel of your toolbars and title bars.
Interface Language	Specifies the language for all dialog boxes and messages displayed: English, Chinese Simplified.

Related Topics

[Customized Appearance of the Screen](#)

Customize Dialog Box, Toolbars and Menus Tab

To access: Tools > Customize menu item > Toolbars and Menus tab

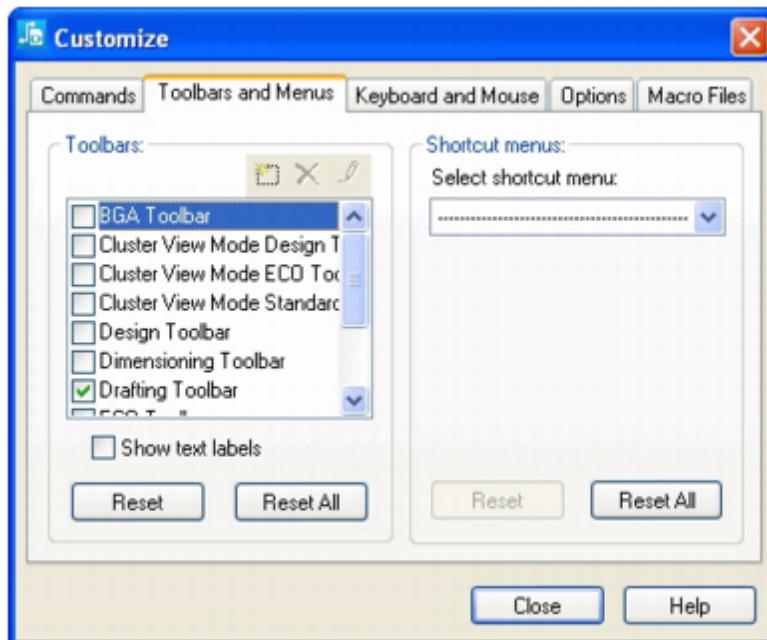
Use the **Toolbars and Menus** tab on the Customize dialog box to create custom toolbars and shortcut menus.



Tip

To create a custom main menu, use the **Commands** tab on the Customize dialog box. See [Creating a Custom Menu](#).

Figure 62. Toolbars and Menus Tab



Objects

Table 97. Toolbars and Menus Tab Contents

Name	Description
Toolbars list	Specify which toolbars to display in the main window.
Add toolbar	Add a new toolbar, delete a toolbar you've added, or rename a toolbar you've added.
Show text labels	Shows the text label on the button in addition to the icon.
Select shortcut menus	Specifies the shortcut menu you want to customize.

Table 97. Toolbars and Menus Tab Contents (continued)

Name	Description
	 Restriction: SailWind Router only.
Reset button	Sets the selected toolbar or shortcut menu to the default settings.
Reset All button	Sets all toolbars or shortcut menus back to their default settings.

Related Topics

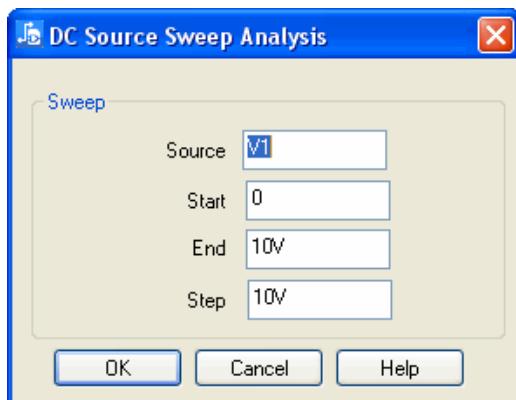
[Customizing the SailWind Interface](#)

DC Source Sweep Analysis Dialog Box

To access: Tools > SPICE Netlist menu item > Simulation Setup button> DC Sweep button

Use the DC Source Sweep Analysis dialog box to set options specifically for a DC Sweep analysis.

Figure 63. DC Source Sweep Analysis Dialog Box



Objects

Table 98. DC Source Sweep Analysis Dialog Box Contents

Name	Description
Source	Specifies the name of the voltage or current source.
Start	Specifies the starting voltage for the sweep.
End	Specifies the stopping voltage for the sweep.
Step	Specifies the incrementing values for the sweep.

Related Topics

[Creating a SPICE Netlist](#)

[Setting Up DC Source Sweep Analysis](#)

Default Rules Dialog Box

To access: **Setup > Design Rules** menu item > **Default** button

Use the Default Rules dialog box to define rules which apply to all objects that are not included in any other rule within the hierarchy.

Figure 64. Default Rules Dialog Box



Objects

Table 99. Default Rules Dialog Box Contents

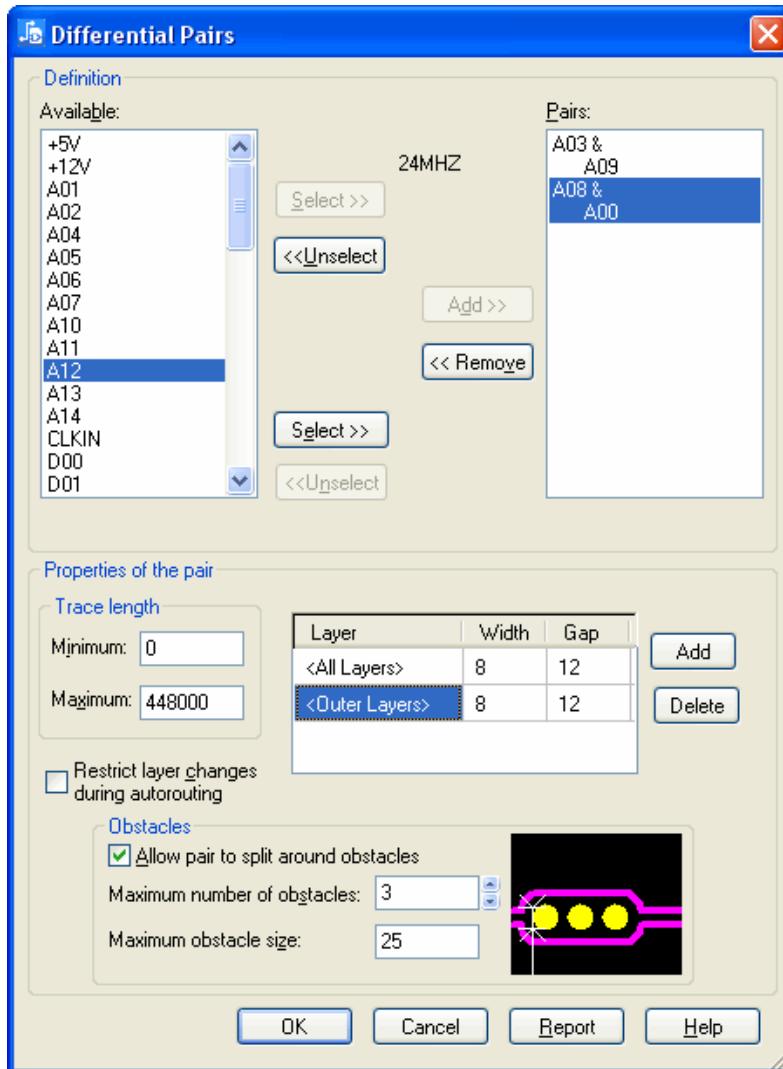
Name	Description
Clearance button	Opens the Clearance Rules Dialog Box .
Routing button	Opens the Routing Rules Dialog Box .
Hi Speed button	Opens the HiSpeed Rules Dialog Box .
Report button	Opens the Rules Report Dialog Box .

Differential Pairs Dialog Box

To access: **Setup > Design Rules** menu item > **Differential Pairs** button

Use the Differential Pairs dialog box to identify nets or pin pairs that behave electrically as differential pairs, and to define differential pair design rules.

Figure 65. Differential Pairs Dialog Box



Objects

Table 100. Differential Pairs Dialog Box Contents

Name	Description
Available list	Lists nets that have not been assigned to a differential pair.

Table 100. Differential Pairs Dialog Box Contents (continued)

Name	Description
Selected list	<p>Lists the nets that have been selected.</p> <p>Tip Click Add to move the selected nets to the Pairs list.</p>
Select >> button	Moves the selected net to the Selected list.
<<Unselect button	Moves the net that was previously selected back to the Available list.
Add>> button	Moves the two nets in the Selected list to the Pairs list.
<<Remove button	Moves the two nets in the Pairs list to the Available list.
Pairs list	Lists the differential pairs.
Trace length area	Specifies the minimum and maximum trace length values.
Restrict layer changes during autorouting	<p>Forces the pair to be routed on a single layer.</p> <p>Tip This setting does not restrict layer changes when routing interactively.</p>
Properties of the pair table	Sets the width and gap per layer.
Add button	Adds a row to the Properties of the pair table.
Delete button	Removes the selected row from the Properties of the pair table.
Obstacles area	<p>Specifies to allow routing around an obstacle in the controlled gap area by temporarily exceeding the pair routing gap.</p> <p>Tip This setting applies to autorouting and does not restrict splitting around obstacles when routing interactively. Also specifies the maximum number of obstacles and the maximum obstacle size.</p> <p>Tip Obstacles in the start zone or end zone are not counted.</p>
Report button	Opens the Rules Report Dialog Box .

Display Colors Dialog Box

To access: **Setup > Display Colors** menu item

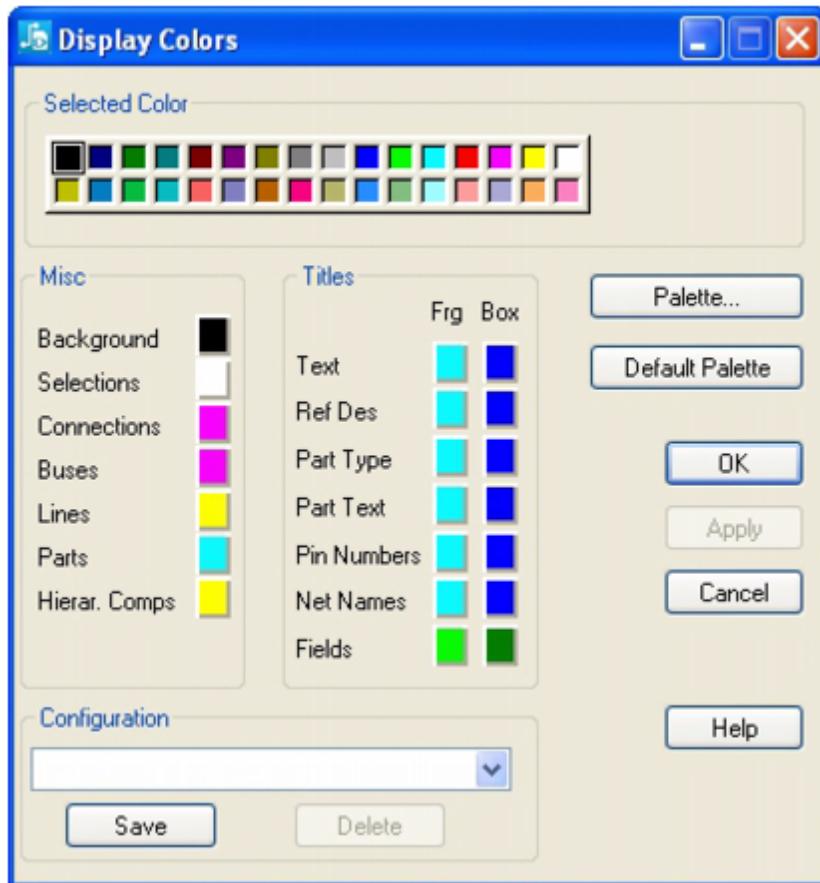
Use the Display Colors Setup dialog box to Set display colors, save them, and restore them; Change the color palette; make objects visible; and make objects invisible.



Tip

Changes you make to the color configuration in the Display Colors Setup dialog box do not apply to disabled layers.

Figure 66. Display Colors Setup Dialog Box



Objects

Table 101. Display Colors Setup Dialog Box Contents

Name	Description
Selected Color area	Select a color from the palette to assign to items on a layer. Once you select a color here, click the tile in the Color by Layer area of the item to which you want to assign the color.
Palette button	Opens the Color dialog box where you can choose to use new colors or customize colors you want to use.
Default Palette button	Reassigns all colors and settings to the default settings.  Tip You can change the default settings by saving a configuration and naming it default.
Misc area	Apply a color to objects in the Misc area to change the color of that object in the workspace.
Titles area	To make a text string visible in the part editor, select the check box beside the string type in the Titles area.
Configuration list	The list of saved configurations.
Save button	Opens the Save Configuration Dialog Box .
Delete button	Removes the selected configuration from the Configuration list.

Related Topics

[Display Colors Dialog Box - Part Editor](#)

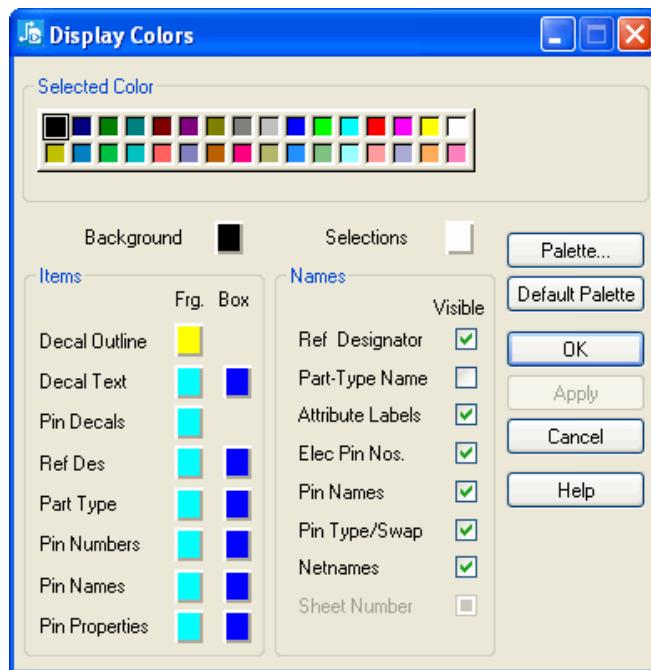
[Setting Display Colors](#)

Display Colors Dialog Box - Part Editor

To access: In the Part Editor, **Setup > Display Colors** menu item

Use the Display Colors dialog box while in the Part Editor or the CAE Decal Editor to control the colors of objects and the working area. SailWind Logic saves the color settings for part editor items with the schematic. This dialog box is similar to the one used in the schematic editor, with check boxes to display text items.

Figure 67. Display Colors Dialog Box in the Part Editor



Objects

Table 102. Display Colors Dialog Box contents

Name	Description
Selected Color	Select a color in this area to apply to tiles in the Items area.
Palette button	Click to open the Color dialog box where you can specify new colors or customize colors that appear in the Selected Color area.
Default Palette button	Click Default Palette to restore the default color settings in the Selected Color area.
Background	Apply a color to this color tile to change the background color or work area surface in the Part Editor workspace.



Tip

Refer to the Microsoft Windows Help for more information about changing the Color Palette.

Table 102. Display Colors Dialog Box contents (continued)

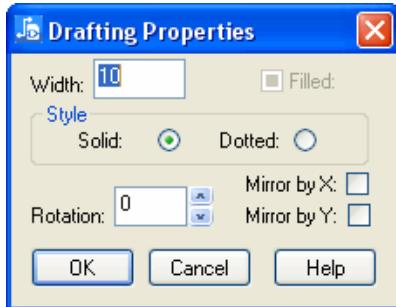
Name	Description
	 Tip <ul style="list-style-type: none"> • To make an item invisible, set it to the background color. • The color selection for displaying items or making them invisible does not affect plotting of the schematic.
Selections	Apply a color to this color tile to change the color of objects that you select in the workspace.
Items area	<p>Apply a color to objects in the Items area to change the color of that object in the workspace.</p>  Tip <p>Some of the items have a color setting in the Box column. Box indicates the color of the box that is drawn around the text item. This box serves two purposes: it indicates the exact size of the text item when it is plotted, thereby helping you avoid overlaps while moving the item; and it provides visibility of the text item at very small zoom levels.</p>
Names area	<p>To make a text string visible in the part editor, select the check box beside the string type in the Names area.</p>  Restriction: <ul style="list-style-type: none"> • This area is only available when editing a CAE decal, pin decal, or when editing the graphics of individual connector, or off-page type symbols. • When editing a CAE decal - Sheet Number is unavailable. • When editing a pin decal - Reference Designator, Part-Type Name, Attribute Labels, and Sheet Number are unavailable. • When editing the pin decal of an off-page, power or ground symbol - only Netnames and Sheet Number are available. • When editing the graphics of a single connector pin - only Ref Designator, Part-Type Name, and Attribute Labels are available.

Drafting Properties Dialog Box

To access: Select a drafting object > right-click > **Properties** menu item

Use the Drafting Properties dialog box to modify the line width, style, and orientation of selected drafting objects.

Figure 68. Drafting Properties Dialog Box



Objects

Table 103. Drafting Properties Dialog Box Contents

Field	Description
Width	Specifies the line width for the drafting object. The current line width of the selected drafting object is automatically displayed, change it if necessary. Tip Use the Line Widths tab in the Options dialog box to change the default line width.
Filled	Specifies to create a filled shape from a selected polygon. Tip This option is grayed for circles, paths, and if you used Pull Arc to modify the polygon.
Style area	Specifies the line style option for the selected drafting object: Solid or Dotted.
Rotation	Specifies the degree of rotation from the Rotation list. Tip <ul style="list-style-type: none"> • Rotation can be 0 or 90 degrees. • The point used when selecting the object is the also the point of rotation.
Mirror by X	Specifies to mirror the selected drafting object in the X (horizontal) direction.
Mirror by Y	Specifies to mirror the selected drafting object in the Y (vertical) direction.

Related Topics

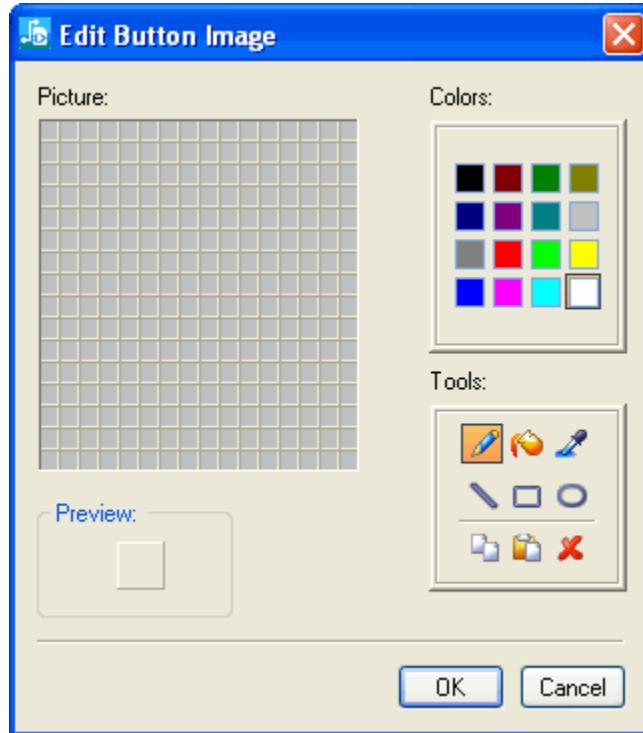
[Modifying Drafting Objects](#)

Edit Button Image Dialog Box

To access: **Tools > Customize** menu item > **Commands** tab > **New** button > Select User-Defined Image > **New** or **Edit** button

Use the Edit Button Image dialog box to create or edit button icons.

Figure 69. Edit Button Image Dialog Box



Objects

Table 104. Edit Button Image Dialog Box Contents

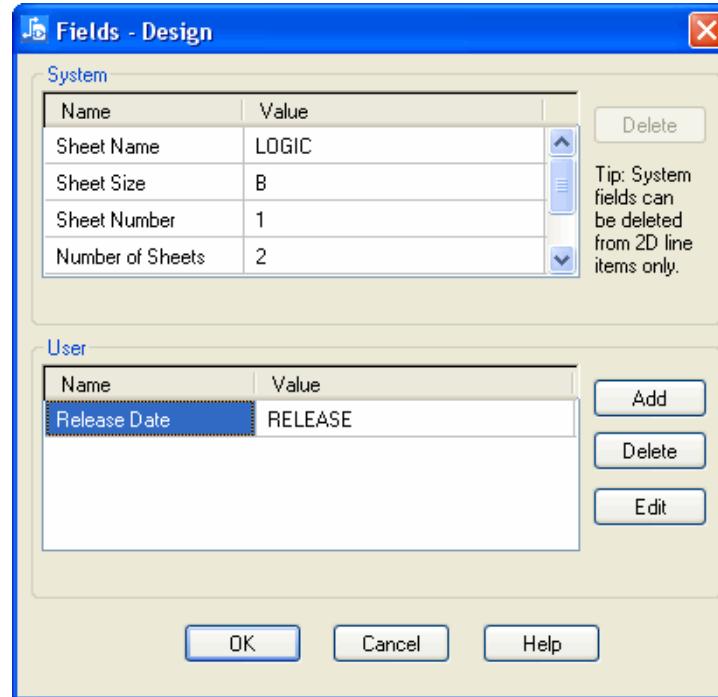
Field	Description
Colors area	Select a color to use with the tools
Tools area	Select a tool to draw/edit the picture or icon of the button

Fields Dialog Box

To access: Select nothing or a 2D line object > right-click > **Fields** menu item

Use the Fields dialog box to manage multiple fields. You can manage the fields in the entire schematic or in a 2D line object.

Figure 70. Fields Dialog Box



Objects

Table 105. Fields Dialog Box Contents

Field	Description
System	Displays the name and value for all system fields in the design.
Delete button	Removes the selected row. Restriction: System fields can be deleted from 2D line items only.
User	Displays the name and value for all user fields in the design
Add button	Inserts a row at the bottom of the list where you can add a new field.
Delete button	Removes the selected row.
Edit button	Makes the selected cell available for editing.

Table 105. Fields Dialog Box Contents (continued)

Field	Description
	 Tip You can also double-click a cell to edit the contents.

Related Topics

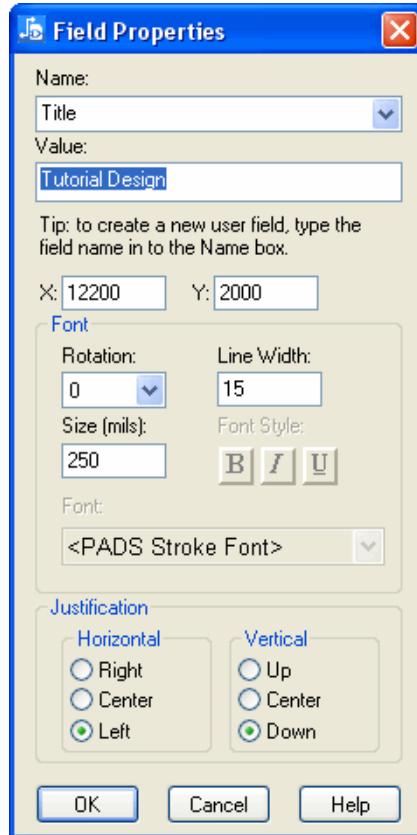
[Managing Fields](#)

Field Properties Dialog Box

To access: Select a field > right-click > **Properties** menu item

Use the Field Properties dialog box to modify a field name or change its text size, orientation, or justification.

Figure 71. Field Properties Dialog Box



Objects

Table 106. Field Properties Dialog Box Contents

Name	Description
Name	The name of the field. Type a new or select an existing field name to change it.
Value	Specifies the text you want in the schematic.
X/Y	Type coordinates to place the text in a specified location.
Rotation	Specifies the rotation of the text: 0 or 90 degrees.
Line Width	Specifies the line width for stroke fonts only.

Table 106. Field Properties Dialog Box Contents (continued)

Name	Description
	 Stroke Line Width
Size	<p>Specifies the size of the font.</p> <p>Size (pts): This is font size in points and appears for system fonts</p> <p>Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.</p>  Stroke Font - Size
Font Style	<p>Enables you to change the font style to bold, italic, and underlined.</p> <p> Restriction: System fonts only.</p>
Font list	<p>The fonts available to you. This lists either stroke fonts or system fonts. You choose which type of font to use in the Fonts Dialog Box.</p> <p> Tip</p> <ul style="list-style-type: none"> • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

[Modifying Fields](#)

[Add Field Dialog Box](#)

Font Replacement Dialog Box

To access: When you open a design created with fonts that are not installed on your system, the Font Replacement dialog box opens automatically.

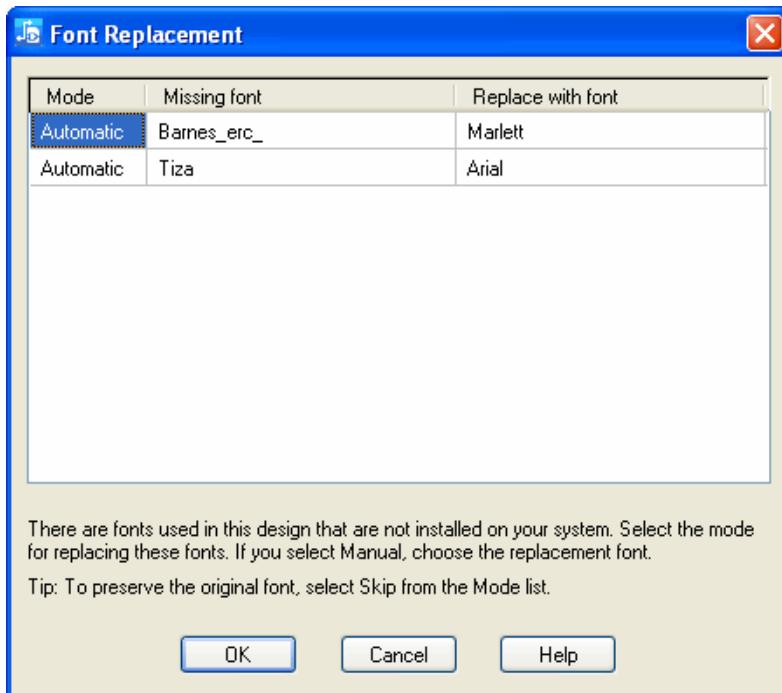
Use the Font Replacement dialog box to manage how missing fonts are replaced in your design.



Tip

If the design uses fonts or character sets that are not installed on your system, empty boxes will appear where you expect to find text or symbols. Once the font replacement process completes, the symbols display properly.

Figure 72. Font Replacement Dialog Box



Objects

Table 107. Font Replacement Dialog Box Contents

Name	Description
Mode	<p>Specifies the mode to replace this font.</p> <ul style="list-style-type: none">• Automatic — Specifies to replace the font automatically with the one selected by SailWind Layout.• Manual — Specifies to replace the font with one you select from the Replace with font list.• Skip — Specifies to preserve the original font.

Table 107. Font Replacement Dialog Box Contents (continued)

Name	Description
Missing font	The name of the font in this design that is missing from your system.
Replace with font	If you chose Manual, lists the fonts available for you to replace the missing font. If you chose Automatic, lists the font SailWind Layout chose to replace the missing font.

**Tip**

When replacing fonts, observe the following:

- You can select some fonts for automatic replacement, select others for manual replacement, and choose that other font replacements be skipped entirely.
- You can have a combination of stroke font and system fonts within the same design.
- You must set up fonts for each text string and/or label you create in your design. Once you set up fonts for a text string or label, you can then use the Properties dialog to apply a font and font characteristics to all objects that you select for modification with the Properties dialog box.

**Restriction:**

When working with fonts note the following restrictions:

- If the design uses fonts or character sets that are not installed on your system, a font substitution process begins automatically when the file is loaded. During this process, you are asked to choose fonts to substitute for those that are missing from your system.
- System font text is supported in RS274X Gerber format when Fill mode is on. System font text is output to Gerber format as a set of filled polygons.
- System fonts are not supported in the RS-274D CAM output format. If you attempt to use this format with system fonts, the program displays a warning message. If you proceed, system fonts will not be output. Instead, you should use the 274X format with system fonts.
- Type 1 fonts are not supported.

Related Topics

[Managing Font Replacement](#)

Fonts Dialog Box

To access: **Setup > Fonts** menu item

Use the Fonts dialog box to set up or change the fonts to be used in your design.



Restriction:

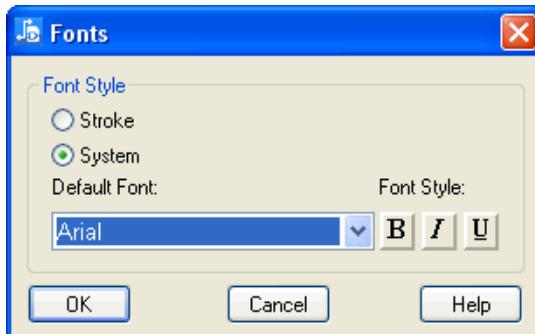
If the schematic uses fonts or character sets that are not installed on your system, a font substitution process begins automatically when the file is loaded. During this process, you are asked to select fonts to substitute for those that are missing from your system.



Tip

If you want to change font sizes, see [Modifying Part Type Labels](#).

Figure 73. Fonts Dialog Box



Objects

Table 108. Fonts Dialog Box Contents

Name	Description
Font Style	The fonts available to you: <ul style="list-style-type: none">• Stroke• System
Default Font list	Specifies the system font you want to use.  Tip You can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.

Related Topics

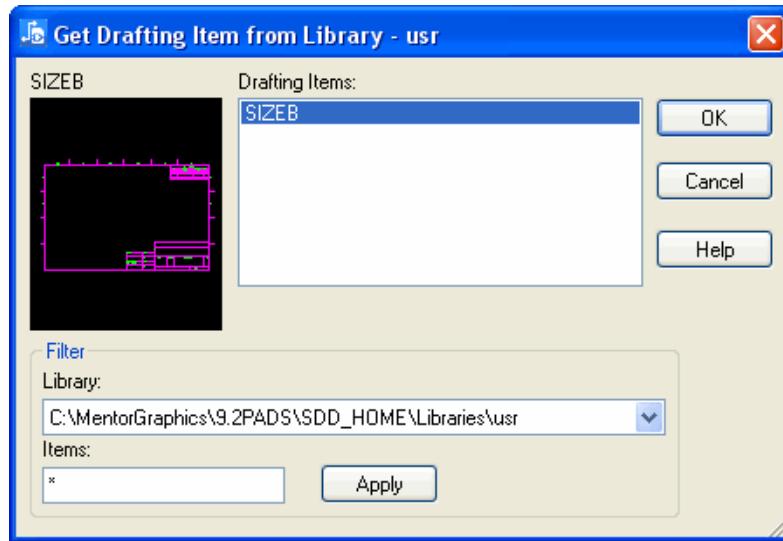
[Setting Fonts](#)

Get Drafting Items From Library Dialog Box

To access: Schematic Editing toolbar > **Add 2D Line from Library** button

Use the Get Drafting Item from Library dialog box to load a 2D line item from the available libraries to the current schematic.

Figure 74. Get Drafting Items From Library Dialog Box



Objects

Table 109. Get Drafting Items From Library Dialog Box Contents

Field	Description
Preview area	Displays the selected gate decal
Drafting Items	Lists the drafting items available based on the filter settings.
Library list	Specifies the library you want to use.
Items	Narrows the search. You can use wildcards or expressions on page 105. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.

Related Topics

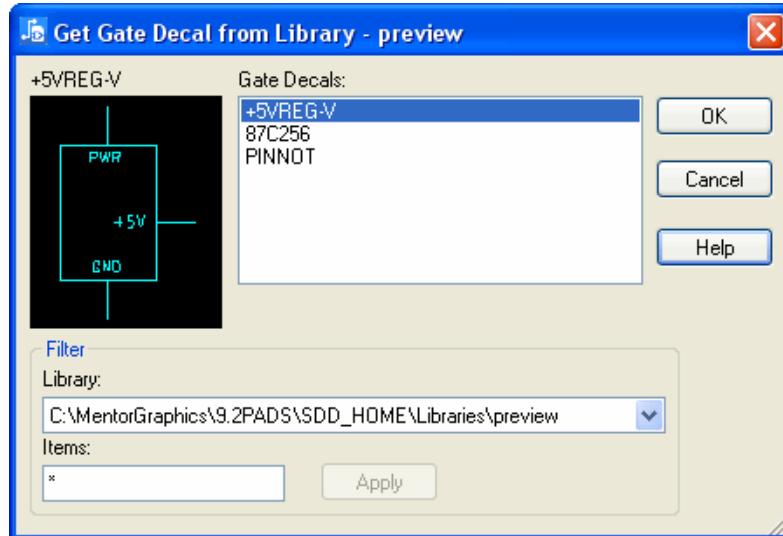
[Adding Drafting Items From a Library](#)

Get Gate Decal From Library Dialog Box

To access: Tools > Part Editor menu item > Open button > Select CAE Decal > OK button

Use the Get Gate Decal from Library dialog box to open an existing CAE Decal in the Part Editor.

Figure 75. Get Gate Decals From Library Dialog Box



Objects

Table 110. Get Gate Decals From Library Dialog Box Contents

Field	Description
Preview area	Displays the selected gate decal
Gate Decals	Lists the gate decals available based on the filter settings.
Library list	Specifies the library you want to use.
Items	Narrows the search. You can use wildcards or expressions on page 105. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.

Related Topics

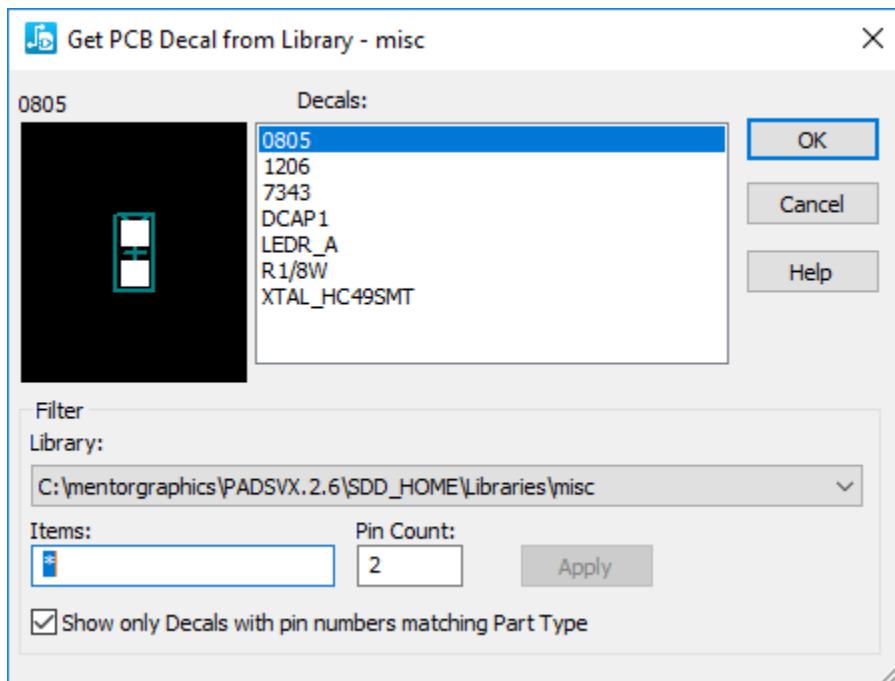
[Getting Gate Decals From the Library](#)

Get PCB Decal From Library Dialog Box

To access: Select a part > right-click > **Properties**> **PCB Decals** button > **Browse** button

Use the Get PCB Decal from Library dialog box to search a library for a PCB decal.

Figure 76. Get PCB Decal From Library Dialog Box



Objects

Table 111. Get PCB Decal From Library Dialog Box Contents

Name	Description
Picture Area	Displays the PCBdecal highlighted in the Decals area.
Decals	Lists the available PCB decals in the selected library or all libraries.
Library list	Lists the libraries available to you.
Filter	Searches the chosen library (or libraries) for a specific part or item name, or names that match a wildcard or expression on page 105. Use the Library dropdown list to select specific library directories or the All Libraries setting. Type * to view all parts or items in the chosen libraries. Click Apply to search the libraries and display the search results.

Related Topics

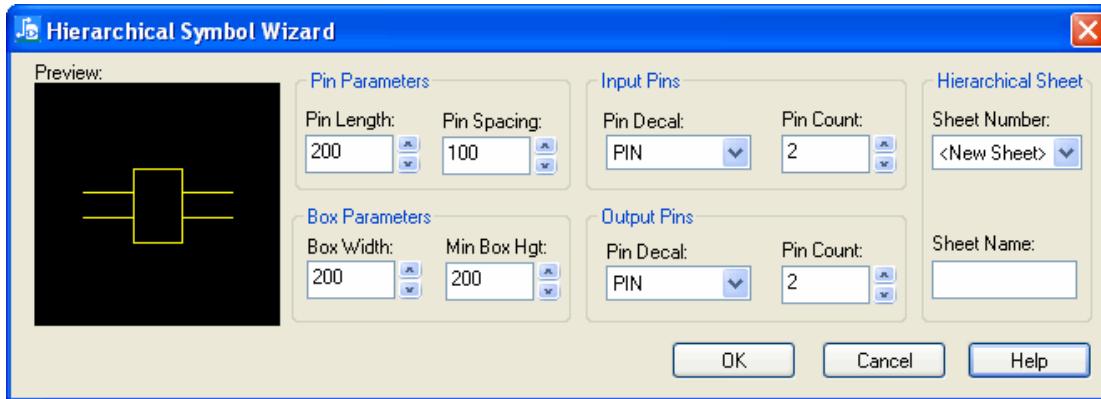
[Searching the Library for a Decal](#)

Hierarchical Symbol Wizard Dialog Box

To access: Schematic Editing toolbar > **New Hierarchical Symbol** button

Use the Hierarchical Symbol Wizard dialog box to create a new hierarchical symbol in either a top-down or bottom-up design structure. The symbol creation methodology varies, depending on the structure you use.

Figure 77. Hierarchical Symbol Wizard Dialog Box



Objects

Table 112. Hierarchical Symbol Wizard Dialog Box Contents

Name	Description
Preview	Displays the symbol outline before the symbol appears in the Part Editor. Display is based on the current settings.
Pin Parameters area	<ul style="list-style-type: none"> Pin Length — Sets the distance from the terminal connection point and the decal outline. This option does not adjust the length of the pin decal. Pin Spacing — Specifies the spacing between adjacent pins.
Box Parameters area	<ul style="list-style-type: none"> Box Width — Sets the width of the decal outline. Pin decals are moved left or right to accommodate the box width. Min Box Hgt. — Sets the minimum height of the decal outline. If you enter a value larger than needed to accommodate the number of input or output pins, space is added to the bottom of the decal. <p>Tip Type values or use the arrow buttons.</p>
Input Pins area	<ul style="list-style-type: none"> Pin Decal — Specifies the type of pin decal. Pin Count — Specifies the number of input pins for a hierarchical symbol of a new sheet during top-down design. <p>Tip Type values or use the arrow buttons.</p>

Table 112. Hierarchical Symbol Wizard Dialog Box Contents (continued)

Name	Description
Output Pins area	<ul style="list-style-type: none"> • Pin Decal — Specifies the type of pin decal. • Pin Count — Specifies the number of output pins for a hierarchical symbol of a new sheet during top-down design. <p> Tip Type values or use the arrow buttons.</p>
Hierarchical Sheet area	<ul style="list-style-type: none"> • Sheet Number — Lists the sheets in the set or lists only the <New Sheet> option for top-down design. <p> Restriction: For bottom-up design, this list never displays the current sheet since you can place the symbol that represents the current sheet on the current sheet.</p> <ul style="list-style-type: none"> • Sheet-name — Specifies the name of the hierarchical symbol. <p> Restriction: <ul style="list-style-type: none"> ◦ Sheet Name has a maximum of 37 characters. ◦ You can only use the first 32 characters for the textual portion of the name, the remaining five are reserved for the numeric portion. </p>

Related Topics

[Creating a Top-Down Hierarchy](#)

[Creating a Bottom-Up Hierarchy](#)

HiSpeed Rules Dialog Box

To access: **Setup > Design Rules** menu item > a rule hierarchy button > **HiSpeed** button

Use the HiSpeed Rules dialog box to define rules for Parallelism, Tandem, Shielding, Routed Length, Stub Length, Delay, Capacitance, Impedance, and Matched Length.

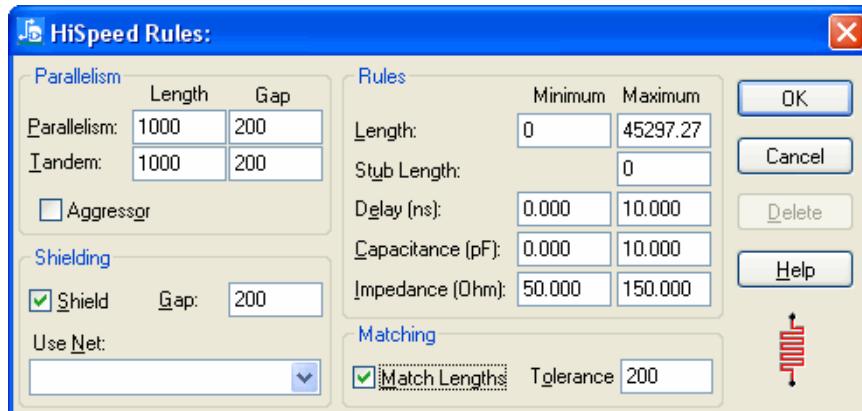


Tip

When working with HiSpeed Rules, observe the following:

- When imported into SailWind, the EDC (Electrodynamic Checking) routine checks to see if rules are met correctly after routing (except shielding and matched length).
- You can use the [Conditional Rule Setup Dialog Box](#) to save a high speed configuration as a set, or to apply for a selected item only when it comes in contact with different level of the [hierarchical order](#) on page 305.

Figure 78. HiSpeed Rules Dialog Box



Objects

Table 113. HiSpeed Rules Dialog Box contents

Name	Description
Parallelism	Restricts the distance that traces in different nets on the same layer can run together. Tip <ul style="list-style-type: none">• Length defines the maximum allowable parallel/tandem distance.• Gap defines the minimum gap between traces below which the parallel/tandem rules apply.
Tandem	Restricts the distance that traces in different nets on different layers can run together.

Table 113. HiSpeed Rules Dialog Box contents (continued)

Name	Description
	 Tip <ul style="list-style-type: none"> Length defines the maximum allowable parallel/tandem distance. Gap defines the minimum gap between traces below which the parallel/tandem rules apply.
Aggressor	Specifies if a net is a source of interference, during parallel/tandem checks.
Rules area	<p>Specifies the minimum and maximum values for:</p> <ul style="list-style-type: none"> Length — Defines a minimum and maximum length. Stub Length — Specifies a maximum stub length. The stub length is the distance from a T-point to the end of the route. Delay — Defines a minimum and maximum delay time in nanoseconds. Capacitance — Defines a minimum and maximum capacitance in picofarads. Impedance — Defines a minimum and maximum impedance in ohms.  Tip <p>These text boxes restrict the trace width to a range of values. Recommended is the width you want to assign to the trace when routing begins. The Minimum and Maximum values are respected by routing routines which must use trace width to achieve some high-speed routing functions, such as impedance matching.</p>
Shield	<p>Specifies to arrange certain nets as shielding others; the Net in the Use Net list box is routed up and down both sides of a selected net to provide protection from interference.</p>  Tip <p>You can only assign nets associated with plane layers in the Layer Definition dialog box to shield other nets. If there are no plane layers, the Shield area is grayed out.</p>
Gap	Specifies the value of the shield gap.
Use Net	Specifies the net to use as the shield.
Match Lengths	Specifies that you want to use Length Matching.
Tolerance	Specifies the maximum difference allowed between the shortest length and longest length in the matched length group.
Delete button	<p>Removes this set of High Speed rules from your rules hierarchy.</p>  Tip <p>You cannot delete the Default High Speed rules.</p>

Related Topics

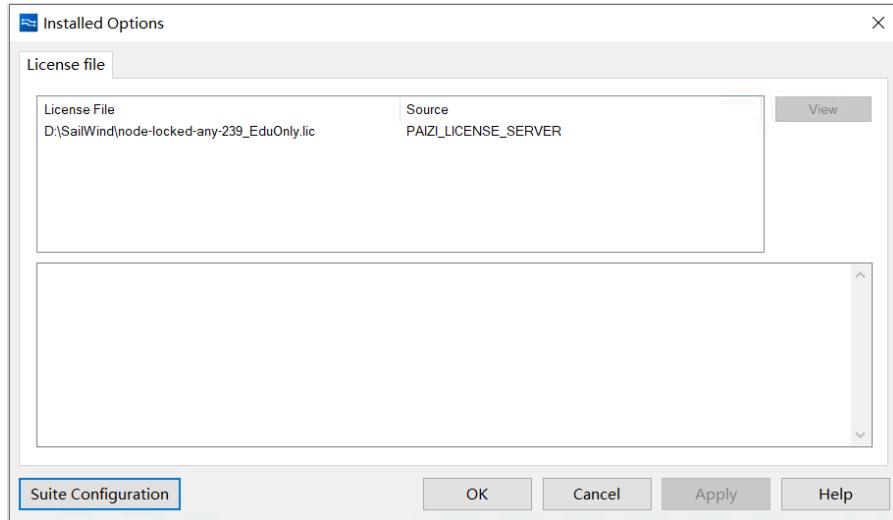
[Setting Up High-Speed Rules](#)

Installed Options Dialog Box

To access: **Help > Installed Options** menu item > **License File** tab

Use the **License File** tab to select and then view license file information, either the actual license file (for node-locked licensing) or the feature usage status associated with a server license (for floating licensing).

Figure 79. Installed Options Dialog Box, License File Tab



Objects

Table 114. Installed Options Dialog Box, License File Tab contents

Name	Description
License File	Displays the location of the license file.
Source	Displays the source of the license.
View button	Specifies to display the selected license file in the view area. Exception: For Node-locked licenses only.
Status button	Specifies to display the selected feature usage information. Exception: For Floating licenses only.
View area	Displays the selected license file information after you click View (Node-locked license) or Status (Floating license).
Suite Configuration button	Opens the SailWind Suite Configuration Dialog Box . ■ Restriction: Available only with floating/server-based licenses, a mix of different SailWind Suites, or a mix of unbundled licenses and suites.

Related Topics

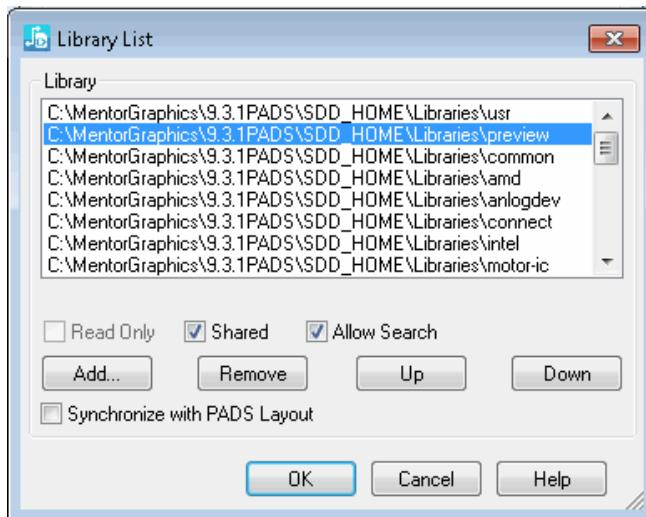
[Viewing a License File or License Status](#)

Library List Dialog Box

To access: **File > Library** menu item > **Manage Lib. List** button

When you add a part to the design, you retrieve information about that part from a library. The Library List dialog box contains a list of libraries to search. The libraries are searched in the order in which they are listed, beginning with the first library in the list.

Figure 80. Library List Dialog Box



Objects

Table 115. Library List Dialog Box Contents

Name	Description
Library list	The libraries currently listed in the Library Manager Library list.
Read Only	A status indicator only; this box is always unavailable.
Shared	Shares the library over the network. This enables more than one user to access the library file at the same time.
Allow Search	Includes the library when performing operations that involves libraries, such as adding parts.
Add button	Adds a library to the Library list.
Remove button	Removes a library from the Library list.
Up/Down buttons	Moves the order of the libraries in the Library list.
Synchronize with SailWind Layout	Specifies to push the library settings to SailWind Layout from SailWind Logic.

Related Topics

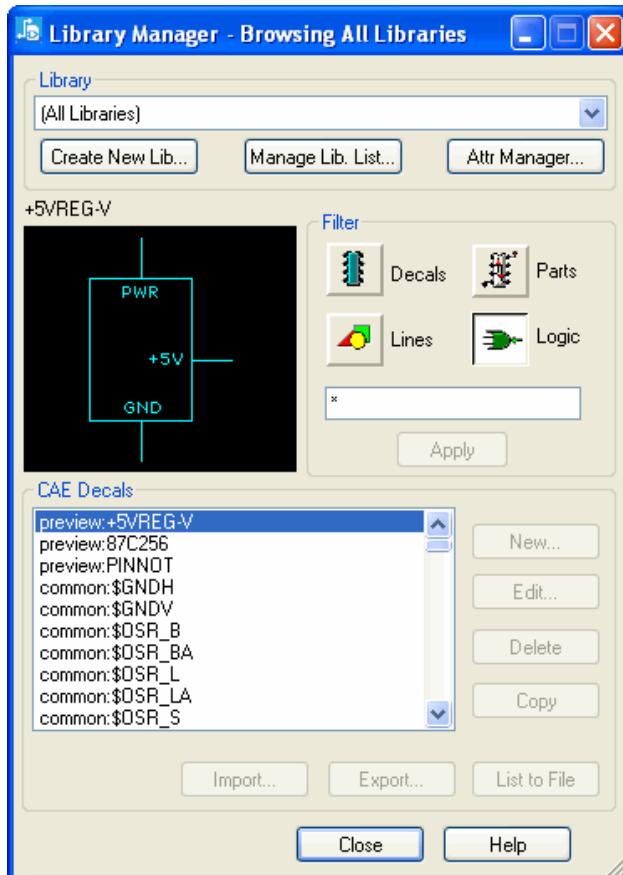
[Setting the Library List Order](#)

Library Manager Dialog Box

To access: **File > Library** menu item

Use the Library Manager to perform operations associated with editing and copying the contents of libraries.

Figure 81. Library Manager Dialog Box



Objects

Table 116. Library Manager Dialog Box Contents

Name	Description
Library list	The list of libraries available to you.
Create New Lib button	Opens the New Library window where you can specify a new library name and location.
Manage Lib. List button	Opens the Library List Dialog Box .
Attr Manager button	Opens the Manage Library Attributes Dialog Box .

Table 116. Library Manager Dialog Box Contents (continued)

Name	Description
Preview area	Shows the item selected in the Filter list.
Filter area	Narrows down the Filter list by Decals, Parts, Lines, or Logic. You can further narrow the list using wildcards on page 105 in the Filter box.  Tip Add an asterisk "*" to the box to display all items.
Filter list	The results from your filter area selections.
New button	The action taken is dependent on the filter. <ul style="list-style-type: none"> • Decals — Opens the PCB Decal Editor on a new decal. • Parts — Opens the Part Information Dialog Box, Gates Tab on an unnamed part. • Lines — Unavailable. There is no special library lines editor. Use drafting tools to create or edit lines and save them to the library. • Logic — Unavailable. Use SailWind Logic to create or edit CAE decals.  Restriction: This button is unavailable when the Library is set to (All Libraries). See also Adding Items to a Library .
Edit button	The action taken is dependent on the filter. <ul style="list-style-type: none"> • Decals — Opens the PCB Decal Editor on the selected decal. • Parts — Opens the Part Information Dialog Box, Gates Tab on the selected part. • Lines — Unavailable. There is no special library lines editor. Use drafting tools to create or edit lines and save them to the library. • Logic — Unavailable. Use SailWind Logic to create or edit CAE decals.  Restriction: This button is unavailable when the Library is set to (All Libraries). See also Editing Items in a Library .
Delete button	Removes the selected item from the library.  Restriction: This button is unavailable when the Library is set to (All Libraries). See also Deleting Items From a Library .
Copy button	Copies the selected item to another name or another library.

Table 116. Library Manager Dialog Box Contents (continued)

Name	Description
	 Restriction: This button is unavailable when the Library is set to (All Libraries). See also Copying a Library Item .
Import button	Import library data from an ASCII file. The file type is dependent on the filter.  Restriction: This button is unavailable when the Library is set to (All Libraries). See also Importing Library Data .
Export button	Export library data to an ASCII file. The file type is dependent on the filter.  Restriction: This button is unavailable when the Library is set to (All Libraries). See also Exporting Library Data .
List to File button	The action taken is dependent on the filter. <ul style="list-style-type: none"> • Decals — Generates a list of PCB Decals in a single library. • Parts — Generates a list of Parts in a single library or all libraries along with chosen attributes. • Lines — Generates a list of line items in a single library. • Logic — Generates a list of CAE decals or Logic symbols in a single library.  Restriction: When the Library is set to (All Libraries), this button is unavailable for all but Parts.

Log Test Dialog Box

To access: type BLT > press Enter. If nothing happens, close SailWind Logic and restart it.

Use BLT to replay session playback media created by BMW.

Figure 82. Log Test Dialog Box



Objects

Table 117. Log Test Dialog Box contents

Name	Description
Media Directories	Lists the session playback media files.
New name	Specifies to rename the selected media directory.
Rename button	Renames the selected media directory to the name in the New name box.

Related Topics

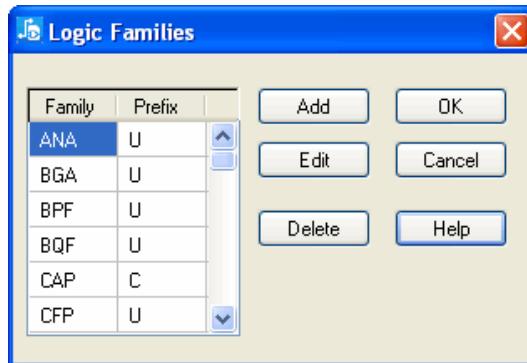
[Replaying Session Playback Media With BLT](#)

Logic Families Dialog Box

To access: Tools > Part Editor menu item > Edit Electrical button > General tab > Families button

Use the Logic Families dialog box to add, delete, or modify logic family names and default reference designator prefixes.

Figure 83. Logic Families Dialog Box



Objects

Table 118. Logic Families Dialog Box Contents

Name	Description
Family column	Lists the logic family.
Prefix column	Lists the prefix for the logic family
Add button	Adds a row to the table.
Edit button	Makes the selected field editable.
Delete button	removes the selected row.

Related Topics

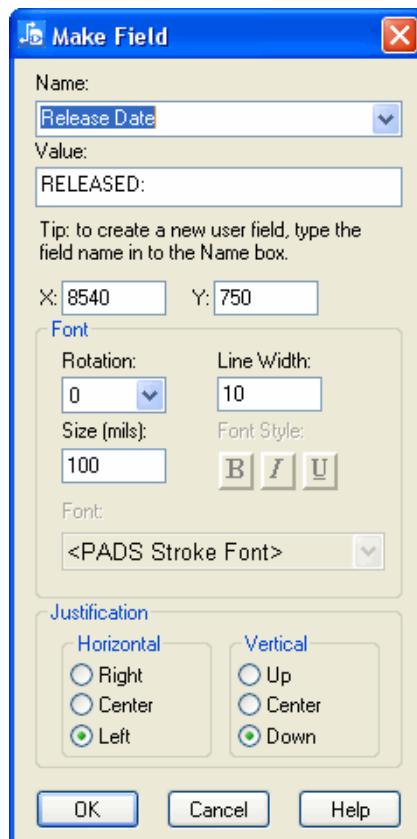
[Editing Logic Families](#)

Make Field Dialog Box

To access: Select a text string > right-click > **Make Field** menu item

Use the Make Field dialog box to change an existing text string into a field.

Figure 84. Make Field Dialog Box



Objects

Table 119. Make Field Dialog Box Contents

Name	Description
Name list	Lists all of the fields available to you.
Value	The value of the field.  Restriction: The Value box is unavailable for system fields since the value is derived from your system.
X/Y	Type coordinates to place the field in a specified location.

Table 119. Make Field Dialog Box Contents (continued)

Name	Description
	 Tip Leave these blank to attach the field to your pointer and click to indicate the location.
Rotation	Specifies the rotation of the text: 0 or 90 degrees.
Line Width	Specifies the line width for stroke fonts only.  Stroke Line Width
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.  Stroke Font - Size
Font Style	Enables you to change the font style to bold, italic, and underlined.  Restriction: System fonts only.
Font list	The fonts available to you. This lists either stroke fonts or system fonts. You choose which type of font to use in the Fonts Dialog Box .  Tip <ul style="list-style-type: none"> Select stroke font or a system font. For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

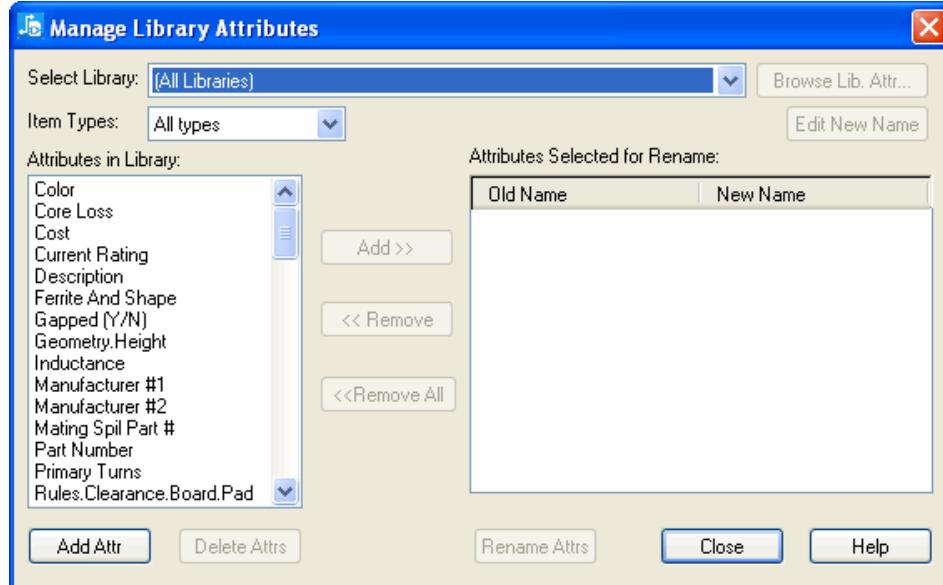
[Changing a Text String Into a Field](#)

Manage Library Attributes Dialog Box

To access: **File > Library** menu item > **Attr Manager** button

Use the Manage Library Attributes dialog box to list the attribute names in your libraries and to add an attribute to, rename an attribute in, or delete an attribute from, the library.

Figure 85. Manage Library Attributes Dialog Box



Objects

Table 120. Manage Library Attributes Dialog Box Contents

Name	Description
Select Library list	The list of libraries available to you.
Item Types list	Filters the type of items in the Attributes in Library list.
Browse Lib. Attr button	Opens the Browse Library Attributes Dialog Box . Tip This is available only when an attribute in the New Name column in the Attributes Selected for Rename list is selected.
Edit New Name button	Makes the selected attribute Name editable. Tip This is available only when an attribute in the New Name column in the Attributes Selected for Rename list is selected.
Attributes in Library list	The list of attributes in the selected library.
Add >> button	Adds the selected attribute to the Rename list.

Table 120. Manage Library Attributes Dialog Box Contents (continued)

Name	Description
<< Remove button	Removes the selected attribute from the Rename list.
<< Remove All button	Removes all of the attributes from the Rename list.
Attributes Selected for Rename list	The list of attributes you've selected to rename.
Add Attr button	Opens the Add New Attribute to Library Dialog Box .
Delete Attrs button	Deletes the selected attribute from the selected library.
Rename Attrs button	Renames all of the attributes you gave a new name to in the selected library.

Related Topics

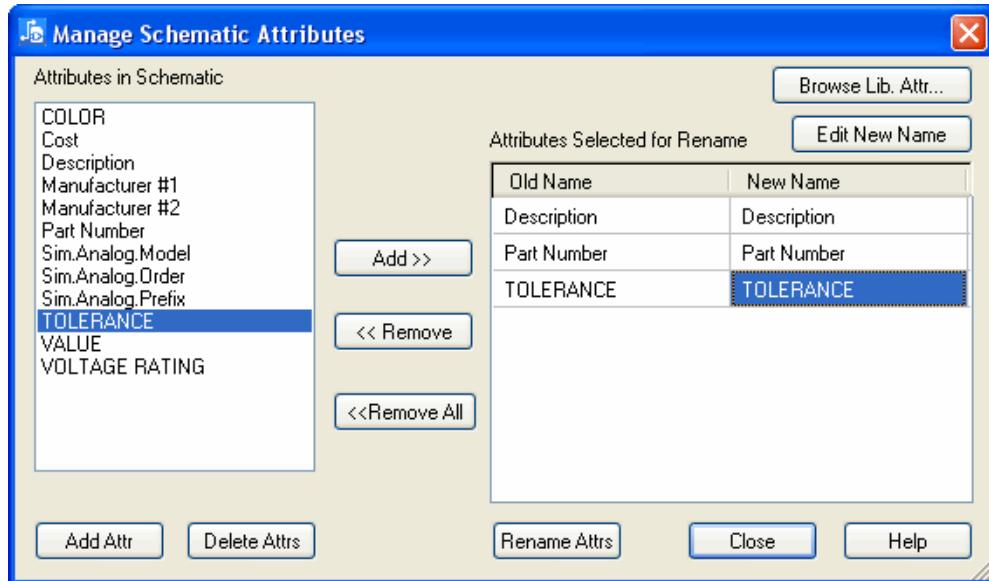
[Managing Library Attributes](#)

Manage Schematic Attributes Dialog Box

To access: **Edit > Attribute Manager** menu item

Use the Manage Schematic Attributes dialog box to manage attributes at the schematic level. You can create a new attribute and automatically assign it to every part in your design. You can also rename an attribute in, or delete an attribute from, the schematic. All parts are automatically updated.

Figure 86. Manage Schematic Attributes Dialog Box



Objects

Table 121. Manage Schematic Attributes Dialog Box Contents

Name	Description
Attributes in Schematic list	The list of attributes in the schematic.
Add >> button	Adds the selected attribute to the Rename list.
<< Remove button	Removes the selected attribute from the Rename list.
<< Remove All button	Removes all of the attributes from the Rename list.
Attributes Selected for Rename list	The list of attributes you've selected to rename.
Browse Lib. Attr button	Opens the Browse Library Attributes Dialog Box .
Edit New Name button	Makes the selected attribute Name editable. Tip This is available only when an attribute in the New Name column in the Attributes Selected for Rename list is selected.

Table 121. Manage Schematic Attributes Dialog Box Contents (continued)

Name	Description
Add Attr button	Opens the Add New Attribute to Library Dialog Box .
Delete Attrs button	Deletes the selected attribute from the selected library.
Rename Attrs button	Renames all of the attributes you gave a new name to in the selected library.

Related Topics

[Manage Attributes in a Schematic](#)

Media Wizard Dialog Box

To access: type BMW > press Enter

BMW (Basic Media Wizard) is a tool that you can use to record and play back SailWind Logic, SailWind Layout and SailWind Router sessions. It is particularly useful as a means of supplying information to SailWind Technical Support engineers trying to identify and resolve any problematical behavior you may encounter.

Figure 87. Media Wizard Dialog Box



Objects

Table 122. Media Wizard Dialog Box contents

Name	Description
Media Wizard area	Specifies what you want the Media Wizard to do: <ul style="list-style-type: none">• Create Media from Current Session — Use this procedure when the session you are recreating did not cause a SailWind tool crash.• Create Media from Previous Session — Use this procedure when the session you are recreating caused the SailWind tool to crash, and the automatic procedure described in Automatically Creating Session Playback Media for a Crashed Session cannot be used due to one of the restrictions listed in that section.• Stop Logging — Specifies to stop the Media Wizard from logging any further actions.
User Initials	Specifies your initials. They are included in the playback media filenames to identify the files as yours.
Delete Actions Before Last Save	Specifies to delete all entries in the session log file between the first Open and the last Save command. You can do this to eliminate any actions you may have performed before beginning the series of actions that produced the problematical behavior. This makes it easier for the Tech Support engineer to identify the problem.

Related Topics

[BMW and BLT](#)

Modeless Commands and Keyboard Shortcuts

You can set or change some settings and functions at any time using a code letter for the command, entering the new value, and clicking Enter. This is called a Modeless Command.

Modeless Commands usually apply to values that you change frequently during design. Use the Modeless Command G, for example, to change the grid setting. Type G, the new setting, and click Enter. Below is a summary of the Modeless Commands in SailWind Logic:

To show this help topic at any time while SailWind Logic is running, type ? and click Enter.



Tip

(X,Y) = coordinates; (s) = text; (n) = number.

Table 123. Modeless Command and Shortcut Key Table Conventions

Convention	Description
< >	A variable, or something you can type.
{ }	An optional command argument.
click	Click the left mouse button.
middle-click	Click the middle mouse button or wheel.
right-click	Click the right mouse button.
wheel forward	Rotate the wheel forward, where the top of the wheel rotates away from your palm.
wheel back	Rotate the wheel backward, where the top of the wheel rotates toward your palm.



Tip

Spaces have significance in modeless commands and shortcut keys. For example, SS W1 and S SW1 have different meanings. SS W1 means to search for and select W1, while S SW1 means to search for SW1.

Modeless Commands

The following is a complete list of all of the modeless commands:

Table 124. Modeless Commands for Grid Settings

Name	Command	Description
Global Grid Setting	G<n>	Sets the Design grid, for example G50.

Table 124. Modeless Commands for Grid Settings (continued)

Name	Command	Description
Dot Grid Setting	GD<n>	Sets the Displayed (Dot) grid, for example, GD100.

Table 125. Modeless Commands for 2D Line Angles

Name	Command	Description
Any Angle	AA	Any angle mode. Sets the line angle for drafting objects to Any angle (no angle restrictions).
Diagonal Angle	AD	Diagonal angle mode. Sets the line angle for drafting objects to Diagonal (45 degree angles only).
Orthogonal Angle	AO	Orthogonal angle mode. Sets the line angle for drafting objects to Orthogonal (90 degree angles only).

Table 126. Modeless Commands for Line Width Settings

Name	Command	Description
Set Minimum (Real) Display Width for Paths	R<width>	Sets the minimum (Real) display width for paths.
Change Current Line Width	W<width>	Changes the current line width to the number <n> you enter, for example W 5.

Table 127. Modeless Commands for Searching

Name	Command	Description
Search Absolute	S<x> <y>	Search absolute. Moves the pointer to the specified X and Y coordinates, for example S 1000 1000.
Search for Named Item (Pin/Part/Net)	S<string>	Search for named item (pin, part or net), for example SU1.
Search Relative	SR<x><y>	Search relative. Moves the pointer by the specified X and Y offset, for example SR -100 -50.
Search Relative X	SRX<x>	Search relative X at current Y. Moves the pointer by the specified X offset, for example SRX 300.
Search Relative Y	SRY<y>	Search relative Y at current X. Moves the pointer by the specified Y offset, for example SRY 400.

Table 127. Modeless Commands for Searching (continued)

Name	Command	Description
Absolute Move to <n>, Current Y	SX<x>	Search absolute X at current Y. Moves the pointer to the specified X coordinate and the current Y coordinate, for example SX 300.
Absolute Move to <n>, Current X	SY<y>	Search absolute Y at current X. Moves the pointer to the specified Y coordinate and the current X coordinate, for example SY 400

Table 128. Modeless Commands for Drafting Shape Control

Name	Command	Description
Circle shape draw mode.	HC	Sets drawing mode to circle shape.
Path shape draw mode.	HH	Sets drawing mode to path shape.
Polygon shape draw mode.	HP	Sets drawing mode to polygon shape.
Rectangle shape draw mode.	HR	Sets drawing mode to rectangle shape.

Table 129. Modeless Commands for Measurement

Name	Command	Description
Quick Measure	Q	Quick Measure with a dynamic ruler. Attaches a measurement line to the pointer and displays dx, dy, and hypotenuse information, depending on pointer movement. Place the pointer at the starting point, then type the "q" modeless command. Drag the pointer to create a line between the start and end point of your measurement. Snaps to the design grid when the Snap to grid is on. Measurements are gridless when Grid Snap is off. Dynamically reports delta x, delta y and delta x,y in current design units.

Table 130. Modeless Commands for Hierarchical Design

Name	Command	Description
Hierarchical Push	HI	Invokes Hierarchical Push.
Hierarchical Pop	HO	Invokes Hierarchical Pop.

Table 131. Modeless Commands for Undo/Redo

Name	Command	Description
Undo	UN	Undo
Redo	RE	Redo

Table 132. Miscellaneous Modeless Commands

Name	Command	Description
Open File <name>	F<name>	Open file <name>, where <name> is the path and name of the file to open (for example, F demo.eco).
Select Sheet Name or Number	SH<sheet>	Selects the sheet name or number you type, for example SH3.
Database Integrity Test	I	Runs the Database Integrity Test.
Help	?	Show Help topic.

Table 133. Modeless Commands for Basic Media Wizard/Log Test

	Command	Description
Basic Log Test	BLT	Basic Log Test. Opens the Log Test Dialog Box. BLT finds and runs BMW session playback media. See also Crash Detection, BMW and BLT .
Open Basic Media Wizard	BMW	Opens the Basic Media Wizard dialog box. <ul style="list-style-type: none"> • BMW records session playback media for a problematic SailWind Logic session. It can create playback media based on your last SailWind Logic session or your current session. This playback media can be replayed using the BLT modeless command. • BMW is also a command line option. See also Crash Detection, BMW and BLT .
Start BMW Session Logging	BMW ON	Starts BMW session logging.
Stop BMW Session Logging	BMW OFF	Stops BMW session logging.

Function Keys

The following is a complete list of all of the function key command assignments:

Table 134. Function Key Command Assignments

Function Key	Description
F1	Open Help (context sensitive)
F2	Add Connection
F3	Unassigned
F4	Unassigned
F5	Unassigned
F6	Unassigned
F7	Unassigned
F8	Unassigned
F9	Unassigned
F10	Unassigned
F11	Unassigned
F12	Unassigned

Keypad Keys

The following is a complete list of all of the keypad key command assignments:

Table 135. Keypad Key Command Assignments

Keypad Keys	Description
(Number Keys) with NumLock On	
Keypad (0)	Center the view using the pointer location
Keypad (1)	Redraw
Keypad (2)	Pans the workspace down one increment
Keypad (3)	Zooms out at the pointer
Keypad (4)	Pans the workspace left one increment
Keypad (5)	Starts Zoom from center
Keypad (6)	Pans the workspace right one increment
Keypad (7)	Zoom to the Sheet
Keypad (8)	Pans the workspace up one increment

Table 135. Keypad Key Command Assignments (continued)

Keypad Keys	Description
Keypad (9)	Zoom in at the pointer location
Keypad (.)	Starts Zoom from corner (Zoom Mode only)
(Command Keys) with NumLock Off	
Insert	Centers the view using the pointer location
End	Redraw
Down Arrow	Moves the pointer down one design grid
Page Down	Zooms out at the pointer
Left Arrow	Moves the pointer left one design grid
Right Arrow	Moves the pointer right one design grid
Home	Zooms to the Sheet
Up Arrow	Moves the pointer up one design grid
Page Up	Zooms in at the pointer
Delete	Delete the selected object

Shortcut Keys

For mouseless operation, use keyboard shortcuts to start commands for selected items and change some system settings. Following are the shortcut assignments for SailWind Logic:

Table 136. Keyboard Shortcuts for Panning, Zooming and Navigation

Name	Shortcut Keys	Description
Zoom to Sheet	<home>	Zooms to sheet. Fits the sheet border into the workspace.
Zoom to Sheet	Ctrl + B	Zooms to sheet. Fits the sheet border into the workspace.
Zoom Extents	Ctrl+Alt + E	Zooms to extents. Fits all objects in the design into the workspace.
Zoom Area In/Out	MMB (drag)	Zooms area in or out. Drag pointer up to zoom in. Drag pointer down to zoom out.
Start Zoom from Corner	Shift + MMB (Drag)	Starts Zoom from Corner.

Table 136. Keyboard Shortcuts for Panning, Zooming and Navigation (continued)

Name	Shortcut Keys	Description
Zoom to Selection	Alt + Z	Zooms to selection. Fits the selected objects into the workspace.
Zoom Mode On/Off	Ctrl + W	Toggles Zoom Mode On/Off.
Center View (Using Pointer Location)	MMB	Centers the view at the pointer.
Center View (Using Pointer Location)	<Insert>	Centers the view at the pointer.
Zoom In at Pointer (Zoom Mode)	LMB Click	Zooms in at the pointer (zoom mode).
Zoom Out at Pointer (Zoom Mode)	RMB Click	Zooms out at the pointer (zoom mode).
Zoom In at Pointer (Zoom Mode)	<spacebar>	Zooms in at the pointer (zoom mode).
Zoom In at Pointer	<PgUp>	Zooms in at the pointer.
Zoom Out at Pointer	<PgDn>	Zooms out at the pointer.
Zoom In at Pointer	Ctrl + Wheel Fwd	Zooms in at the pointer.
Zoom Out at Pointer	Ctrl + Wheel Back	Zooms out at the pointer.
Move Pointer Down (One Design Grid)	<Down Arrow>	Pointer moves down one design grid.
Move Pointer Up (One Design Grid)	<Up Arrow>	Pointer moves up one design grid.
Move Pointer Left (One Design Grid)	<Left Arrow>	Pointer moves left one design grid.
Move Pointer Right (One Design Grid)	<Right Arrow>	Pointer moves right one design grid.
Dynamic Panning	Alt + MMB (Drag)	Pans the sheet area below the pointer to the center of the workspace.
Pan Workspace Down (One Line)	Wheel Back	Pans workspace down one line.
Pan Workspace Up (One Line)	Wheel Fwd	Pans workspace up one line.
Pan Workspace Right (One Line)	Shift + Wheel Back	Pans workspace right one line.
Pan Workspace Left (One Line)	Shift + Wheel Fwd	Pans workspace left one line.

Table 136. Keyboard Shortcuts for Panning, Zooming and Navigation (continued)

Name	Shortcut Keys	Description
Pan Workspace Down (One Pixel)	Ctrl-Alt + Wheel Back	Pans workspace down one pixel.
Pan Workspace Up (One Pixel)	Ctrl-Alt + Wheel Fwd	Pans workspace up one pixel.
Pan Workspace Right (One Pixel)	Alt-Shift + Wheel Back	Pans workspace right one pixel.
Pan Workspace Left (One Pixel)	Alt-Shift + Wheel Fwd	Pans workspace left one pixel.

Table 137. Keyboard Shortcuts for Selection

Name	Shortcut Keys	Description
Filter	Ctrl-Alt + F	Opens the Selection Filter.
Select	LMB Click	Selects an object.
Select	<Spacebar>	Selects an object.
Select All	Ctrl + A	Selects all objects on the current sheet based upon the selection filter choices.
Select All on Schematic	Ctrl-Shift + A	Selects all objects on the schematic based upon the selection filter choices.
Area Select	LMB (Start Drag)	Starts an area selection.
Area Complete	LMB (End Drag)	Completes an area selection.
Cancel Area Selection	LMB (Cancel Drag)	Cancels area selection.
Toggles/Multiple Selection	Ctrl + LMB Click	Toggles object selection or selects multiple objects.
Duplicate	Ctrl + LMB (Drag)	Duplicates selected object and attaches to pointer.

Table 138. Keyboard Shortcuts for File Operations

Name	Shortcut Keys	Description
Create New Design File (Blank)	Ctrl + N	Creates a new blank design file.
Open File (Design)	Ctrl + O	Opens a design file.
Print/plot	Ctrl + P	Print/plot

Table 138. Keyboard Shortcuts for File Operations (continued)

Name	Shortcut Keys	Description
Save (Quick Save)	Ctrl + S	Saves the design file

Table 139. Keyboard Shortcuts for Opening Menus and Dialog Boxes

Name	Shortcut Keys	Description
Open File Menu	Alt + F	Opens the File menu.
Open Edit Menu	Alt + E	Opens the Edit menu.
Open View Menu	Alt + V	Opens the View menu.
Open Setup Menu	Alt + S	Opens the Setup menu.
Open Tools Menu	Alt + T	Opens the Tools menu.
Open Help Menu	Alt + H	Opens the Help menu.
Open Shortcut Menu	RMB	Opens the Shortcut menu.
Toggle Menu Bar	Ctrl-Alt + M	Toggles the visibility of the Menu Bar.
Properties (Current Object)	LMB DblClick	Displays Properties for the currently selected object.
Properties (for Selected)	Alt+<Enter>	Displays Properties for the currently selected object.
Properties (for Selected)	Ctrl + Q	Displays Properties for the currently selected object.
Options	Ctrl + <Enter>	Opens the Options dialog box.
Options	Ctrl-Alt + G	Opens the Options dialog box.
Display Colors	Ctrl-Alt + C	Opens the Display Colors dialog box.
Status Window	Ctrl-Alt + S	Opens the Status Window.

Table 140. Keyboard Shortcuts for Placement Operations

Name	Shortcut Keys	Description
Move Selected Object(s)	Ctrl + E	Moves the selected object(s).
Rotate Selected (90)	Ctrl + R	Rotates the selected object (90 degrees).
Flip Selected on X Axis	Ctrl + F	X Mirror (flips selected object on X axis).

Table 140. Keyboard Shortcuts for Placement Operations (continued)

Name	Shortcut Keys	Description
Flip Selected on Y Axis	Ctrl-Shift + F	Y Mirror (flips selected object on Y axis)
Draw Group While in Move Group Mode	Ctrl-Shift + D	Draw Group while in Move Group Mode.
Vertically Justify Text During Move	Ctrl-Shift + J	Vertically justify text during move.
Horizontally Justify Text During Move	Ctrl + J	Horizontally justify text during move.
Connect to Layout for Cross-Probing	Ctrl-Shift + O	Connect to Layout for cross-probing.

Table 141. Keyboard Shortcuts for Connection Operations

Name	Shortcut Keys	Description
Add Corner	LMB Click	Adds a new connection corner.
Add Corner	<Spacebar>	Adds a new connection corner.
Backup	<Backspace>	Removes the last connection corner on a connection line or the last corner on a 2D line (in polygon or path drawing mode).
Complete Bus	<Enter>	Completes the bus.
Complete Bus	LMB DblClick	Completes the bus.
Rename	Ctrl + L	Rename.
Reset DxDy	Ctrl + <PgDn>	Reset delta coordinates to measure from current position.
Add Ground Symbol	Ctrl + <spacebar>	Adds a Ground Symbol while in Add Connection mode.
Add Power Symbol	Shift + <spacebar>	Adds a Power Symbol while in Add Connection mode.
Add Off-page Symbol	Alt + <spacebar>	Add an Off-page Symbol while in Add Connection mode.
Cycle Alternate Gate Decals	Ctrl + <Tab>	Cycles through alternate gate decals.

Table 142. Keyboard Shortcuts for Editing

Name	Shortcut Keys	Description
Cut	Ctrl + X	Cut
Copy	Ctrl + C	Copy
Paste	Ctrl + V	Paste
Redraw	<End>	Redraw
Redraw	Ctrl + D	Redraw
Delete	<Delete>	Delete
Cancel	<Escape>	Cancel
Redo	Ctrl + Y	Redo
Undo	Ctrl + Z	Undo

Table 143. Keyboard Shortcuts for Viewing

Name	Shortcut Keys	Description
Next View	Alt + N	Displays the next view.
Previous View	Alt + P	Displays the previous view.

Table 144. Mouse Button Substitutions

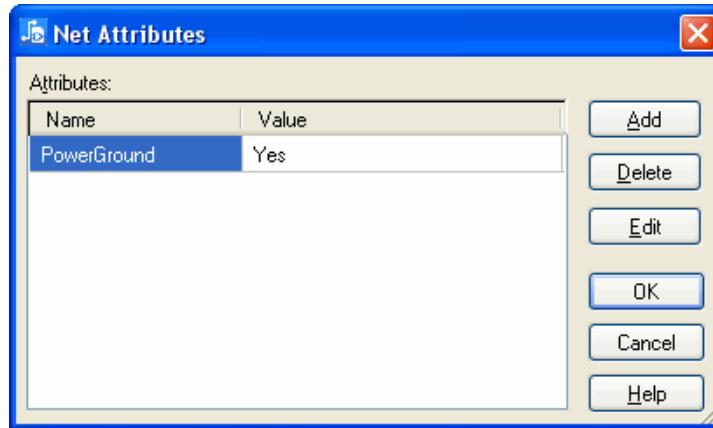
Name	Shortcut Keys	Description
Activate Right Click Popup Menu (Right Mouse Button)	M	Activates the shortcut menu for the current mode. Same as right-click.
Left Mouse Click	<Spacebar>	Activates a left mouse button click (to add corners, select items, complete, etc.) at the current pointer location.

Net Attributes Dialog Box

To access: Select a net > right-click > **Attributes** menu item

Use the Net Attributes dialog box to associate information with a single net or set of nets on the schematic. Attributes are made of two parts, an attribute name and its corresponding value. If connecting to SailWind Layout, these attributes are passed along with the rest of the schematic.

Figure 88. Net Attributes Dialog Box



Objects

Table 145. Net Attributes Dialog Box

Name	Description
Attributes table	Lists the name and value of the net selected in the schematic.
Add button	Adds a new row below the selected row.
Delete button	Removes the selected row from the Attributes table.
Edit button	Makes the selected cell available for editing.

Related Topics

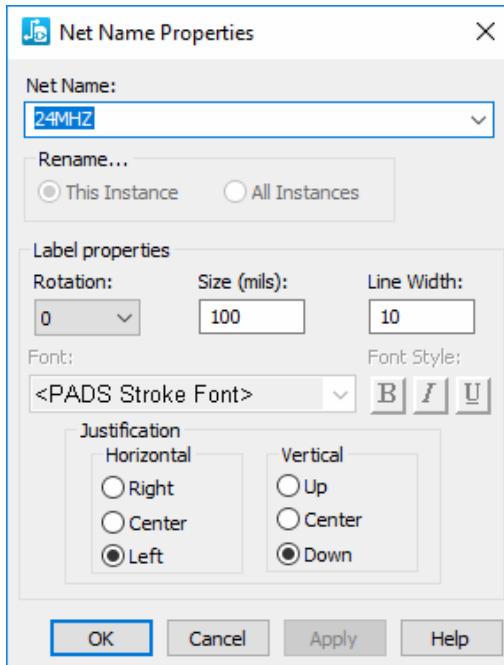
[Creating Net Attributes](#)

Net Name Properties Dialog Box

To access: Select a net name label > right-click > **Properties** menu item

Use the Net Name Properties dialog box to provide or change text and font settings for one or more net name labels.

Figure 89. Net Name Properties Dialog Box



Objects

Table 146. Net Name Properties Dialog Box Contents

Name	Description
Net Name	The name of the selected net.
Rename	Specifies to rename the selected net name or all instances of the net name.
Rotation	Specifies the rotation of the label: 0 or 90 degrees.
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.

Table 146. Net Name Properties Dialog Box Contents (continued)

Name	Description
	 Stroke Font - Size
Line Width	Specifies the line width for stroke fonts only.  Stroke Line Width
Font list	The fonts available to you. This lists either stroke fonts or system fonts. You choose which type of font to use in the Fonts Dialog Box . i Tip <ul style="list-style-type: none">• Select stroke font or a system font.• For system fonts, you can also click a font style button, or any combination of styles:B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

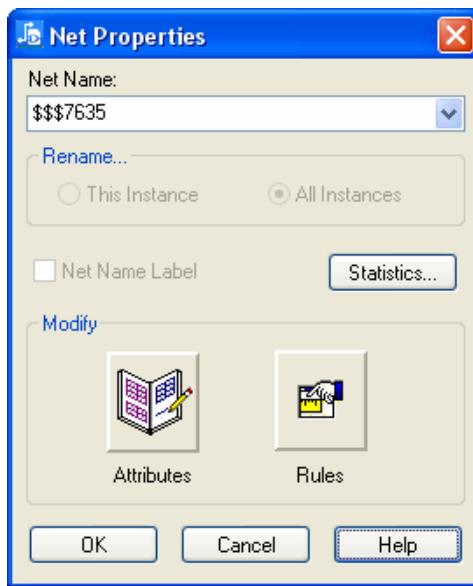
[Modifying Part Type Labels](#)

Net Properties Dialog Box

To access: Select a net > right-click > **Properties** menu item

Use the Net Properties dialog box to access net specific properties.

Figure 90. Net Properties Dialog Box



Objects

Table 147. Net Properties Dialog Box Contents

Name	Description
Net Name	The name of the selected net. Type or select one from the list to create a new one.
Rename area	Specifies to rename the selected net or all instances of the net. Tip Selecting This Instance will cause the net to be split into two separate nets.
Net Name Label	Specifies to add a visible label to the selected net segment. Restriction: You must select the connection segment where it enters or exits a part, or the Net Name Label check box will be unavailable. Tip Clearing the Net Name Label check box removes the visible label. The net and all subnets retain the net name.
Statistics button	Displays connection information in your default text editor.

Table 147. Net Properties Dialog Box Contents (continued)

Name	Description
Attributes button	Opens the Net Attributes Dialog Box .
Rules button	Opens the Net Rules Dialog Box .

Related Topics

[Modifying Nets](#)

Net Rules Dialog Box

To access:

- **Setup > Design Rules** menu item > **Net** button
- Select a net > right-click > **Show Rules** menu item
- Select a net > right-click > **Properties > Rules** button

Use the Net Rules dialog box to define design rules that apply to nets.

Figure 91. Net Rules Dialog Box



Objects

Table 148. Net Rules Dialog Box

Name	Description
Nets list	Lists all nets in the design.
Show Nets with rules	Specifies to show only nets that have rules.
Clearance button	Opens the Clearance Rules Dialog Box .
Routing button	Opens the Routing Rules Dialog Box .
HiSpeed button	Opens the HiSpeed Rules Dialog Box .
Report button	Opens the Rules Report Dialog Box .
Picture below rule button	The picture below each type of rule button identifies which rules hierarchy level is used for that rule type. The meaning of the picture corresponds to the button in the Hierarchy area of the Rules dialog box. For example, if you select a class in the Class list and a green polygon

Table 148. Net Rules Dialog Box (continued)

Name	Description
	appears below the Clearance button, then the default values apply to the class.
Selected	Lists the net(s) selected in the Nets list.
Default button	Removes non-default rules from the selected nets, so that only default rules apply.

Netlist to PCB Dialog Box

To access: Tools > Layout Netlist menu item

Use the Netlist to PCB dialog box to create a netlist for import to SailWind Layout.



Tip

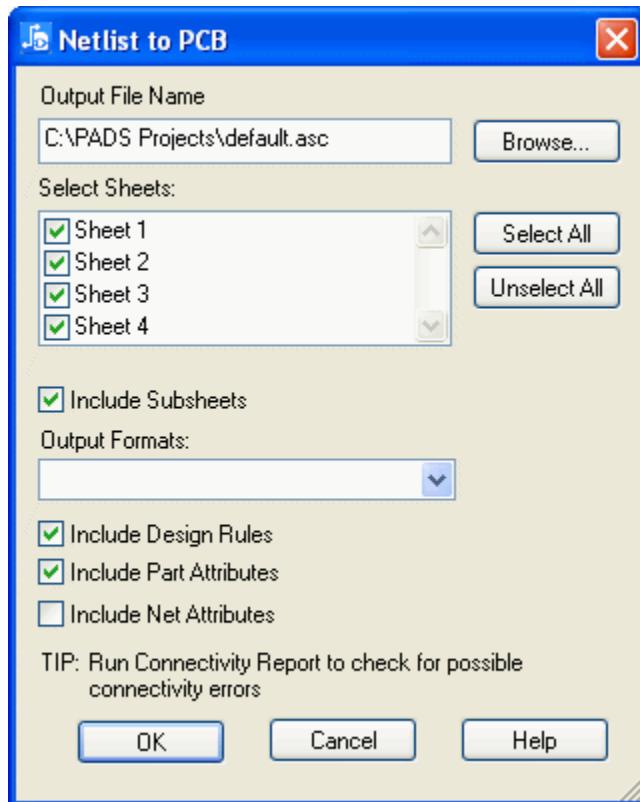
If you use both SailWind Logic and SailWind Layout on the same computer, there is a more automated method to pass the netlist. For more information, see the **Send Net list** button on the [SailWind Layout Link Dialog Box, Design Tab](#).



Restriction:

Transferring non-ECO-registered parts and non-electrical parts is constrained by settings in the Options. See [Options Dialog Box, Design Category](#) on page 591 for details.

Figure 92. Netlist to PCB Dialog Box



Objects

Table 149. Netlist to PCB Dialog Box Contents

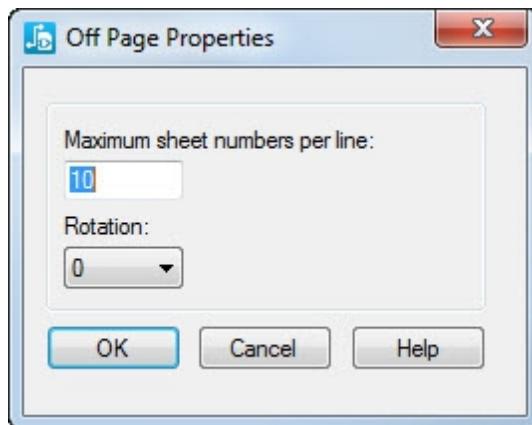
Name	Description
Output File Name	Accept the default name, type, or browse to the location and name of the netlist (.asc) file you are creating.
Select Sheets area	Specifies the schematic sheets to include in the netlist. You can use the Select All and Unselect All buttons as shortcuts.
Include Subsheets	Specifies to include any connections to hierarchical symbols in the netlist.
Output Formats	Specifies to output netlists compatible with the latest or older database formats.
Include Design Rules	Specifies to include design rules and trace width settings in the netlist.
Include Part Attributes	Specifies to include part attributes in the netlist.
Include Net Attributes	Specifies to include net attributes in the netlist
TIP: Run Connectivity Report to check for possible connectivity errors	To run the Connectivity Report, in SailWind Logic, click the File > Reports menu item. In the Reports Dialog Box , select the Connectivity check box and click OK .

Off-Page Properties Dialog Box

To access: Select an off-sheet label > right-click > **Properties** menu item

Use the Off-Page Properties dialog box to set the properties of off-sheet labels.

Figure 93. Off-Page Properties Dialog Box



Objects

Table 150. Off-Page Properties Dialog Box

Name	Description
Maximum sheet numbers per line	Enter the maximum number of sheet numbers allowed per line.
Rotation	Select 0 or 90 degrees.

Options Dialog Box

Using the Options dialog box, you can preset options for commands in SailWind Logic, setting up how those SailWind Logic commands will work and overriding the default settings in the default.txt file. Setting options enables you to set up a working environment that suits your design and the way you work.

To access: **Tools > Options** menu item

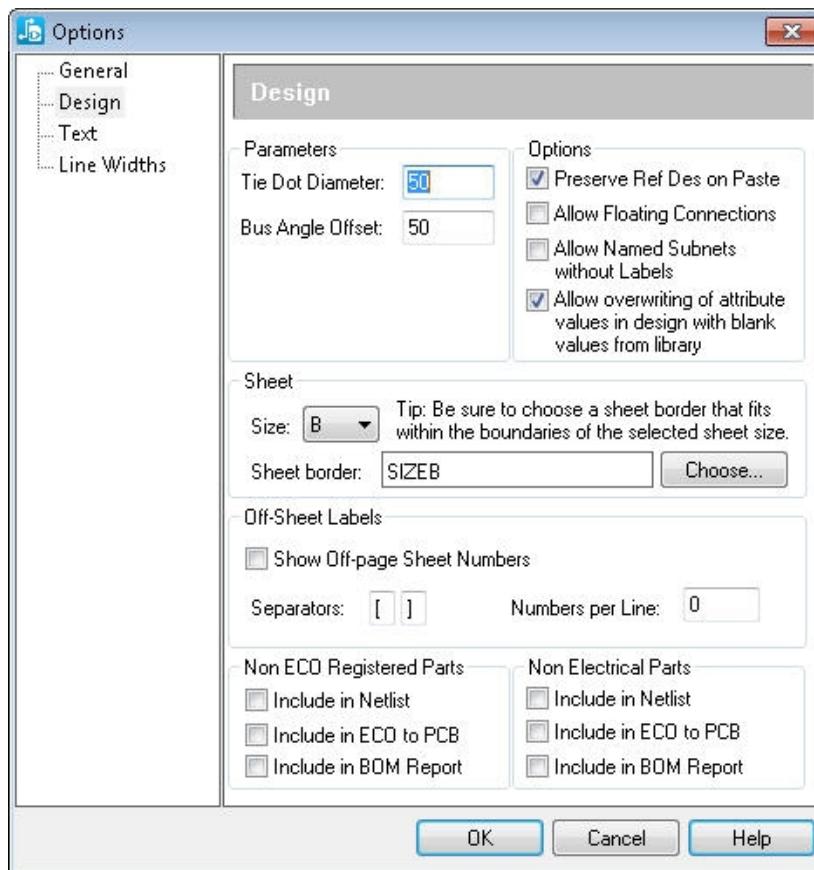
- [Options Dialog Box, Design Category](#)
- [Options Dialog Box, General Category](#)
- [Options Dialog Box, Line Widths Category](#)
- [Options Dialog Box, Text Category](#)

Options Dialog Box, Design Category

To access: Tools > Options menu item > Design category

There are six categories of general options for the Schematic Editor; they are organized into six areas in the Design category labeled Parameters, Options, Sheet, Off-sheet Labels, Non-ECO-Registered Parts, and Non-Electrical Parts.

Figure 94. Options Dialog Box, Design



Objects

Table 151. Options Dialog Box, Design Category

Name	Description
Parameters area	
Tie Dot Diameter	Specifies the diameter of tie dots. The value must be from 0 to 100.
Bus Angle Offset	Defines the starting point distance from the bus for the bus tap. The value must be from 0 to 250.

Table 151. Options Dialog Box, Design Category (continued)

Name	Description
	 Tip The bus tap joins connections to the bus.
Options area	
Preserve Ref Des on Paste	Specifies to use a group's current reference designator assignment when possible.  Tip Click to clear this check box to assign the pasted group's reference designators the first or next available number. See also Preserving Reference Designators .
Allow Floating Connections	Specifies to create connections without terminations.  Tip Click to clear this check box to prevent creating additional Floating Connections. Disabling the option does not remove existing Floating Connections. See also Working With Floating Connections .
Allow Named Subnets without Labels	Specifies to allow deletion of net name labels in all cases except that of a power symbol connected to a net that overrides its default net name. When this box is checked, the current net name is not changed under any circumstances. When this box is cleared, you cannot delete labels tied to bus rippers or off-page symbols. You can delete net labels tied to component pins; this causes the net to be renamed to a system-generated name, but only if both the following conditions apply: <ul style="list-style-type: none"> • The net is not connected to a bus or off-page symbol. • The deleted label was the only label on the net.  Tip It is possible to have a named subnet without a label even if the "Allow Named Subnets without Labels" check box is unchecked. If you change the name of a subnet with a system net name in the Net Properties dialog box, and click OK without checking the Net Labels check box, the result will be a named subnet without a label.
Allow overwriting of attribute values in design with blank values from library	Specifies how design attribute values should be handled by operations that change design part types, such as Update From Library, Update Selected Part Type from Library, Change Type (in the Part Type Properties dialog), ECO Import, and automated operations. <ul style="list-style-type: none"> • Select the check box to allow the value of an attribute in the design to be overwritten by a blank value from the library or other update source. • Clear it to prevent overwriting these values.
Sheet area	
Size list	Specifies the size sheet you want.

Table 151. Options Dialog Box, Design Category (continued)

Name	Description
Sheet border	Specifies the current sheet boarder.
Choose button	Opens the Get Drafting Item from Library dialog box to change the sheet boarder. i Tip Be sure to select a sheet border that fits within the boundaries of the selected sheet size. See also Adding Drafting Items From a Library .
Off-Sheet Labels area	
Show Off-page Sheet Numbers	Displays sheet reference numbers that are adjacent to off-page reference symbols and bus names.
Separators	Specifies the open and close characters you want to enclose sheet numbers. For example, "" {} or [] .
Numbers per Line	Specifies how many sheet numbers to display per line. The value must be from 0 to 99. Depending on the value you specify, you can stack or line up the sheet number references. i Tip <ul style="list-style-type: none"> • This only applies when Show Off-page Sheet Numbers is selected. • The sheet numbers appear on a line below the net name.
Non-ECO-Registered Parts area	
Include in Netlist	Specifies to include the part in the netlist. i Tip To include a part that is both non-ECO-registered and non-electrical in a netlist, select the check boxes in both Part areas.
Include in ECO to PCB	Specifies to include a part in the ECO file that is forwarded to the PCB. i Tip To include a part that is both non-ECO-registered and non-electrical in a Forward ECO to PCB, select the check boxes in both Part areas.
Include in BOM report	Specifies to include a part in the Bill of Materials (BOM) report. i Tip To include a part that is both non-ECO-registered and non-electrical in a BOM report, select the check boxes in both Part areas.
Non-Electrical Parts area	
Include in Netlist	Specifies to include the part in the netlist.

Table 151. Options Dialog Box, Design Category (continued)

Name	Description
	<p> Tip To include a part that is both non-ECO-registered and non-electrical in a netlist, select the check boxes in both Part areas.</p>
Include in ECO to PCB	<p>Specifies to include a part in the ECO file that is forwarded to the PCB.</p> <p> Tip To include a part that is both non-ECO-registered and non-electrical in a Forward ECO to PCB, select the check boxes in both Part areas.</p>
Include in BOM report	<p>Specifies to include a part in the Bill of Materials (BOM) report.</p> <p> Tip To include a part that is both non-ECO-registered and non-electrical in a BOM report, select the check boxes in both Part areas.</p>

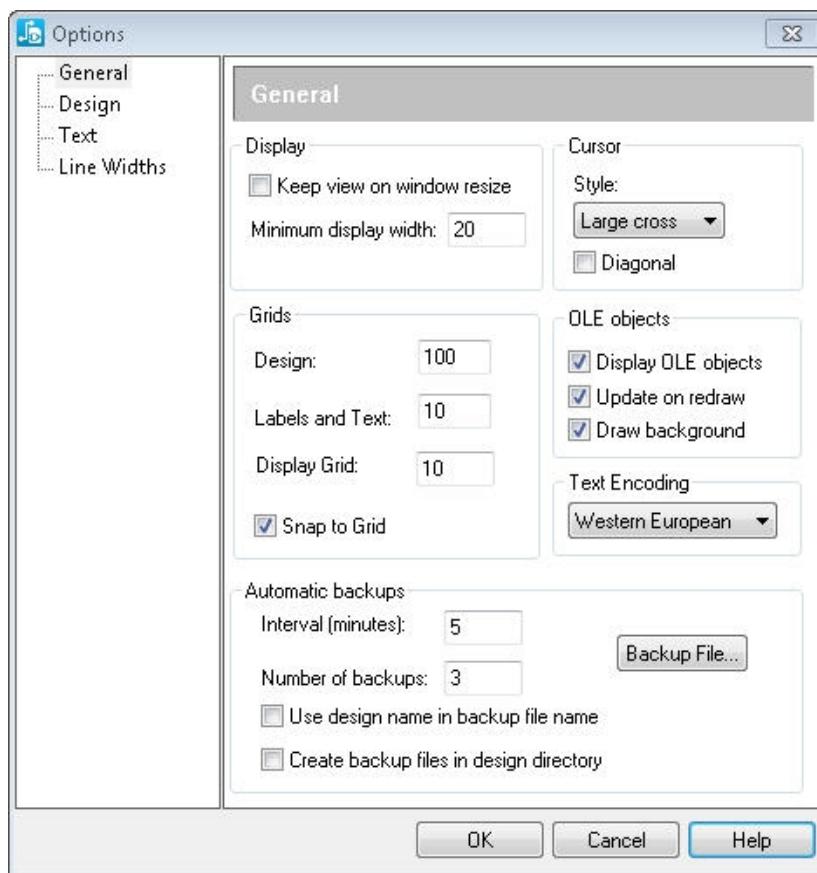
Related Topics

[Setting Schematic Editor Options](#)

Options Dialog Box, General Category

To access: Tools > Options menu item > General category

There are six categories of general options for the Schematic Editor; they are organized into six areas in the General category labeled Display, Cursor, Grids, OLE objects, Text Encoding, and Automatic Backups.



Objects

Table 152. Options Dialog Box, General Category

Name	Description
Display area	
Keep view on window resize	Specifies to maintain the area view of the design when you resize the SailWind Logic window, by automatically zooming in or out.
Minimum display width	Specifies the minimum width, in current design units, of lines you want to draw at actual width. Lines smaller than this width are drawn only as centerlines to save memory and redraw time.

Table 152. Options Dialog Box, General Category (continued)

Name	Description
	 Tip <ul style="list-style-type: none"> Set this value to zero to display all lines at actual width. Larger values reduce redraw times.
Cursor area	
Style list	<p>Specifies the cursor shape:</p> <ul style="list-style-type: none"> Normal — Arrow Small cross — Small plus sign + Large cross — Large plus sign + Full screen — Full screen crosshair  Tip <p>If you want the cursor resemble an x, select the Diagonal check box.</p>
Grids area	
Design	Specifies a grid size for the Design grid which determines object placement. The value must be from 2 to 2000 and a multiple of 2 (for example, 2, 4, 50, 100).
Labels and Text	Specifies a grid size for the Labels and text grid which determines placement of all labels, fields, names, attributes, and text. The value must be from 2 to 2000 and a multiple of 2 (for example, 2, 4, 50, 100).
Display Grid	<p>Specifies a grid size for the Display grid which is a visible guide for drawing lines, decals, connections, and more. The value must be from 10 to 9998 and a multiple of 2 (for example, 2, 4, 50, 100).</p>  Tip <ul style="list-style-type: none"> The Display grid is independent of the system grid displayed on the status line. If the grid is not visible when the work area is redrawn, you probably have set an interval that is too small for display at all zoom levels. You will have to zoom in several times before the display grid becomes visible.
Snap to Grid	<p>Specifies to snap an item or object to the grid during editing.</p> <p>See also Work Area and Grid Settings.</p>
OLE Objects area	
Display OLE objects	<p>Specifies to display linked and embedded objects in the work area.</p>  Tip <p>The OLE display settings affect SailWind Logic only when it is embedded in another application.</p>
Update on redraw	Specifies to update the SailWind Layout linked or embedded object in the container application.

Table 152. Options Dialog Box, General Category (continued)

Name	Description
	<p> Restriction: This option applies only when you are editing the SailWind Logic object in a separate window and you click the Redraw button in the separate window.</p> <p> Tip</p> <ul style="list-style-type: none"> • To increase performance, disable this option. • The OLE display settings affect SailWind Logic only when it is embedded in another application.
Draw background	<p>Specifies to draw the SailWind Logic background color in the linked or embedded SailWind Logic object.</p> <p> Tip When this option is disabled, the background of the SailWind Logic object is transparent and you can see through the object at the container application's background.</p>
Text Encoding area	
Text Encoding list	<p>Specifies the language encoding to use in text strings and labels displayed in the design screen.</p> <p> Tip Encoding choices include those supported in the current localized version of SailWind Logic, plus any default encoding already in use in the design.</p> <p>Example: A letter with a 0x20 code displays as a Latin capitol letter A with a grave accent in a design using Western encoding, and as a Cyrillic capital letter А in a design with Cyrillic encoding.</p> <p> Restriction: The default text encoding cannot be changed. It is automatically set by the Regional and Language settings of the operating system.</p>
Automatic backups area	
Interval	Specifies the time in minutes between automatic backups to a file.
Number of backups	<p>Specifies the quantity (1-9) of different backup files to create.</p> <p> Tip Backup files are named <filename>#.sch, where # is a sequential number. For example, logic1.sch, logic2.sch, and so on.</p>
Backup File button	Changes the folder or name of the backup file
Use design name in backup file name	<p>Specifies to use the design name instead of the product name as the file name.</p> <p>Example: preview_logic1.sch, preview_logic2.sch instead of logic1.sch, logic2.sch.</p>
Create backup files in design directory	Specifies to place all of your backup files in the same directory as the design.

Table 152. Options Dialog Box, General Category (continued)

Name	Description
	 Tip Click to clear if you want your backup files in one, common backup directory.

Related Topics

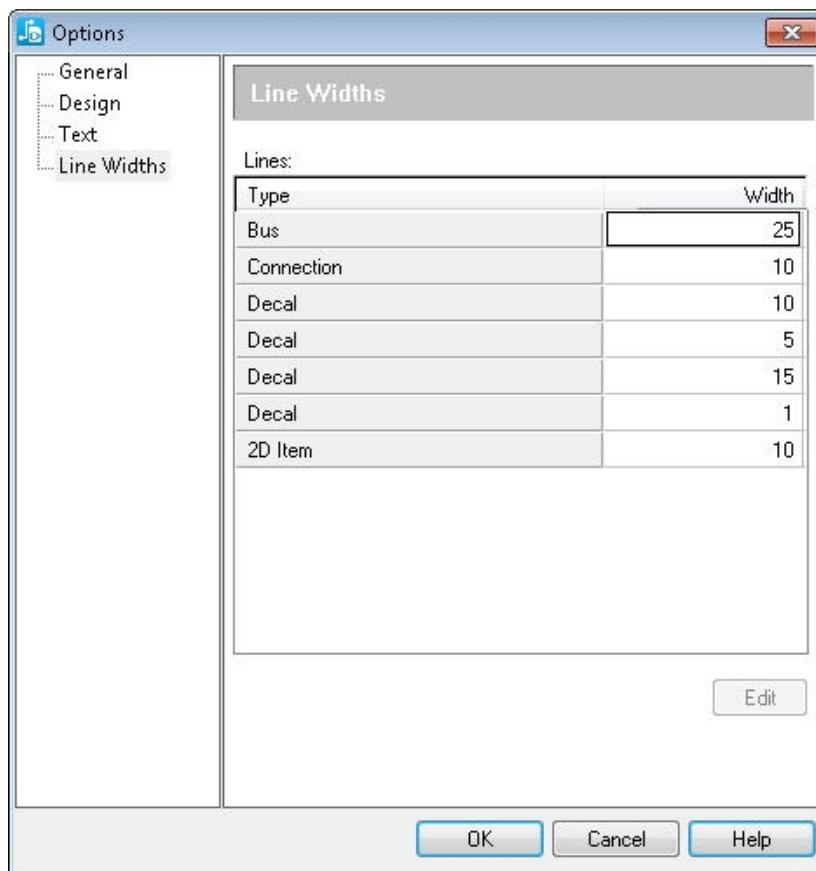
[Setting Schematic Editor Options](#)

[Creating a Backup File](#)

Options Dialog Box, Line Widths Category

To access: Tools > Options menu item > Line Widths category

Use the Line Widths category to change the size of line widths in the workspace.



Objects

Table 153. Options Dialog Box, Line Widths Category

Name	Description
Type	Lists the types of lines available in the schematic.
Width	Specifies the width of the line type.
Edit button	Makes the Width column of the selected row editable.

Related Topics

[Setting Line Widths](#)

Options Dialog Box, Text Category

To access: **Tools** menu item > **Text** category

The Text category differs depending on the selection you made in the Fonts dialog box: Stroke or System font.

Figure 95. Options Dialog Box, Text Page - System Font

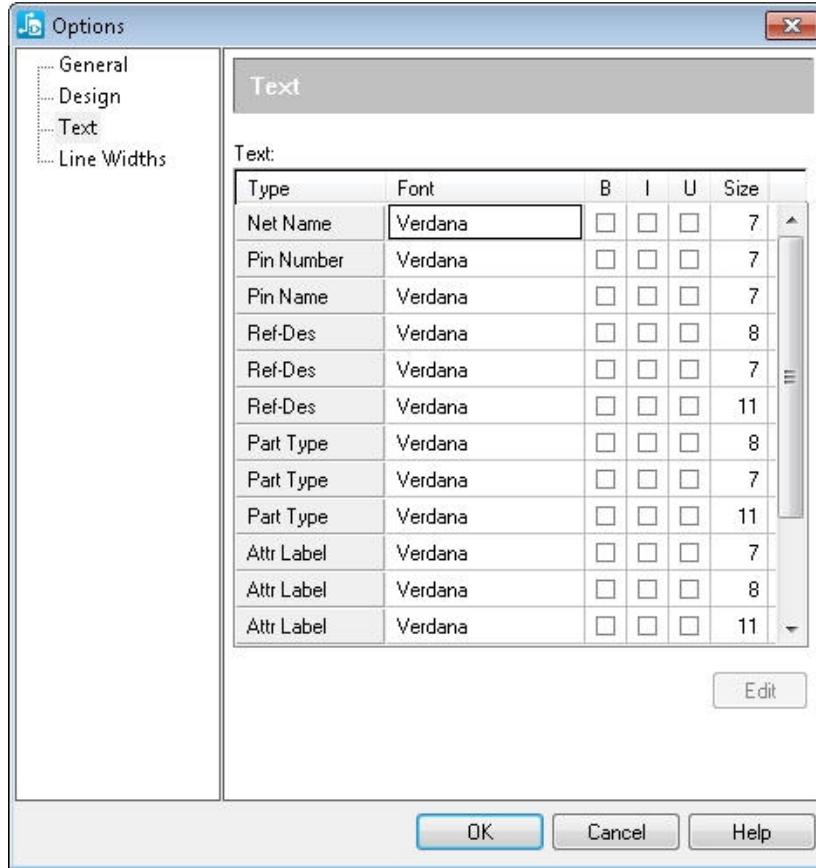
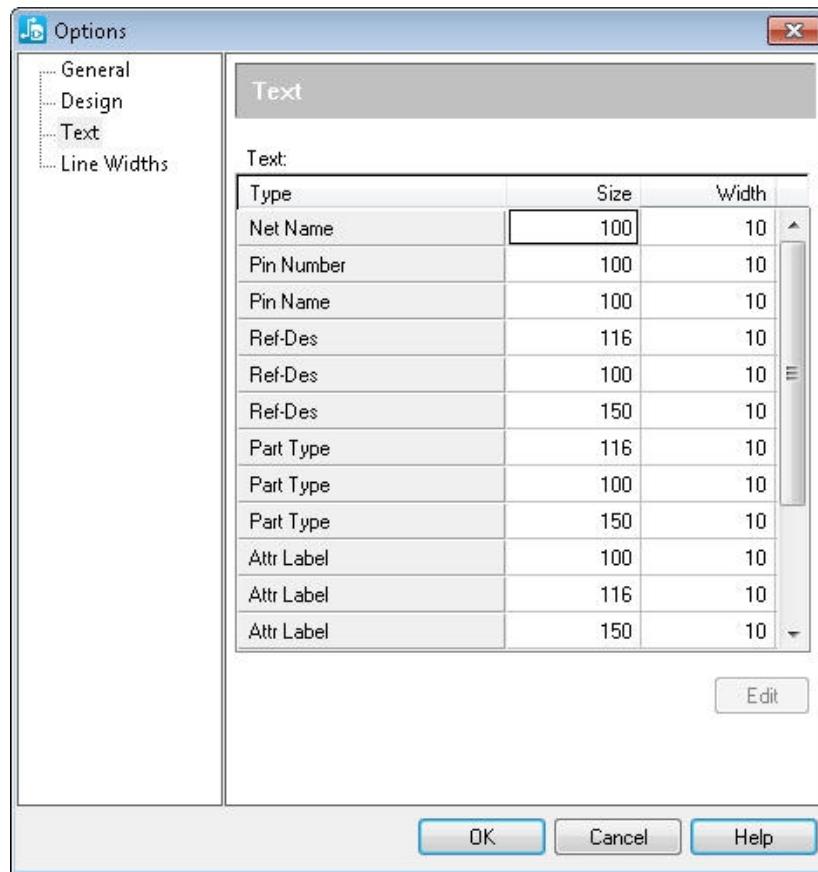


Figure 96. Options Dialog Box, Text Page - Stroke Font



Objects

Table 154. Options Dialog Box, Text Category

Name	Description
Type	Displays the type of font used in the schematic.
Font	Specifies the system font used. [File] Restriction: System font only.
B	Specifies if this font is bold. Click to make the font bold; click to clear to make it regular weight. [File] Restriction: System font only.
I	Specifies if this font is italicized. Click to make the font italic; click to clear to make it regular. [File] Restriction: System font only.

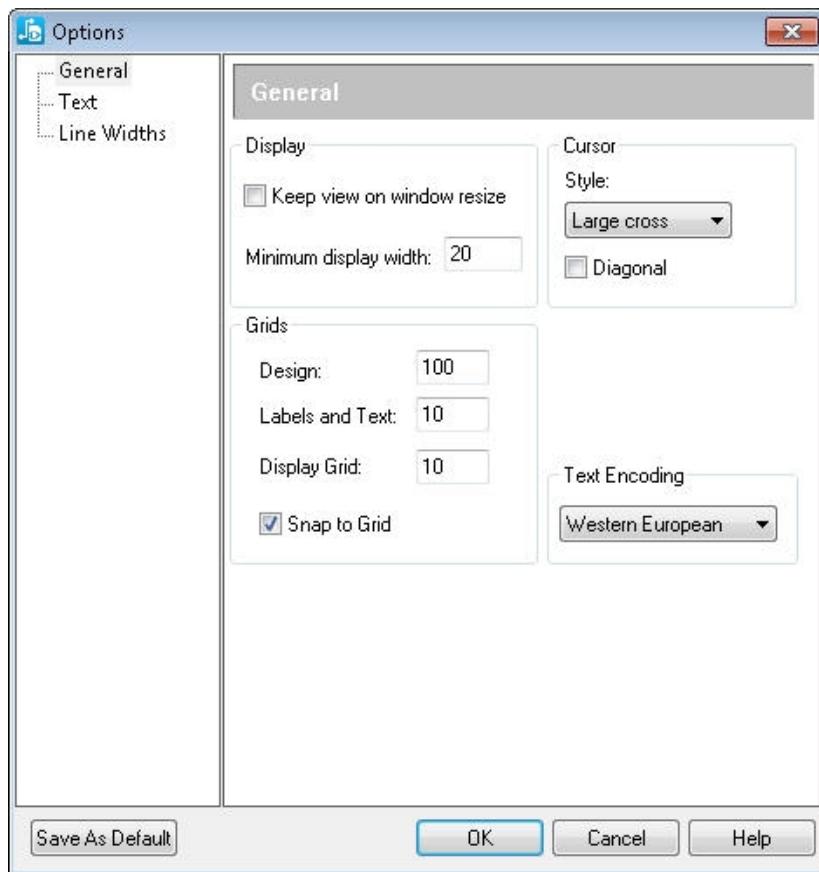
Table 154. Options Dialog Box, Text Category (continued)

Name	Description
U	Specifies if this font is underlined. Click to underline the font; click to clear to make it regular.  Restriction: System font only.
Size	Specifies the size of the font.  Restriction: System font only.
Width	Specifies the width of the font.  Restriction: Stroke font only.
Edit button	Makes the selected field editable.  Restriction: You cannot edit fields in the Type column.

Options Dialog Box - Part Editor, General Category

To access: While in the Schematic Editor, click the **Tools > Part Editor** menu item. In the Part Editor, click the **Edit > CAE Decal Editor** menu item. In the CAE Decal Editor, click the **Tools > Options** menu item, then click the **General** category.

There are four categories of general options for the Part Editor; they are organized into four areas on the General category labeled Display, Cursor, Grids, and Text Encoding.



Objects

Table 155. Options Dialog Box - Part Editor, General Category

Name	Description
Display area	
Keep view on window resize	Specifies to maintain the area view of the design when you resize the SailWind Logic window, by automatically zooming in or out.

Table 155. Options Dialog Box - Part Editor, General Category (continued)

Name	Description
Minimum display width	<p>Specifies the minimum width, in current design units, of lines you want to draw at actual width. Lines smaller than this width are drawn only as centerlines to save memory and redraw time.</p> <p>i Tip</p> <ul style="list-style-type: none"> • Set this value to zero to display all lines at actual width. • Larger values reduce redraw times.
Cursor area	
Style list	<p>Specifies the cursor shape:</p> <ul style="list-style-type: none"> • Normal — Arrow • Small cross — Small plus sign + • Large cross — Large plus sign + • Full screen — Full screen crosshair <p>i Tip</p> <p>If you want the cursor to resemble an x, select the Diagonal check box.</p>
Grids area	
Design	Specifies a grid size for the Design grid which determines object placement. The value must be from 2 to 2000 and a multiple of 2 (for example, 2, 4, 50, 100).
Labels and Text	Specifies a grid size for the Labels and text grid which determines placement of all labels, fields, names, attributes, and text. The value must be from 2 to 2000 and a multiple of 2 (for example, 2, 4, 50, 100).
Display Grid	<p>Specifies a grid size for the Display grid which is a visible guide for drawing lines, decals, connections, and more. The value must be from 10 to 9998 and a multiple of 2 (for example, 2, 4, 50, 100).</p> <p>i Tip</p> <ul style="list-style-type: none"> • The Display grid is independent of the system grid displayed on the status line. • If the grid is not visible when the work area is redrawn, you probably have set an interval that is too small for display at all zoom levels. You will have to zoom in several times before the display grid becomes visible.
Snap to Grid	<p>Specifies to snap an item or object to the grid during editing.</p> <p>See also Work Area and Grid Settings.</p>
Text Encoding area	
Text Encoding list	Specifies the language encoding to use in text strings and labels displayed in the design screen.

Table 155. Options Dialog Box - Part Editor, General Category (continued)

Name	Description
	<p> Tip Encoding choices include those supported in the current localized version of SailWind Logic, plus any default encoding already in use in the design. Example: A letter with a 0x20 code displays as a Latin capital letter A with a grave accent in a design using Western encoding, and as a Cyrillic capital letter A in a design with Cyrillic encoding.</p>

Related Topics

[Setting Part Editor Options](#)

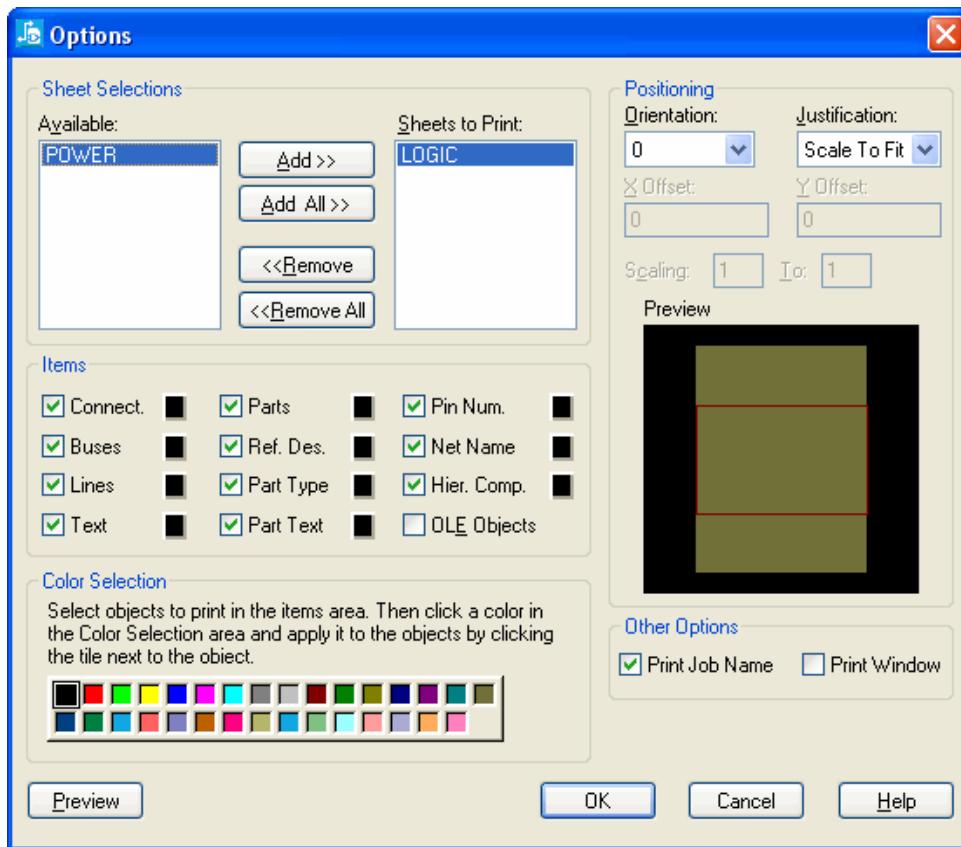
Options Dialog Box - Print/Plot

To access:

- **File > Plot** menu item > **Options** button
- **File > Print** menu item > **Options** button

Use the Options dialog box to set the output options for printing or plotting. Setting options enables you to set the position, orientation, and color of selected sheets and objects.

Figure 97. Options Dialog Box - Print/Plot



Objects

Table 156. Options Dialog Box - Print/Plot

Name	Description
Available list	Lists the available sheets to print.
Add >> button	Moves the selected sheet to the Sheets to Print list.
Add All >> button	Moves all sheets to the Sheets to Print list.

Table 156. Options Dialog Box - Print/Plot (continued)

Name	Description
<< Remove button	Moves the selected sheet to the Available list.
<< Remove All button	Moves all sheets to the Available list.
Sheet to Print list	Lists off the sheets you want to print.
Items area	Specifies the items you want to include in your output.
Color Selection area	Specifies the color you want for the different items in the Items area. Click a color tile, then in the Items area, click the tile to the right of the item to change its color.
Orientation	Specifies the orientation angle for the output.
Justification	Specifies the justification position for the output.
X/Y Offset	Specifies the offset values. i Tip <ul style="list-style-type: none"> You can only set offsets if you clicked something other than Scale to Fit and Centered in the Justification list. The print or plot location is calculated after the plot is rotated and scaled.
Scaling	Specifies the type the plot size to actual size ratio. Example: A 2 to 1 scaling results in a printout or plot that is twice the actual size.
Preview area	Graphically shows the position, orientation, justification, and scaling of the output.
Plot Job Name	Specifies to output the schematic name, time, and date.
Plot Window	Specifies to print the displayed window.
Preview button	Opens the Selections Preview Dialog Box .

Output Window

Use the Output window for displaying reports and session logs, macro editing and debugging, and custom programming and debugging, as well as CIS configuration.

To access: Click the **Output Window** button.

The Output window is located in the lower left section of the display window. You can dock or float the Output window. You can also open or close the Output window.

The Output window has three tabs described below.

Status Tab

The **Status** tab displays information about the current session. It specifies the filename of the opened PCB file and the name of the test integrity file that is saved. It also reports routing statistics and messages when routing a board.

If the **Status** tab is closed, and you get an error while autorouting - or performing other tasks - the Output window opens with the **Status** tab active and the error appearing in red. The Output window reappears in its most recent state (floating or docked).

- **Output Window** button > **Status** tab

Macro Tab

You can edit, run, and debug macro scripts in the **Macro** tab. You can open multiple macros and nest macros using the macro editor.

A macro is any combination of commands, keystrokes, and mouse clicks that you record to replay as a single action. You can record virtually any set of procedural steps for replay, thereby simplifying redundant activities, such as setting preferences and layer/display settings.

- **Output Window** button > **Macro** tab

CIS Tab

The **CIS** tab provides a graphic interface to view part information in the database, to which you connect. You can select and add parts to the design, specify the part information to display, and export database configuration.

- **Output Window** button > **CIS** tab

[Library Config Dialog Box](#)

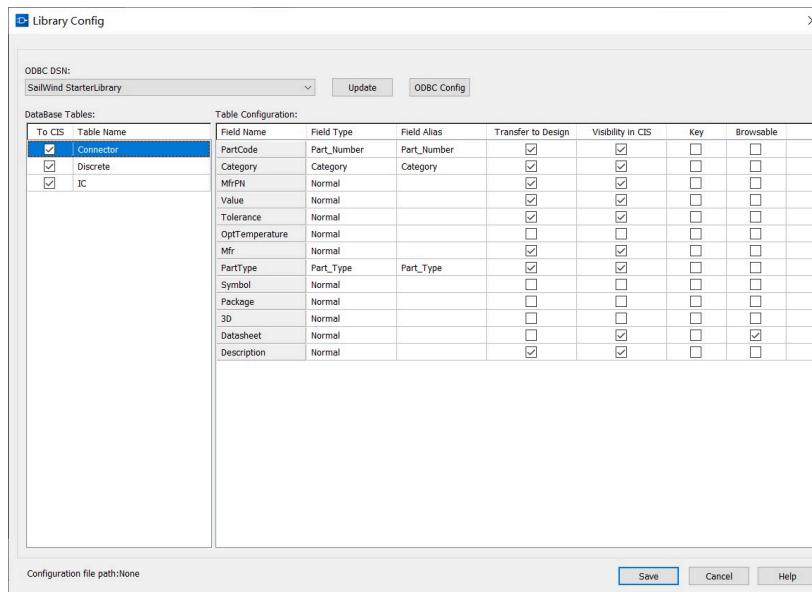
[Part Manager Dialog Box](#)

Library Config Dialog Box

To access: **Output Window > CIS tab > New/Edit button**

Use the Library Config dialog box to specify the database from which to load part information and specific part information displayed in the **CIS** tab.

Figure 98. Library Config Dialog Box



Objects

Table 157. Library Config Dialog Box Contents

Name	Description
ODBC DSN area:	
ODBC DSN	Selects the database from which to load part information from the drop-down list. Tip SailWind StarterLibrary is connected by default.
ODBC Config	Adds a new database. Tip Click Update to make the newly added database available.
Update	Makes the newly added database available in the ODBC DSN drop-down list.
Database Tables area	
To CIS	Specifies database table(s) to display in the CIS tab.

Table 157. Library Config Dialog Box Contents (continued)

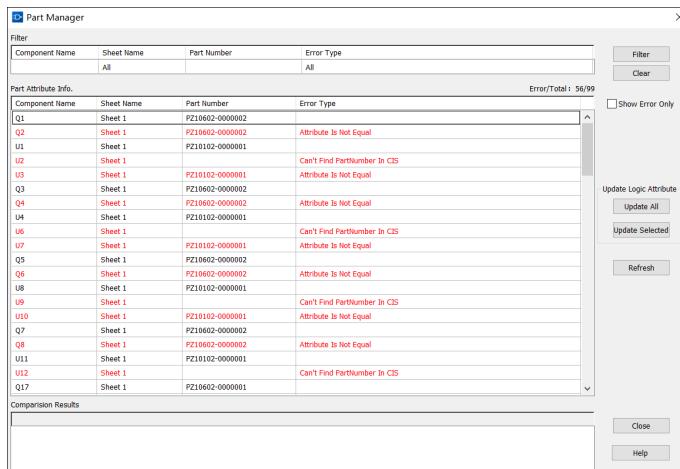
Name	Description
Table Name	Displays all the table(s) in the selected database.
Table Configuration area	
Field Name	Lists all fields in the table selected on the left.
Field Type	<p>Specifies what type the table fields belong to from the drop-down list, wherein:</p> <ul style="list-style-type: none"> Part_Type is mandatory, based on which to load data into the CIS tab. Besides, you can see schematic symbol and PCB decal assigned to the part type in the local libraries from the CIS preview window. Part_Number is mandatory, based on which to check whether part attribute values in design are identical with those in CIS. Category allows to show table structure hierarchically by subcategories in the CIS tab. All field types except Normal must be unique.
Field Alias	<p>Specifies the table heading for each field to display in the CIS tab.</p> <ul style="list-style-type: none"> Field aliases corresponding to Field Type "Part_Type" and "Part_Number" are defined by default, and no modification is allowed. If nothing is set, Field Name will be used instead.
Transfer to Design	<p>Specifies to add the Field Name to the part attributes. If yes, you can see it in the Part Attributes list by checking part properties in the design.</p> <p> Tip When set, Field Alias will be used instead.</p>
Visibility in CIS	Specifies to display Field Name in the CIS tab. When set, Field Alias will be used instead.
Key	Reserved
Browsable	Specifies to add hyperlinks to the field contents in the CIS tab, which often links to such reference files as datasheets and drawings.
Property Checking	<p>Specifies attribute(s) to compare for consistency checking in the Part Manager Dialog Box.</p> <p> Tip No checkbox selected means comparing all attributes by default.</p>

Part Manager Dialog Box

To access: **Output Window > CIS tab > Part... button**

Use the Part Manager dialog box to compare part attributes in design with those in CIS for consistency checking. For the inconsistent attribute values, you can update from CIS with multiple options.

Figure 99. Part Manager Dialog Box



Objects

Table 158. Part Manager Dialog Box Contents

Name	Description
Filter area	<ul style="list-style-type: none"> Specifies the search filter. Note that filtering by Component Name or Part Number is case-sensitive and no wildcard or expression is currently supported. Provides functionality buttons as follows: <ul style="list-style-type: none"> Filter: Used to activate the filter Clear: Used to reset the search filter settings Show Error Only: Used to show parts found with errors
Part Attribute Info. area	<p>Displays the search result. For schematic parts whose attributes are not identical with those in CIS, they are highlighted in red.</p>
Update the Selected button	<p>Updates the inconsistent attributes of the selected schematic part(s) from CIS. Takes effect only on schematic parts found with "Attribute is not equal" error.</p>
Update All button	Updates inconsistent attributes of all schematic parts found with "Attribute is not equal" error from CIS.

Table 158. Part Manager Dialog Box Contents (continued)

Name	Description
Refresh button	Manually updates the comparision result if you change part attributes in the design.
Comparision Results area	Displays what assigned to the part attributes in design and CIS respectively, with differences highlighted in red. You can: <ul style="list-style-type: none">• Update the selected attribute: Right-click on the attribute cell and click the Update Selected Attribute From CIS popup menu item• Update all attributes of the schematic part: Right-click and click the Update Selected Part From CIS popup menu item

SailWind Layout Link Dialog Box

The SailWind Layout Link dialog box enables you to connect to SailWind Layout to automate sending the netlist and passing design change annotations, and cross-probe (receive and send selections). This feature also automates certain tasks for you, such as netlist comparisons and rules exports. Once you connect to SailWind Layout, you can select objects in SailWind Layout and the object is automatically selected and shown on its SailWind Logic schematic sheet (and vice versa).

See also [Cross-Probe Between Sailwind Products](#).

The options you set in the SailWind Layout Link dialog box are saved to the registry. These options are restored the next time you open the SailWind Layout Link.



Note:

The exception is the selection made in the Sent Selection list on the **Selection** tab, and the document pathname on the **Document** tab.

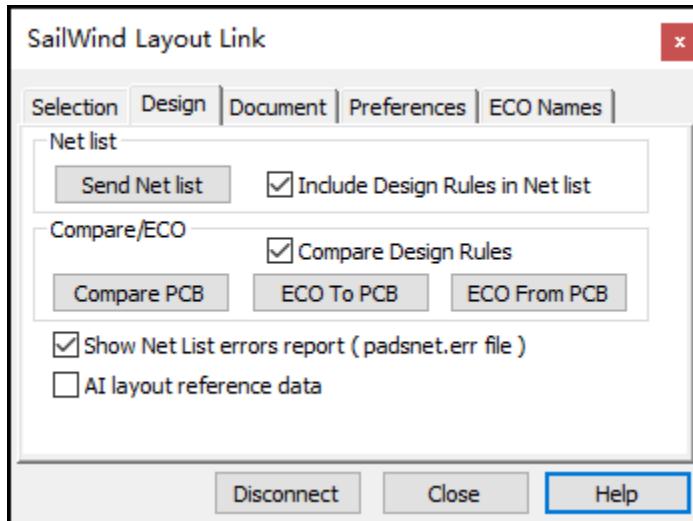
SailWind Layout Link dialog box tabs:

- [SailWind Layout Link Dialog Box, Design Tab](#)
- [SailWind Layout Link Dialog Box, Document Tab](#)
- [SailWind Layout Link Dialog Box, ECO Names Tab](#)
- [SailWind Layout Link Dialog Box, Preferences Tab](#)
- [SailWind Layout Link Dialog Box, Selection Tab](#)

SailWind Layout Link Dialog Box, Design Tab

To access: **Tools > SailWind Layout** menu item > connect to SailWind Layout using the [Connect to SailWind Layout Dialog Box](#) > **Design** tab

Use the **Design** tab of the SailWind Layout Link dialog box to export a netlist to SailWind Layout, compare the schematic to the layout design, or forward and backward annotate design data.



Objects

Table 159. SailWind Layout Link Dialog Box, Design Tab Contents

Name	Description
Net list area	
Send Netlist button	<p>Exports a netlist from the current schematic to the current SailWind Layout design.</p> <p>Results:</p> <ul style="list-style-type: none">If no errors are found in the netlist, SailWind Layout is updated.If errors are found in the netlist, the error report file is displayed in a Notepad window, a link to the error file is displayed in the Output Window, and you are asked whether you want to continue.<ul style="list-style-type: none">If you click Yes, SailWind Layout is updated.If you click No, the operation is canceled and SailWind Layout is not updated.
Include Design Rules in Netlist check box	Select the check box to include design rules in the exported netlist.

Table 159. SailWind Layout Link Dialog Box, Design Tab Contents (continued)

Name	Description
Compare/ECO area	
Compare PCB button	<p>Compares the netlist of the current SailWind Logic schematic to the netlist of the current SailWind Layout design.</p> <p>Results:</p> <ul style="list-style-type: none"> If no errors are found in either of the netlists, any differences are reported in the default text editor. If errors are found in either of the netlists, the error report file is displayed in a Notepad window, a link to the error file is displayed in the Output Window, and you are asked whether you want to continue. <ul style="list-style-type: none"> If you click Yes, the comparison operation continues. If you click No, the operation is canceled. <p>See also SailWind Layout Link Dialog Box, Preferences Tab, SailWind Layout Link Dialog Box, ECO Names Tab.</p>
ECO To PCB button	<p>Compares the netlist in the current SailWind Logic schematic to the part and netlist in the current SailWind Layout design.</p> <p>Results:</p> <ul style="list-style-type: none"> If no errors are found in either of the netlists, the SailWind Layout design is updated. <p> Tip Overwriting of non-blank attribute values with blank attribute values from the ECO file is allowed/prevented by the “Allow overwriting of attribute values in design with blank values from library” check box in the Design category on page 591 of the Options dialog box.</p> <ul style="list-style-type: none"> If errors are found in either of the netlists, the error report file is displayed in a Notepad window, a link to the error file is displayed in the Output Window, and you are asked whether you want to continue. <ul style="list-style-type: none"> If you click Yes, the SailWind Layout design is updated. If you click No, the operation is canceled, and SailWind Layout is not updated. <p>See also SailWind Layout Link Dialog Box, Preferences Tab, SailWind Layout Link Dialog Box, ECO Names Tab.</p>
ECO From PCB button	<p>Compares the netlist in the current SailWind Logic schematic to the part and netlist in the current SailWind Layout design.</p> <p>Results:</p> <ul style="list-style-type: none"> If no errors are found in either of the netlists, the SailWind Logic schematic is updated. <p> Tip Overwriting of non-blank attribute values with blank attribute values from the ECO file is allowed/prevented by the “Allow overwriting of attribute values in design with blank values from library attributes” check box in the Design category on page 591 of the Options dialog box.</p>

Table 159. SailWind Layout Link Dialog Box, Design Tab Contents (continued)

Name	Description
	<ul style="list-style-type: none">If errors are found in either of the netlists, the error report file is displayed in a Notepad window, a link to the error file is displayed in the Output Window, and you are asked whether you want to continue.<ul style="list-style-type: none">If you click Yes, the SailWind Logic schematic is updated.If you click No, the operation is canceled, and SailWind Logic is not updated. <p>See also SailWind Layout Link Dialog Box, Preferences Tab, SailWind Layout Link Dialog Box, ECO Names Tab.</p>
Compare Design Rules check box	Includes design rules in the comparison that occurs when you select Compare PCB , ECO to PCB , or ECO from PCB .
Show Net List errors report check box	Select the check box to open the Net List errors report once it has been generated. The filename is padsnet.err.  Tip If you clear the check box, you can still open the file from the link in the Output window.
AI layout reference data	Select the check box in order for the AI Intelligent Layout feature to work in SailWind Layout.
Disconnect button	Breaks the connection with SailWind Layout and closes the dialog box.

Related Topics

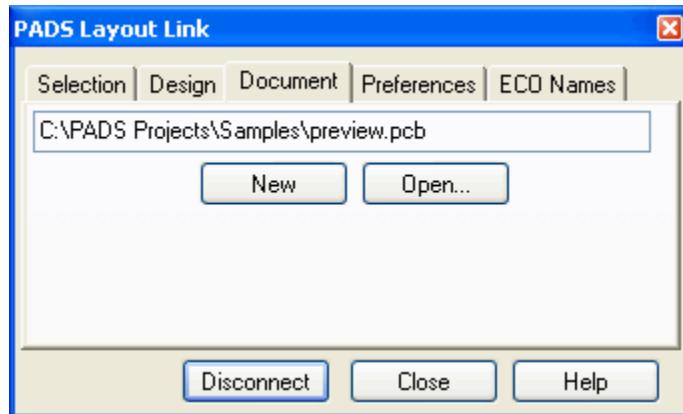
[SailWind Layout Link Dialog Box](#)

SailWind Layout Link Dialog Box, Document Tab

To access: **Tools > SailWind Layout** menu item > connect to SailWind Layout using the [Connect to SailWind Layout Dialog Box](#) > **Document** tab

Use the **Document** tab of the SailWind Layout Link dialog box to open and connect to a different design or a new design file within the current SailWind Layout session. The current design path and filename is listed at the top of this tab.

Figure 100. SailWind Layout Link Dialog Box, Document Tab



Objects

Table 160. SailWind Layout Link Dialog Box, Document Tab Contents

Name	Description
Pathname field	Displays the path and filename of the current SailWind Layout design.
New button	Creates a new file within the current SailWind Layout session. SailWind Logic automatically connects to this file. Tip If the current SailWind Layout design has unsaved changes, a prompt window appears, asking, "Remote PCB document is modified - Do you want to save it?"
Open button	Enables you to locate and open an existing SailWind Layout file. SailWind Logic automatically connects to this file. Tip If the current SailWind Layout design has unsaved changes, a prompt window appears, asking, "Remote PCB document is modified - Do you want to save it?"
Disconnect button	Breaks the connection with SailWind Layout and closes the dialog box.

Related Topics

[SailWind Layout Link Dialog Box](#)

SailWind Layout Link Dialog Box, ECO Names Tab

To access: **Tools > SailWind Layout** menu item > connect to SailWind Layout using the [Connect to SailWind Layout Dialog Box](#) > **ECO Names** tab

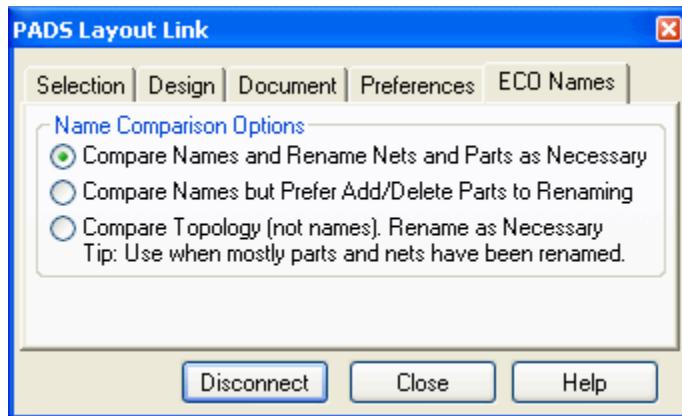
Use the **ECO Names** tab of the SailWind Layout Link dialog box to manage options for the compare and ECO functions on the **Design** tab.



Note:

See also [Design tab](#) on page 614.

Figure 101. SailWind Layout Link Dialog Box, ECO Names Tab



Objects

Table 161. SailWind Layout Link Dialog Box, ECO Names Tab Contents

Name	Description
Compare Names and Rename Nets and Parts as Necessary	Compare differences using reference designators and net names. i Tip <ul style="list-style-type: none">• Best used to minimize changes to routed traces.• Selecting this option may result in the positional swapping of parts.
Compare Names but Prefer Add/Delete Parts to Renaming	Compare differences using reference designators and net names on the basis that few reference designators have been renamed and nets have not been renamed. i Tip Best used to minimize the positional swapping of parts, and the design disruption that may result.
Compare Topology (not names). Rename as Necessary.	Compare differences without using reference designators or net names. Compare differences using pin names, part type names, and so on.

Table 161. SailWind Layout Link Dialog Box, ECO Names Tab Contents (continued)

Name	Description
	 Tip Best used to compare designs when parts and nets have been renamed, and minimal interconnect changes have been performed. For example, only an auto renumber has been performed on the design.
Disconnect button	Breaks the connection with SailWind Layout and closes the dialog box.

Related Topics

[SailWind Layout Link Dialog Box](#)

SailWind Layout Link Dialog Box, Preferences Tab

To access: Tools > SailWind Layout menu item > connect to SailWind Layout using the [Connect to SailWind Layout Dialog Box](#) > **Preferences** tab

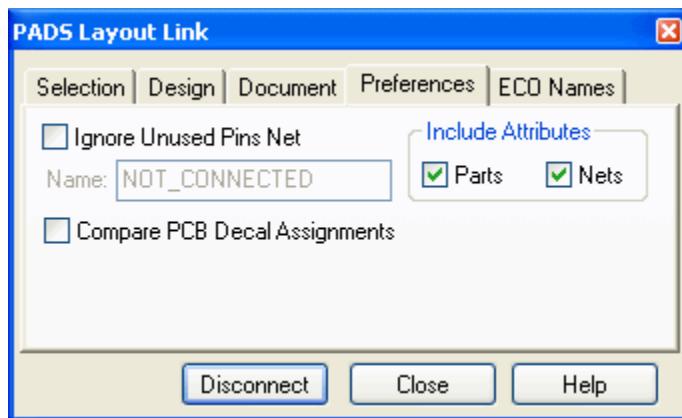
Use the **Preferences** tab of the SailWind Layout Link dialog box to manage options for the compare and ECO functions on the **Design** tab.



Note:

See also [Design tab](#) on page 614.

Figure 102. SailWind Layout Link Dialog Box, Preferences Tab



Objects

Table 162. SailWind Layout Link Dialog Box, Preferences Tab Contents

Name	Description
Ignore Unused Pins Net check box	Select to ignore the unused pins net in the original design and preserve the existing unused pins net in the design you're updating.  Tip <ul style="list-style-type: none">If you clear this option, the unused pins net may be deleted.The unused pins net contains pins that have no logical net association.An unused pins net may be created by other software tools in the PCB design process.
Name box	Type the name of the unused pins net.  Restriction: This box only becomes available if you select the Ignore Unused Pins Net check box.

Table 162. SailWind Layout Link Dialog Box, Preferences Tab Contents (continued)

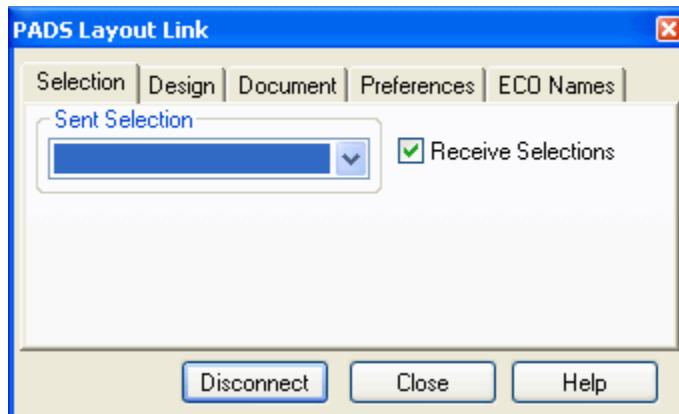
Name	Description
	 Tip <ul style="list-style-type: none"> The maximum netname length is 47 characters. You can use any alphanumeric characters except curly braces { }, asterisks *, or spaces. The default name is NOT_CONNECTED.
Include Attributes area	<p>Parts — Controls whether part attributes are included in the netlist. SailWind Logic evaluates parts that have different attributes as different part types. Therefore, if you select Include Part Attributes when you generate the netlist, you must also select Include Part Attributes when you perform an ECO comparison. Otherwise the comparison considers the part types to be different and reports errors.</p> <p>Nets — Controls whether net attributes are included in the netlist. SailWind Logic checks net names on the schematic against the SailWind Layout design. Therefore, if you select Include Net Attributes when you generate the netlist, you must also select Include Net Attributes when you perform an ECO comparison. Otherwise the comparison considers the net types to be different and reports errors.</p>
Compare PCB Decal Assignments	Compares PCB decal assignments between the schematic and the design in SailWind Layout. Updates decal assignments in SailWind Layout.
Disconnect button	Breaks the connection with SailWind Layout and closes the dialog box.

SailWind Layout Link Dialog Box, Selection Tab

To access: **Tools > SailWind Layout** menu item > connect to SailWind Layout using the [Connect to SailWind Layout Dialog Box](#) > **Selection** tab

Use the **Selection** tab of the SailWind Layout Link dialog box to manage the selections sent to and received from SailWind Layout.

Figure 103. SailWind Layout Link Dialog Box, Selection Tab



Objects

Table 163. SailWind Layout Link Dialog Box, Selection Tab Contents

Name	Description
Sent Selection list	Contains a list of the objects currently selected in SailWind Logic.
Receive Selections check box	Select to enable the SailWind Logic to automatically locate the item corresponding to the SailWind Layout selection by changing schematic sheets and selecting the item in SailWind Logic.
Disconnect button	Breaks the connection with SailWind Layout and closes the dialog box.

Related Topics

[SailWind Layout Link Dialog Box](#)

SailWind Router Link Dialog Box

You can link to and cross-probe with SailWind Router using the SailWind Router Link dialog box. This function enables you to receive and send selections and start a new SailWind Router session or document. Once you connect to SailWind Router, you can select objects in SailWind Router and the object is automatically selected and shown on its SailWind Logic schematic sheet (and vice versa).

See also “[Cross-Probe Between Sailwind Products](#)” on page 324.

SailWind Router Link dialog box tabs:

[SailWind Router Link Dialog Box, Document Tab](#)

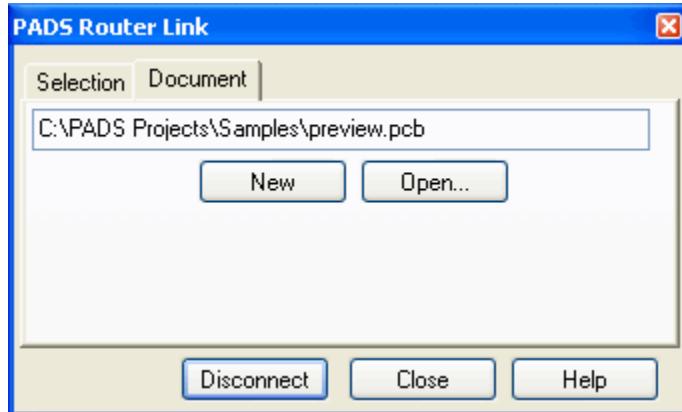
[SailWind Router Link Dialog Box, Selection Tab](#)

SailWind Router Link Dialog Box, Document Tab

To access: **Tools > SailWind Router** menu item > connect to SailWind Router using the [Connect to SailWind Router Dialog Box](#) > **Document** tab

Use the **Document** tab of the SailWind Router Link dialog box to open and connect to a different design or a new design file within the current SailWind Router session. The current design path and filename is listed at the top of this tab.

Figure 104. SailWind Router Link Dialog Box, Document Tab



Objects

Table 164. SailWind Router Link Dialog Box, Document Tab Contents

Name	Description
Pathname field	Displays the path and filename of the current SailWind Router design.
New button	Creates a new file within the current SailWind Router session. SailWind Logic automatically connects to this file. Tip If the current SailWind Router design has unsaved changes, a prompt window appears, asking, "Remote PCB document is modified - Do you want to save it?".
Open button	Enables you to locate and open an existing SailWind Router file. SailWind Logic automatically connects to this file. Tip If the current SailWind Router design has unsaved changes, a prompt window appears, asking, "Remote PCB document is modified - Do you want to save it?".
Disconnect button	Breaks the connection with SailWind Router and closes the dialog box.

Related Topics

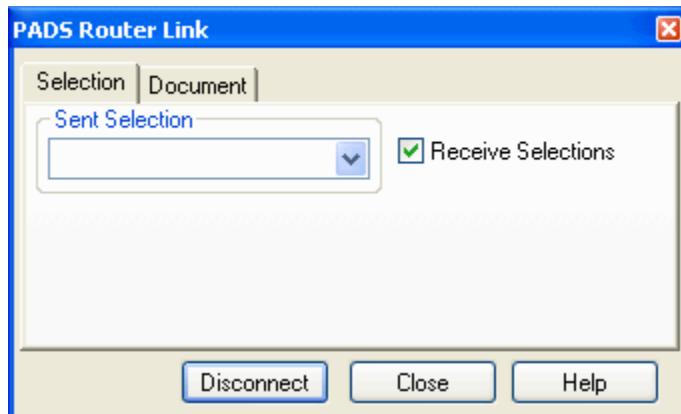
[SailWind Router Link Dialog Box](#)

SailWind Router Link Dialog Box, Selection Tab

To access: **Tools > SailWind Layout** menu item > connect to SailWind Router using the [Connect to SailWind Router Dialog Box](#) > **Selection** tab

Use the **Selection** tab of the SailWind Router Link dialog box to manage the selections sent to and received from SailWind Router.

Figure 105. SailWind Router Link Dialog Box, Selection Tab



Objects

Table 165. SailWind Router Link Dialog Box, Selection Tab Contents

Name	Description
Sent Selection list	Contains a list of the objects currently selected in SailWind Logic.
Receive Selections check box	Select to enable the SailWind Logic to automatically locate the item corresponding to the SailWind Router selection by changing schematic sheets and selecting the item in SailWind Logic.
Disconnect button	Breaks the connection with SailWind Router and closes the dialog box.

Related Topics

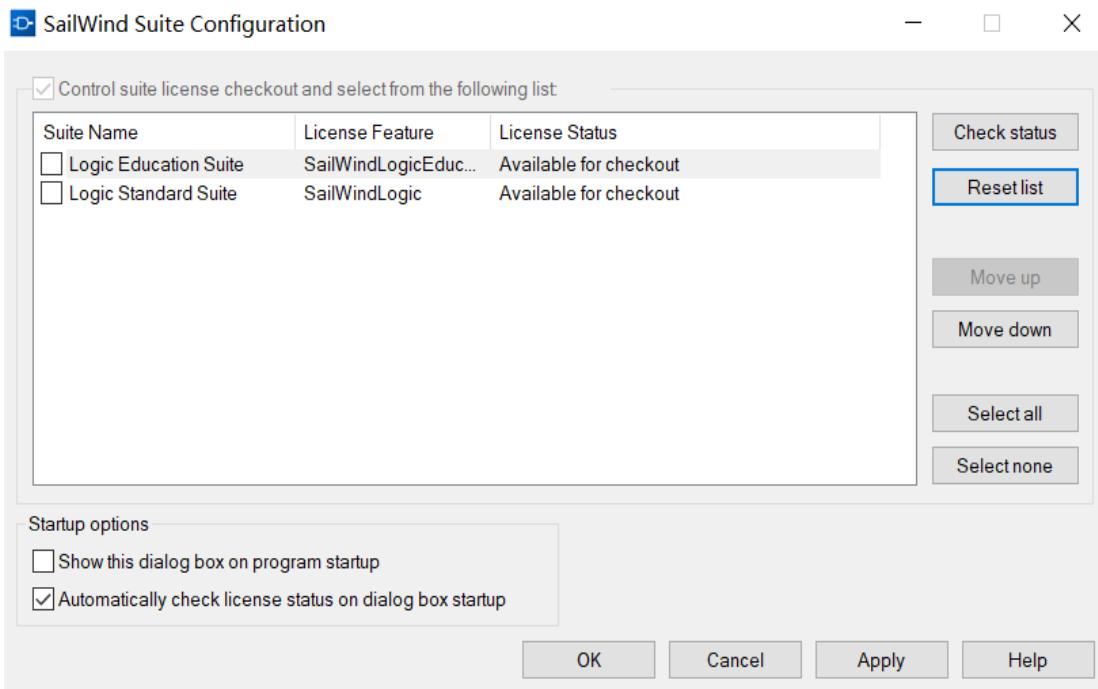
[SailWind Router Link Dialog Box](#)

SailWind Suite Configuration Dialog Box

To access:

- **Help > Installed Options** menu item > **Suite Configuration** button
- Optional: Opens on program startup

Use the SailWind Suite Configuration dialog box to manage SailWind Suite (composite) licenses.



Objects

Field	Description
Control suite license checkout and select from the following list	Enables the Suite License table for you to control checkouts.
Suite Name column	Lists the name of the suite for which the license works.
License Feature column	Lists the specific features available for each license.
License status column	Lists the status of the licenses when you click the Check status button.
Check Status button	Specifies to check the status of all licenses listed in the table and displays the status in the License status column.

Field	Description
Reset List button	Specifies to reset the list of suite licenses to only those detected in your licensing environment.
Move up button	Moves the selected license up one row.
Move down button	Moves the selected license down one row.
Select all button	Selects all of the listed licenses.
Select none button	Deselects all of the listed licenses.
Show this dialog box on program startup	Specifies to open the SailWind Suite Configuration dialog box when SailWind Layout starts.
Automatically check license status on dialog box startup	Specifies to check the status of the licenses when you open the SailWind Suite Configuration dialog box.

Part Attributes Dialog Box

To access: Select a part > right-click > **Attributes** menu item

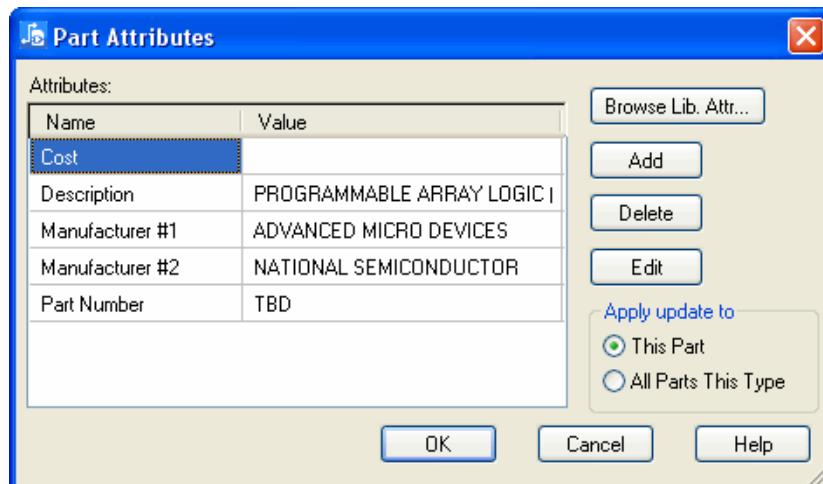
Use the Part Attributes dialog box to assign or modify part attributes, which is information about the part such as manufacturer and cost.



Tip

Adding or changing attributes in a part at the schematic level does not update the part type (in the library). Edit the part in the Part Editor to update the library part.

Figure 106. Part Attributes Dialog Box



Objects

Table 166. Part Attributes Dialog Box Contents

Name	Description
Attributes table	Lists the name and value of the attributes of the selected part.
Browse Lib Attr button	Opens the Browse Library Attributes Dialog Box .
Add button	Adds a row to the end of the Attributes table where you can add a new attribute.
Delete button	Removes the selected row from the Attributes table.
Edit button	Makes the selected cell available for editing.
Apply update to area	Specifies how parts are updated: <ul style="list-style-type: none">• This Part — Only updates the selected part.• All Parts This Type — Updates all matching parts in the design

Related Topics

[Modifying Part Attributes](#)

Part Information Dialog Box

Use the part information dialog box to create or edit part types.

Part Information dialog box tabs:

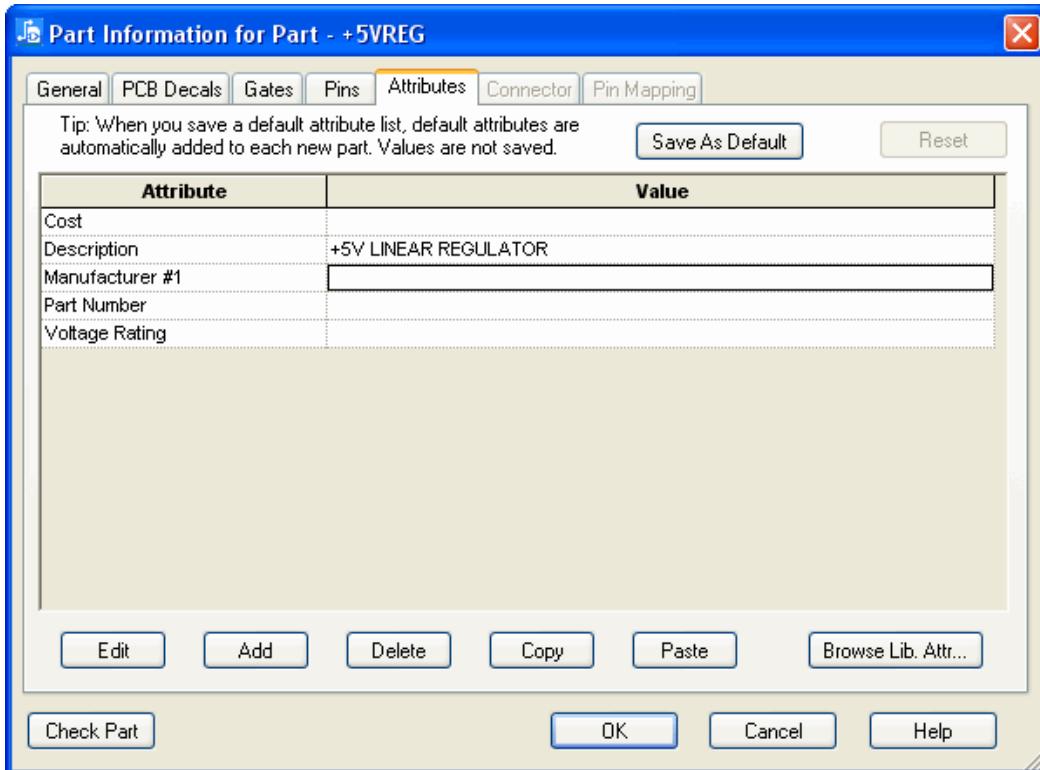
- [Part Information Dialog Box, Attributes Tab](#)
- [Part Information Dialog Box, Connector Tab](#)
- [Part Information Dialog Box, Gates Tab](#)
- [Part Information Dialog Box, General Tab](#)
- [Part Information Dialog Box, PCB Decals Tab](#)
- [Part Information Dialog Box, Pin Mapping Tab](#)
- [Part Information Dialog Box, Pins Tab](#)

Part Information Dialog Box, Attributes Tab

To access: Tools > Part Editor menu item > Edit Electrical button > Attributes tab

Use the **Attributes** tab of the Part Information dialog box to manage attributes for the selected part, and to define default attributes for new parts.

Figure 107. Part Information Dialog Box - Attributes Tab



Objects

Table 167. Part Information Dialog Box - Attributes Tab Contents

Name	Description
Attribute table	<ul style="list-style-type: none"> Attribute column — The name of the Attribute. Value column — The value of the Attribute.
Save As Default button	Saves the current attribute list as the default for all new attributes. i Tip The Attribute name is saved, but not the Attribute value.
Reset button	Undoes all changes you made in the Attributes tab.
Edit button	Makes the selected cell available for editing.

Table 167. Part Information Dialog Box - Attributes Tab Contents (continued)

Name	Description
	<p>Result: The attribute is edited for only the selected part. To manage attributes design-wide or in all libraries, use the Manage Library Attributes Dialog Box.</p> <p>Tip You can also double-click a cell to edit its contents. See also Managing Attributes.</p>
Add button	<p>Adds a new row for a new attribute at the bottom of the table.</p> <p>Result: The new attribute is added to the Part Type Level of the attribute hierarchy. For more information, see Attribute Hierarchy in the <i>SailWind Layout Guide</i>.</p> <p>Tip After you type the new attribute name, press the Tab key or double-click in the value field to type the value.</p>
Delete button	Removes the selected row.
Copy button	<p>Places the selected cell information in the paste buffer.</p> <p>Tip You can also copy from Microsoft Excel.</p>
Paste button	Pastes the information from the paste buffer. The selected cell in the table is the paste origin. Data is pasted below and to the right of the paste origin.
Browse Lib. Attr button	Opens the Browse Library Attributes Dialog Box where you can browse for an existing library attribute.
Check Part button	<p>Checks for missing or inconsistent information.</p> <p>Tip Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.</p>

Related Topics

[Managing Attributes](#)

Part Information Dialog Box, Connector Tab

To access: Tools > Part Editor menu item > Edit Electrical button > Connector tab

Use the **Connector** tab to define the alternate Logic decals to display in a schematic. Decals are referred to as Special Symbols. You can associate a logical Pin Type with each alternate so that you can have a graphical indication of the connector pin function in the schematic.



Restriction:

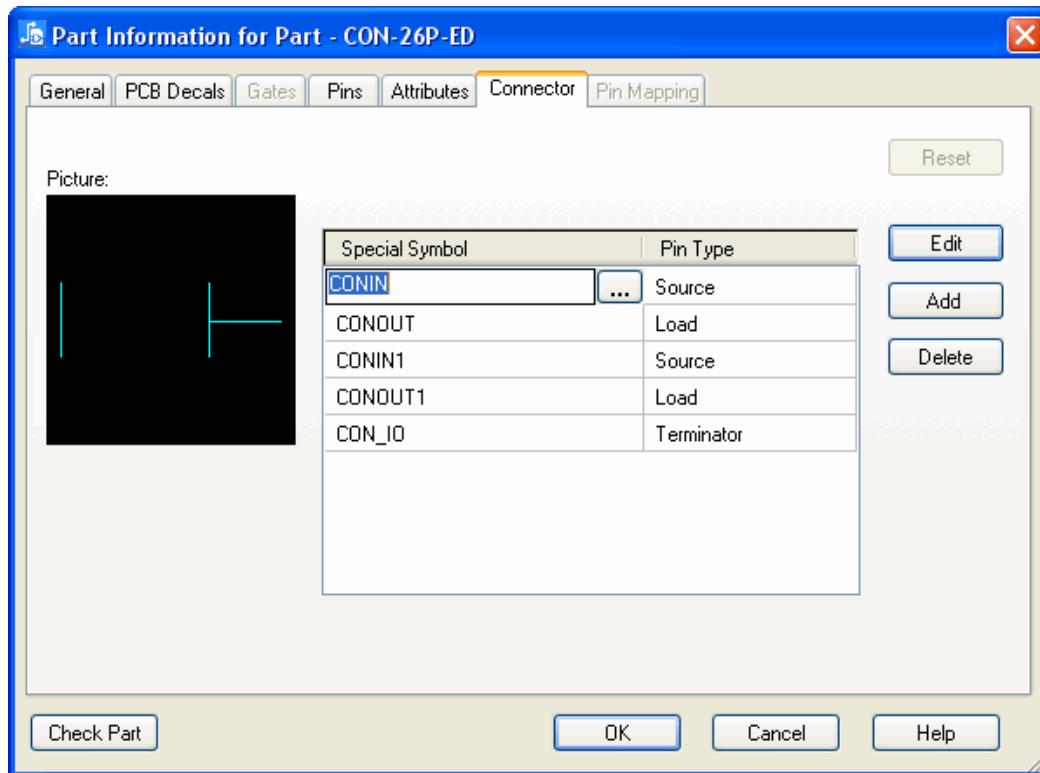
The **Connector** tab is available only when you open an existing connector or create a new connector.



Tip

Many users like to use a different symbol, or decal, to distinguish between input (Source) and output (Load) pins. You may define multiple symbols for each of the ten different pin types.

Figure 108. Part Information Dialog Box - Connector Tab



Objects

Table 168. Part Information Dialog Box - Connector Tab Contents

Name	Description
Picture	Displays a picture of the selected Special Symbol.
Attribute table	<ul style="list-style-type: none">• Special Symbol — The name of a connector pin decal for use in the schematic.• Pin Type — The function of the special symbol.
 ...	Opens the Browse for Special Symbols Dialog Box where you can browse for a pin decal.
Reset button	Undoes all changes you made in the Connector tab.
Edit button	Makes the selected cell available for editing.  Tip You can also double-click the cell to edit the contents.
Add button	Adds a new row at the bottom of the table.
Delete button	Removes the selected row.
Check Part button	Checks for missing or inconsistent information.  Tip Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.

Related Topics

[Assigning Alternate Logic Decals for Connector Symbols](#)

Part Information Dialog Box, Gates Tab

To access: Tools > Part Editor menu item > Edit Electrical button > Gates tab

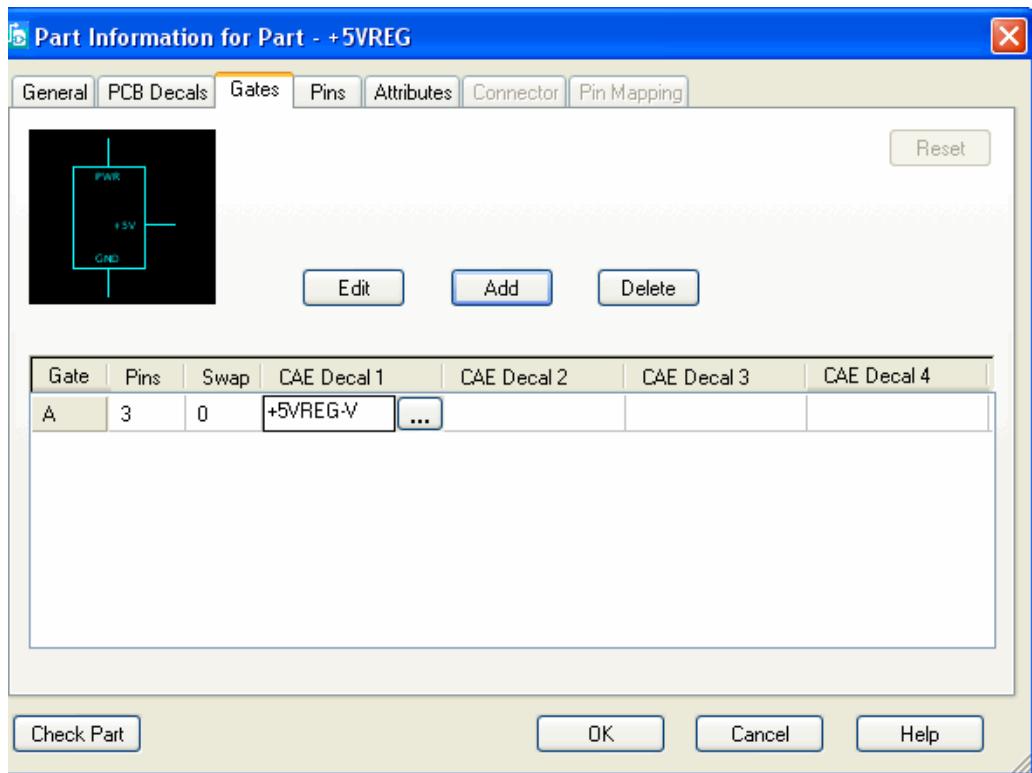
Use the **Gates** tab of the Part Information dialog box to assign gate information to a part, including the number of gates, gate swap information, and CAE Decals for the part.



Tip

A space and a period (.) are illegal characters for pin names.

Figure 109. Part Information Dialog Box - Gates Tab



Objects

Table 169. Part Information Dialog Box - Gates Tab Contents

Name	Description
Preview area	Shows the item selected in the Decal cell.
Reset button	Undoes all changes you made in the Gates tab.
Edit button	Makes the selected cell available for editing. Also displays the Browse button.

Table 169. Part Information Dialog Box - Gates Tab Contents (continued)

Name	Description
	Opens the Assign Decal to Gate Dialog Box . i Tip This button is available only in the CAE Decal columns, and only when the cell is available for editing.
Add button	Adds a new row with the next Gate letter at the bottom of the Gate table.
Delete button	Removes the selected row from the Gate table.
Gate table	<ul style="list-style-type: none"> • Gate column — Displays the letter of the gate. • Pins column — Displays the number of pins for the gate. Gate pins are added on the Pins tab. • Swap column — Displays the swap ID from 0 to 100. To uncross connections and facilitate routing, gates with the same swap ID (except for 0) can be swapped within a part or with another part of the same type. To disable swapping, type 0. • CAE Decal N column — Displays the CAE Decal name. The decal listed for CAE Decal 1 is the default decal and is used when you add the part to the schematic. Additional decals are alternates. You can assign up to four CAE decals to a part. i Tip Double-click to Type a decal name or click the “...” (Browse) button to search for a decal from a library
Check Part button	Checks for missing or inconsistent information. i Tip Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.

Related Topics

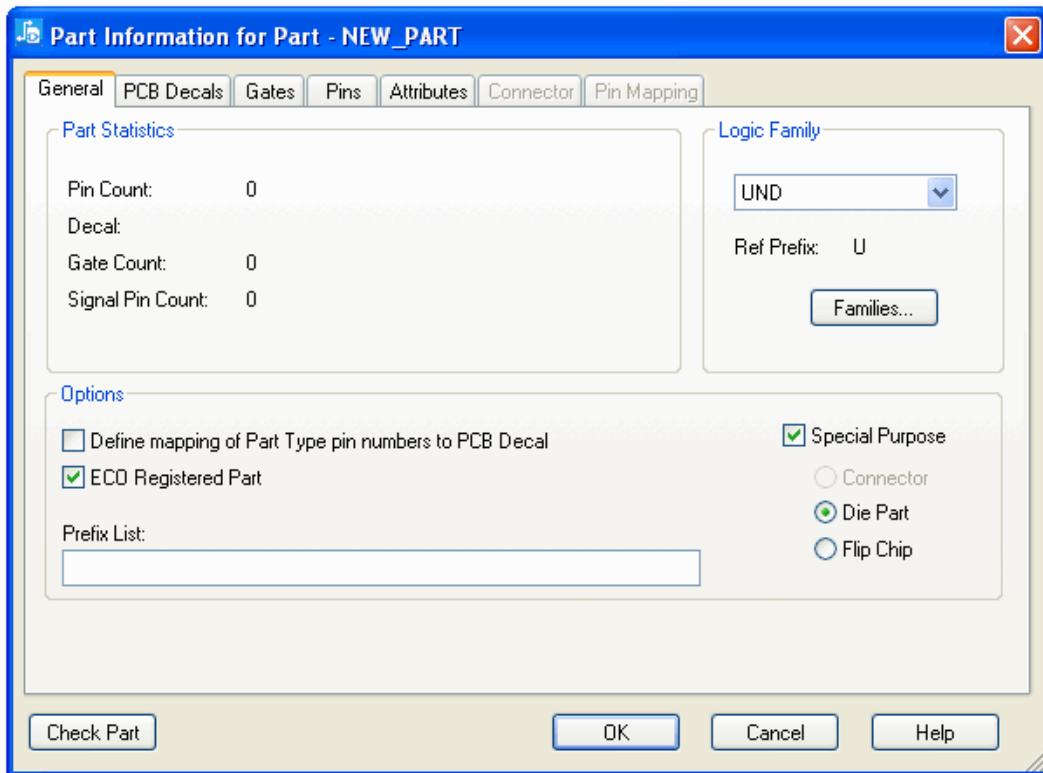
[Assigning Gates to Parts](#)

Part Information Dialog Box, General Tab

To access: Tools > Part Editor menu item > Edit Electrical button > General tab

Use the **General** tab of the Part Information dialog box to view part statistics and set family information.

Figure 110. Part Information Dialog Box - General Tab



Objects

Table 170. Part Information Dialog Box - General Tab Contents

Name	Description
Part Statistics area	
Pin Count	Displays the total number of pins in the part. Includes gate pins, signal pins, and unused pins. If multiple decals are assigned with different pin counts, a range of smallest to largest decal pin counts is shown.
Decal	Displays the name of the decal, as chosen on the PCB Decals tab.
Gate Count	Displays how many gates exist in the part.
Signal Pin Count	Displays the number of signal pins in the part.

Table 170. Part Information Dialog Box - General Tab Contents (continued)

Name	Description
Logic Family area	
Logic Family list	<p>Specifies the Logic Family (reference designator prefix) to use for the part. You can also create a new logic family or edit the existing reference designator prefix designations by clicking the Families... button.</p> <p> Note: Beginning with PADS 9.0, die parts and flip chips are no longer identified by their family designations (DIE or FLP), but instead by the Special Purpose settings on this tab. With this change, you can assign any reference designator (logic family) to a die part or flip chip without losing the special properties of these parts (such as the ability to move the part's substrate bond pads in the schematic).</p>
Ref Prefix	Displays the reference designator prefix for the selected logic family.
Families button	Opens the Logic Families dialog box on page 561 where you can add, edit, or delete a logic family.
Options area	
Define mapping of Part Type pin numbers to PCB Decal	<p>Activates the Pin Mapping tab where you can map logical pin numbers to different physical pin numbers.</p> <p> Restriction:</p> <ul style="list-style-type: none"> The check box is unavailable once you add one or more alphanumeric decals to the part type. Remove the assigned alphanumeric decal to make the check box available. The check box also becomes available if you assign a numeric decal. However, you will still need to remove the alphanumeric decal from the list to make the part valid. You must assign a decal to use the Pin Mapping tab. Only decals with sequential numerical pin numbers can be used with pin mapping.
ECO Registered Part	<p>Enables a part to be passed between the design and schematic file for forward and backward annotation. By default, all existing part types in your design are ECO registered.</p> <p> Tip Typically you do not select this check box for non-electrical parts. For example, if you create and add a mounting hole to your design in the layout software, you would not need the part (mounting hole) to pass back to your schematic when you perform a backward annotation of the design.</p>
Prefix List	Specifies to apply the part information edits to other parts in the library. Type the prefixes and wildcards that match the names of the other parts to update.

Table 170. Part Information Dialog Box - General Tab Contents (continued)

Name	Description
	<p> Note: Examples:</p> <ul style="list-style-type: none"> • Question mark ? in a prefix acts as a wildcard for one character. The prefix "?4" is the equivalent of "54" or "74". • If you type "\02" as the suffix, the edits are applied to all parts ending in 02. <p> Note: Warning: The contents of the Prefix List box are applied when you click OK or Save As on other tabs in the Part Information dialog box.</p>
Special Purpose	<p>Identifies the part as one of the following types:</p> <ul style="list-style-type: none"> • Connector — In contrast to other part types, connectors do not require a prefix list, or gate definitions. <p> Restriction:</p> <ul style="list-style-type: none"> ◦ This check box is automatically selected when you create or modify connectors. It is unavailable if you open a part other than a connector. ◦ This check box is unavailable when the part already has gate or pin assignments. ◦ The Gate Decals tab is unavailable when the Connector check box is selected. ◦ Some Pins tab controls not applicable to connector parts are disabled. <ul style="list-style-type: none"> • Die Part — See the following note. • Flip Chip — See the following note. <p> Note: Beginning with PADS 9.0, die parts and flip chips are identified by the Special Purpose settings in the Part Type rather than by the DIE and FLP logic families. With this change, any reference designator (logic family) can be assigned to a die part or flip chip.</p>
Check Part button	<p>Checks for missing or inconsistent information.</p> <p> Tip Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.</p>

Related Topics

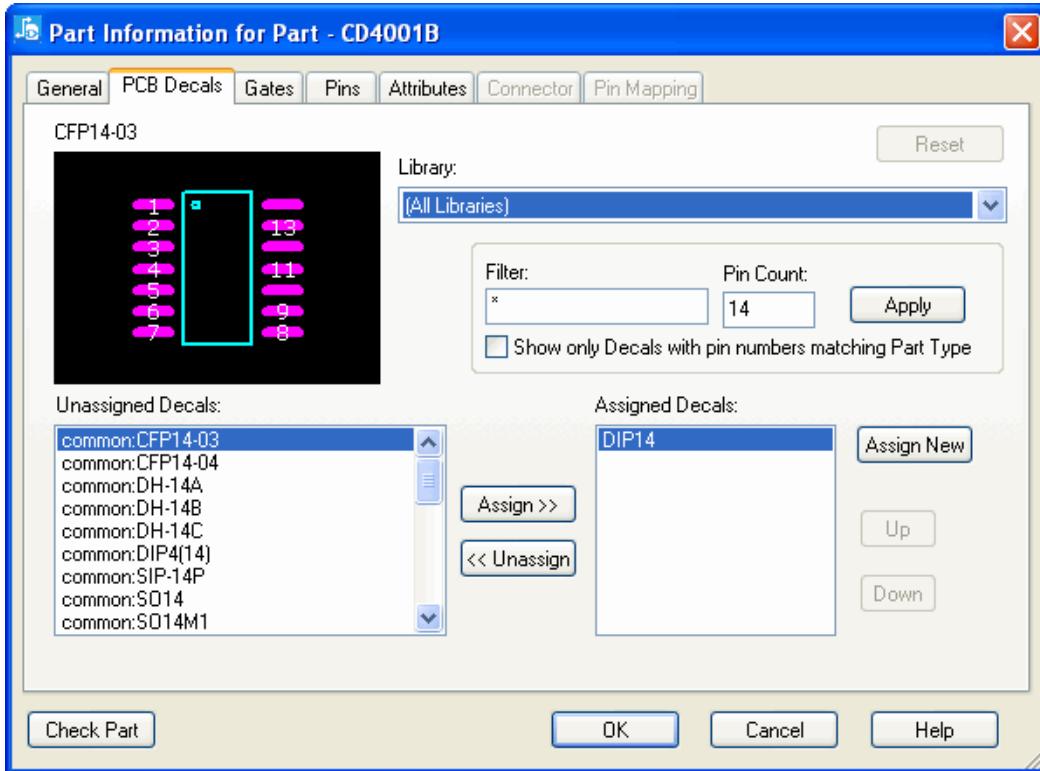
[Viewing and Setting General Part Information](#)

Part Information Dialog Box, PCB Decals Tab

To access: Tools > Part Editor menu item > Edit Electrical button > PCB Decals tab

Use the **PCB Decals** tab of the Part Information dialog box to assign decals, or footprints, to library parts.

Figure 111. Part Information Dialog Box - PCB Decals Tab



Objects

Table 171. Part Information Dialog Box - PCB Decal Tab Contents

Name	Description
Preview area	Displays the item selected in the Assigned Decals list.
Library list	Lists all your available libraries. Filters the Unassigned Decals list to only the selected library.
Filter	Searches the chosen library (or libraries) for a specific part or item name, or names that match a wildcard or expression on page 105. Type * to view all parts or items in the chosen libraries. Click Apply to search the libraries and display the search results.
Pin Count	Narrows down your unassigned decals list by displaying only the decals with the specified number of pins.

Table 171. Part Information Dialog Box - PCB Decal Tab Contents (continued)

Name	Description
	 Tip Delete all numbers in the Pin Count box to display all decals. This box is always available as a filter to enable decals of differing pin counts to be assigned.
Show only Decals with pin numbers matching Part Type	Filters out decals that do not have pin numbers matching existing gate and signal pins on the Pins tab, or the physical pin numbers on the Pin Mapping tab.
Unassigned Decals list	Lists all of the available decals to assign to the part.
Assign >> button	Moves the selected decal from the Unassigned Decals list to the Assigned Decals list.  Restriction: <ul style="list-style-type: none"> You must assign a decal with enough pins for all the defined gate pins and signal pins on the Pins tab. Only decals with sequential numerical pin numbers can be used with pin mapping.
<< Unassign button	Moves the selected decal from the Assigned Decals list to the Unassigned Decals list.
Assigned Decals list	Lists all assigned decals. Assigned PCB decals can have a different number of pins, but you must assign a decal with enough pins for all the defined gate pins and signal pins on the Pins tab. You can assign up to 16 PCB decals to a part.  CAUTION: Decals are switched to alternates using the Component Properties dialog box and can be changed outside of ECO mode. An .eco file created by the ECO toolbar will not contain decal changes to alternates. Use the Compare/ECO dialog box to create an .eco file that lists changes to alternate decals.
Assign New button	Opens the New PCB Decal dialog box on page 469 where you can type the name for a decal that does not yet exist in a library. To use this part in a design, you must either acquire or create this decal.  Restriction: When you assign a decal that exists, it pre-populates the Pins tab on page 645 with pin numbers. When you assign a new PCB decal, you must enter the pin numbers manually.
Up/Down buttons	Moves the selected Decal up or down.

Table 171. Part Information Dialog Box - PCB Decal Tab Contents (continued)

Name	Description
Check Part button	Checks for missing or inconsistent information.  Tip Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.

Related Topics

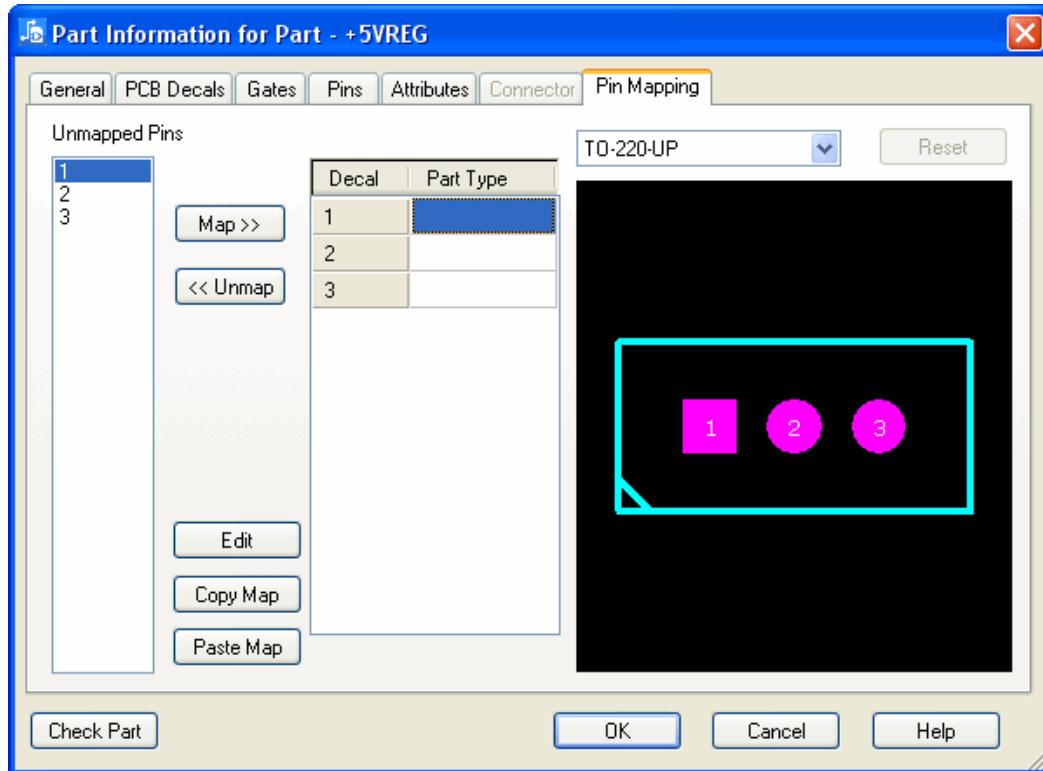
[Assigning PCB Decals](#)

Part Information Dialog Box, Pin Mapping Tab

To access: **Tools > Part Editor** menu item > **Edit Electrical** button > **General** tab. On the **General** tab, select the “Define mapping of Part Type pin numbers to PCB Decal” check box to make the **Pin Mapping** tab available. On the **PCB Decals** tab, assign a decal with sequential numerical pin numbers to use the **Pin Mapping** tab. The decal determines the number of pins in the part.

Use the **Pin Mapping** tab of the Part Information dialog box to overlay alphanumeric pin numbers onto numeric PCB decal pins. Prior to PADS 2007, alphanumeric pin numbers could not be saved in PCB decals.

Figure 112. Part Information Dialog Box - Pin Mapping Tab



Objects

Table 172. Part Information Dialog Box - Pin Mapping Tab Contents

Name	Description
Decal list	Lists the decals available to you for which you can map alphanumeric pins.
Reset button	Undoes all changes you made in the Pin Mapping tab.
Unmapped Pins list	Lists all unmapped pins available to map in the Mapping table.
Map >> button	Moves the selected pin from the Unmapped pins list to the selected cell in the Mapping table.

Table 172. Part Information Dialog Box - Pin Mapping Tab Contents (continued)

Name	Description
<< Unmap button	Moves the selected decal number from the Mapping table to the Unmapped Pins list.
Mapping table	<ul style="list-style-type: none"> Decal column — The number of the Decal. Part Type column — The value of the Attribute.
Edit button	Makes the selected cell available for editing.  Tip You can also double-click the cell to edit the contents.
Copy Map button	Places the map information into the paste buffer to paste into Microsoft Excel where you can make mass edits.  Restriction: Copy Map only works with the whole pin mapping table and not selective rows.
Paste Map button	Pastes the map information from the paste buffer.  Restriction: Paste Map only works with the whole pin mapping table and not selective rows.
Preview area	Shows the item selected in the Decal list. You can assign unmapped pins to decal pins by selecting the pins in the preview window. Select an alphanumeric in the Unmapped Pins list and double-click the pin in the decal preview window to map the alphanumeric to the pin. The next row in the Unmapped Pins list becomes the next selected alphanumeric for mapping. In the preview window, you can click and drag to define a zoom box, or use Shift+click or Shift+right-click to zoom in or out by a factor of two. You can zoom in up to 16X the original scale. The preview window will only zoom out to fit the decal entirely in the view.
Check Part button	Checks for missing or inconsistent information.  Tip Even if you do not click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.

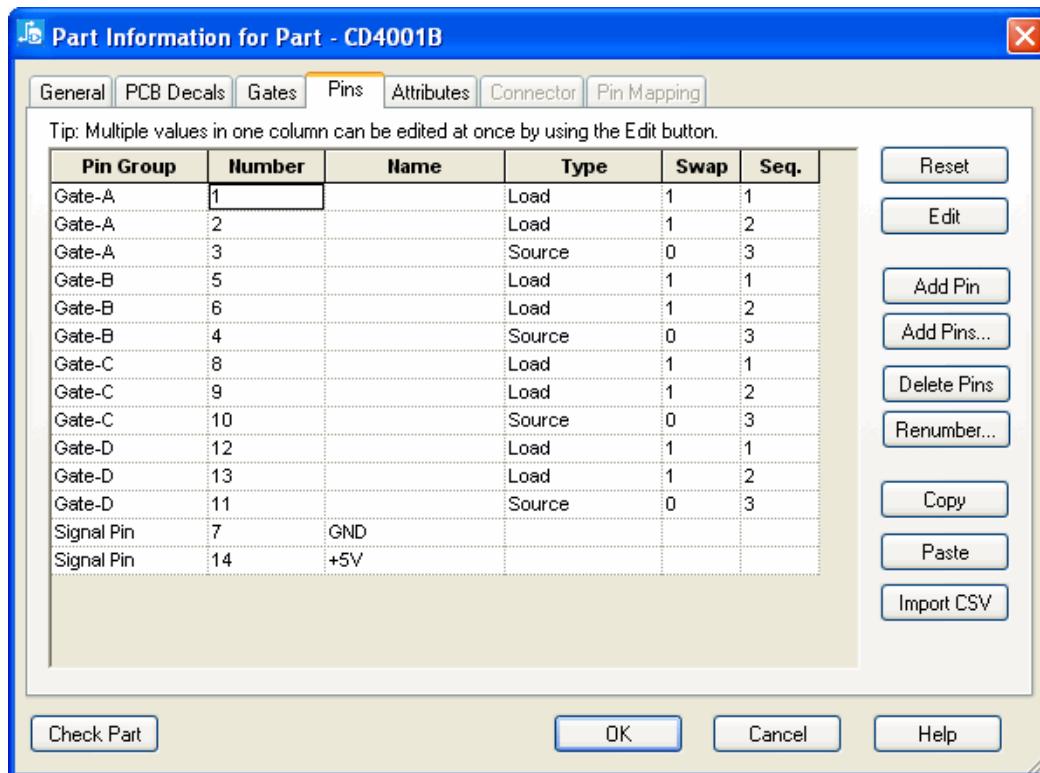
Related Topics

[Part Information - Pin Mapping](#)

Part Information Dialog Box, Pins Tab

To access: Tools > Part Editor menu item > Edit Electrical button > Pins tab

Use the **Pins** tab of the Part Information dialog box to assign gate pins, signal pins, and unused pins to the part. Pin numbers added to the **Pins** tab, must match those of the PCB Decal. Use the **Pin Mapping** tab to overlay logical (schematic) alphanumeric pin numbers onto the physical numeric PCB decal.



Objects

Name	Description
Pins Table	<p>Double-click a column to sort the column by ascending order.</p> <p>Tip Sorting by pin Sequence number or Pin Group has the same effect, it sorts by Pin Group and then by sequence number within each gate.</p>
Pin Group column	<p>Select from Gate, Signal Pin, Connector Pin or Unused Pin.</p> <ul style="list-style-type: none"> • Gate Pins — Assign to gates you've added to the part on the Gates tab on page 635. • Signal Pins — Assign to implicit pins (pins which are not displayed on any gate in the schematic). Typically, ground and power pins are the only implicit pins. You are not required to use Signal Pins. Instead, you can add power and ground pins to a gate or create a separate gate for power and/or ground

Name	Description
	<p>pins. For the parts in the libraries shipped with SailWind Logic, the standard ground signal name is GND. The standard power signal name is +5V.</p> <ul style="list-style-type: none"> • Connector Pins — Assign to connector pins instead of using Gate pins. With a connector, every pin is its own gate in order to spread each connector pin throughout the schematic as needed, instead of having to create gates for each pin. You designate a part type as a connector on the General tab on page 637. • Unused Pins — You can assign a pin to be an unused pin. An unused pin is a pin that is defined in a PCB decal but has no electrical function in the part type. The unused pin information is not saved in the part type, but is derived automatically based on the number of assigned gate and signal pins to the number of pins in the assigned PCB decal.
Number column	<p>Specifies the pin number for the pin.</p> <p> Note: Requirement: The pin number must match the PCB decal. For example, alphanumeric to alphanumeric.</p>
Name column	<p>Specifies the pin signal or function name of the pin.</p> <p> Note: Requirement: The pin must have a name to be valid.</p>
Type column	<p>Specifies the type of pin.</p> <p> Tip This column is used with gate pins only.</p>
Swap column	<p>Specifies an identical number between pins that can be swapped.</p> <p> Tip Type 0 to disable swapping.</p>
Seq. column	<p>Specifies the sequence number.</p> <p> Tip The sequence number determines the mapping of CAE gate pins to PCB decal pins. The sequence is automatically shared with alternate CAE decals. For example, it shows how pin numbers appear on the CAE gate decal; therefore, in Gate A, sequence number 1 could be pin 1, but in Gate B, sequence number 1 would be pin 4.</p>
Reset button	Undoes all changes you made in the Pins tab.
Edit button	<p>Makes the selected cell available for editing. Press Ctrl or Shift and click to select multiple cells from the same column, then click Edit to make the same changes to the selected cells. The action of the Edit button depends on the cells you select:</p> <ul style="list-style-type: none"> • Pin Group column — Opens the Update Pin Gate dialog box. • Number column — Opens the Renumber Pins dialog box.

Name	Description
	<ul style="list-style-type: none"> • Name column — Opens the Update Pin Name dialog box. • Type column — Opens the Update Pin Type dialog box. • Swap column — Opens the Update Pin Swap dialog box. • Seq. column — Not available for multiple cell edits.
Add Pin button	<p>Adds a new row below the selected row. If it's the first pin to be added it takes the default of belonging to Gate-A. If pins already exist, the new pin takes the Pin Group of the currently selected pin.</p> <p> Note: Requirement: You must add a pin number to make the pin valid.</p> <p> Tip <ul style="list-style-type: none"> • You can add all pins automatically by adding a decal. • To add a series of pins, click the Add Pins button. • To import pins using a comma separated value (.csv) file, click the Import CSV button. </p>
Add Pins button	<p>Opens the Add Pins Dialog Box.</p> <p> Note: Restriction: Total pins for the part can not exceed 32,767 pins.</p> <p> Tip <ul style="list-style-type: none"> • You can add all pins automatically by adding a decal. • To add a single pin, click the Add Pin button. • To import pins using a comma separated value (.csv) file, click the Import CSV button. </p>
Delete Pins button	Removes the selected row.
Renumber button	Opens the Renumber Pins Dialog Box .
Copy button	<p>Places the selected cell information in the paste buffer.</p> <p> Tip You can also copy from Microsoft Excel.</p>
Paste button	<p>Pastes the information from the paste buffer. The selected cell in the table is the paste origin. Data is pasted below and to the right of the paste origin.</p> <p> Restriction: When the pasted data includes either Pin Group or Pin Number data, extra pin rows are added automatically, otherwise the paste will fail if the number of rows and columns in the pasted data does not match those available in the table below and to the right of the paste origin.</p>
Import CSV button	Opens the Library Import File dialog box where you select the .csv file to import.

Name	Description
	<p> Tip</p> <ul style="list-style-type: none">• The entire contents of the Pins tab table is replaced with the data of the CSV file.• CSV field names must correspond to the column headers in the Pins tab table. Only the first two characters of the header must match. For example, "Pi" for the Pin Group column. Gate or "Ga" are acceptable alternatives to the Pin Group header.• The sample <i>Part_Pins_Template.csv</i> file is located in your <i>\SailWind Projects\Samples</i> folder.
Check Part button	<p>Checks for missing or inconsistent information.</p> <p> Tip</p> <p>Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.</p>

Related Topics

[Part Information - Pins](#)

Part Properties Dialog Box

To access: Select a part > right-click > **Properties** menu item

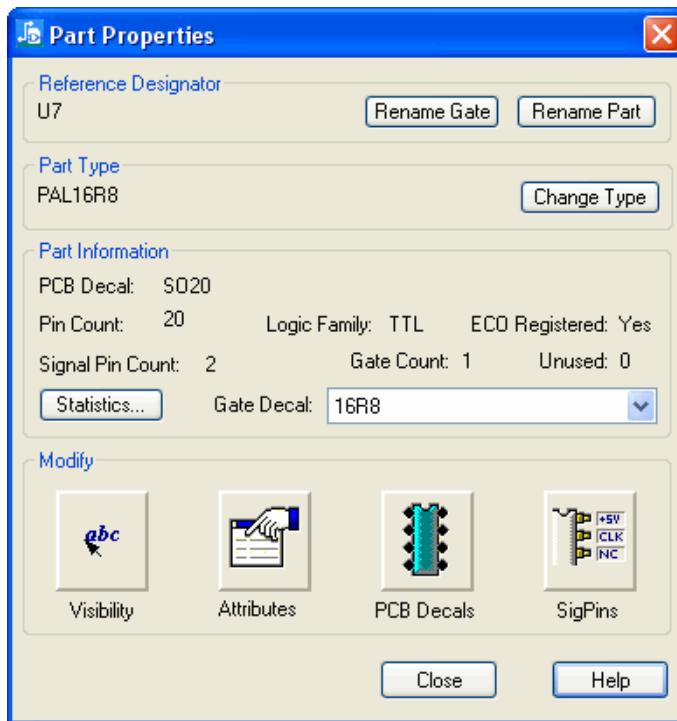
Use the Part Properties dialog box to create and edit part attributes. You can also define signal pins and control the visibility of attributes assigned to the part.



Tip

This dialog box contains several sub-dialog boxes. Before using any of these sub-dialog boxes, changes made in open dialog box must be applied to the design.

Figure 113. Part Properties Dialog Box



Objects

Table 173. Part Properties Dialog Box Contents

Name	Description
Reference Designator	Displays the name of the reference of the selected part.
Rename Gate button	Opens the Rename Gate Dialog Box .
Rename Part button	Opens the Rename Part Dialog Box .
Part Type	Displays the type of the selected part.

Table 173. Part Properties Dialog Box Contents (continued)

Name	Description
Change Type button	Opens the Change Part Type Dialog Box .
Part Information area	Displays the PCB Decal, pin count, Logic Family, if the part is ECO registered, the signal pin count, the gate count, and the number of unused pins.
Statistics button	Displays gate and connection information for the selected part. This information is displayed in the default text editor so you can save the contents to a file.
Gate Decal list	Specifies the gate decal. Select a gate decal name from the Gate Decal list to change the gate decal of the selected gate or part to one of the predefined alternate decals
Visibility button	Opens the Part Text Visibility Dialog Box .
Attributes button	Opens the Part Attributes Dialog Box .
PCB Decals button	Opens the PCB Decal Assignment Dialog Box .
SigPins button	Opens the Part Signal Pins Dialog Box .

Related Topics

[Modifying Parts](#)

Part Signal Pins Dialog Box

To access: Select a part > right-click > **Properties** > **SigPins** button

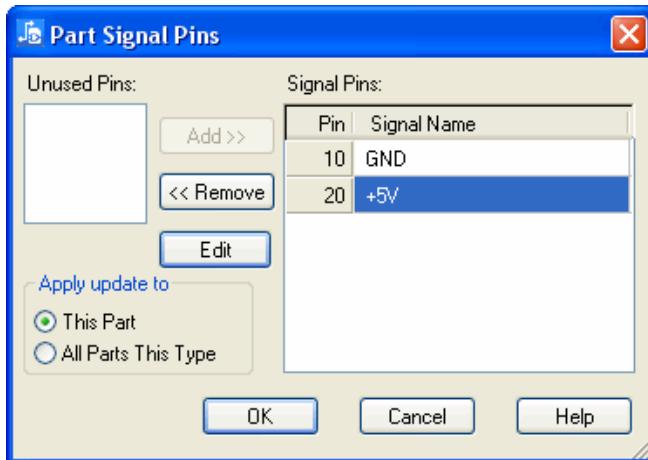
Use the Part Signal Pins dialog box to assign any unused pins as additional signal pins. When the part type is created and stored in the library, the standard power and ground pins for part types are defined. Signal pins assigned during part creation cannot be modified through this dialog box. Instead, use the Assigning Signal Pin Names to Parts dialog box in the Library Manager.



Tip

The Part Signal Pins dialog box lists pins and their corresponding signal names. A signal pin is a pin that has a signal net (GND for example) assigned by a schematic capture program during part type creation.

Figure 114. Part Signal Pins Dialog Box



Objects

Table 174. Part Signal Pins Dialog Box Contents

Name	Description
Unused Pins list	Lists all of the unused pins for the selected part.
Add >> button	Adds the selected unused pin to the Signal Pins table
<< Remove button	Removes the selected Signal Pin to the Unused Pins list.
Edit button	Makes the selected cell available for editing.
Signal Pins table	Lists the signal pin number and corresponding name.
Apply update to area	Specifies how parts are updated:

Table 174. Part Signal Pins Dialog Box Contents (continued)

Name	Description
	<ul style="list-style-type: none">• This Part — Only updates the selected part.• All Parts This Type — Updates all matching parts in the design

Related Topics

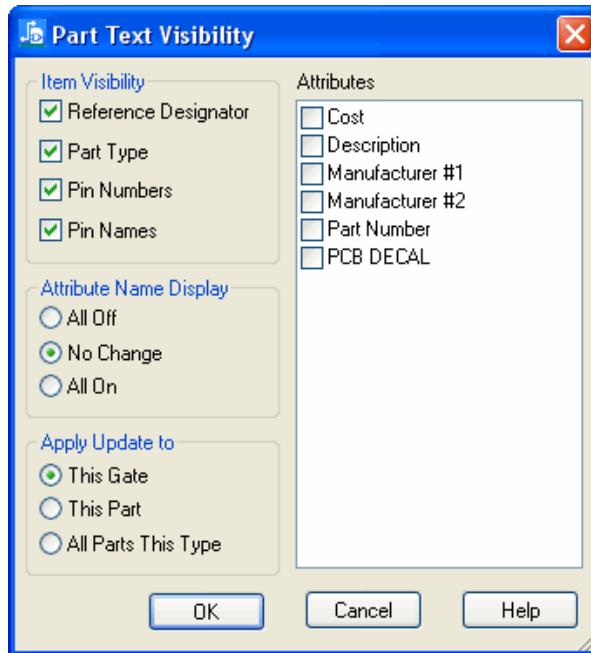
[Modifying Parts](#)

Part Text Visibility Dialog Box

To access: Select a part > right-click > **Visibility** menu item

Use the Part Text Visibility dialog box to control the display of text associated with the selected part. You can control the visibility of one part or all parts of the same type.

Figure 115. Part Text Visibility Dialog Box



Objects

Table 175. Part Text Visibility Dialog Box Contents

Name	Description
Item Visibility area	Specifies the visibility of the items: click to make the item visible; click to clear to make the item invisible.
Attribute Name Display area	Specifies the attribute name option. Display just the attribute's value or display the attribute name and its value. <ul style="list-style-type: none">• All Off — Makes all attribute names invisible, displays only the value.• No Change — Keeps the current attribute name visibility settings.• All On — Displays all attribute names and their values.
Apply Update to area	Specifies the selected part update options.

Table 175. Part Text Visibility Dialog Box Contents (continued)

Name	Description
	<ul style="list-style-type: none">• This Gate — Updates the selected gate.• This Part — Updates a part or all associated gates of a part.• All Part This Type — Updates all matching gates or parts in the design. <p> Restriction: This area is unavailable when more than one part is selected.</p>
Attributes list	Specifies the attributes you want to display in the schematic.

Related Topics

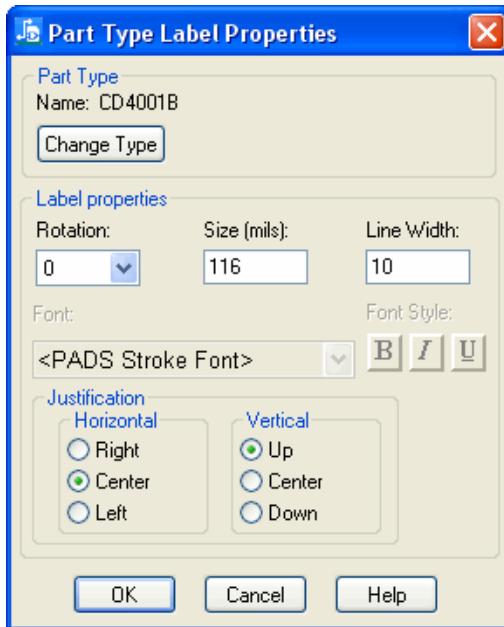
[Controlling Text Visibility for a Part](#)

Part Type Label Properties Dialog Box

To access: Select a part type label > right-click > **Properties** menu item

Use the Part Type Label Properties dialog box to provide text and font settings for one or more part type labels.

Figure 116. Part Type Label Properties Dialog Box



Objects

Table 176. Part Type Label Properties Dialog Box Contents

Name	Description
Name	The name of the selected attribute.
Change Type button	Opens the Change Part Type Dialog Box .
Rotation	Specifies the rotation of the label: 0 or 90 degrees.
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.

Table 176. Part Type Label Properties Dialog Box Contents (continued)

Name	Description
	 Stroke Font - Size
Line Width	Specifies the line width for stroke fonts only.  Stroke Line Width
Font list	The fonts available to you. This lists either stroke fonts or system fonts. You choose which type of font to use in the Fonts Dialog Box . i Tip <ul style="list-style-type: none">• Select stroke font or a system font.• For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

[Modifying Part Type Labels](#)

Pen Plotter Advanced Setup Dialog Box

To access: **File > Plot** menu item > select Pen Plot > **Setup** button > **Advanced** button

Use the Pen Plotter Advanced Setup dialog box to add a new pen plotter to the list of available plotters.

Figure 117. Pen Plotter Advanced Setup Dialog Box



Objects

Table 177. Pen Plotter Advanced Setup Dialog Box Contents

Name	Description
Plotter Device Name	Specifies the name of a different pen plotter you want to use.  Note: Exception: Do not reuse one of the existing, supplied device names.
Device Type list	Specifies the interface language the plotter uses: HPGL or HGML.
Plotter Units area	Specifies the plotter resolution as a scaling ratio using the numbers in the Multiplier and Divisor boxes. The ratio defined is the scale factor to convert from mils (0.001 in) to plotter units.  Note: Example: Most Hewlett-Packard plotters have a resolution of 0.025 mm or 1/40 mm. This means that a distance of one inch (1000 mils) is 1016 plotter units (25.4 X 40). So a ratio of 1016 to 1000 would be defined. The ratio actually used is 254 to 250 which is the same as 1016 to 1000.
Origin at Center	Specifies that the origin of the plotter is at the center of the paper.  Tip Clear this check box if the origin is in the lower left corner or other location.

Related Topics

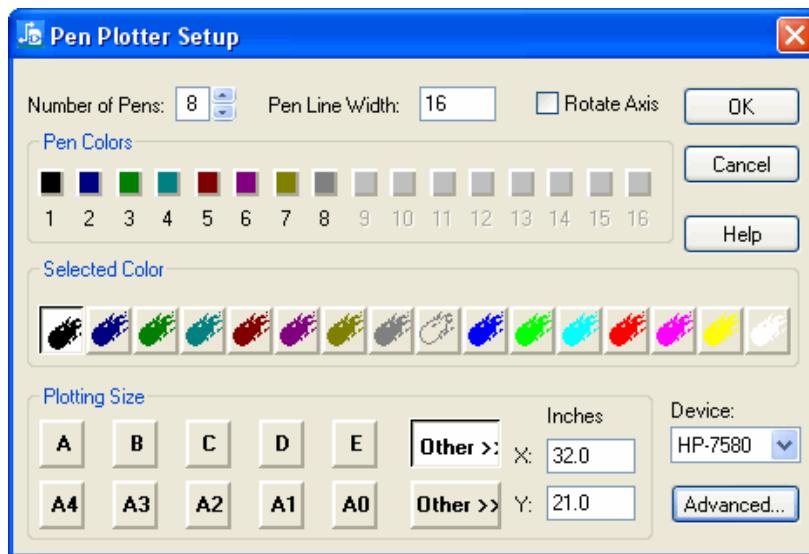
[Pen Plotter Setup Dialog Box](#)

Pen Plotter Setup Dialog Box

To access: **File > Plot** menu item > select Pen Plot > **Setup** button

Use the Pen Plotter Setup dialog box to set various options for the plotter.

Figure 118. Pen Plotter Setup Dialog Box



Objects

Table 178. Pen Plotter Setup Dialog Box Contents

Name	Description
Number of Pens	Specifies the number of pens (1-16) in your device.
Pen Line Width	Specifies the pen line width in mils.
Rotate Axis	Specifies to reverse the X and Y axes of the design.
Pen Colors	Specifies the color of each pen. Tip Select the color in the Selected Color area and then click the tile for each pen number in the Pen Colors area.
Selected Color	The area where you select the color you want for each pen color.
Plotting size area	Specifies the size of the plot. A through E and Other in inches; A4 through A0 and Other in millimeters. Tip If you click Other, specify the X and Y dimensions.
Device	Specifies the Plotter device to use.

Table 178. Pen Plotter Setup Dialog Box Contents (continued)

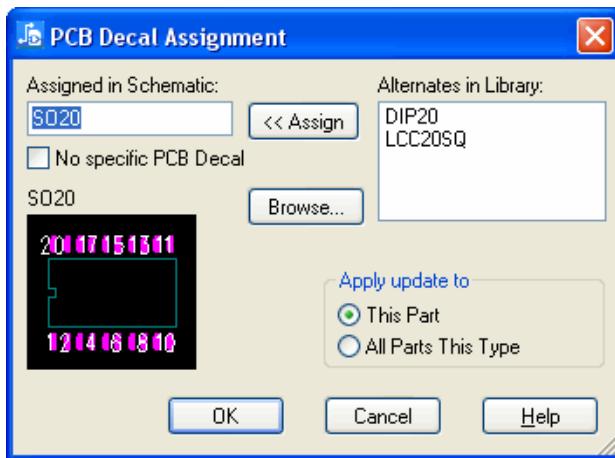
Name	Description
Advanced button	Opens the Pen Plotter Advanced Setup Dialog Box .

PCB Decal Assignment Dialog Box

To access: Select a part > right-click > **Properties** > **PCB Decals** button

Use the PCB Decal Assignment dialog box to assign alternate PCB decals to schematic parts. Decal names are included in the netlist file to display the proper decal, or footprint, when the file is imported into the PCB design file. You can select a decal assigned as an alternate during part creation, override the decal with one from the library, or enter a name for an undefined decal you plan to create later in the PCB design.

Figure 119. PCB Decal Assignment Dialog Box



Objects

Table 179. PCB Decal Assignment Dialog Box Contents

Name	Description
Assigned in Schematic	Displays the name of the currently selected decal as it is assigned to the schematic from the current library.
<< Assign button	Assigns the selected decal in the Alternate in Library list to the part.
Alternates in Library list	Lists alternate decals in the library.
No specific PCB Decal	Specifies to remove the assigned decal. The default decal assigned to the part type is used when the netlist is imported to SailWind Layout; no decal assignment appears in the netlist.
Browse button	Opens the Get PCB Decal From Library Dialog Box .
Picture area	Displays the selected pin.
Apply update to area	Specifies how parts are updated:

Table 179. PCB Decal Assignment Dialog Box Contents (continued)

Name	Description
	<ul style="list-style-type: none">• This Part — Only updates the selected part.• All Parts This Type — Updates all matching parts in the design

Related Topics

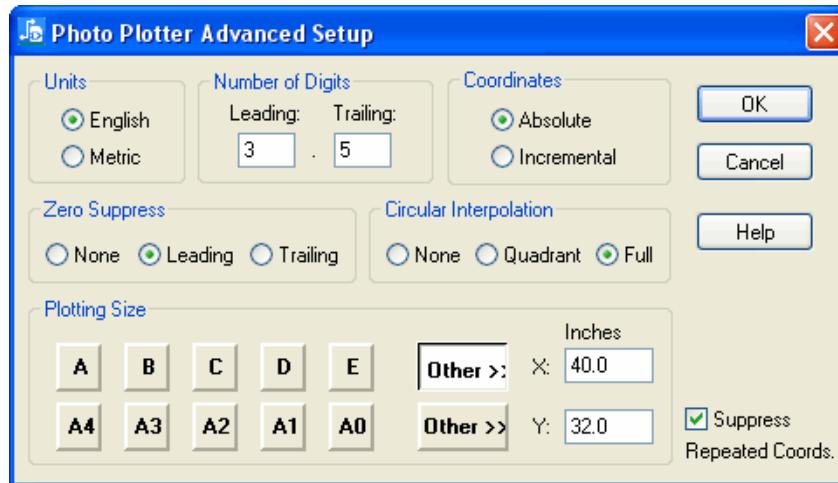
[Modifying Parts](#)

Photo Plotter Advanced Setup Dialog Box

To access: **File > Plot** menu item > select Photo Plot > **Setup** button > **Advanced** button

Use the Photo Plotter Setup dialog box to define the aperture and other photo plotter options.

Figure 120. Photo Plotter Advanced Setup Dialog Box



Objects

Table 180. Photo Plotter Advanced Setup Dialog Box Contents

Name	Description
Units area	Specifies the units to use: <ul style="list-style-type: none">English — milsMetric — millimeters
Number of Digits area	Specifies the precision of the output file coordinates: <ul style="list-style-type: none">Leading — the number of digits that should lead the decimal pointTrailing — the number of digits that should trail the decimal point
Coordinates area	Specifies the coordinates for the output file: <ul style="list-style-type: none">Absolute — absolute coordinatesIncremental — relative coordinates
Zero Suppress area	Specifies how to handle zero suppression in the output file: <ul style="list-style-type: none">None — retains leading and trailing zerosLeading — suppresses zeros before the decimal pointTrailing — suppresses zeros after the decimal point

Table 180. Photo Plotter Advanced Setup Dialog Box Contents (continued)

Name	Description
Circular Interpolation area	Specifies how to draw arcs and circles: <ul style="list-style-type: none">• None — if your photo plotter does not support circular interpolation. Arcs and circles are drawn as small straight-line segments• Quadrant — if your photo plotter does not support full, 360-degree circular interpolation• Full — if your photo plotter supports full, 360-degree circular interpolation
Plotting Size area	Specifies the size of the plot. A through E and Other in inches; A4 through A0 and Other in millimeters.  Tip If you click Other , specify the X and Y dimensions.
Suppress Repeated Coords	Specifies to eliminate repeated coordinates from the output file.

Related Topics

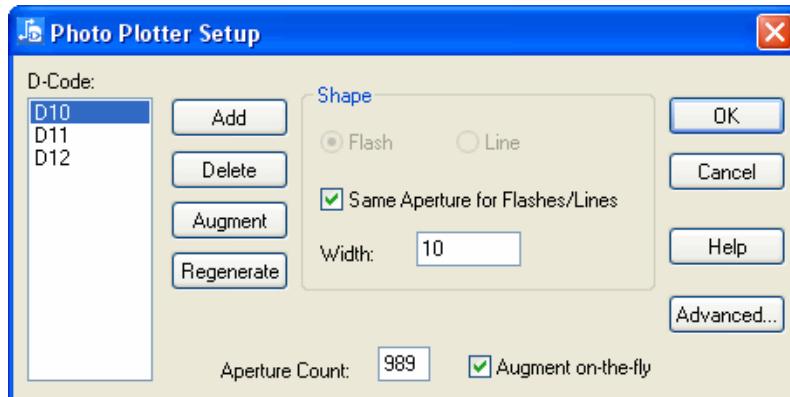
[Photo Plotter Setup Dialog Box](#)

Photo Plotter Setup Dialog Box

To access: **File > Plot** menu item > select Photo Plot > **Setup** button

Use the Photo Plotter Advanced Setup dialog box to define the units for the photo plotter to use.

Figure 121. Photo Plotter Setup Dialog Box



Objects

Table 181. Photo Plotter Setup Dialog Box Contents

Name	Description
D-Code list	Lists the D-codes for the photo plotter.
Add button	Opens the CAM Question dialog box where you can name the new D-Code.
Delete button	Removes the selected code from the D-Code list.
Augment button	Automatically generates D-codes for items in the schematic file that are not in the current list.
Regenerate button	Clears the current D-code list and automatically adds D-codes for all items in the schematic.
Shape area	Specifies the shape for a selected D-code: <ul style="list-style-type: none">• Flash — sets a flash aperture.• Line — set a draw aperture.
Same Aperture for Flashes/Lines	Specifies to draw lines and flashed items with the same aperture. Round and square shapes for lines will be gray.
Width	Specifies the diameter for round shapes. This box is unavailable if a width is not appropriate for the specified shape.
Aperture Count	Specifies the maximum aperture count.

Table 181. Photo Plotter Setup Dialog Box Contents (continued)

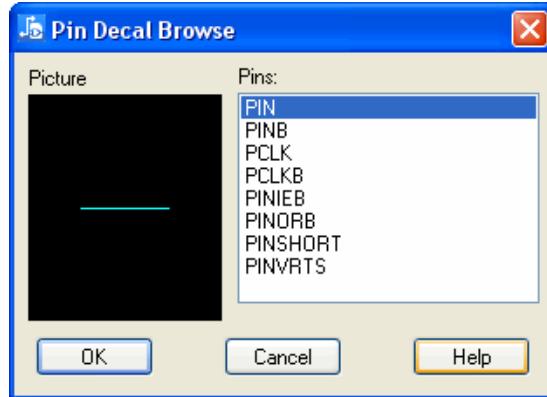
Name	Description
Augment on-the-fly	Specifies to add apertures to the D-code list when photo plots are run if any newly created lines were added to the schematic.
Advanced button	Opens the Photo Plotter Advanced Setup Dialog Box .

Pin Decal Browse Dialog Box

To access: Tools > Part Editor menu item > Decal Editing Toolbar button> Add Terminal or Change Pin Decal button

Use the Pin Decal Browse dialog box to add or change a terminal. A terminal consists of a pin decal and a series of text strings that define the terminal's number, swap data, etc.

Figure 122. Pin Decal Browse Dialog Box



Objects

Table 182. Pin Decal Browse Dialog Box Contents

Name	Description
Picture area	Displays the selected pin.
Pins list	Displays the available pins.

Related Topics

[Adding Terminals](#)

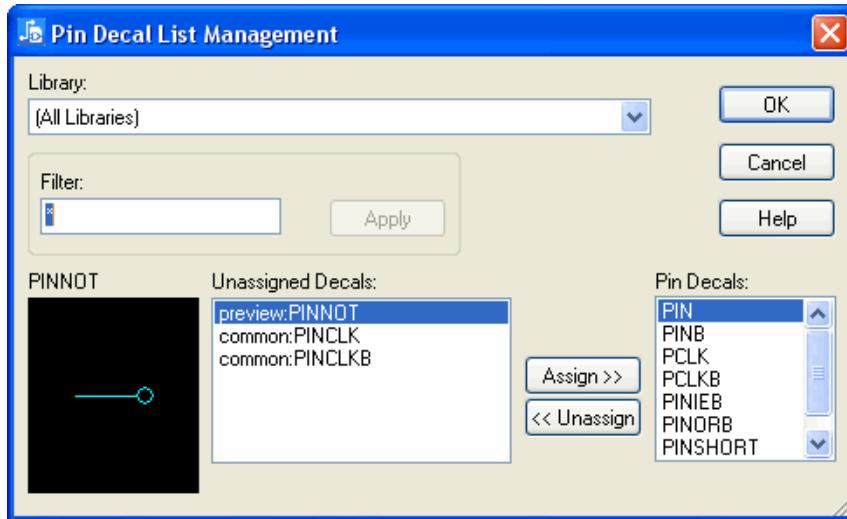
[Modifying Terminals](#)

Pin Decal List Management Dialog Box

To access: Tools > Part Editor menu item > Setup > Pin List Manger menu item

Use the Pin Decal List Management dialog box to control which decals are displayed in the Pin Decal List. The pin decal list contains the decals that can be used as terminal graphics when creating gate or part decals.

Figure 123. Pin Decal List Management Dialog Box



Objects

Table 183. Pin Decal List Management Box Contents

Name	Description
Library list	Lists the libraries available to you.
Filter	Searches the chosen library (or libraries) for a specific part or item name, or names that match a wildcard or expression on page 105. Use the Library dropdown list to select specific library directories or the All Libraries setting. Type * to view all parts or items in the chosen libraries. Click Apply to search the libraries and display the search results.
Preview Area	Displays the pin decal highlighted in the Unassigned Decals area or the Pin Decals area.
Unassigned Decals list	Lists the available pin decals in the selected library or all libraries.
Assign>> button	Moves a decal from the Unassigned Decals list box to the Pin Decals list. Select a decal, then click Assign .
<<Unassign button	Moves a decal from the Pin Decals list box to the Unassigned Decals list box. Select a decal, then click Unassign .

Table 183. Pin Decal List Management Box Contents (continued)

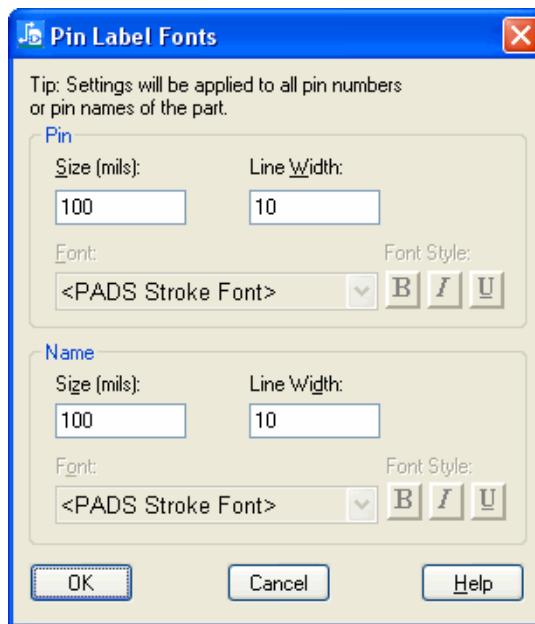
Name	Description
Pin Decals list	Lists the pin decals that are displayed in the dialog boxes that access pin decals, for example, the Pin Decal Browse dialog box. You can display up to 100 pin decals.

Pin Label Fonts Dialog Box

To access: Select a pin > right-click > **Properties** menu item > **Font** button

Use the Pin Label Fonts dialog box to change the fonts of a pin label.

Figure 124. Pin Label Fonts Dialog Box



Objects

Table 184. Pin Label Fonts Dialog Box Contents

Name	Description
Pin Area	
Size	<p>Specifies the size of the font.</p> <p>Size (pts): This is font size in points and appears for system fonts</p> <p>Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.</p>  <p>Stroke Font - Size</p>
Line Width	Specifies the line width for stroke fonts only.

Table 184. Pin Label Fonts Dialog Box Contents (continued)

Name	Description
	 Stroke Line Width
Font list	<p>The fonts available to you. This lists either stroke fonts or system fonts.</p> <p>You choose which type of font to use in the Fonts Dialog Box.</p> <p>Tip</p> <ul style="list-style-type: none"> • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Justification area	<p>Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.</p>
Name Area	
Size	<p>Specifies the size of the font.</p> <p>Size (pts): This is font size in points and appears for system fonts</p> <p>Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.</p>  Stroke Font - Size
Line Width	<p>Specifies the line width for stroke fonts only.</p>  Stroke Line Width
Font list	<p>The fonts available to you. This lists either stroke fonts or system fonts.</p> <p>You choose which type of font to use in the Fonts Dialog Box.</p> <p>Tip</p> <ul style="list-style-type: none"> • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.

Table 184. Pin Label Fonts Dialog Box Contents (continued)

Name	Description
Justification area	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

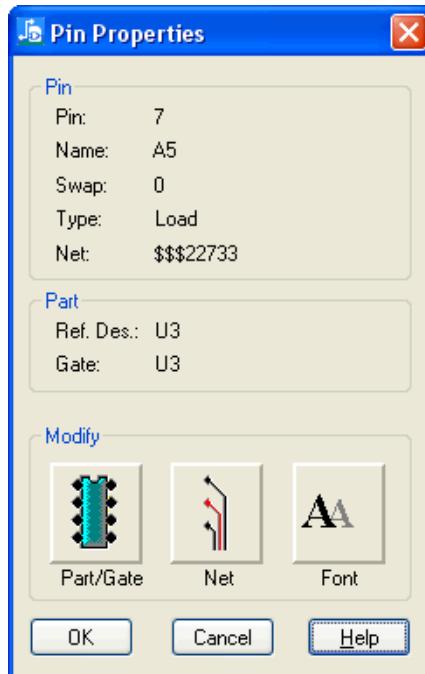
[Modifying Pin Label Fonts](#)

Pin Properties Dialog Box

To access: Select a pin > right-click > **Properties** menu item

Use the Pin Properties dialog box to view pin information, to modify parts and nets to which the selected pin is connected, and also to set font settings for pin number and pin name labels.

Figure 125. Pin Properties Dialog Box



Objects

Table 185. Pin Properties Dialog Box Contents

Name	Description
Pin area	Displays the information about the selected pin: Pin number, name, swap number, type, and the net.
Part area	Displays the information about the part: reference designator and gate number.
Part/Gate button	Opens the Part Properties Dialog Box .
Net button	Opens the Net Properties Dialog Box .
Font button	Opens the Pin Label Fonts Dialog Box .

Related Topics

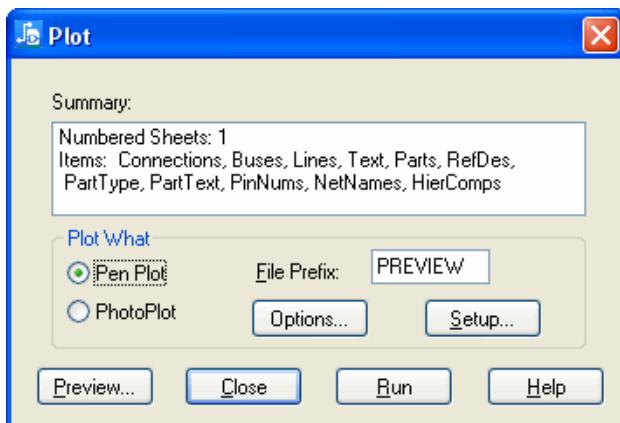
[Modifying Pins](#)

Plot Dialog Box

To access: **File > Plot** menu item

You can output your designs to pen plotters or photo plotters.

Figure 126. Plot Dialog Box



Objects

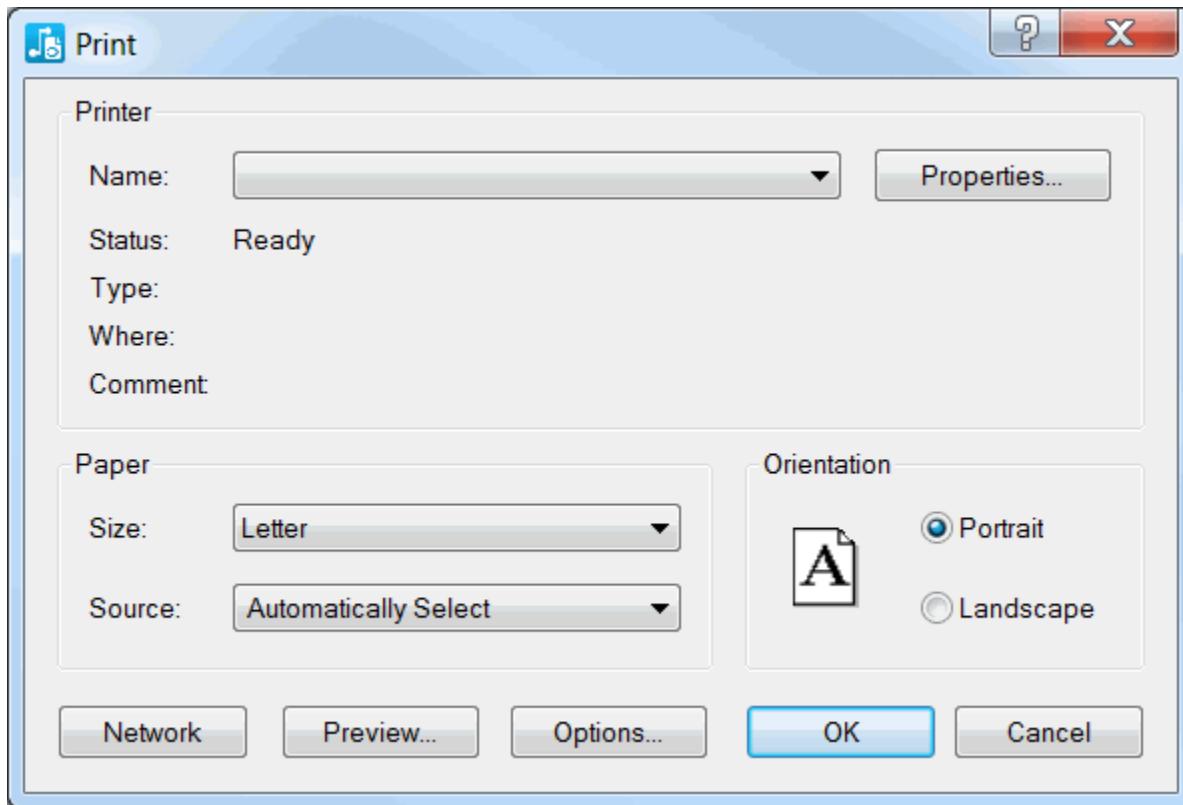
Table 186. Plot Dialog Box Contents

Name	Description
Summary	Lists the numbered available sheets you can plot, and the items contained in the sheets.
Plot What area	Specifies what to plot: Pen or Photo.
File Prefix	Specifies the prefix name of the file you want to plot.
Options button	Opens the Options dialog box on page 606.
Setup button	Depending on what you are plotting, opens the Pen Plotter Setup Dialog Box or the Photo Plotter Setup Dialog Box .
Preview button	Opens the Selections Preview Dialog Box .
Run button	Runs the Plot with what you've set.

Print Dialog Box

To access: **File > Print** menu item

A standard Microsoft Print dialog box with access to Print Preview and Options.



Objects

Object	Description
Preview button	Opens the “Selections Preview Dialog Box” on page 714.
Options button	Opens the “Options Dialog Box - Print/Plot” on page 606.

Project Explorer

To open the Project Explorer, click the **Project Explorer** button.

The Project Explorer shows a hierarchical structure for the objects in your design. It provides access to objects and rules. When you update your design, the hierarchical structure is automatically updated to reflect the changes you make.



Tip

The Hierarchical structure is available only when a design is open.



Restriction:

The Project Explorer is not available in the Part Editor.

Object Types

Objects in the Project Explorer are placed in object groups. Object groups are of two types: primary and secondary as shown in the following table.



Restriction:

Observe the following restrictions:

- You cannot remove or rename primary object groups.
 - Modification of secondary group items is only available in SailWind Router.
-

Table 187. Object Groups and Subgroups

Primary Group	Product Availability	Secondary Group	Description
schematic sheets	SailWind Logic	Sheet names	Lists all parts on the sheet
Layers	SailWind Layout SailWind Router	Electrical layers	Lists all electrical layers, including plane layers and routing layers
		General layers	Lists all other layers except electrical
Components	SailWind Logic SailWind Layout SailWind Router		Lists all components and pin pairs
Part decals	SailWind Router		Lists all part decals in the design or all components that use the selected part decal
Nets	SailWind Logic SailWind Layout		Lists all nets in the design
Net objects	SailWind Router	Net classes	Lists all nets belonging to net classes
		Matched length net groups	Lists all matched length net groups
		Nets	Lists all nets in the design

Table 187. Object Groups and Subgroups (continued)

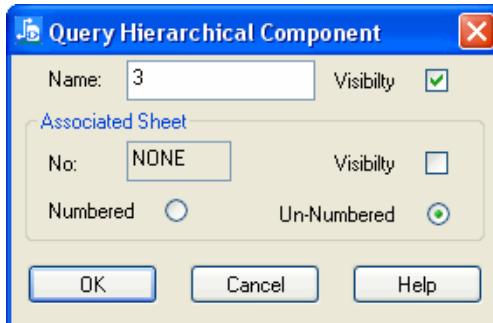
Primary Group	Product Availability	Secondary Group	Description
		Matched length pin pair groups	Lists all matched length pin pair groups
		Pin pair groups	Lists all nets belonging to pin pair groups (containing pin pair rules)
		Conditional rules	Lists all nets with conditional rules
		Differential pairs	Lists all differential pairs
Via types	SailWind Router		Lists the via types used in the design
CAE decals	SailWind Logic		Lists the CAE decals used in the design
PCB decals	SailWind Logic SailWind Layout		Lists the PCB decals used in the design

Query Hierarchical Component Dialog Box

To access: Select a hierarchical component > right-click > **Properties** menu item

Use the Hierarchical Component Properties dialog box to assign a hierarchical component to the next available sheet number to make it accessible from the Sheet list when a sheet other than the parent sheet is displayed. Use the **Setup > Sheets** menu item to modify the sheet name or the numeric order.

Figure 127. Query Hierarchical Component Dialog Box



Objects

Table 188. Query Hierarchical Component Dialog Box Contents

Name	Description
Name	The name of the selected component.
Visibility	Specifies to display the name on top of the hierarchical component in the schematic.
No	The assigned sheet number of the selected component.
Visibility	Specifies to display the sheet number in the schematic.
Numbered	Specifies to assign the hierarchical component the next available sheet number.
UnNumbered	Specifies to remove a sheet number assignment from a hierarchical component.

Related Topics

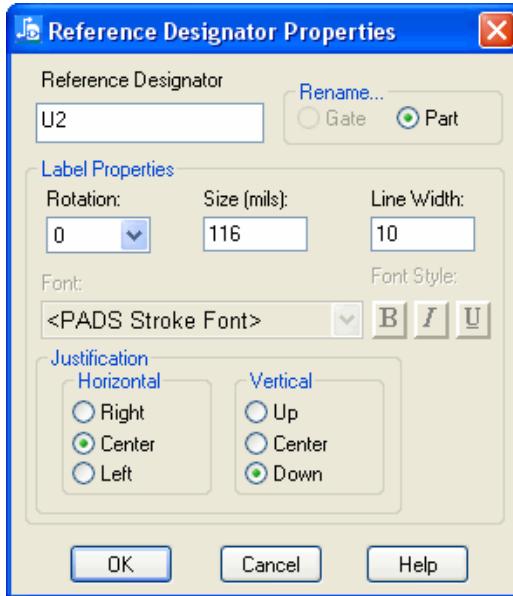
[Modifying Hierarchical Components](#)

Reference Designator Properties Dialog Box

To access: Select a part type label > right-click > **Properties** menu item

Use the Reference Designator Properties dialog box to view and modify label size and justification as well as text and font settings for one or more reference designator labels.

Figure 128. Reference Designator Properties Dialog Box



Objects

Table 189. Reference Designator Properties Dialog Box Contents

Name	Description
Reference Designator	The name of the selected attribute.
Rename area	Specifies to rename just the gate or the entire part.
Rotation	Specifies the rotation of the label: 0 or 90 degrees.
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.  Stroke Font - Size

Table 189. Reference Designator Properties Dialog Box Contents (continued)

Name	Description
Line Width	<p>Specifies the line width for stroke fonts only.</p>  <p>Stroke Line Width</p>
Font list	<p>The fonts available to you. This lists either stroke fonts or system fonts.</p> <p>You choose which type of font to use in the Fonts Dialog Box.</p> <p>i Tip</p> <ul style="list-style-type: none"> • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.

Related Topics

[Modifying Reference Designator Labels](#)

Remap Special Symbols Dialog Box

To access: **Tools > Update from Library** menu item; then, in the Update from Library dialog box, choose the “Update design from library” option and select the Off-page symbols, Ground symbols, or the Power symbols check box.

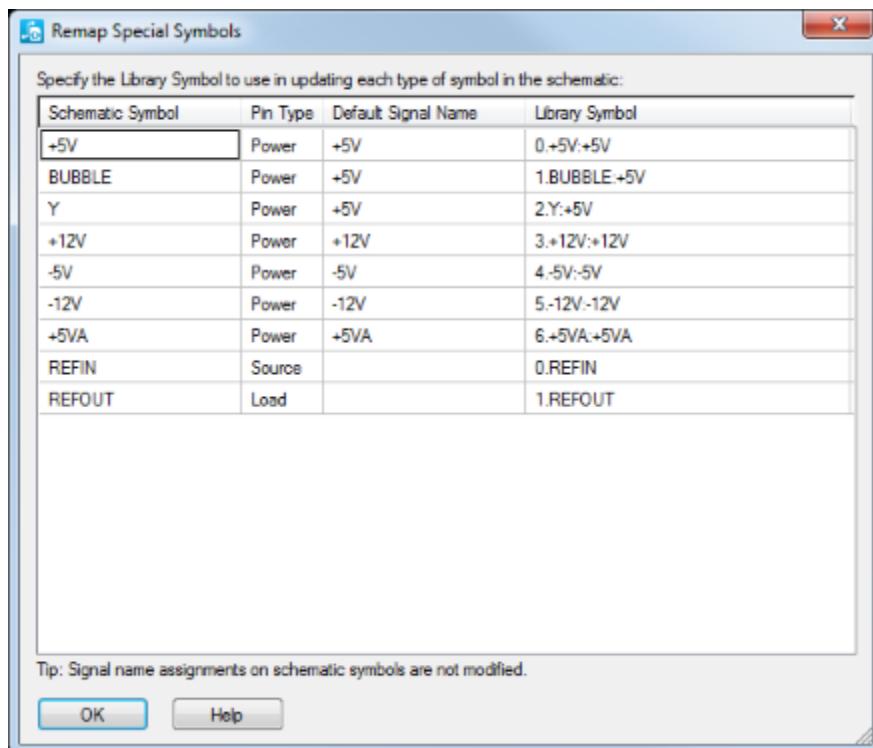
Use the Remap Special Symbols dialog box to make updates to power, ground, or off-page schematic symbols in your design. You can use this dialog box to update your schematic symbols with changes in the library or assign a different library symbol to a schematic symbol in the design.



Note:

The specific signal name assigned to special symbols currently used within the schematic are not modified when performing the “Update from Library” function. For example, the signal name of a power symbol assigned to +5V is not modified in the schematic if updated with another power symbol that uses a different default signal name (such as +12V).

Figure 129. The Remap Special Symbols Dialog Box



Objects

Table 190. The Remap Special Symbols Dialog Box Options

Name	Description
Schematic Symbol	Displays the name power, ground, or off-page symbols currently associated with your design.

Table 190. The Remap Special Symbols Dialog Box Options (continued)

Name	Description
Pin Type	Displays the function of the pin associated with the symbol; for example, load, ground, or power.
Default Signal Name	Displays the name of the signal currently associated with the symbol (as it originally existed in the library); for example, +5VCC, or AGND. This name does not necessarily reflect the signal name used on specific instances of the symbol in the schematic.
Library Symbol	<p>The symbol mapped to the schematic symbol. SailWind Layout initially displays the best-matching symbol in this box, regardless of the current mapping.</p> <p>You can change the mapping by double-clicking in the box to access a dropdown list of available symbols. You can assign the same library symbol to more than one schematic symbol.</p> <p>Only symbols associated with your current library appear in the list.</p> <p>The available library symbols in the dropdown list also depends on the pin type. For example, if the pin type is “ground,” only ground symbols appear in the list.</p> <p>Any additional symbols not specifically mapped to existing symbols become updated to the design and available for use.</p>

Related Topics

[The Update From Library Function](#)

[Updating Special Symbol Mappings](#)

Rename Gate Dialog Box

To access: select a part > right-click > **Properties** menu item > **Rename Gate** button

Use the Rename Gate dialog box to change the reference designator of the selected gate.

Figure 130. Rename Gate Dialog Box



Objects

Table 191. Rename Gate Dialog Box Contents

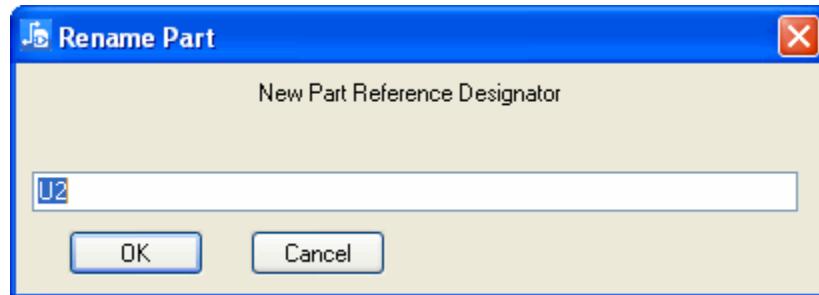
Name	Description
Text box	Type the new gate reference designator information.  Restriction: You are prevented from assigning an already used reference designator or an unused gate of a part with a different part type.

Rename Part Dialog Box

To access: select a part > right-click > **Properties** menu item > **Rename Part** button

Use the Rename Part dialog box to change the reference designator for the selected part or gate.

Figure 131. Rename Part Dialog Box



Objects

Table 192. Rename Part Dialog Box Contents

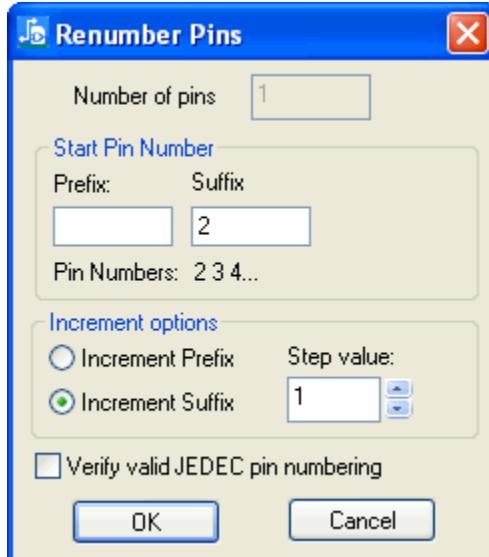
Name	Description
Text box	Type the new part reference designator information.  Restriction: <ul style="list-style-type: none">• All gates are renamed if you change the reference designator of one gate of a multi-gated part.• You are prevented from assigning an already used reference designator.

Renumber Pins Dialog Box

To access: **Tools > Part Editor** menu item > **Edit Electrical** button > **Pins** tab > select pin(s) > **Renumber** button

Use the Renumber Pins dialog box to renumber pins (terminals). You can renumber part type pins on the **Pins** tab of the Part Information dialog box.

Figure 132. Renumber Pins Dialog Box



Objects

Table 193. Renumber Pins Dialog Box Contents

Name	Description
Number of pins	The number of pins available for renumbering.
Prefix/Suffix	For a single pin number, use either Prefix or Suffix box, and void the other box. Use both boxes if you want to increment one of the values. Alphabetic and numeric values can be used in either box.
Pin numbers	A preview of pin numbers based on your prefix/suffix input.
Increment prefix	Sets the prefix as the part of the pin number to increment.
Increment suffix	Sets the suffix as the part of the pin number to increment.
Step value	Sets the step value. Type a positive or negative number by which to increase or decrease the pin number with consecutive or stepped values.

Table 193. Renumber Pins Dialog Box Contents (continued)

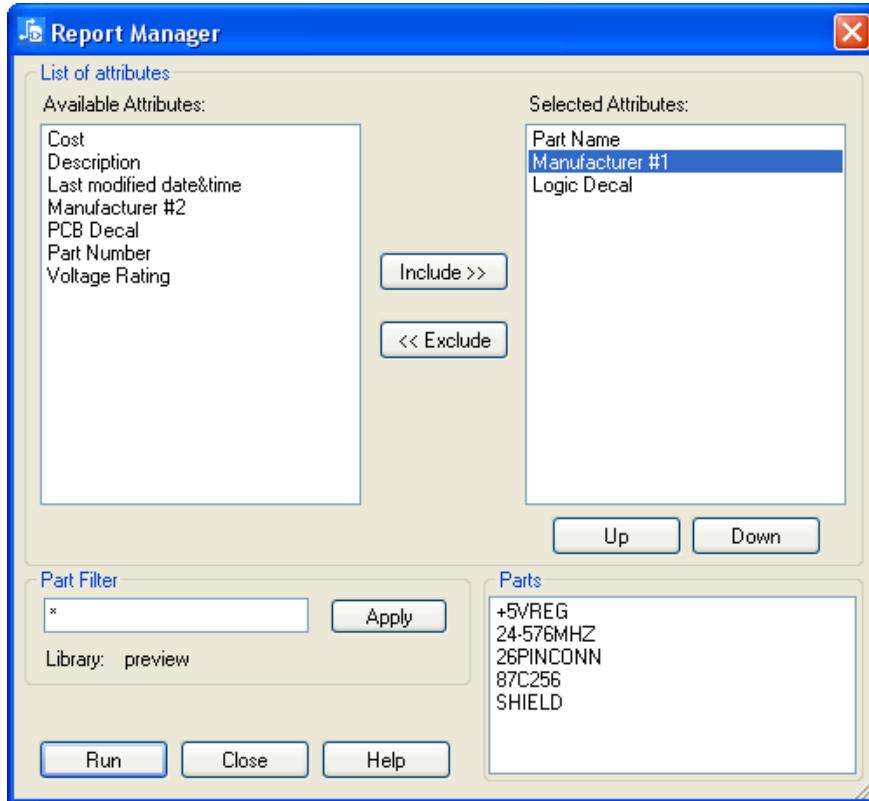
Name	Description
	 Restriction: Step value must be non-zero and be in the range -10 to +10. Zero would replicate a single pin number and is not allowed.
Verify valid JEDEC pin numbering	If using alphanumerics, you can select the Verify valid JEDEC pin numbering check box to ensure that legal alphanumeric values are used.

Report Manager Dialog Box

To access: **File > Library** menu item > **Parts** button > select a part > **List to File** button

Use the Report Manager dialog box to generate a report about the parts in a library. You can specify the parts and the attributes to include in the report.

Figure 133. Report Manager Dialog Box



Objects

Table 194. Report Manager Dialog Box

Field or Button	Description
Available attributes	All attributes of the part types in the selected library. Click an attribute in the list to select it. (To select additional attributes, press CTRL and click each attribute.) Click Include >> to include selected attributes in the report.
Selected attributes	Attributes in the report. Click an attribute in the list to select it. (To select additional attributes, press CTRL and click each attribute.) Click << Exclude to remove selected attributes from the report.

Table 194. Report Manager Dialog Box (continued)

Field or Button	Description
	The order of attributes in the list is the order of columns in the report. Select an attribute and click Up or Down to change the order.
Include >> button	Includes the selected attributes in the report (moves the attributes to the Selected attributes list). Select one or more attributes on the Available attributes list and click Include >> .
<< Exclude button	Excludes the selected attributes from the report (moves the attributes from the Selected attributes list back to the Available attributes list). Select one or more attributes on the Selected attributes list and click << Exclude .
Up / Down buttons	Moves a selected attribute up or down on the Selected attributes list. List order determines the order in which columns appear in the report.
Part Filter	Specifies the part types to include in the report. Type a part type name in the field or use wildcards (*) to specify a group of part types. For example: * Specifies all part types in the library. +5* Specifies all part types that begin with the characters +5, such as +5volt and +5LS07.
Apply button	Filters the part types.
Parts	Lists part types included in the report (as determined by the Part Filter).
Run button	Generates the report and lets you save it either in lst for viewing or printing or in csv format for use with MS Excel.
Close button	Cancels the operation and closes the dialog box.

Related Topics

[Creating a Report of the Parts in a Library](#)

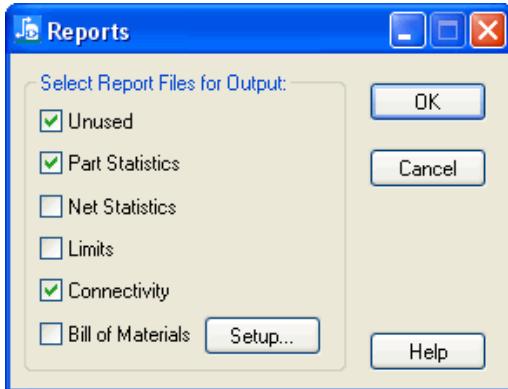
[Creating a Report of Decals, Lines or Logic Symbols in a Library](#)

Reports Dialog Box

To access: **File > Reports** menu item

Use the Reports dialog box to produce any of six different types of reports on the current schematic. You can save these reports as text files on your hard disk or output them to a printer.

Figure 134. Reports Dialog Box



Objects

Table 195. Reports Dialog Box Contents

Name	Description
Select Report Files for Output area — Select the report(s) you want to run.	
Unused	<p>The Unused report has two parts - the Unused Gate List, followed by the Unused Pins List. Use this report for troubleshooting and to maximize part usage.</p> <ul style="list-style-type: none">• Unused Gate List — lists all part types with multiple gates and if there are any unused gates, it lists the specific gate(s) under the part type name. If there are no unused gates then it only lists the part type name.• Unused Pins List — lists parts with pins unused in the schematic. It lists the unused pins under the part type name. If there are no unused pins there will be no part types listed. <p>When you run this report, a link to the <i>UnusedGatesPins.rep</i> appears in the Output Window. Click the link to open the report in your default text editor.</p>
Part Statistics	<p>The Part Statistics report lists information about all parts in the schematic. The report includes the reference designator name and part type for each part in the schematic, and for each pin on the part, the pin type, sheet location, and signal name. Use this report to locate possible errors in the schematic.</p> <p>When you run this report, a link to the <i>PartStatistics.rep</i> appears in the Output Window. Click the link to open the report in your default text editor.</p>
Net Statistics	<p>The Net Statistics report lists information for each net in the schematic. The report includes the reference designator name and pin number for all parts in the net. SailWind Logic flags possible errors (for example, nets with no inputs</p>

Table 195. Reports Dialog Box Contents (continued)

Name	Description
	<p>or no outputs, nets with multiple outputs, etc.) for further examination. Use this report to locate possible errors in the schematic.</p> <p>Error messages that can occur:</p> <ul style="list-style-type: none"> • Net has only one pin: a net going to an off-page flag not connected elsewhere. • Net has no defined source: there is no pin in the net that has a pin type of S. • Net has no defined loads: there is no pin in the net that has a pin type of L. • Net has multiple sources- be sure they are tieable: the net has more than 1 pin with the pin type S. <p>When you run this report, a link to the <i>NetStatistics.rep</i> appears in the Output Window. Click the link to open the report in your default text editor.</p>
Limits	<p>The Limits report indicates the maximum number of each SailWind Logic data item (parts, nets, text) your system will allow, as well as the current count of each of these items in the schematic. This limit varies depending on the amount of virtual memory that is available.</p> <p>The report has two parts. The first is a list of items whose limits are common to the entire schematic. The second is a count, for each schematic sheet, of the items whose limits apply for each sheet.</p> <p>You should periodically run a Limits report to ensure that you are not approaching the system's limit for any item.</p> <p>If you exceed the Maximum No. of Items for any item, you cannot continue adding those items to the schematic. The solution is to split the design into multiple schematics, run separate netlists for each schematic, then merge the netlists using a text editor.</p> <p>When you run this report, a link to the <i>DesignLimits.rep</i> appears in the Output Window. Click the link to open the report in your default text editor.</p>
Connectivity	<p>The Connectivity report lists the X,Y coordinate location and sheet number of all off-page, ground, and power symbols in the schematic.</p> <p> Tip Use the report to quickly locate an off-page symbol using the S (Search) modeless command.</p> <p>An error message appears when a net contains only one off-page reference. Subnets tied together without a visible net name are identified by flagging them as missing an off-page symbol.</p> <p>When you run this report, a link to the <i>ConnectivityReport.rep</i> appears in the Output Window. Click the link to open the report in your default text editor.</p>
Bill of Materials	<p>The Bill of Materials report produces a user-configurable list of the parts contained in the current schematic. You can direct any part attribute in the schematic to a Bill of Materials report.</p> <p>When you run this report, a link to the <i>BillOfMaterials.rep</i> appears in the Output Window. Click the link to open the report in your default text editor.</p>
Setup button	Opens the Bill of Materials Setup dialog box on page 479. See also Setting up the Bill of Materials Report on page 314.

Related Topics

[Generating Reports](#)

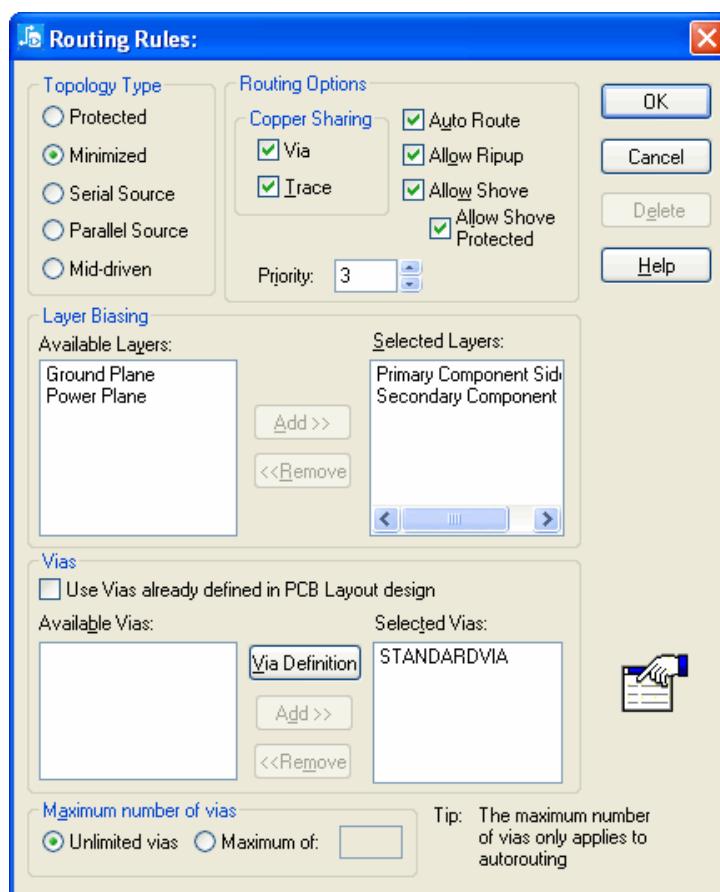
Routing Rules Dialog Box

To access:

- **Setup > Design Rules** menu item > a rule hierarchy button) > **Routing** button
- Select a net, right-click and click **Show Rules**, then click the **Routing** button.

Use the Routing Rules dialog box to specify rules for interactive and automatic routing. You can specify the default set of routing rules and routing rules for specific nets.

Figure 135. Routing Rules Dialog Box



Objects

Table 196. Routing Rules Dialog Box Contents

Name	Description
Topology Type	Specify the topology type to determine the pin-to-pin order when routing the net or moving a part. When routing interactively, a ratsnest guides you as you route from pin to pin.

Table 196. Routing Rules Dialog Box Contents (continued)

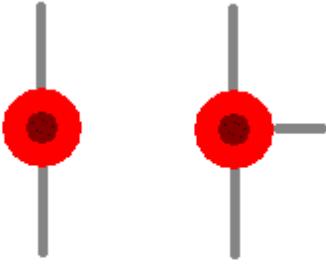
Name	Description
	<p>To specify the topology type, click one of the following options:</p> <ul style="list-style-type: none"> • Protected — Do not change the order of the connectivity in the net. <p>Note that this option disables length minimization.</p> <ul style="list-style-type: none"> • Minimized — Order the net by the shortest distance between pins. Net reorder or reconnect is permitted. • Serial source — Order the net in a series order from source pins to load pins to a terminator. • Parallel source — Same as "Serial source" except order the net with parallel branches for each source-to-load connection. • Mid-driven — Divide the net into two branches and order each branch in a source to load to terminator order.
Routing options	
Via	<p>Vias can share copper with another object.</p>  <p>Non-shared Via One Net Shared Via Multiple Nets</p> <p> Restriction: This rule is used only in SailWind Router, although you can define this rule in SailWind Logic, SailWind Layout, or SailWind Router.</p>

Table 196. Routing Rules Dialog Box Contents (continued)

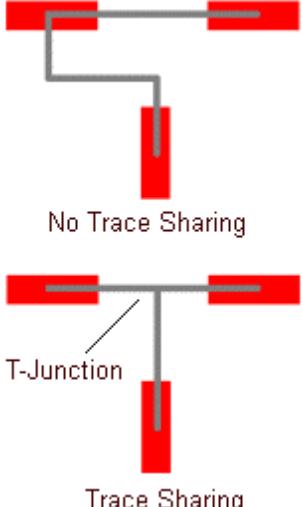
Name	Description
Trace	<p>Traces can share copper with another object.</p>  <p>No Trace Sharing</p> <p>T-Junction</p> <p>Trace Sharing</p> <p>Restriction: This rule is used only in SailWind Router, although you can define this rule in SailWind Logic, SailWind Layout, or SailWind Router.</p>
Priority	<p>Assign priority from 0 to 100. Nets with higher priority are routed first.</p> <p>Restriction: SailWind Router does not use the priority value. This rule applies only to SPECCTRA.</p>
Auto Route	Enables the autorouter to route nets.
Allow Ripup	<p>Unroute existing traces and reroute the nets.</p> <p>i Tip Enable this option to rip up traces while DRC Warn or Prevent is enabled.</p>
Allow Shove	<p>Move unprotected traces aside to create room for new traces.</p> <p>i Tip Enable this option to shove traces while DRC Warn or Prevent is enabled.</p>
Allow Shove Protected	Move protected traces aside to make room for new traces.
Layer biasing	
Available Layers	Lists layers, not already in the Selected Layers list, that can be made available for routing.

Table 196. Routing Rules Dialog Box Contents (continued)

Name	Description
Add >> button	Click to move a selected layer from the Available Layers list to the Selected layers list.
<< Remove button	Click to move a selected layer from the Selected Layers list to the Available layers list.
Selected layers	Lists the layers, that have been selected from all the available design layers, to be available for routing.
Vias	
Use Vias already defined in PCB Layout design	Select the check box to suppress the via biasing in SailWind Logic. Clear the check box to specify via biasing for routing. i Tip <ul style="list-style-type: none"> If vias exist in SailWind Logic that don't exist in SailWind Layout, you are prompted to create those vias before importing the netlist into SailWind Layout. Since there is no Pad Stacks Properties facility for creating vias in SailWind Logic you can only specify the names of via and you must create those vias in SailWind Layout. Since only Default, Net, and Class rules are available in SailWind Logic, any control of vias at higher levels of the rules hierarchy must be done in SailWind Layout.
Available vias	Lists the vias, not already in the Selected Vias list that are available for routing.
Via Definition button	Click to open the Via Setup dialog box to Add, Delete, or Rename vias available for routing. Since there is no Pad Stacks Properties facility for creating vias in SailWind Logic, any vias added must be created in SailWind Layout before importing the netlist in SailWind Layout.
Add >> button	Click to move a selected via from the Available Vias list to the Selected Vias list.
<< Remove button	Click to move a selected via from the Selected Vias list to the Available Vias list.
Selected Vias	Lists the vias, that have been selected from all the available vias, to be available for routing.
Maximum number of vias	
Unlimited vias	Click to give the autorouter unrestricted use of vias during autorouting.
Maximum of	Click to restrict the number of vias the autorouter can add to nets during autorouting. In the box, type the maximum number of vias between 0 and 50000. The autorouter considers this to be a hard rule . Interactive routing and design verification check this rule.

Table 196. Routing Rules Dialog Box Contents (continued)

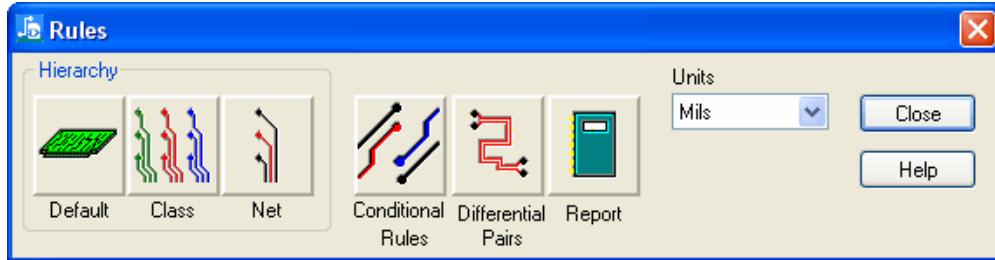
Name	Description
	 Tip An insufficient maximum number of vias might increase autorouting runtime and reduce completion rates.
Delete button	Click to remove the current set of routing rules from the rules hierarchy for the selected nets.  Restriction: The Delete button is unavailable for the default set of routing rules. You cannot delete the Default Routing rules.

Rules Dialog Box

To access: **Setup > Design Rules** menu item

Use the Rules dialog box to enter item-to-item Clearance rules, routing guidelines, and values for the optional High Speed checking commands. You can also indicate the unit of measure for passing rules to SailWind Logic: mils, metric, or inches.

Figure 136. Rules Dialog Box



Objects

Table 197. Rules Dialog Box Contents

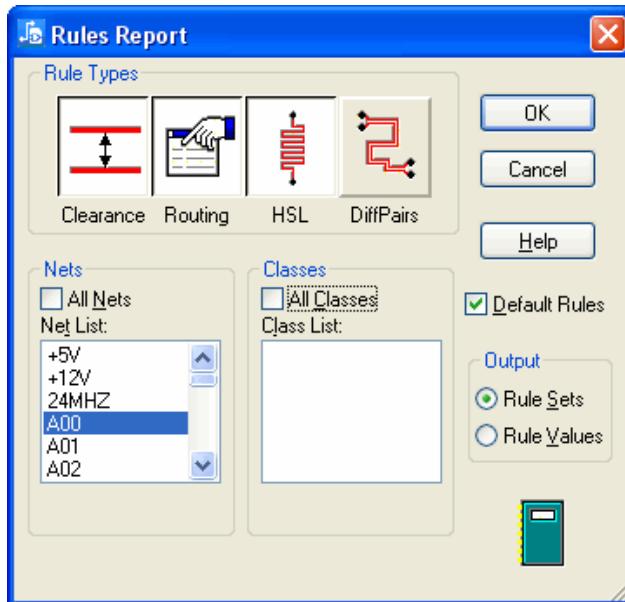
Name	Description
Default button	Opens the Default Rules Dialog Box .
Class button	Opens the Class Rules Dialog Box .
Net button	Opens the Net Rules Dialog Box .
Conditional Rules button	Opens the Conditional Rule Setup Dialog Box .
Differential Pairs button	Opens the Differential Pairs Dialog Box .
Report button	Opens the Rules Report Dialog Box .
Units button	Specifies the units you want: Mils, Metric, or Inches

Rules Report Dialog Box

To access: **Setup > Design Rules** menu item > **Report** button

Use the Rules Report dialog box to produce a report of some or all of the rules you have defined. By default, a complete report of all rules is reported.

Figure 137. Rules Report Dialog Box



Objects

Table 198. Rules Report Dialog Box Contents

Name	Description
Rule Types area	Displays the specified rules for the specified nets and classes. Click any combination of buttons, including Differential Pairs, to report net pairs.
Nets area	Displays the specified rules for every net or selected nets. Click All Nets or select specific nets in the list box.
Classes area	Displays the specified rules for every class or selected classes. Click All Classes or select specific net classes in the list box.
Default Rules	Displays the default rules for the specified nets and classes.
Output area	Specifies how you want your output. <ul style="list-style-type: none"> • Rule Sets — display all rules in the current hierarchy that are unique from the default rules. • Rule Values — display all rules in the current hierarchy level, even if the values are the same as the default rules.

Related Topics

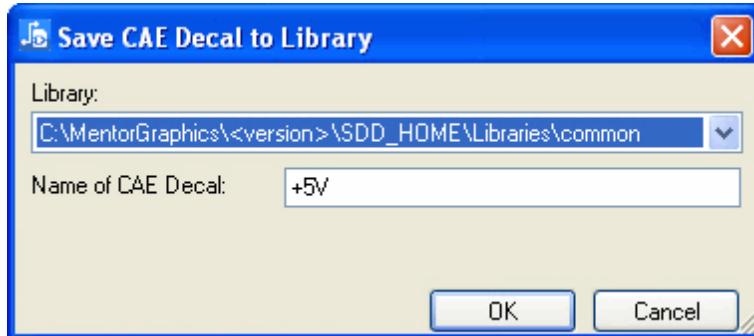
[Creating a Rules Report](#)

Save CAE Decal to Library Dialog Box

To access: **File > Library** menu item > select a library > **Logic** filter type > select a CAE decal > **Copy** button

Use the Save CAE Decal to Library dialog box to copy a CAE decal to another name or another library.

Figure 138. Save CAE Decal to Library Dialog Box



Objects

Table 199. Save CAE Decal to Library Dialog Box Content

Name	Description
Library	Select the library for the copied CAE decal.
Name of CAE Decal	Type the name for the copied CAE decal.

Related Topics

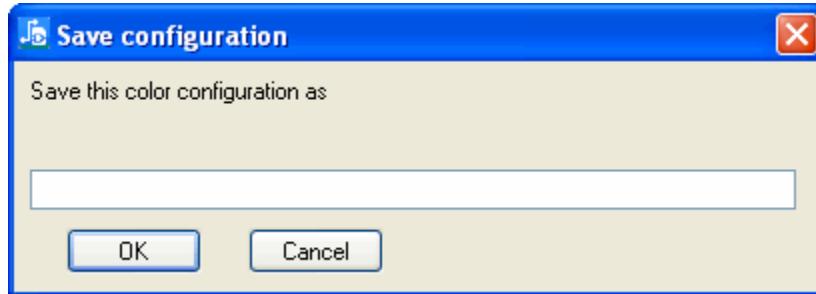
[Copying a Library Item](#)

Save Configuration Dialog Box

To access: **Setup > Display Colors** menu item > **Save** button

Use the Save Configuration dialog box to save the color assignments and settings you've made in the Display Colors dialog box.

Figure 139. Save (color) configuration Dialog Box



Objects

Table 200. Save configuration Dialog Box Content

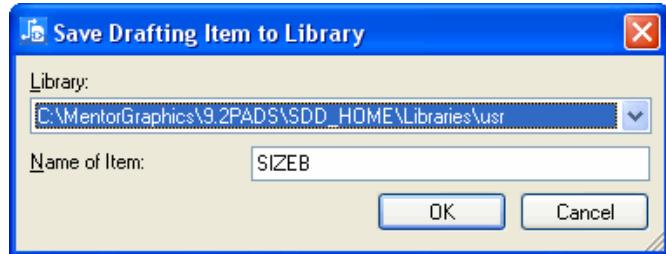
Name	Description
Text box	Type the name of the new color configuration. The name will appear in the Configuration list in the Display Colors Dialog Box .

Save Drafting Item to Library Dialog Box

To access: Select an item > right-click > **Save to Library** menu item

Use the Save Drafting Item to Library dialog box save 2D line items or complex 2D line item in the schematic to a library.

Figure 140. Save Drafting Item to Library Dialog Box



Objects

Table 201. Save Drafting Item to Library Dialog Box Contents

Name	Description
Library list	Specifies the library you want to save the item to.
Name of Item	The name of the item you want to save in the selected library.

Related Topics

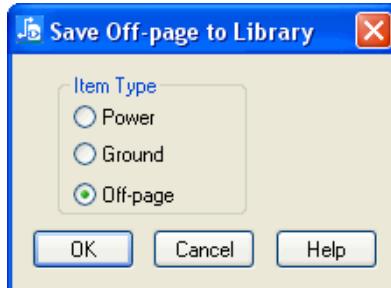
[Adding Drafting Items to a Library](#)

Save Off-Page to Library Dialog Box

To access: Tools > Save Off-page to Library menu item

Use Save Off-page to Library to update the off-page, ground, or power symbols in the library with the current version(s) in the schematic.

Figure 141. Save Off-page to Library Dialog Box



Objects

Table 202. Save Off-page to Library Dialog Box Contents

Name	Description
Item Type area	Specifies the item you want to update in the library with the current version in the schematic.

Related Topics

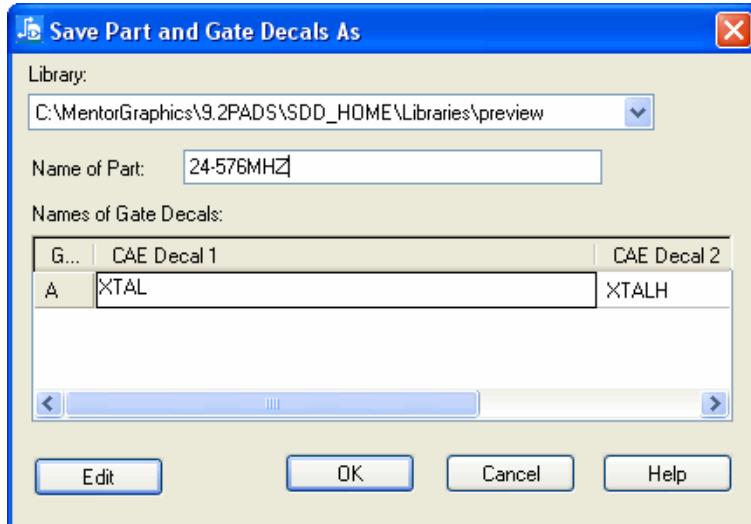
[Saving Off-Page to Library](#)

Save Part and Gate Decals As Dialog Box

To access: Tools > Part Editor menu item > (in the Part Editor) File > Save as menu item

The Save Part and Gate Decals As dialog box saves a new or modified part type to the library. You can also rename a modified decal to prevent other parts that use the same decal from being affected.

Figure 142. Save Part and Gate Decals As Dialog Box



Objects

Table 203. Save Part and Gate Decals As Dialog Box Contents

Name	Description
Name of Part	Defines the part type name in the library.
Library	The library folder to which information is saved.
Name of Gate Decals	Displays the name of decal and any alternate decals associated with the part type. Select a decal name and click Edit to rename the decal.
Edit button	Makes the highlighted field in the table editable. You can also double click a text field to edit it.

Related Topics

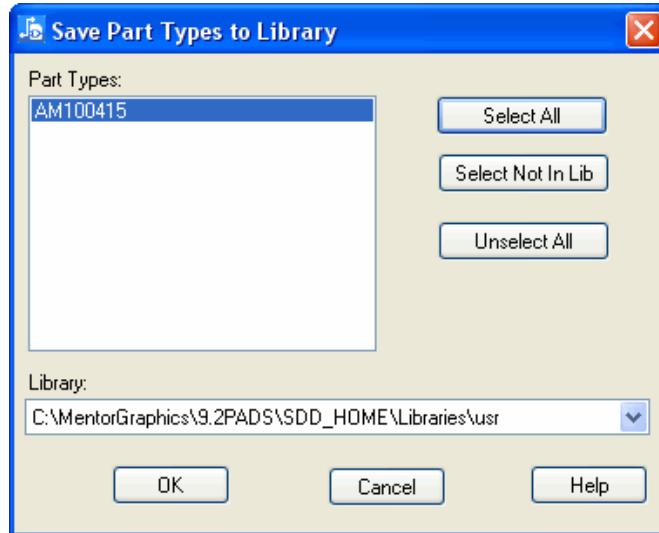
[Saving Part Types](#)

Save Part Types to Library Dialog Box

To access: Select a part > right-click > **Save to Library** menu item

Use the Save Part Types to Library dialog box to copy the part types of the schematic. If the original part type is deleted from the library, get another copy from the schematic.

Figure 143. Save Part Types to Library Dialog Box



Objects

Table 204. Save Part Types to Library Dialog Box Contents

Name	Description
Part Types list	The part(s) you selected in the schematic.
Select All button	Selects all of the items in the Part Types list.
Select Not in Lib button	Selects only those part types that are not currently saved in the library.
Unselect All button	Clears the selection of any or all part types in the Part Types list.
Library list	Specifies the library you to which you want to save the part types.

Related Topics

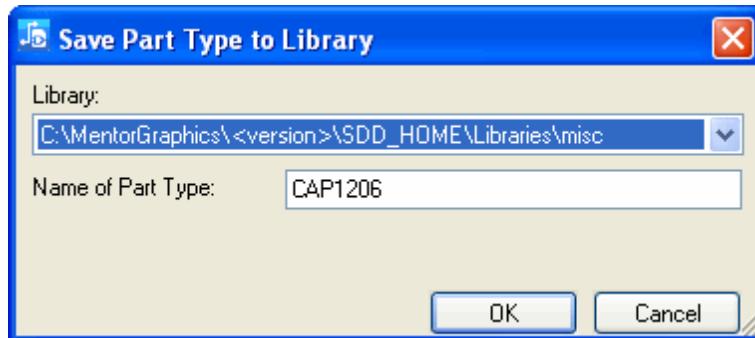
[Saving Part Types to a Library](#)

Save Part Type to Library Dialog Box

To access: **File > Library** menu item > select a library > **Parts** filter type > select a part type > **Copy** button

Use the Save Part Type to Library dialog box to copy a Part Type to another name or another library.

Figure 144. Save Part Type to Library Dialog Box



Objects

Table 205. Save Part Type to Library Dialog Box Content

Name	Description
Library	Select the library for the copied part type.
Name of Part Type	Type the name for the copied Part Type.

Related Topics

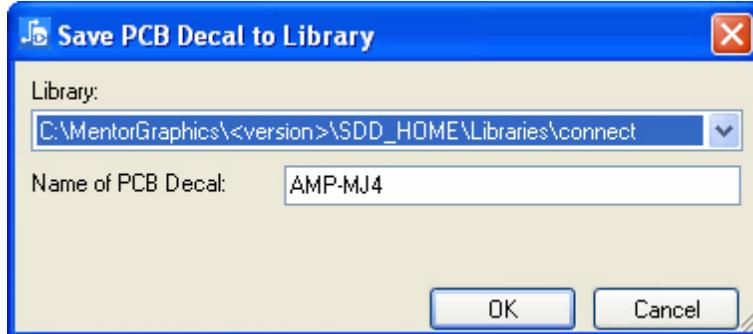
[Copying a Library Item](#)

Save PCB Decal to Library Dialog Box

To access: **File > Library** menu item > select a library > **Decal** filter type > select a PCB decal > **Copy** button

Use the Save PCB Decal to Library dialog box to copy a PCB decal to another name or another library.

Figure 145. Save PCB Decal to Library Dialog Box



Objects

Table 206. Save PCB Decal to Library Dialog Box Content

Name	Description
Library	Select the library for the copied PCB decal.
Name of PCB Decal	Type the name for the copied PCB decal.

Related Topics

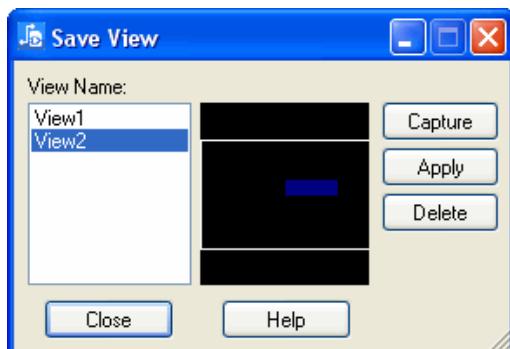
[Copying a Library Item](#)

Save View Dialog Box

To access: **View > Save View** menu item

The Save View dialog box to save a work area view for easy restoration.

Figure 146. Save View Dialog Box



Objects

Table 207. Save View Dialog Box Contents

Name	Description
View Name list	The views you have already saved.
Preview area	Shows you the location of the selected view in relation to the extents of the design.
Capture button	Opens the Capture a New View Dialog Box where you can name the view you want to save. Tip You can create up to nine views. The view names appear at the bottom of the View menu.
Apply button	Applies the selected, previously saved view to the work area.
Delete button	Removes the selected view from the View Name list.

Related Topics

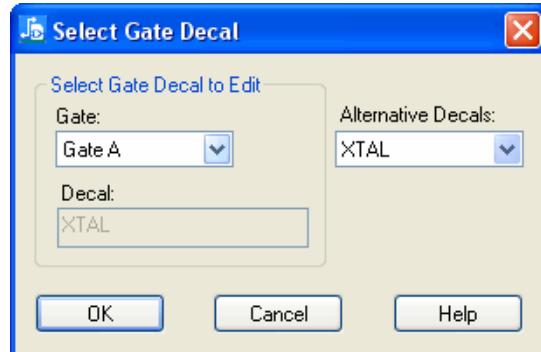
[Saving and Restoring Views](#)

Select Gate Decal Dialog Box

To access: **Tools > Part Editor** menu item > open a part type > **Edit Graphics** button

Use the Select Gate Decal dialog box to select the gate decal you want to use.

Figure 147. Select Gate Decal Dialog Box



Objects

Table 208. Sheets Decal Dialog Box Contents

Name	Description
Gate list	Lists the gates available to you.
Decal	Displays the decal assigned to the selected gate.
Alternative Decals list	Lists the alternative decals available.

Related Topics

[Assigning Pin Information to the CAE Decal](#)

Select Pin Decal Dialog Box

To access: **Tools > Part Editor** menu item > open an Off-page type (Off-page, Power, or Ground) > **Edit Graphics** button

Use the Select Pin Decal dialog box to access and create graphics for a new Special Symbol. This dialog box is displayed when you use the Extended Selection command from the popup menu to edit a gate for a Special Symbol.

Figure 148. Select Pin Decal Dialog Box



Objects

Table 209. Select Pin Decal Dialog Box Contents

Name	Description
Pin Decal list	Lists the Pin Decals available to you.

Related Topics

[Creating New Special Symbols](#)

Select Type of Editing Item Dialog Box

To access: Tools > Part Editor menu item > New or Open button

Use the Select type of editing item dialog box to select the type of library item to create or modify.

Figure 149. Select Type of Editing Item Dialog Box



Objects

Table 210. Select Type of Editing Item Dialog Box Contents

Name	Description
Part Type	Click Part Type to create or modify an existing part.
Connector	Click Connector to create or modify a connector.
CAE Decal	Click CAE Decal to create or modify a new CAE decal.
Pin Decal	Click Pin Decal to create or modify a pin decal, the information and appearance of a terminal pin. A number of different pin decals are provided.
Off-page	Click Off-page to modify the off-page reference symbols. SailWind Logic allows only one part definition in the library for off-page reference symbols, so this option grays when you select the File > New menu item. You can modify the existing symbols or add new symbols. Refer to the Special Symbols on page 167 topic for additional information.
Power	Click Power to modify an existing off-page reference symbol. SailWind Logic allows only one part definition in the library for power symbols, so this option grays when you select the File > New menu item. You can modify the existing symbols or add new symbols. Refer to the Special Symbols on page 167 topic for additional information.
Ground	Click Ground to modify an existing off-page reference symbol. SailWind Logic allows only one part definition in the library for ground symbols, so this option grays when you click the File > New menu item. You can modify the existing symbols or add new symbols. Refer to the Special Symbols on page 167 topic for additional information.

Related Topics

[Getting Gate Decals From the Library](#)

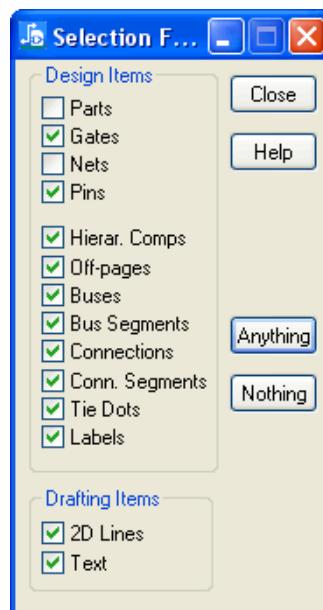
Selection Filter Dialog Box

To access:

- **Edit > Filter** menu item
- With nothing selected, right-click > **Filter** menu item

Use the Selection Filter to specify which objects you can select. Select a check box to enable the object for selection or clear the check box to disable the object for selection.

Figure 150. Selection Filter Dialog Box



Objects

Table 211. Selection Filter Dialog Box Contents

Name	Description
Design Items	Specifies the design items you want to be able to select in the design.
Drafting Items	Specifies the design items you want to be able to select in the design.
Anything button	Specifies that you want to select anything in the design. Exception: Clusters, unions, stitching vias, pin pairs, nets, and board outline shapes are not selected.
Nothing button	Specifies that you don't want to select anything in the design.

Related Topics

[Using the Selection Filter](#)

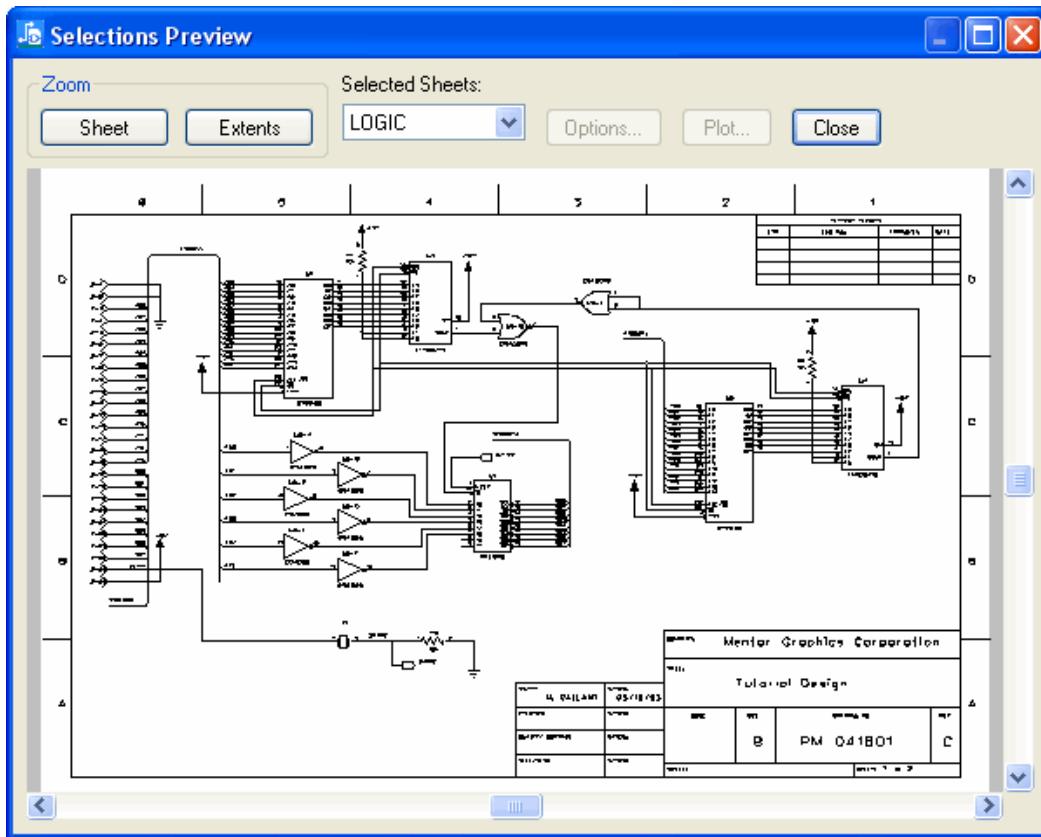
Selections Preview Dialog Box

To access:

- **File > Plot** menu item > **Preview** button
- **File > Print** menu item > **Preview** button

Use the Selections Preview dialog box to preview your options and output.

Figure 151. Selections Preview Dialog Box



Objects

Table 212. Selections Preview Dialog Box

Name	Description
Sheet button	Displays the entire sheet in the window.
Extents button	Zooms to the extents on the sheet.
Selected Sheets list	Specifies the sheet you want to preview.

Table 212. Selections Preview Dialog Box (continued)

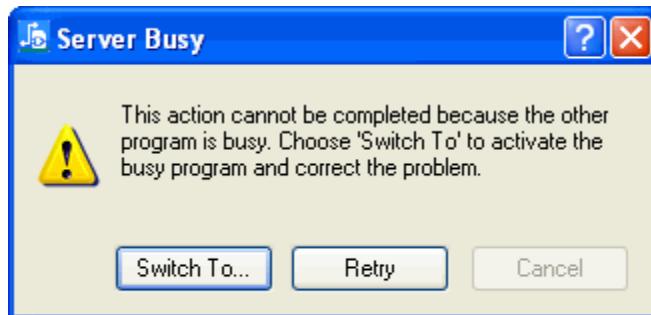
Name	Description
Options button	Opens the Options dialog box on page 606.
Plot/Print button	Sends the output to the printer or plotter.
Preview area	Graphically shows what you will print or plot.

Server Busy Dialog Box

To access: There is no sure way to access this dialog box, but it may appear when you attempt to connect with one of the other SailWind software applications, for example, when you click one of the buttons in the [Connect to SailWind Layout Dialog Box](#).

The Server Busy dialog box appears when you are launching another application from SailWind Logic and for some reason, the other application is slow to respond.

Figure 152. Server Busy Dialog Box



Objects

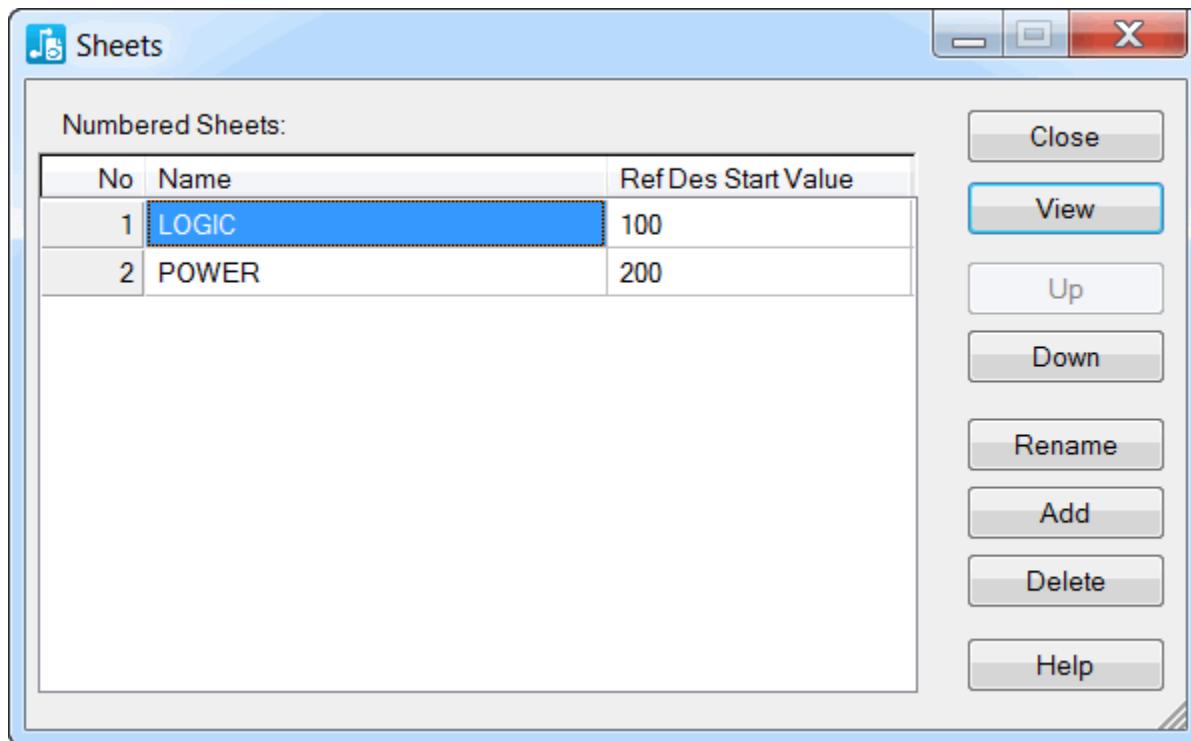
Table 213. Server Busy Dialog Box Contents

Name	Description
Switch To button	Switches to the application being launched. This is typically required when a prompt window in the other application is waiting for your input before you can connect to it with SailWind Logic.
Retry button	Attempts to connect to the other application again. This is typically required when there has been a delay in launching the other application.  Tip Wait until you see the application appear in your Windows Taskbar then click Retry .

Sheets Dialog Box

To access: **Setup > Sheets** menu item

Use the Sheets dialog box to edit the sheet set of the current schematic in the work area. Using Sheets enables you to add and delete sheets from the set and to modify sheet names and the numeric order of the set. You can create up to 1024 sheets.



Objects

Table 214. Sheets Dialog Box Contents

Name	Description
Numbered Sheets table	<ul style="list-style-type: none"> Name — Type a name for the schematic sheet. RefDes Start Value — The minimum number to use for reference designators of new components or copied/pasted components. The first available number equal to or greater than this value is used. <p>The RefDes Start Value stays with the sheet if the order of sheets is changed. For more information, see “Setting Reference Designators by Sheet in a New Schematic” on page 281</p> <p>Bottom up hierarchy sheets are displayed but Top down hierarchy sheets are not displayed. If you need to renumber reference designators on a Top down sheet, see “Automatically Renumbering Reference Designators” on page 280.</p>

Table 214. Sheets Dialog Box Contents (continued)

Name	Description
View button	Specifies to show the selected sheet in the view area.
Up/Down buttons	Moves the selected sheet up or down in the table.  Tip The sheet value changes as you move it up or down.
Rename button	Makes the selected cell available for editing.  Tip Spaces are not valid characters in a sheet name.
Add button	Adds a new sheet as a row to the bottom of the table.
Delete button	Removes the selected sheet.

Related Topics

[Editing Sheets](#)

Signal Pin Nets Dialog Box

To access:

- **Edit > Select Signal Pin Nets** menu item
- With nothing selected > right-click > **Select Signal Pin Nets** menu item

Use the Signal Pin Nets dialog box to help you find and select specific signal pins in the schematic.

Figure 153. Signal Pin Nets Dialog Box



Objects

Table 215. Signal Pin Nets Dialog Box Contents

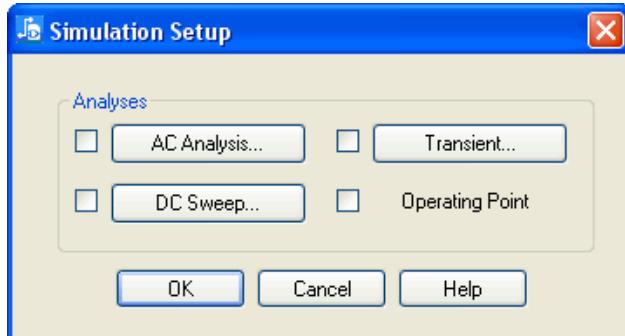
Name	Description
Select Nets list	Selecting nets in this list box selects signal pin nets within the schematic across all components and sheets. Using the right-click menu, you can perform object mode commands on the selected signal pin nets.
Refresh button	Updates the list of signal pin nets (in the Select Nets list box) to include any recently created signal pin nets. For example, when you add part 7400 to an empty design, you create two signal pin nets: GND and +5V. Selecting Refresh includes the new signal pin nets in the Select Nets list box.

Simulation Setup Dialog Box

To access: **Tools > SPICE Netlist** menu item > **Simulation Setup** button

After you add parts (with SPICE attributes) to your schematic, or add SPICE attributes to existing parts, you can create a SPICE netlist in preparation for simulation.

Figure 154. Simulation Setup Dialog Box



Objects

Table 216. Simulation Setup Dialog Box Contents

Name	Description
AC Analysis button	Opens the AC Analysis Dialog Box .
DC Sweep button	Opens the DC Source Sweep Analysis Dialog Box .
Transient button	Opens the Transient Analysis Dialog Box .
Operating Point	Directs the SPICE simulator to determine the DC operating point of the circuit.

Related Topics

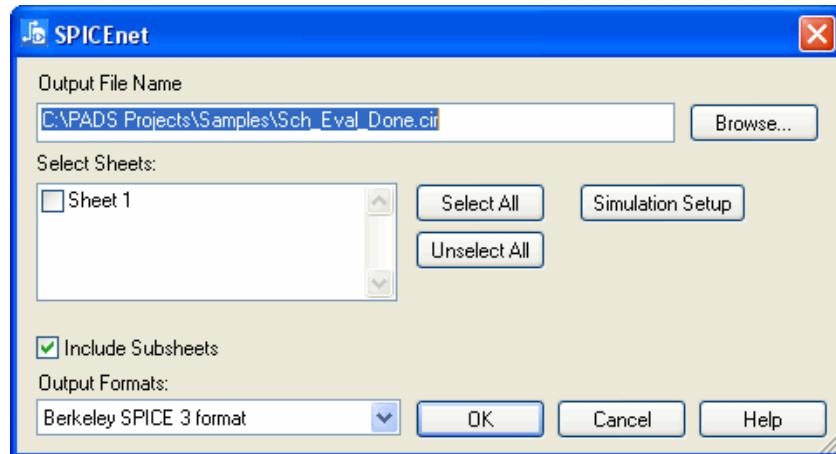
[Creating a SPICE Netlist](#)

SPICEnet Dialog Box

To access: Tools > SPICE Netlist menu item

After you add parts (with SPICE attributes) to your schematic, or add SPICE attributes to existing parts, you can create a SPICE netlist in preparation for simulation.

Figure 155. SPICEnet Dialog Box



Objects

Table 217. SPICEnet Dialog Box Contents

Name	Description
Output File Name	Specifies the path and name of the SPICE netlist file. Tip Type the path or use the Browse button to search for a path.
Select Sheets list	Specifies the sheets to include in the SPICE netlist.
Select All button	Specifies to select all sheets in the Select Sheets list.
Unselect All button	Specifies to clear all the selected sheets in the Select Sheets list.
Simulation Setup button	Opens the Simulation Setup Dialog Box .
Include Subsheets	Specifies to include any underlying hierarchy if the design is hierarchical.
Output Formats list	Specifies the target SPICE software.

Related Topics

[Creating a SPICE Netlist](#)

Step and Repeat Dialog Box

To access: Select an object during an add or duplicate operation > right-click > **Step and Repeat** menu item. When adding a new object in the Schematic Editor, you must place the first object manually before you can use Step and Repeat.

Use Step and Repeat to multiply objects as you place them during an add or duplicate operation. Step and Repeat is available in the Schematic Editor and the Decal Editor. In the Schematic Editor the Step and Repeat command copies parts, connections, text, or drafting items. In the Decal Editor the Step and Repeat command copies terminals, text, or drafting items.

Figure 156. Step and Repeat Dialog Box



Objects

Table 218. Step and Repeat Dialog Box Contents

Name	Description
Direction	Specifies the direction of placement for the array.
Count	Specifies the number of objects to place.
Distance	Specifies the distance between objects. Tip If you place a second object and then Step and Repeat, the spacing between the objects will become the default value in the Distance box and will repeat the pattern you've started.
Preview button	Displays the placement of the multiple objects based on the options you set. The placement of the objects is based on the location of the original object selected. Tip Zoom Mode is available during Step and Repeat.

Related Topics

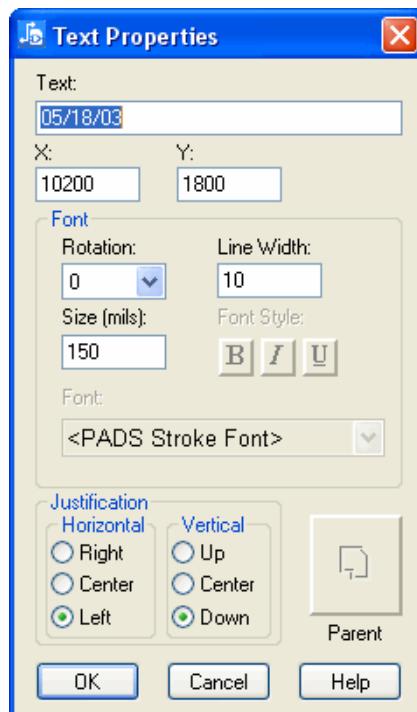
[Step and Repeat](#)

Text Properties Dialog Box

To access: Select text > right-click > **Properties** menu item

Use the Text Properties dialog box to edit the properties of free text in the design.

Figure 157. Text Properties Dialog Box



Objects

Table 219. Text Properties Dialog Box Contents

Name	Description
Text	Specifies the text that you are editing.
X/Y	Type coordinates to place the text in a specified location. Negative coordinates are not permitted. If you want to place text outside the sheet, you must move it there with the cursor.
Rotation	Specifies the rotation of the text: 0 or 90 degrees.
Line Width	Specifies the line width for stroke fonts only.  Stroke Line Width

Table 219. Text Properties Dialog Box Contents (continued)

Name	Description
Size	<p>Specifies the size of the font.</p> <p>Size (pts): This is font size in points and appears for system fonts</p> <p>Size (mils): This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters.</p>  <p>Stroke Font - Size</p>
Font Style	<p>Enables you to change the font style to bold, italic, and underlined.</p> <p> Restriction: System fonts only.</p>
Font list	<p>The fonts available to you. This lists either stroke fonts or system fonts.</p> <p>You choose which type of font to use in the Fonts Dialog Box.</p> <p> Tip</p> <ul style="list-style-type: none"> • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: B for bold, I for italic, or U for underlined.
Horizontal/Vertical Justification	<p>Specifies the horizontal (Right, Center, Left) justification and the vertical (Up, Center, Down) justification of the text.</p>
Parent button	<p>Opens the Drafting Properties Dialog Box.</p> <p> Restriction: Available only if the text had been combined with a drafting object.</p>

Related Topics

[Modifying Text](#)

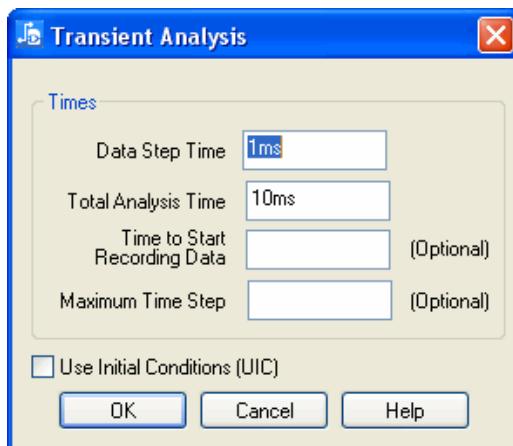
[Add Free Text Dialog Box](#)

Transient Analysis Dialog Box

To access: Tools > SPICE Netlist menu item > Simulation Setup button> Transient button

Use the Transient Analysis dialog box to set options specifically for a Transient analysis.

Figure 158. Transient Analysis Dialog Box



Objects

Table 220. Transient Analysis Dialog Box Contents

Name	Description
Data Step Time	Specifies the increment for the analysis.
Total Analysis Time	Specifies the time to end the analysis.
Time to Start Recording Data	Specifies the time to start recording data from the analysis. i Tip Use this if your simulation files become too large and you are not interested in data from the beginning of the analysis.
Maximum Time Step	Specifies the maximum time step value.
Use Initial Conditions (UIC)	Specifies to use SPICE to solve for the quiescent operating point before beginning the transient analysis. SPICE uses the values specified using IC=... on the various elements as the initial transient condition and proceeds with the analysis.

Related Topics

[Creating a SPICE Netlist](#)

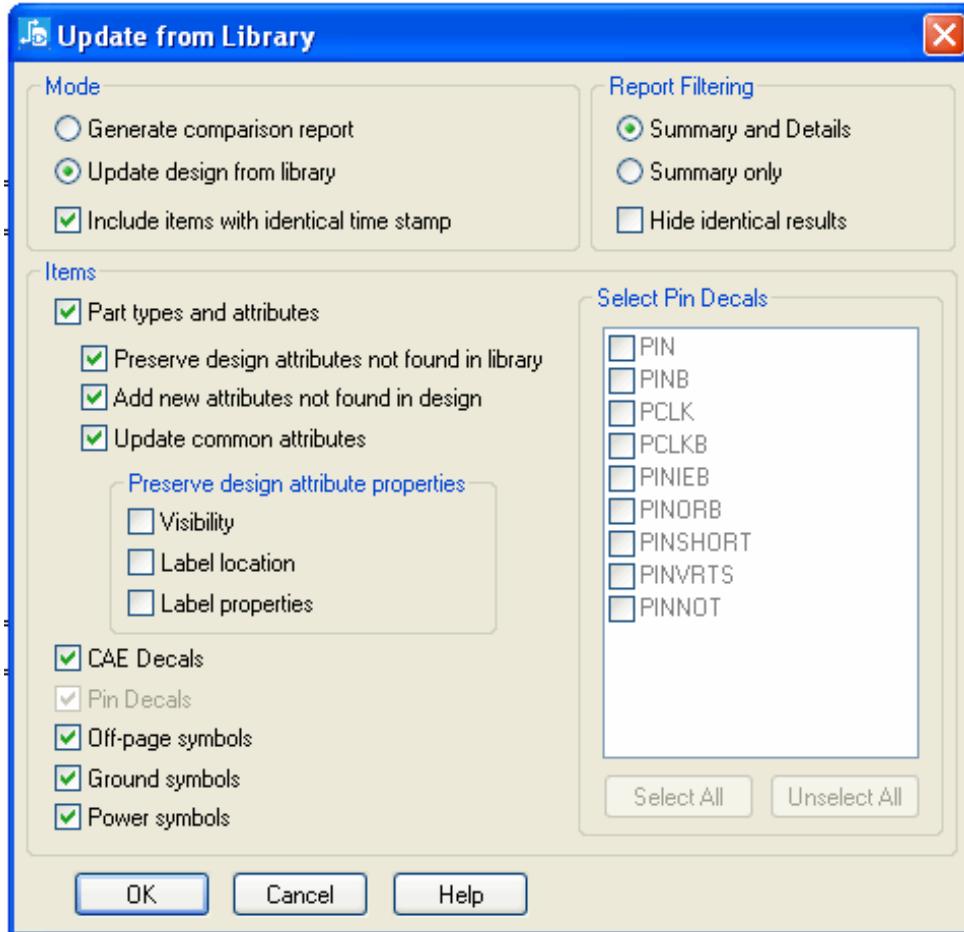
[Setting Up Transient Analysis](#)

Update From Library Dialog Box

To access: Tools > Update from Library menu item

Use the Update from Library dialog box to update a schematic from the library, or to compare items in a schematic with those in the library.

Figure 159. Update From Library Dialog Box



Objects

Table 221. Update From Library Dialog Box Controls

Control	Description
Mode area — Choose compare or update mode.	
Generate comparison report	Select this check box to compare library and schematic items and generate a report file.

Table 221. Update From Library Dialog Box Controls (continued)

Control	Description
Update design from library	Select this check box to compare library and schematic items, update the schematic from the library, and generate a report file.
Include items with identical time stamp	<p>Timestamps are assigned and updated in the SailWind Logic Library routines, but it is possible for items with identical timestamps to have different content in the library and schematic <i>if the item is edited outside the Library routines</i>.</p> <p>For example, if you export a schematic to an ASCII file, manually edit a part type in the ASCII file, and import the schematic back into SailWind Logic, the timestamp of the part type will be unchanged, but the content will be different.</p> <p>Use this check box to specify whether to compare/update items whose timestamps are the same in the library and schematic.</p> <p>Items with identical timestamps are <i>not</i> compared or updated unless this check box is selected.</p>
Report Filtering area — Specify what you want to see in the report.	
Summary and details	Select Summary and details or Summary only.
Summary only	Select Hide identical results to see only the differences between library and design items in the report.
Items area — Specify the items you want to compare or update.	
Part types and attributes	<p>Select this check box to include part types and their attributes in the comparison or update.</p> <p>Tip</p> <ul style="list-style-type: none"> Attributes don't have timestamps, so they can't be updated independently of their part types. If a schematic attribute's corresponding attribute in the library is a placeholder attribute (with a blank value), it will be updated only if the "Allow overwriting of attribute values in design with blank values from library" check box in the Options Dialog Box, Design Category.
Preserve design attributes not found in library	<p>Use this check box to specify what you want to do with attributes that are found in the schematic but not in the library:</p> <ul style="list-style-type: none"> Select the check box to keep these attributes in the updated parts. Clear it to remove them from the updated parts.

Table 221. Update From Library Dialog Box Controls (continued)

Control	Description
Add new attributes not found in design	<p>Use this check box to specify what you want to do with attributes that are found in the library but not in the schematic.</p> <ul style="list-style-type: none"> • Select the check box to add these attributes to the updated parts. • Clear it to not add them to the updated parts.
Update common attributes	<p>Use this check box to specify what you want to do with attributes that are found in both the schematic and the library.</p> <ul style="list-style-type: none"> • Select the check box to update the parts with the library versions. • Clear it to leave the part attributes as they are (that is, do not update them).
Preserve design attribute properties Visibility Label location Label properties	<p>Select the attribute properties you want to preserve in the schematic when updating the attributes from the library.</p>
CAE Decals	<p>Select this check box to include CAE decals in the comparison/update.</p> <ul style="list-style-type: none"> • If you want to update CAE decal assignments in the part types, you must also select Part types and attributes. • Pin decals are updated as part of a CAE decal update, so when this check box is selected, Pin Decals is automatically selected also.
Pin Decals	<p>Select this check box to include pin decals in the comparison/update.</p> <p> Tip</p> <ul style="list-style-type: none"> • Only the decal is updated; pin names and numbers are not changed. • Pin decals are updated as part of a CAE decal update, so when CAE Decals is selected, this check box is automatically selected and unavailable.
Select Pins Decals List	<p>This list is populated with check boxes representing each of the pins available for comparison or update. Enable the check box for each of the pins you want to compare or update.</p>

Table 221. Update From Library Dialog Box Controls (continued)

Control	Description
	 Tip: Because <i>all</i> pin decals are compared/updated as part of a CAE decal update, these check boxes are unavailable and ignored when CAE Decals is selected.
Off-page symbols	Select this check box to include off-page symbols in the comparison/update. Selecting this check box opens the Remap Special Symbols Dialog Box .  Restriction: The off-page symbol update will fail if there is not at least one off-page symbol in the current schematic.
Ground symbols	Select this check box to include ground symbols in the comparison/update. Selecting this check box opens the Update From Library Dialog Box .  Restriction: The ground symbol update will fail if there is not at least one ground symbol in the current schematic.
Power symbols	Select this check box to include power symbols in the comparison/update. Selecting this check box opens the Update From Library Dialog Box .  Restriction: The power symbol update will fail if there is not at least one power symbol in the current schematic.

Related Topics

[The Update From Library Function](#)

[How to Read the Update Report](#)

[The Compare/Update Process](#)

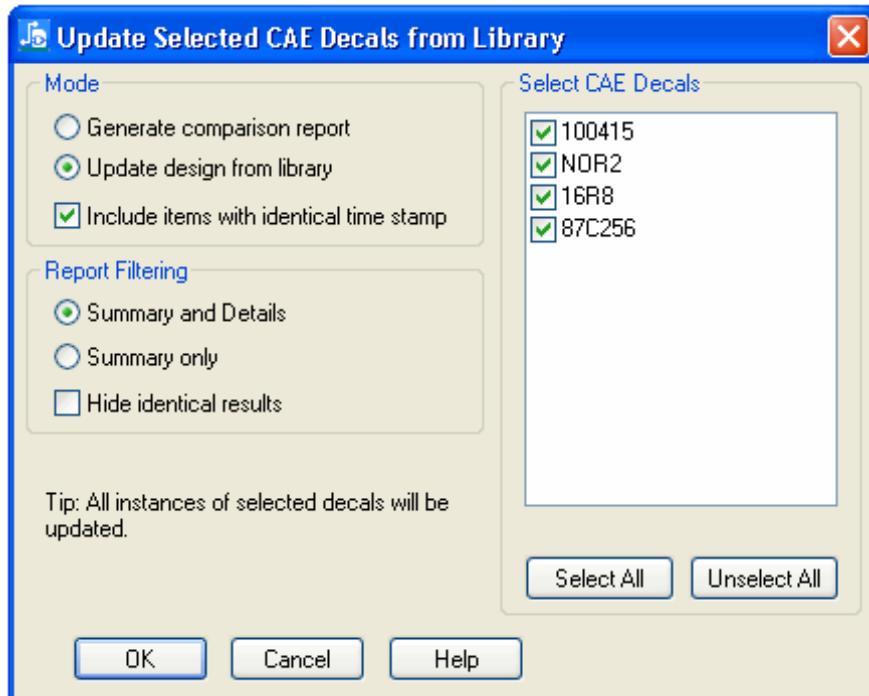
[Updating a Schematic From the Library](#)

Update Selected CAE Decals From Library Dialog Box

To access: Select a part > Right-click > **Update** > **CAE Decal** menu item

Use the Update Selected CAE Decals from Library dialog box to update selected CAE decals in a schematic from the library. All instances of the selected CAE decals are updated.

Figure 160. Update Selected CAE Decals From Library Dialog Box



Objects

Table 222. Update Selected CAE Decals From Library Dialog Box Controls

Control	Description
Mode area — Choose compare or update mode.	
Generate comparison report	Select this check box to compare library and design CAE decals and generate a report file.
Update design from library	Select this check box to compare library and schematic CAE decals, update the schematic from the library, and generate a report file

Table 222. Update Selected CAE Decals From Library Dialog Box Controls (continued)

Control	Description
	 Tip <ul style="list-style-type: none"> • This procedure updates the pin assignments in the selected CAE decals, but doesn't update the pin decals themselves. Use one of the procedures in Updating Selected Pin Decals From the Library to update the pin decals' geometries. • As a corollary, if, for instance, CAE decal X is updated to use PINB instead of PINA, it uses the version of PINB geometry currently in the schematic. If there is no PINB in the schematic, it is installed from the library.
Include items with identical time stamp	<p>Timestamps are assigned and updated in the SailWind Logic Library routines, but it is possible for items with identical timestamps to have different content in the library and schematic <i>if the item is edited outside the Library routines</i>.</p> <p>For example, if you export a schematic to an ASCII file, manually edit a CAE decal in the ASCII file, and import the schematic back into SailWind Logic, the timestamp of the decal will be unchanged, but the content will be different.</p> <p>Use this check box to specify whether to compare/update items whose timestamps are the same in the library and schematic.</p> <p>Items with identical timestamps are <i>not</i> compared or updated unless this check box is selected.</p>
Select CAE Decals Area — Specify which CAE decals to update.	
Select CAE Decals List	This list is populated with check boxes representing each of the selected CAE decals. Enable the check box for each of the CAE decals that you would like to update.
Report Filtering area — Specify what you want to see in the report.	
Summary and details Summary only Hide identical results	Select Summary and details or Summary only. Select Hide identical results to see only the differences between library and schematic items in the report. This will shorten the report.

Related Topics

[The Update From Library Function](#)

[How to Read the Update Report](#)

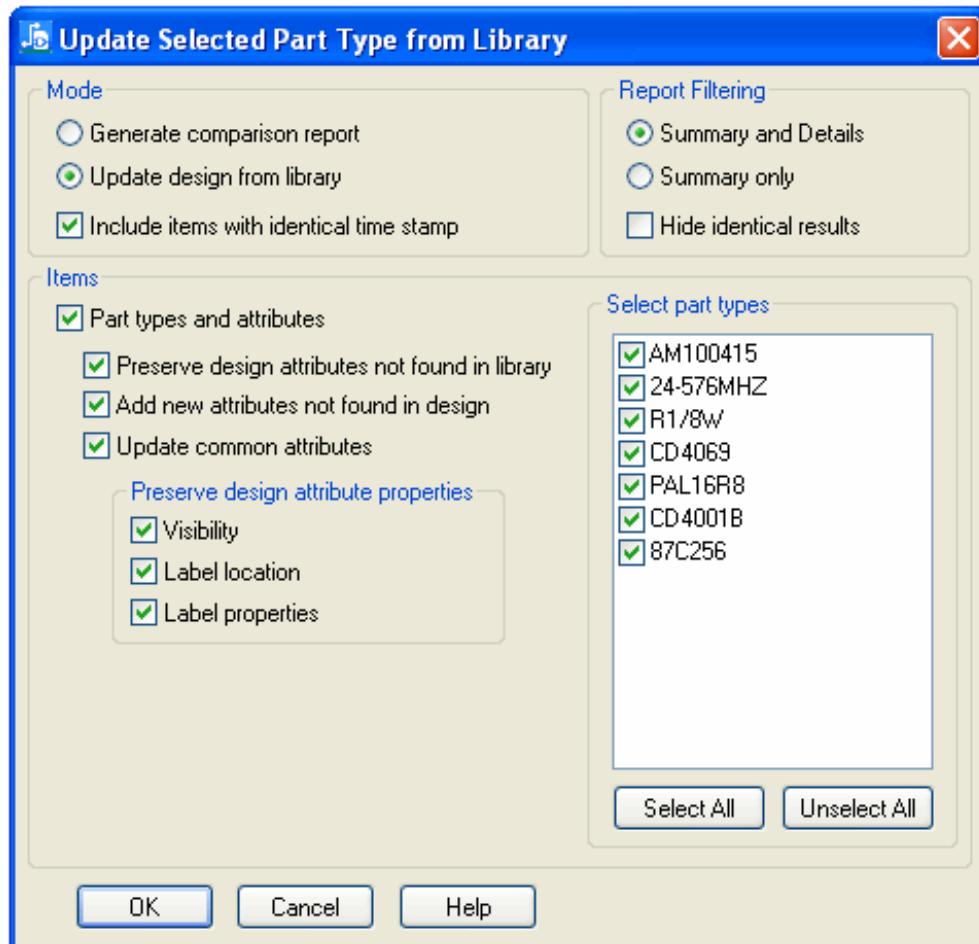
[The Compare/Update Process](#)

[Updating Selected CAE Decals From the Library](#)

Update Selected Part Type From Library Dialog Box

To access: In the Schematic Editor, select the part types you want to compare/update. Right-click > **Update > Part Type** menu item

Use the Update Selected Part Type from Library dialog box to update selected parts in a schematic from the library, or to compare selected parts in a schematic with those in the library.



Objects

Table 223. Update Selected Part Type From Library Dialog Box Controls

Control	Description
Mode area — Choose compare or update mode.	
Generate comparison report	Select this check box to compare library and schematic part types and generate a report file.

Table 223. Update Selected Part Type From Library Dialog Box Controls (continued)

Control	Description
Update design from library	Select this check box to compare library and schematic part types, update the schematic from the library, and generate a report file.  Restriction: All parts having the same part type as a selected part are updated, but attributes are updated only for the parts actually selected.
Include items with identical time stamp	Timestamps are assigned and updated in the SailWind Logic Library routines, but it is possible for items with identical timestamps to have different content in the library and schematic <i>if the item is edited outside the Library routines</i> . For example, if you export a schematic to an ASCII file, manually edit a part type in the ASCII file, and import the schematic back into SailWind Logic, the timestamp of the part type will be unchanged, but the content will be different. Use this check box to specify whether to compare/update items whose timestamps are the same in the library and schematic. Items with identical timestamps are <i>not</i> compared or updated unless this check box is selected.
Report Filtering area — Specify what you want to see in the report.	
Summary and details	Select Summary and details or Summary only.
Summary only	Select Hide identical results to see only the differences between library and design items in the report. This will shorten the report.
Items area — Specify the items you want to compare or update.	
Part types and attributes	Make sure that this check box is selected.  Tip <ul style="list-style-type: none"> Attributes don't have timestamps, so they can't be updated independently of their part types. If a schematic attribute's corresponding attribute in the library is a placeholder attribute (with a blank value), it will be updated only if the Allow overwriting of attribute values in design with blank values from library check box in the Options Dialog Box, Design Category.
Preserve design attributes not found in library	Use this check box to specify what you want to do with attributes that are found in the schematic but not in the library: <ul style="list-style-type: none"> Select the check box to keep these attributes in the updated parts.

Table 223. Update Selected Part Type From Library Dialog Box Controls (continued)

Control	Description
	<ul style="list-style-type: none"> • Clear it to remove them from the updated parts.
Add new attributes not found in design	<p>Use this check box to specify what you want to do with attributes that are found in the library but not in the schematic.</p> <ul style="list-style-type: none"> • Select the check box to add these attributes to the updated parts. • Clear it to not add them to the updated parts.
Update common attributes	<p>Use this check box to specify what you want to do with attributes that are found in both the schematic and the library.</p> <ul style="list-style-type: none"> • Select the check box to update the parts with the library versions. • Clear it to leave the part attributes as they are (that is, do not update them).
Preserve design attribute properties Visibility Label location Label properties	Select the attribute properties you want to preserve in the schematic when updating the attributes from the library.
Select Part Types Area — Specify which part types to update.	
Select part types list	This list is populated with check boxes representing each of the selected part types. Enable the check box for each of the part types that you would like to update.

Related Topics

[The Update From Library Function](#)

[How to Read the Update Report](#)

[The Compare/Update Process](#)

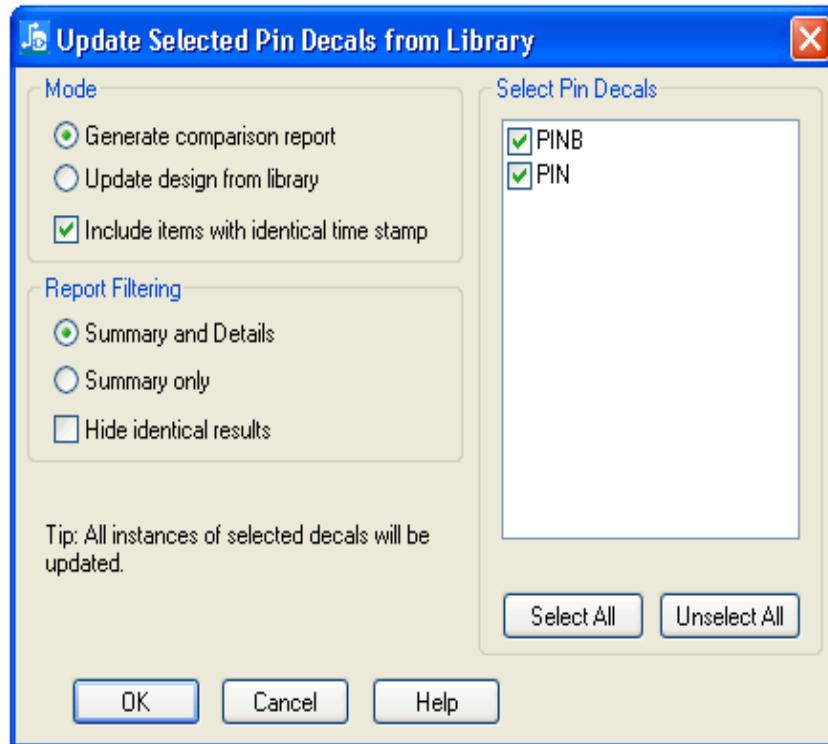
[Updating Selected Part Types From the Library](#)

Update Selected Pin Decals From Library Dialog Box

To access: Select a pin > Right-click > **Update Pin Decal** menu item

Use the Update Selected Pin Decal from Library dialog box to update selected pin decals from the library, or to compare selected pin decals in a schematic with those in the library. All instances of the selected pin decals are updated.

Figure 161. Update Selected Pin Decals From Library Dialog Box



Objects

Table 224. Update Selected Pin Decals From Library Dialog Box Controls

Control	Description
Mode area — Choose compare or update mode.	
Generate comparison report	Select this check box to compare library and schematic pin decals and generate a report file.
Update design from library	Select this check box to compare library and schematic pin decals, update the schematic from the library, and generate a report file.
Include items with identical time stamp	Timestamps are assigned and updated in the SailWind Logic Library routines, but it is possible

Table 224. Update Selected Pin Decals From Library Dialog Box Controls (continued)

Control	Description
	<p>for items with identical timestamps to have different content in the library and schematic <i>if the item is edited outside the Library routines</i>.</p> <p>For example, if you export a schematic to an ASCII file, manually edit a pin decal in the ASCII file, and import the schematic back into SailWind Logic, the timestamp of the decal will be unchanged, but the content will be different.</p> <p>Use this check box to specify whether to compare/update items whose timestamps are the same in the library and schematic.</p> <p>Items with identical timestamps are <i>not</i> compared or updated unless this check box is selected.</p>
Select Pin Decals Area — Specify which pin decals to update.	
Select Pin Decals List	<p>This list is populated with check boxes representing each of the selected pin decals. Enable the check box for each of the pin decals that you would like to update.</p>
Report Filtering area — Specify what you want to see in the report.	
Summary and details Summary only Hide identical results	<p>Select Summary and details or Summary only.</p> <p>Select Hide identical results to see only the differences between library and schematic items in the report. This will shorten the report.</p>

Related Topics

[The Update From Library Function](#)

[How to Read the Update Report](#)

[The Compare/Update Process](#)

[Updating Selected Pin Decals From the Library](#)

Glossary

absolute coordinates

Coordinates of a location based upon their distance from the origin (coordinates 0,0) of the design area.

accelerator keys

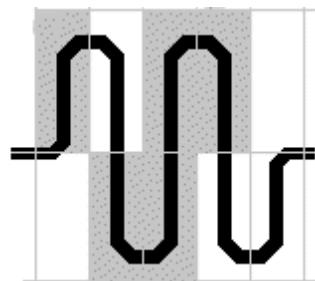
Key sequences used to invoke commands and change system settings without using the mouse. Accelerator keys are called shortcut keys in the SailWind product documentation.

accessible nets

Nets for which you can define test points. DFT Audit analyzes all nets. If DFT Audit determines that test probes can access them, the nets are accessible (also called adaptable).

accordion

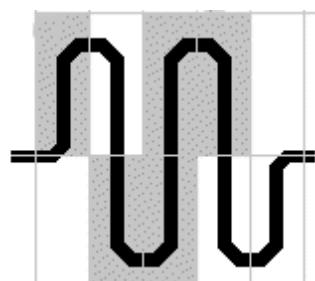
A trace pattern resembling a signal wave that adds length to traces. The trace patterns are contiguous and do not include layer changes.



accordion gap

The gap of an accordion sets the pitch between chords. The gap is a user-definable number multiplied by the same net trace-to-corner clearance.

If the same-net trace-to-corner distance equals zero, then Trace Width is used for the gap calculation.



See also [accordion](#), [amplitude](#), [pair routing gap](#)

acid trap

An acid trap is a location where acid gets trapped in an area due to the surface tension of the etching. This acid causes over-etching, which hurts yield.

active component

The active substituted component in an assembly variant. Active means that this substitution of the component is used in the current variant.

See also [default.asc](#)

active layer

The design layer to which new information is added. You select the active layer by choosing the layer in the Layer list on the Standard Toolbar. You can also do this by using the L modeless command.

ACTM#

The 16-digit number found on your security key.

adaptable nets

See [accessible nets](#)

adhesive

A substance used to attach the bodies of devices to a PC board.

aggressor nets

When using the Electrodynamic Checking program (EDC), a net or pin pair that is considered a source of interference.

align

To reposition placed parts to match the alignment of another part.

alignment tool

A small, temporary marker at each location where dimensioning occurs.

alpha pins

Pins with descriptive letters that are substituted for pin numbers. For example, GND for the ground pin. Alphanumeric pin assignments are made in the Library Manager's part type editor.

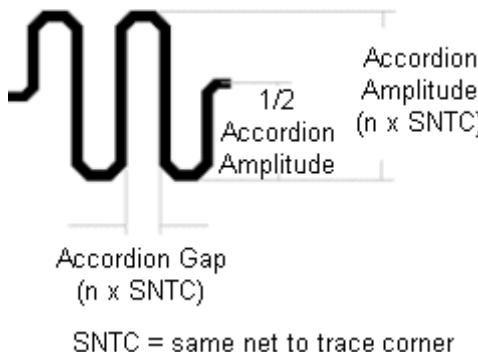
alphanumeric pins

Pins with alphanumeric pin numbers. An alphanumeric name consists of a prefix and suffix. The prefix or the suffix can contain either alpha letters or numeric numbers. For example, A1, 1A, or even DATA07 (consists of the prefix "DATA" and the suffix "07").

amplitude

The amplitude of an accordion sets the accordion height (for horizontal accordions) or accordion width (for vertical accordions). The amplitude is a user-definable number, multiplied by the same net trace-to-corner clearance.

If the same-net trace-to-corner distance equals zero, then Trace Width is used for the amplitude calculation.



See also [accordion, gap \(accordion\)](#)

analog board

A board with mostly discrete components and minimal integrated circuits.

analog circuit

A design composed of discrete components such as capacitors, resistors, and diodes.

angstrom

1/10,000 of a micrometer (10-4um).

annotation (forward and backward)

Forward annotation refers to the process of updating the design file to match the schematic file.
Backward annotation refers to updating the schematic file to match the design file.

annular pad

A pad shape that enables you to specify an inside and an outside diameter. This creates a donut shape because the inner hole was used to center the drill bit when boards were hand-drilled on a drill press. Though obsolete, the annular pad is still offered for special circumstances.

annular ring

The conductive pad material surrounding the hole. The annular ring radius = pad diameter-(finished hole size) / 2.

antipad

For plane layers, a slightly oversized pad diameter that plots as a clearance for through-hole pins that should not connect to the plane.

any-angle coupling trace

Part of a route that connects SBP fanouts to serpentine routes.

aperture

A uniquely shaped window or hole that is attached to an aperture wheel on a photoplotting machine.

aperture table

A table that matches the line widths necessary to print your design with the plotter setup. SailWind Layout can prepare the table automatically, or you can prepare it manually.

Artwork for printed circuit manufacturing is created by exposing clear film to light that is passed through the aperture. Although the aperture wheel has been made obsolete by laser plotters, an aperture table is still necessary to drive laser plotters.

apl.dcr

A setup file for Novell network security.

application-specific integrated circuit

An IC designed to meet a specific customer requirement.

area select

A method for selecting an object or a group of objects. If you enable area select by clicking Filter on the Edit menu, a selection rectangle is created and all items within the rectangle are selected.

array

A group of items, such as bonding pads, that are arranged in rows and columns.

artwork

Clear film with darkened areas representing pads and connecting traces, and used for manufacturing a printed circuit board. Each layer of a design has its own unique artwork, such as silkscreen and solder mask.

.asc

The file extension used to identify a proprietary PADS-format ASCII file.

ASCII format

A translation format that uses ASCII text to define the PCB design. ASCII format is widely used to list the parts and connections in a design, to import and export design items, and to check the design for binary corruption.

ASIC

An acronym for *application-specific integrated circuit*.

assembly drawing

A final design document that provides the part name, type, and orientation for each device on a printed circuit board. An assembly drawing is used for assembly of the final product.

assembly variant

A specific manufacturing configuration of a PCB. Assembly variants specify which components are used, which are not used, and which are substituted with a different decal part type. Several assembly variants can exist for a single PCB.

associated net

See [electrical net](#).

associating component

A component through which an electrical net passes.

associating copper

Copper combined with the terminals in the PCB Decal Editor.

attribute groups

A group of structured attributes. For example, the DFT group includes the following attributes:

- DFT.Nail Count Per Net
- DFT.Nail Number
- DFT.Nail Diameter

attributes

Attributes contain information you have associated with an object in your design. Attributes contain the types of part information that can be included in the parts library description and exported to a parts list. Examples are part manufacturer, package type, order number, and so on.

Auto Dimensioning tab

The tab on the Options dialog box that determines the appearance of newly created dimensions.

automation

A way for heterogeneous applications to communicate with each other. SailWind products make some data, such as the database in use, and some functionality, such as opening files or selecting objects, available to other applications.

autorouter pass types

Pass types are part of an autorouting strategy that determines how the autorouter routes a design.

Pass	Description
Center	Places traces equidistant from component pins or vias and each other to evenly distribute any available space in the channel.
Fanout	Places vias for inaccessible SMD component pins and routes from the vias to the pins.
Miters	Converts all route corners of a specified angle to diagonal corners.
Optimize	Analyzes each trace and tries to improve the quality of the route pattern by removing extra segments, reducing via usage, and shortening trace lengths.
Patterns	Searches for groups of unrouted connections that can be completed using typical C routing patterns, Z routing patterns, and memory patterns and then routes them.
Route	Sequentially routes each unroute until all connections are attempted.

Pass	Description
Test Point	Analyzes the testability of the design, determines which nets require testing, adjusts the routes, and inserts test points to improve testability.
Tune	Adjusts the length of length-controlled traces. The Tune pass tunes all routed traces with length rules, and automatically adjusts length-controlled traces to meet design rules.

axial lead

A connection pin that protrudes straight out from the component body and bends at 90 degrees for insertion into the PC board. An axial lead is usually associated with discrete components such as resistors, capacitors, or diodes.

back-annotate

Update a schematic file to match its design file.

ball bonding

A bonding technique that provides increased contact between a gold wire and a chip bond pad. This method uses thermal compression to melt gold wire to form a ball.

ball grid array

A packaging method that uses a substrate to interconnect one or more die to an array of solder alloy spheres.

base option

The Base Option, in Assembly Variants, contains all of the common components in all of the existing variants; in other words, it contains a filtered database. If you uninstall or substitute components in a variant, they are removed from the Base Option. Therefore, the Base Option, because it contains only installed options, is also a subset of the raw database. You can use the Base Option to view all of the items in all of the variants, or the base of all variants.

The Base Option always exists; you cannot delete it.

base part

When making a union, the part type of the first selected part. Base parts can either be left in position and joined by secondary parts, or repositioned to imitate the first selected prototype part.

baseline dimensioning

A type of dimensioning in which a series of dimensions have a common start point, such as datum dimensioning.

basic units

A basic unit is the smallest unit of measurement in a SailWind database. All values in the database are stored in binary format basic unit and are converted to the current user units (mils, mm, or inches) for screen display. If you need to re-import the information to .pcb format, export in basic units.

Conversions are:

- 1 mil = 38100 basic units
- 1 millimeter = 1500000 basic units

BGA

An acronym for *ball grid array*.

BGA fanout

A single-segment fanout that connects BGA array pads to BGA vias. This single-segment fanout always ends in a via.

BGA/PGA decals

A full matrix decal for BGAs and PGAs, including staggered array patterns.

biased pin pair

A layer biased pin pair is any pin pair with a design rule specifying a layer bias to one or more, but not all, electrical routing layers.

blind via

A via that connects an outer layer to one or more inner layers, without passing through all other layers of a printed circuit board.

bmp

An image file that can be pasted into documents or other programs such as Microsoft Word. SailWind products use the Copy Bitmap command to capture these as screen images.

board markings

Designers usually include identification information on a board. These may include the board part number, the assembly part number, the company name, the product name, the revision level, the serial number, the copyright notice, an anti-static symbol, warning messages, UL labels, test labels and many other types of information. This information may be in ink on the silkscreen layers, in copper on the top and/or bottom layer or some combination of the two. These are typically referred to as board markings.

Add text to an electrical layer and it will be created in copper. Add text to a Fabrication, Assembly, and Documentation Layer and it will be created during the silkscreen process.

Use the Text command to add board markings to your design.

See also Adding Free Text

board outline

The actual shape of the printed circuit board, defined by line segments and arcs. The board outline is entered on layer 0 and displayed on all layers.

bonding pads

Metallization areas placed around the perimeter of the integrated circuit die, to which aluminum or gold wires connect the die to the component package.

bounding rectangle

The smallest rectangle that encloses all nontext graphics on all layers.

breakpoint marker

A small brown dot in the Output window gutter that indicates a breakpoint in a script or macro.

bumped chip

A die or chip that has been specifically processed with buffer metals over the I/O pads, followed by an addition of solder or gold bumps to provide bonding areas for direct chip attachment onto a substrate.

buried via

A via that connects only inner layers.

bus

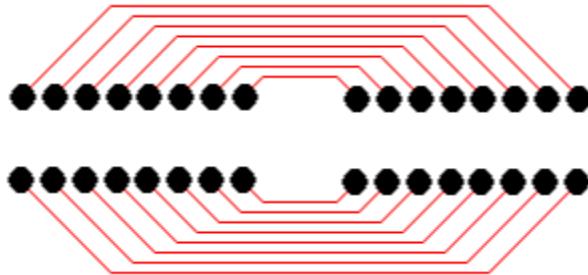
A series of connections that share a common use, such as memory array or data array, and are usually routed parallel to each other.

bus routing

Routing two or more pin pairs simultaneously and in close proximity to each other in neat, flowing patterns.

C routing pattern

A collection of routes that form a pattern resembling the letter C.



CAD

An acronym for Computer-Aided Design or Computer-Aided Drafting.

CAE

An acronym for Computer-Aided Engineering.

CAE Decal

The graphical representation of schematic symbols in SailWind products.

CAM

An acronym for Computer-Aided Manufacturing.

CAM document

A combination of plot type and output device you create and save with the design. For example, you can include "Silkscreen Top, Photoplot" and "Silkscreen Top, Laser Printer" on your CAM Documents List and run them selectively when needed.

CAMDⁱr

The *SailWindpcb.ini* file entry that enables you to specify the CAM master folder for creating CAM output.

capacitance

The ratio of charge within a trace that is a factor of the trace length and signal delay.

CBGA

An acronym for ceramic ball grid array.

CBP

An acronym for chip bond pad.

center pass

An autorouting pass that places traces equidistant from component pins or vias and each other to evenly distribute any available space in the channel.

CGA

An acronym for *column grid array*.

chamfered

A rectangle with the square corners cut off to create beveled edges on the corners.

chamfered path

A solid filled copper that, like a trace, acts as a conductor connecting pins and vias similar to a trace. But unlike a trace, which is created with a round aperture producing rounded outside corners, chamfered path copper allows for sharp specific outlines with a filled interior. When creating a chamfered path, you set options to create shapes with square or chamfered corners. The copper created by chamfered path has a Solid Copper property which overrides the Copper Hatch Grid and Drafting Line Width settings to make it a solid fill. Clearance rules for the chamfered path copper are also changed to match the clearance rules of a trace.

checking

Verifying the design meets previously defined rules, such as clearance and connectivity.

chip

An integrated circuit without packaging. A chip is also called a *die*.

chip bond pad

Interconnect areas on the die on which wire bonds are connected to the substrate.

chip carrier

A square or rectangular IC package, with I/O connections on four sides.

chip on board

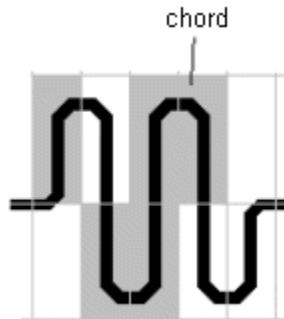
The packaging configuration in which a chip is bonded directly to a circuit board or substrate.

Chip Scale Package

A packaging configuration in which the dimension of the substrate is 1.2 times larger than the die.

chord

Half of an accordion.



See also [accordion](#), [amplitude](#), [gap \(accordion\)](#)

clam shell fixing

A test fixture that tests both the top and bottom side of the PCB.

class

A collection of nets with a common set of design rules.

clearance

The measured space between routed objects such as trace-to-trace, trace-to-pad, or pad-to-pad.

closed cluster

Clusters that you cannot delete or replace during automatic cluster creation.

cluster

In Cluster Placement, a group of parts that must be placed close to each other.

CMOS

An acronym for Complementary Metal Oxide Semiconductor.

COB

An acronym for Chip On Board.

coefficient of thermal expansion

A quantity used to determine the length change of a material due to temperature change. Thermal expansion differences between the die and substrate must be considered for quality assurance.

collapse

To relocate the members of a cluster from their current placement to the center of the cluster.

column grid array

Similar to a ball grid array, but columns are used to improve the stresses of different thermal expansion between the board and the component.

Com port

Abbreviation for communications port. This port provides a connection between your computer and peripheral devices, such as plotters, modems, and other computers.

combine

Joining lines, or lines and text, together as one selectable object.

component side

The top or front side of a printed circuit board where devices are normally mounted.

composite fanout

A fanout from a pin that is common to two subnets. Often created by autorouting operations.

Composite fanouts provide access to component pins that may otherwise be inaccessible.

See also [fanout](#), [subnets](#)

composite rule trace

A trace that is attached to a pin (typically an SMD) shared by two subnets. This type of trace is typically created by autorouting operations.

See also [composite fanout](#), [subnets](#)

conditional rules

Rules placed on a signal that apply only if the signal is routed near another specified signal. Conditional rules are also known as against rules.

conductor

A material that causes heat or electrical current flow. For printed circuit design, a conductor is a piece of metal that connects pins of components together.

connected islands

A maximum set of subnet items already connected by a trace, copper unroute, or jumper.

See also [subnets](#), [subnet](#)

connections

Points of connectivity, such as a pin pair or a net.

connector

A unique component used to connect a portion of a printed circuit board with other devices.

Constraint Manager

The Constraint Manager is a separate application used to input rules for SailWind Layout instead of the rules dialogs within the SailWind Layout application. The Constraint Manager is only used in the *integrated SailWind Designer project*.

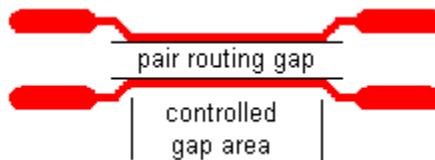
container application

An application that can incorporate embedded or linked items into its own documents. The documents managed by a container application must be able to store and display both OLE components, and data created by the application itself. A container application must also allow users to insert new items or edit existing items.

When you insert objects into a SailWind product, the SailWind product is the container application. When you insert a SailWind file into another application, the other application is the container application.

controlled gap area

The part of the differential pair where the traces are drawn routed in parallel and separated by the pair routing gap. The controlled gap zone area starts at the gathering point and ends at the split point.



See also [gathering point](#), [pair routing gap](#), [split point](#)

controlled gap length

For a differential pair, the ratio of the controlled gap area routing length to the overall routing length, in percentage.

See also [differential pairs](#)

controlled length net

A net that has length rules, or contains pin pairs that have length rules.

The following high-speed rules are net length rules:

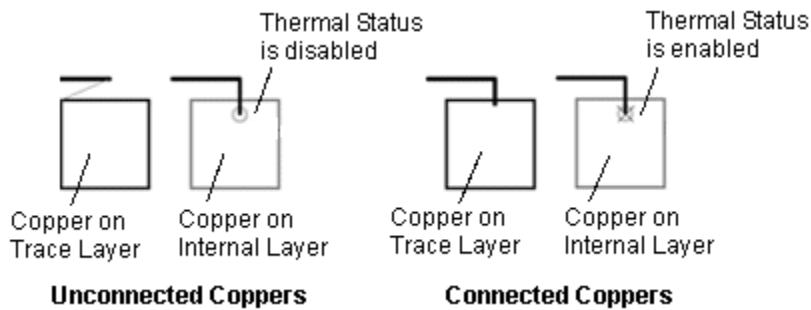
- Minimum/maximum length
- Matched length
- Differential pairs

converting database

The process that converts a non-native file, such as an *.asc* file or a *.dxf* file, to a SailWind native format, or *.pcb*, file.

copper connectivity

Means unroutes are always connected to a copper at some point in the copper outline. A copper outline can include arcs. The following graphic illustrates how copper connects to a net.



See also [coppers](#), [overlapping coppers](#)

copper plane

A copper shape with insulation areas around traces and pins that pass through the copper, but are not attached or connected to the copper. Can be placed on any layer, except CAM plane layers.

coppers

Polygons on an electrical layer representing an area of the PCB to fill with metal.

When a copper is assigned to a net, it is joined to the net with a trace or via. Coppers are obstacles to net objects unless the copper and the net belong to the same net.

See also [overlapping coppers](#), [copper connectivity](#)

copy route

The duplication of a trace or series of traces, using copy and paste.

corner

Point where a trace or line changes direction. The Selection Filter enables or disables picking geometric or route corners.

cost

Reduces usage of a layer. The higher the cost, the less a layer is used for routing.

cross-probing

Uses a link between SailWind programs to reflect, in one SailWind application, selections made in another SailWind application.

CSP

An acronym for chip scale package.

CTE

An acronym for coefficient of thermal expansion. It is also referred to as TCE.

cutouts

A closed polygon in a copper, copper plane, or board outline. In copper or copper planes, a cutout results in an area absent of copper.

See also [overlapping cutouts](#)

cycle picking

To sequentially select objects in the vicinity of the selection point using the Tab key.

dangling route

Dangling routes are stubs or spurs off of traces that are not tied to any pin by a ratsnest. See also [partial route](#).

database units

The use of mils, metric, or inches within a design.

datum dimensioning

A style of dimensioning in which all dimensions are measured from a common starting point. The origin extension line is marked as zero, with each dimension reflecting the measurement from that point.

See also [Creating Baseline Dimensions](#)

D-codes

Specific numbers assigned to photoplot machine apertures for program identification. D-CODES are included in the aperture table.

decal

The physical representation, or footprint, of a part.

decal copper

Open, closed, or associated copper produced within the physical representation of a component.

decal text

Documentation text produced within the physical representation of a component.

default component

The original component, before being replaced in the current assembly variant. The default component is always in the raw database, but not necessarily in the Base Option.

See also [active component](#)

default layer mode

A layer mode in which a design can consist of up to 30 electrical layers, or a combination of electrical and nonelectrical layers. You change from default layer mode to increased layer mode by clicking the Max Layers button in the Layers Setup dialog box.

default.asc

The ASCII file accessed for new file creation. This file provides startup design information such as grid sizes, default colors, or other information.

defaults

Conditions or options that are set when the SailWind product starts.

delay

The time it takes for a signal to travel through a trace.

delete

To remove information from a design.

design area

The actual work area where a design is created.

design on the fly

To use ECO Operations to create a new design without first providing a netlist or parts list from schematic software. This can also be called design.

Design category

The Options category that controls design conditions, general routing conditions, and certain display and part movement method settings.

design rules

Established spacing and general routing constraints for electrical properties, or conductors, which are verified by clicking Verify Design from the Tools menu.

devicesn.dat

A file usually found in the *C:\<install_folder>\<version>\Programs* folder that contains CAM printer and plotter driver data. This file must exist in the same folder specified by the UserDir *SailWindpcb.ini* variable.

DFM

An acronym for Design for Manufacturing.

DFT Audit

DFT Audit analyzes every net for accessibility (adaptability) and creates a board report that identifies all inaccessible (non-adaptable) nets.

dice

The plural of *die*.

die

A single square or rectangular piece of semiconductor material into which a specific electrical circuit has been fabricated.

die bonding

To attach the semiconductor die to the package substrate with epoxy adhesives, gold eutectic, or solder alloy. It is also referred to as Die Attachment.

die flag

Metal shapes placed under a die for thermal management and/or electrical connection; also referred to as a *flower pattern*.

die side of CBP

The side of the die on which the CBP lies. Usually the die side of the CBP is the same as its fanout side, but in some cases more complex patterns of wire bond fanout may mean that the two sides are not the same.

Die Wizard

This feature creates die part definitions parametrically or imports the die description using GDSII or formatted ASCII files. The Die Wizard replaces Component IQ by providing die capture directly in the Advanced Packaging Toolkit layout editor. This eliminates the need to transfer .cig files.

dielectric

A non-conductor of current; an insulator.

dielectric constant

A value given for manufacturing materials, such as FR-4, to describe electrical characteristics.

differential pairs

A group of two nets or two pin pairs routed side-by-side and separated by the pair routing gap for as much of the overall length as practical. A differential pair typically transmits two electrical signals that are driven 180 degrees out of phase from each other.

See also *pair routing gap*

digital board

A board with mostly integrated circuits in proportion to the analog components.

DIP

An acronym for Dual In-line Package.

DisableCaching

A *SailWindpcb.ini* file entry that, when set at 1, shuts off graphics optimization and, when set at 0, enables graphics optimization.

discrete device

A device that contains one circuit element. For example, a resistor or toggle switch.

disperse

A command that is active on several levels of Cluster Placement. When selected, it clears the board of all parts or clusters that are not glued down, and arranges them around the outside of the board outline according to decal type.

dispersion routes

Partial routes, ending in vias, which tie surface mount components to plane layers.

do file

The SPECCTRA router ASCII setup file that contains user-defined router commands to initiate batch routing.

dock

To take an isolated application dataset and pull the changes within the dataset into the main design project. Any conflicts with the merged data must be manually resolved.

documentation layers

Layers higher than the electrical layers in a SailWind Layout database that contain text and lines to illustrate assembly, annotation, and provide instructions for manufacturing.

double-click

Two mouse clicks, in immediate succession, that usually initiate an edit action or complete the current action.

double-sided board

A printed circuit board made up of two routing layers, and which has no internal layers.

double-sided die

A die that has substrate bond pads on one side, and a BGA grid array on the other side. The two sides are connected through vias.

See also [single-sided die](#)

drafting operations

Any operation that involves adding nonelectrical information, not associated with placement or routing, to a design.

drawn pads

Photoplot pads, usually finger pads, that are produced by opening the aperture and moving the board, with the aperture remaining open, to produce a pad shape.

DRC

An acronym for Design Rules Check.

drill chart

A diagram, produced on a drill drawing, that shows drill symbols matched with drill hole sizes. This is also referred to as a drill legend.

drill oversize

A factor applied to plated through holes for DRC purposes to account for drill oversizing during the PCB fabrication process.

drill pairs

Primarily for buried and blind vias, drill pairs define which layers are to be drilled and plated together during the fabrication process.

drill symbols

Unique symbols on a drill drawing plot that represent the various drill hole locations and sizes.

drill.dat

A user-definable ASCII file that determines settings for NC Drill output format options. This file must exist in the same folder specified by the UserDir variable in the *SailWindpcb.ini*.

DXF

An acronym for Data eXchange Format, a standard ASCII format for sharing graphics database files between different environments.

dxfset.dat

A file that contains the information for drill size and library name equivalents in basic units for the DXF Setup dialog box.

dynamic route

To create a route using the Dynamic Route tool, which automatically creates turns and pushes other routes aside to complete the connection.

ecad hint.map

A user-defined text file that you create, edit, and maintain. This file enables the replacement of approximated parts from SailWind Layout, with geometrically accurate components previously modeled in Pro/ENGINEER. This file must exist in either the current working folder or in the Pro/ENGINEER software loadpoint\text folder.

ECO

An acronym for Engineering Change Order. This refers to a file with netlist changes that needs to be annotated to update either the schematic or layout that has become out of sync with the new design changes.

ECO mode

A mode that SailWind Layout enters when the ECO Toolbar is open. Changes that affect the connection list or parts list can be recorded in a file for backward annotation.

See also [ECO](#)

ECO Options

The setup choices available for the ECO output file in the ECO Options dialog box.

ECO registration of attributes

Only ECO-registered attributes, set on the Objects tab of the Attribute Properties dialog box, can be added, deleted, or changed during the ECO process. Via attributes are not registered attributes and cannot be added, deleted, or changed during the ECO process.

You can modify ECO-registered attributes only in ECO mode.

Non-ECO-registered attributes are never recorded in an .eco file during ECO operations.

To compare ECO-registered attributes, use the Compare Only ECO Registered Attributes option on the Comparison tab in the Compare/ECO Tools dialog box.

EDA

An acronym for Electronic Design Automation.

EDC

An acronym for Electrodynanic Checking.

edge

One side of a polygon.

edge die

The two or three rows of dice along the outer circumference of a wafer.

edges

The Selection Filter preference that enables or disables selection of geometric segments.

editing

Any action that modifies a design.

electrical layers

Layers enabled for routing that are checked by DRC.

electrical net

A series of nets connected by one or more components. Length, differential pair and matched length rules can be applied to an electrical net as though it were a single net.

embedded objects

An object, including all of its data and the information needed to manage the object, that is contained within the framework of, and is a part of, the container application document.

See also [linked objects](#)

EnableMacroLanguage

The *SailWindpcb.ini* file entry that, when equal to one, enables loading of all macro parameters on startup and, when equal to zero, disables loading of macro parameters upon startup.

end component

A component having at least one pin which is a final pin of an electrical net.

end no via

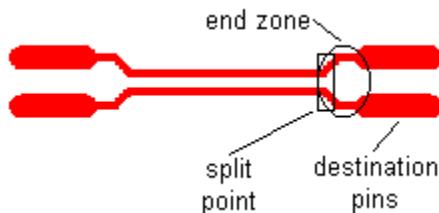
The mode initiated in the routing shortcut menus that, while routing, ends a partial route without a via.

end via

The mode initiated in the routing shortcut menus that, while routing, ends a partial route with a via.

end zone

The part of the differential pair between the split point and destination pins.



The labels in the above graphic correspond to routing that starts at the left-hand set of pins and ends at the right-hand set of pins. The label positions are reversed if the routing starts at the right-hand set of pins.

See also [differential pairs](#), [split point](#)

ending layer

The finishing layer for a drill pair or via definition. Enter information about ending layer in the Pad Stacks Properties dialog box.

engineering change order (ECO) operations

Any processes that modify the connection list or parts list.

entry angle

The angle at which a route enters a pad.

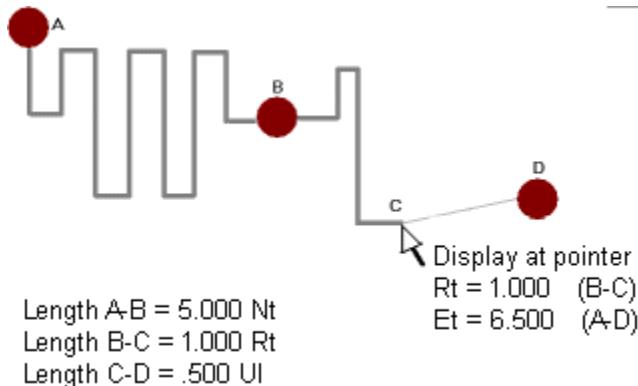
Esc

To use the Escape or Cancel keys to stop a current action.

estimated total length

The trace length monitor calculates estimated length as the combined total of routed length (R_t), plus the routed length for the entire net—including overlapping segments—(N_t), the unrouted length (U_1) of the trace being routed, and includes half the Discrete length value of each connected pin of components that have a Discrete length assigned (not shown).

Overlapping segments are counted only once.



See also [routed length](#), [unrouted length](#)

eutectic solder

A tin/lead alloy (63% tin, 37% lead) that melts at optimum temperatures.

export

The translation command used to convert a design file into PADS-format ASCII or DXF.

extended rules

Clearance, routing, and high-speed rules consisting of classes (one or more nets), groups (one or more pin pairs), individual pin pairs, decals, components and differential pairs. Without the Extended Rules option, you can assign rules on the net level only.

extension lines

Lines extending from the points being measured.

extents

The limits of the x and y coordinate area that is occupied by all items within a design. This includes information external to the board outline, such as dimensions or fabrication notes.

Fabless

A semiconductor company that subcontracts wafer manufacturing because it does not have its own wafer manufacturing facility.

fabrication

With semiconductor manufacturing, the front-end process of making devices in semiconductor wafers only, not the package assembly or back-end stages.

fanout

A segment of trace or copper shape added to SMD pads to facilitate routing. A fanout typically consists of one or more trace segments connecting a component pad to a via, enabling the signal on an outer layer to connect to one or more internal signal layers or planes. A specialized repeated pattern is often necessary to break out multiple pads on the same component far enough from the component to enable easy routing.

Use fanouts to:

- enable on-grid access by autorouters that cannot handle off-grid pads.
- make routing easier, and ensure connections are made.
- connect SMD pins to an inner plane layer using vias.
- connect an SMD pad to an inner signal layer where more routing space is available.

fanout pass

An autorouting pass that places vias for inaccessible SMD component pins and routes, from the vias to the pins.

fanout side of CBP

The side of the SBP Guide to which the CBP should be wire bonded. Usually the fanout side of the CBP is the same as its die side, but in some cases more complex patterns of wire bond fanout may mean that the two sides are not the same.

FCBGA

An acronym for flip chip ball grid array.

feature size

The smallest line width or spacing between lines or features on a semiconductor die.

feed-through hole

A drilled and plated hole that passes conductivity from one layer to another. This is also called a via.

fiducials

Fiducials are alignment marks, a type of target, used for calibration before placing objects.

There are at least three types of fiducials:

- **Panel fiducials** — used for alignment and calibration of images on a multi-board panel.
- **Board fiducials** — used align components on a specific board (on or off a panel). Fiducials are (typically) round solid targets placed near three corners of each board on each side of the board that will receive components. The pick and place system scans the board for these targets (shiny circles approximately .040" in diameter) and uses them to align the machine before it starts placing parts.
- **Component (local) fiducials** — used for close tolerance placement of high pin-count components with fine pitch leads. The footprint (PCB decal) of a fine pitch component will typically contain two component fiducials at opposite corners of the footprint. This enables the pick and place machine to align the fine pitch component exactly on the footprint.

field upgrade

Programming options on your security key by entering in a key unlock code using license.exe or equivalent.

file sharing

Multiple users accessing the same file or files through a network.

file.dir

The *SailWindpcb.ini* file entry that specifies the default location of your design files.

filter

A settings dialog box within that controls which types of objects can be selected.

find

The SailWind command that locates, and optionally selects, an object or group of objects in the database.

finger pad

One of many long pads placed in a series to represent an edge connector.

finished hole size

The size of a drilled or routed hole after plating and/or solder reflow has been applied.

flashed pads

Pads produced on a photoplotter by opening the aperture momentarily, without moving the board, to produce a pad shape.

flat pack

A component package where the leads extend away from the component and remain on a parallel plane with the base of the component.

flip

The command that moves the selected items to the opposite side of the board.

flip chip

An IC designed for face-down mounting by means of controlled-collapse solder pillars on a device's I/O bonding pads.

floating license

A method of licensing where a central security server manages a pool of licenses for use by a large number of clients.

floating toolbars

Toolbars you can undock from the sides of the application window and place anywhere on screen.

flood

To fill a previously defined copper plane.

flower pattern

Metal shapes placed under a die for thermal management and/or electrical connection; also referred to as a die flag.

A ceramic, surface-mounted hermetic package.

FlushUndoBeyondSize

The *SailWindpcb.ini* file entry that determines the maximum size of the undo buffer before SailWind Layout removes previous commands from the undo buffer to make room for the current command. If adding the current command causes the undo buffer to exceed this maximum size, SailWind Layout removes previous commands until the undo buffer can store the current command.

follow route

The connections or pin pairs that are part of the bus routing, and which are routed following the guide route's path.

footprint

The arrangement of pads for a given part decal. For example, the footprint of a fourteen DIP is two rows of seven pads, spaced 100 mils in the Y direction, and 300 mils in the X direction.

forward-annotate

Update a design file using data from a schematic.

FR-4

An acronym for Fire Retardant Number Four, an epoxy-resin substrate material used in laminate applications.

free copper

Open or closed copper that is not associated to other copper or pads.

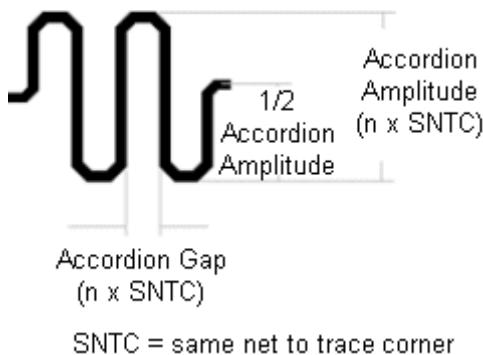
free disk space

The physical amount of space available on your hard drive that is available for use by programs.

gap (accordion)

The gap of an accordion sets the pitch between chords. The gap is a user-definable number multiplied by the same net trace-to-corner clearance.

If the same-net trace-to-corner distance equals zero, then Trace Width is used for the gap calculation.



See also [accordion](#), [amplitude](#), [pair routing gap](#)

gate

An element of an electronic circuit whereby one or more signals are input, with one output being dependent on the state of the input(s) and the type of logic used to interpret the input.

Pin swapping involves exchanging like inputs

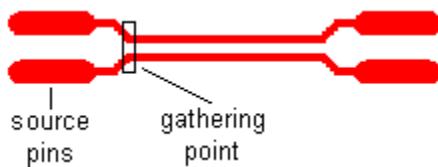
Gate swapping involves exchanging the entire element for a like element.

gate array

An IC consisting of a regular arrangement of gates that are interconnected to provide custom functions.

gathering point

The point near the source pins where differential pair traces can start to be routed together at the pair routing gap.



The labels in the above graphic correspond to routing that starts at the left-hand set of pins and ends at the right-hand set of pins. The label positions are reversed if the routing starts at the right-hand set of pins.

See also [differential pairs](#)

GDI memory

Memory reserved for Windows devices and graphics.

Geometry.Height

This attribute is used to indicate the height of the part. The attribute enables SailWind Layout to prevent the component from being placed in an area of the PCB which is height restricted.

In SailWind Layout, you can set board height restrictions for the top and bottom layers in the Drafting Options or area height restrictions using a Keepout area with a "Component height" restriction. This attribute is also passed from SailWind Layout to mechanical tools where it can be used in 3D simulations to determine whether the part will meet spatial requirements.

In addition to the value, use one of the following units:

- Use the quotation symbol " for inches. The SailWind Layout Attribute Dictionary specifies the following limits of acceptable values. Min=0.00000", Max=25.00000".

Example: GEOMETRY.HEIGHT=3.26548"

- Use the abbreviation mil in upper or lower case. The SailWind Layout Attribute Dictionary specifies the following limits of acceptable values. Min=0.00mil, Max=25000.00mil

Example: GEOMETRY.HEIGHT=12654.83mil

- Use the abbreviation mm in upper or lower case. The SailWind Layout Attribute Dictionary specifies the following limits of acceptable values. Min=0.00000mm, Max=635.00000mm.

Example: GEOMETRY.HEIGHT=123.21348mm

Gerber

The language used to drive a photoplot machine. This language is an ASCII file with instructions for selecting an aperture, moving the light source, and turning the light source on and off.

Global tab

Options tab that includes settings that affect an entire design, such as units of measurement and pointer size.

Glue

Anchors component(s) in their current location so they cannot be moved

grab bars

The two vertical or horizontal bars to the left or top of the window.

graphics cache

The SailWind setting used to optimize graphics. This is handled by the DisableCaching entry in the *SailWindpcb.ini* and *SailWindlogic.ini* files.

green dot

The status indicator located in the upper left corner of the workspace. It is green when the system is idle or ready for operation. It is red when the workspace cannot receive user input, such as when producing CAM drawings.

grid

A division of the workspace into measurement steps to facilitate accurate spacing between placed parts and routed lines. Also refers to the display; small white dots locating the measurement steps

ground plane

A design layer completely filled with copper, except for clearances around nonconnected pads and vias.

group

A collection of pin pairs that share common design rules.

grow

An cluster placement feature that adds additional parts to an existing cluster.

guard band

An octagonal shape that appears at the end of a trace during routing operations whenever the head of the trace meets a clearance obstacle that it cannot shove. The guard band only appears when online design rules are enabled.

**gui**

An acronym for Graphical User Interface. The GUI includes such things as menus and commands that allow for interaction between the user and the software program.

guide route

A route segment that is used for the first connection and that is the lead for laying down two or more pin pairs simultaneously in neat flowing patterns.

hard rule

A rule that is always followed. See also [soft rule](#), and Hard and Soft Rules.

hard breakout

Use of associated copper within a surface mount decal to simulate a dispersion route. The disadvantage to this method is that routing channels will possibly be blocked.

hatch

A copper fill pattern that uses horizontal and vertical lines at a specified width and spacing.

HDI

An acronym for High Density Interconnect.

heat dissipating component

While all components generate heat, these components generally have a published wattage rating. Care must be taken to ensure components or materials adjacent to these heat generating components do not exceed their max temperature ratings. If more than one of these components are used in a design, they should be spread out and should also be positioned to not impede airflow.

heat sink

An assembly that serves to dissipate, carry away, or radiate heat into the surrounding atmosphere.

high density interconnect

A class of packaging involving boards, substrates, and components using extremely small trace and spacing dimensions.

highlight

A user-defined color, usually white, used to denote that an object is selected.

high-speed checking

Using the Electrodynamic Checking utility. A simulator-type check that finds traces that may run parallel to each other close enough, and for a long enough distance, to cause cross talk.

hole plating

A fabrication process where solder flows through a drilled hole to connect the pads on either side of the hole, to provide connectivity between two or more layers.

HPGL

An acronym for Hewlett Packard Graphics Language, a standard pen plotter interface language.

IC (Integrated Circuit)

An acronym for Integrated Circuit.

IDF

Intermediate Data Format. An industry standard format used for exchanging data between electrical and mechanical design systems.

IMAPS

An acronym for International Microelectronics and Packaging Society.

impedance

Resistance to the flow of current in a trace. Measured in ohms.

in circuit testing

An exhaustive and thorough test of a PCB in final production that tests nets and unused pins for such things as correct voltage, correct parts, or bridging. Test point placement is critical for in circuit testing.

inaccessible nets

Nets for which you cannot define test points. DFT Audit analyzes all nets. If DFT Audit determines that test probes cannot access them, the nets are inaccessible (also called non-adaptable).

INI file

An ASCII file, with the *.ini* filename extension, that contains startup parameters.

An INI file for Windows might contain the following information: graphics drivers, mouse drivers, fonts, and so on.

An INI file for programs might contain the following information: folder structure, display colors, default editors, and so on.

inner layers

Design layers other than those on the top or bottom of a printed circuit board. Inner layers may be routing layers, plane layers, or a combination of both.

installed options

SailWind product features that you have bought and installed as part of the software package.

instruction pointer

A small yellow arrow in the Output window gutter that indicates the current line in a script or macro.

insulator

A material used to inhibit heat or electrical properties, such as current flow.

integrated SailWind Designer project

A SailWind Designer to SailWind Layout project that uses a common database to share design information instead of using netlists to pass the design information between the software applications.

integrity check

A database check runs whenever a *.job*, *.dxf*, or *.asc* file loads. You can also initiate an integrity check while you are working. Type the *I* modeless command, then press the Enter key.

intensity

A value assigned to objects such as vias to weigh decisions made during the autorouting process in SailWind Router. The higher the intensity, the less the item is used. For example, set a high intensity for via usage to minimize the amount of vias added to the design.

interconnect

A conductive connection between two or more circuit elements.

IPC

An acronym for Interprocess Communications within the SailWind product.

irregular trace length

Sections or segments of differential pair traces not routed at the pair routing gap.

See also [differential pairs](#)

islands

Small, isolated sections of copper plane that are not attached to anything.

JEDEC

An acronym for Joint Electron Device Engineering Council.

Joint Electron Device Engineering Council

JEDEC is the semiconductor engineering standardization body of the Electronic Industries Alliance, a trade association that represents all areas of the electronics industry.

jumper

A physical part used to cross over traces on most one layer PCB designs. Jumpers can be 0 Ohm resistors or wires stretched between jumper pads.

keepout areas

Areas that automatically ban objects. Depending on the keepout Properties, these areas may be set to prevent: placement of components, components that exceed a specified height, component drill holes, traces and copper, copper planes, vias and jumpers, and test points.

keyview.exe

An executable file used to list the options programmed into your key. When you run keyview.exe, it creates a file named *keyview.txt* that contains a listing of your key options.

label

A label is a display instance of a component or jumper attribute. If you want to make an attribute visible in the design, you must instantiate it as a label associated with a component or jumper. You can do this in the Design Editor or the Decal Editor. You can have multiple labels based on the same attribute. An attribute has two parts—a name, and a value. A label of an attribute can have one of four visibility settings—None, Value, Name and Value, and Full Name and Value.

Latinum rules

Latinum rules are for advanced functionality in SailWind Router. Some constraints that you can set in Layout are only used by SailWind Router. For examples, see the Tune/Diff Pairs options, Fanout Rules, Pad Entry Rules. The Latium checks include:

- component clearance rules
- component routing rules
- differential pair rules
- via at SMD rules

layer biased net

A layer biased net is any net with a design rule specifying a layer bias to one or more, but not all, electrical routing layers

layer pair

The assignment of two routing layers to switch between using the Layer Toggle command. On two layer boards, the toggle is automatically set between 1 (top) and 2 (bottom).

layer toggle

To switch between layer pairs while routing.

layers

A standard CAD database feature that separates graphical information into sheets of similar information such as dimensions, construction lines, or text. For PCB applications, this enables the various fabrication layers to be created and output separately.

layout-driven design

A PCB design process in which no schematic is created, and both the logical (netlist) and physical conformations of the board are defined in a layout tool. Also, a design created using this process. See also [schematic-driven design](#).

Layout.rep

The error report file that is created by the database integrity test and which is written to the *\SailWind Projects* folder.

LCC

An acronym for *Leadless Chip Carrier*.

lead frame

A sheet metal framework that is etched to form an array of metal traces.

lead pitch

The sum of the lead width and lead spacing.

lead spacing

The distance between a component's adjacent leads.

Leadless Chip Carrier

(LCC) Ceramic IC package with no physical lead. There are only pads on the bottom of the package around the edges.

length matching

A same-length requirement where the entered value represents a minimum/maximum length tolerance for nets belonging to the same class.

length minimization

A routing feature that configures unroutes to the shortest available distances, or in a specific topology, to facilitate high speed routing.

LGA

An acronym for land grid array.

LibDir

The *SailWindpcb.ini* file entry that specifies the location of your library files.

libraries

The collection of part types, part decals, and drawn items included with a SailWind product or created by the user.

Library Manager

The SailWind feature that provides access to, and allows for modifying, the library of parts.

linked objects

When an object is linked, a presentation of the object and link to the source is contained within the framework of, and is a part of, the *container application* document. The object is linked to its source, and the source continues to physically reside wherever it was initially created. Therefore, the file that contains the object is smaller than if the object were an embedded object. See also *embedded objects*.

Whenever you open the file that contains the linked object, the object checks the source to see if it changed since you last saved the file. If the source changed, then the linked object automatically updates.

LogCompressionMode

A *SailWindpcb.ini* file entry that controls recorded mouse movements in a log file. When set to one, the default, recording of compressed mouse movement is enabled. When set to zero, recording of all mouse movement is enabled.

logic family

The assignment of an electrical type by name, such as CAP (capacitor) or RES (resistor), to indicate the appropriate reference designator prefix such as C or R.

LOGMode

A *SailWindpcb.ini* file entry for online macro recording to a log file. When set to 0, the default, recording is disabled. When set to one, macro recording is enabled, and the *next.log* file is created.

loop

A pin pair that contains a route that branches off the original route, then branches back into the same route to form a loop.

loop routing

Used to create a loop in an existing route.

LPT port

A parallel printer port, usually referred to as LPT1 or LPT2.

macro

Internal objects that are handled using the macro engine vocabularies, and may or may not have the automation interface.

Manhattan distance (delta x + delta y)

Used to approximate unrouted net length for BoardSim. Add a percentage multiplier to account for indirect routing paths.

masking

The inhibiting of electrical interference between two traces on different layers due to separation by a ground or power plane.

material condition

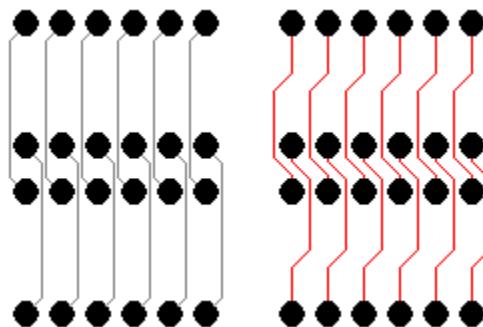
There are three material conditions when creating component decals. They are Maximum (providing the most robust solder joint), Nominal (providing a general purpose solder joint) and Minimum (providing the least possible solder joint for very dense designs).

MCM

An acronym for multichip module.

memory pattern

A collection of routes between memory devices that form a distinctly repeatable pattern.



menufile.dat

The file containing the structure and text of all lists and shortcut menus. The *menufile.dat* file must be located in the same folder as *SailWindPCB.exe*.

micrometer

One-millionth (10^{-6}) of a meter; about 40 millionths of an inch. Micrometer is synonymous with micron.

micron

A term used for micrometer. One-millionth (10^{-6}) of a meter; 25.4 microns = 1 mil.

microvia

Vias that have a narrow drill hole. Because of their specific diameter to depth ratio, they are typically blind or buried vias and do not pass through many layers of the design.

minimum geometry

The smallest line width or spacing between lines or features on a semiconductor die.

miter

A diagonal segment or arc that replaces a corner.

miters pass

An autorouting pass that converts all 90 degree route corners to diagonal corners.

mixed plane layer

A plane layer that contains obstacles other than pads, such as routes, copper, or text.

modeless command

A command invoked through the keyboard. Commands include display options, design settings, and mouse click substitutions.

modify

To change information for a selected object.

moiré

Target-shaped objects located in the corners of finished artwork that are used to properly align each layer to others for design verification and fabrication.

monolithic device

A device whose circuitry is completely contained on a single die or chip.

mounted side

The side of the printed circuit board, either front or back, on which components are mounted.

mounting holes

Many (but not all) boards have mounting holes. Mounting holes are typically located around the perimeter of a board (most often in the corners). They are drilled holes, used to mount a printed circuit board to the finished product (for example, a mother board mounted to the computer casing), or used to attach bolt-on components to the printed circuit board (for example, stiffeners and ejector tabs).

There are two types of mounting holes: plated and non-plated. Plated mounting holes have copper inside the hole and usually have a large annular ring of copper on both sides of the board connected by this copper cylinder (plating) inside the hole. These holes are typically connected to the GROUND bus or plane on the board and provide a method for grounding the board circuitry to the enclosure (for shielding purposes). The mounting hole ring diameter is usually slightly larger than the diameter of the head of the screw that will be used to fasten the board to the mounting device within the enclosure. Non-plated mounting holes are used for the same purpose, the only difference being that they are not internally plated and do not have a copper ring, therefore they are not used for grounding the board to the enclosure.

Plated mounting holes cannot be used as tooling holes as the thickness of the copper plating can vary and violate the close tolerance required by a tooling hole. Non-plated mounting holes can sometimes work double duty as tooling holes because there is no internal plating, therefore the tolerance of the hole size can be more closely controlled and fit within the requirements of a tooling hole.

Use the Decal Editor and the Pad Stacks dialog box to create tooling holes. Save the single-terminal object as a part to the library for reuse.

See also Creating and Adding Board Mounting Holes in the *SailWind Layout User's Guide and Reference Manual*

multichip module

A package with multiple dice that is 20% or more silicon, has 100 or more I/O on a substrate, and four or more layers.

multilayer PC board

A design that contains routing and/or plane layers, in addition to those on the front and back side.

nail diameter

The diameter of the test probe.

NC drill

An abbreviation for numerical control drill. This technology involves producing an output file containing the x-y location and drill size for each hole, then feeding this information into a machine for automated hole drilling.

negative

A photographically produced reverse image of a plane layer. This allows cleared areas, or airgaps, to be created using normal drawing techniques. When reversed, all areas not drawn for clearance become the actual planes.

nested embedding

Nested embedding occurs when you insert an object using OLE into another object. For example, inserting a SailWind Logic schematic into your SailWind Layout Design or inserting a Microsoft Word document into a schematic.

nested macros

Macros called from other macros.

net

All pin pairs composing one individual signal. Nets contain at least one subnet, but may contain more than one.

See also [subnets](#)

net class

A collection of nets with a common set of design rules.

net length rules

Rules that control a net's or pin pair's routing length.

The following high-speed rules are examples of net length rules: minimum/maximum length, matched length, and differential pairs.

The phrase controlled length net refers to nets that have length rules, or nets with pin pairs that have length rules.

net name

A specific name given to a net to describe its function; for example, GND, PWR, or DATA0.

The maximum net name length is 47 characters. You can use any alphanumeric characters except { } * and space.

netlist

A point-to-point connection list for each signal in a design, providing the reference designator (part name) and pin number.

netlist file

A SailWind ASCII file containing all of the nets in a design, including all component pins that make up the nets. The file may also contain a list of all parts in a design, and/or the settings that control the substrate bond pad numbers and functions for newly created substrate bond pads.

netlist.fmt

The ASCII setup file for the report format that produces a netlist without pin information.

network security

Use of one security key, programmed with multiple options, for network use with one or more systems at a time.

next.ini

A file produced in the C:\<install_folder>\<version>\Programs folder when the *SailWindpcb.ini* entry LOGMode is equal to one. The *next.ini* file is a copy of your *SailWindpcb.ini* file at the time *next.log* is written.

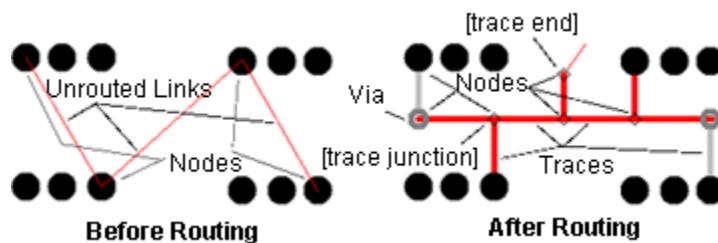
next.log

A file produced in the C:\<install_folder>\<version>\Programs folder when the *powerpcb.ini* entry LOGMode is equal to one. The *next.log* file records all activities within a SailWind Layout session so that they can be replayed to reproduce a series of steps or used to illustrate a problem.

node

A point along a trace where traces join other traces (T junction), where traces transition to other layers, or where traces end at pins, virtual pins, vias, or floating endpoints. Specifically, a node can be any pin, virtual pin, via, copper, trace junction, virtual point, or trace end.

See also [virtual point](#)



node-locked

A license for a specific Host ID.

non-ECO-registered parts

These parts are found in the schematic and layout design. Parts not selected as an ECO-Registered Part on the General tab of the Part Information dialog box are non ECO registered parts.

- A schematic non-ECO-registered part is required in the schematic but has no place in the layout of the circuit board. For example, a chip socket shown in the schematic for inventory tracking in the bill of materials.
- A layout non-ECO-registered part is required in the layout design but has no place in the schematic. For example, a plated and grounded mounting hole.

non-electrical parts

Parts with no pins. For example, a mounting screw shown in the schematic for inventory tracking in the bill of materials.

non-plated holes

Pads that are not reflowed with solder, usually reserved for mounting holes. Non-plated holes are not drilled with an oversize to accommodate the solder flow.

To determine plating status, in SailWind Layout, use the Pad Stacks Properties dialog box. In SailWind Router, use the Pad Stack tab in the Pin Properties dialog box.

nudge

A placement feature that relocates parts to make room for new parts being placed. Movement is based on previously defined clearance rules.

object

One discrete item in the design. For example, an object may be a route segment, a part, a drawing line, or a via.

object mode

Start a command by selecting one or more objects and then selecting the command to perform on them.

See also *verb mode*

obstacles

Objects that block routing, for example protected pins, vias, traces, keepouts, and board outlines.

odd pad shape

A pad that requires a special aperture, or plot sequence, to create. In SailWind Layout, in the Pad Stacks Properties dialog box, the odd shape setting should not be confused with trying to create a custom shaped pad which is accomplished by drawing copper in the decal and associating it to the pad.

offline plot

A plot that is sent to a file before it is copied to a printer or plotter for processing.

offset

The distance by which rectangular or oval pads are moved away from the electrical center of the pad stack.

offset pads

Rectangular or oval pads moved off the electrical center of the pad stack to facilitate identification/selection, or for a special design consideration.

one pin nets

A net that contains only one pin. Also called single pin net. In SailWind products, a net must have a minimum of two pins.

online DRC

A SailWind feature that actively checks established-design rules during routing or placement operations.

online plot

A plot sent directly to a printer or plot.

open cluster

Clusters that you can delete or replace during automatic cluster creation.

optimization

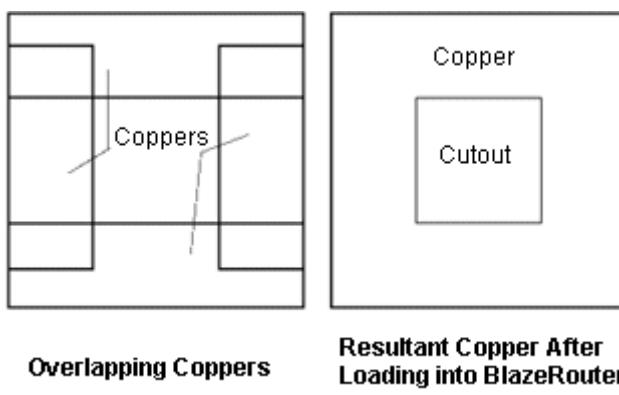
Rearranging placed parts and/or swapping pins and gates on parts to minimize trace lengths and reduce the number of vias required for routing.

optimize pass

An autorouting pass that analyzes each route and tries to improve the quality of the route pattern by removing extra segments, reducing via usage, and shortening routed trace lengths. This pass includes glossing and smoothing processes.

overlapping coppers

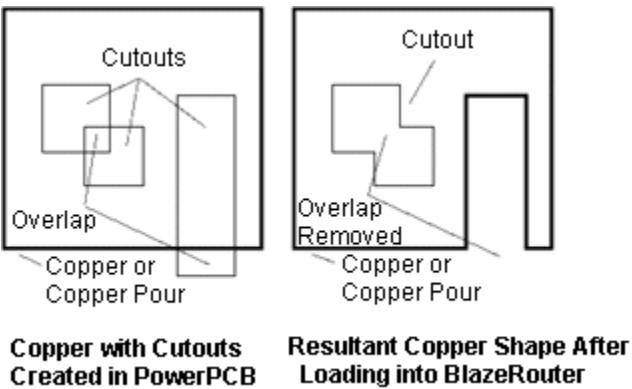
Overlapping coppers are combined into one copper area, with possible cutouts.



See also [coppers](#), [copper connectivity](#)

overlapping cutouts

Overlapping cutouts are combined into one cutout area.



See also [cutouts](#)

overlapping segments

Multiple trace segments stacked on top of one another on one layer.

package

The protective container for an electronic component with terminals to provide electrical access to the die components inside.

pad entry

The point where a trace entering or exiting a pin first crosses the edge of a pad.

pad entry angle

The command in SailWind Layout and Router that establishes the angle at which a trace enters a pad. This may be orthogonal (90 degrees), diagonal (45 degrees), or any angle.

pad function

The die signal name to which the component bond pad is connected.

pad number

The number of the component bond pad.

pad oversize

On plane layers, pads that are larger than normal, to generate proper clearances when the image of the pad is printed in a negative format.

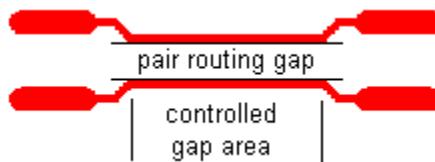
Pad oversize is measured from the center of the pad, not the perimeter. For example, if you have a 3 mil oversize, the measurement is actually 1.5 mils in each direction from the center of the pad.

pad stacks

The combination of pads, drills, and pastes, for example, on a pin or via, for each layer of a design, stacked directly on top of one another.

pair routing gap

The fixed edge-to-edge clearance between the traces in the controlled gap area for a differential pair.



See also [controlled gap area](#), [differential pairs](#).

paired layer

The start and end layers used by the layer toggle command when changing layers while routing. It defines the default layers to use when you make layer changes.

palette

A user-definable color chart in the Display Colors dialog box.

pan

Up and down or side-to-side movement of the screen without zoom or redraw. Use the scroll bars or postage stamp to pan.

panning

Moving the view horizontally or vertically without changing the size of the design on your screen.

parallel port

A printer port, usually referred to as LPT1 or LPT2.

parallelism

Traces on the same layer that are checked for running parallel to each other.

The traces are subject to crosstalk if they run parallel to each other too long and the gap between them is too short.

parasitic

An undesirable stray capacitance, inductive coupling, resistance leakage, or undesired transistor actions.

parent object

The object to which individual design elements, such as lines, arcs, or corners, belong.

part decal

The physical representation of a part, or footprint, assigned to the part type.

part list

An output listing of all parts belonging to the same design. This normally includes the reference designation, part name, and part type, and total number of each type.

part name

The text for each part that indicates the reference designator.

part outline width

The line width of 2D line shapes, created in the PCB Decal Editor, that represent silkscreen or documentation data within a part decal. The shapes do not include text, reference designators, or copper with the decal.

partial route

Partial routes are uncompleted routes where the ratsnest flightline is still visible. This occurs when you click End while routing or bus routing, or when you delete a trace segment. See also [dangling route](#).

partial via

A via that does not travel through all of the board's electrical layers. The *blind via* and *buried via* are both types of partial vias.

parts1.fmt

An ASCII part list format file for the report file generator that consists of a reference designator, part type, and logic type.

parts2.fmt

An ASCII part list format file for the report file generator that consists of a part type, reference designator, and part description.

paste

A substance used to attach each pin of a surface mount device to a PC board.

paste mask

An artwork layer with a paste location for all pads of surface mount components.

patterns pass

An autorouting pass that searches for, and routes, groups of unrouted connections that can be completed using a typical *C routing pattern*, *Z routing pattern*, and *memory pattern*.

PBGA

An acronym for plastic ball grid array.

PDF configuration

A set of PDF Configuration dialog box control settings. A PDF configuration can be saved as a *.pdc* file and reused to create PDF documents for multiple designs.

PGA

An acronym for pin grid array.

photoplotting

Using a machine to create printed circuit board fabrication artwork. The machine creates artwork by exposing clear film to light or by rasterizing an image onto clear film.

physical design reuse

A collection of design objects that you want to reuse, which are associated with one another. The collection of objects can be saved to a file.

physical design reuse elements

The objects that compose the reuse. They can include components, routes, vias, text items, and other elements.

pick and place

An automatic printed circuit board assembly machine, driven by outputting the part type, location, and orientation of suitable parts from a design.

pin

The through-hole or surface mount terminal that represents a connection to a part. Pins are also referred to as pads in pad stacks.

pin array/pin grid array

A package with pins distributed over much or all of the bottom surface of the package in rows and columns.

For more information, see Decal Wizard Dialog Box, BGAPGA Tab.

See also [pin types](#).

pin number

Within a component, the numeric or alphanumeric designation that distinguishes pins from each other.

In the Status bar, pins are identified using the following format:

Pin: [Component name].[Pin number].[Pin type]

For example:

Pin:Y1.N.Nonelectrical

See also [pin types](#).

pin pair

The combination of a trace or unroute, and the pins on either side. A net can contain one or more pin pairs.

pin pair group

A collection of pin pairs that share common design rules.

pin type

A designation that indicates the electrical characteristics of the pin such as Source (S), Load (L), Terminator (Z), and Undefined (U). For example, U1.1.S may appear on the status bar.

pin types

Pins and pin pairs can be identified by one of the following pin types:

- Source
- Bidirectional
- Open Collector
- Or-Tieable Source
- Tristate

- Load
- Terminator
- Power
- Ground
- Nonelectrical

Pin types make up the last portion of the pin identifier in the Status bar. For example:

Pin:U10.C.Open Collector

See also [pin number](#)

placement check prints

Generate a CAM Assembly drawing to make a placement check print.

After the PCB Designer receives a schematic and a netlist from an Engineer, they (typically) place the components onto the board in a manner that best suits the routing of the board. Sometimes, the placement better suits the intentions of the Board Designer than the Engineer, so before routing proceeds, the Engineer will request to see a set of Placement Check Prints. Placement Check Prints show the placement of all components on both sides of the board, so the Engineer can review the locations and confirm the Designer has correctly placed the components. These Placement Check Prints typically require agreement from both the Designer and the Engineer before routing can proceed.

placement operations

Operations where parts are relocated or added to a design to optimize an existing placement.

plane hatch outline

The outline of a copper plane shape after it has been flooded to differentiate it from the plane pour outline as originally drawn. When you draw the copper plane pour outline and then flood the shape, the outline often changes to accommodate the modeless command PO.

plane layers

A design layer where the entire surface is covered by copper, except for information not connected to the plane.

plane nets

Nets assigned to plane layers.

plane pour outline

The outline of a plane shape after it has been drawn to differentiate it from the plane hatch outline after it has been flooded. You can switch plane display modes using the shortcut modeless command PO.

plastic ball grid array

A surface mount package with an array of solder sphere-shaped interconnects arranged across the bottom surface of the package substrate.

plastic leaded chip carrier

A common surface mount package with leads on all four sides, used as a socket for devices that cannot withstand the heat of the reflow process, and/or to allow for easy component replacement.

plated holes

Drilled holes that have copper covering the inside surface of the hole, and which are connected to a pad on each side. Plated holes pass connectivity from one layer to others.

plating tail

A route that connects BGA vias to a plating bar or bus bar.

PLCC

An acronym for *plastic leaded chip carrier*.

plicense.exe

The program used to verify and program your security key during a field upgrade process.

polar decal

A single-radius, circular pattern decal with through-hole pins.

polar SMD decal

A single-radius, circular pattern decal with SMD rectangular or finger pads.

polygon

A closed shape consisting of three or more line segments.

positive

An image of a plane layer where cleared areas, or airgaps, are created using normal drawing techniques. When reversed to create a negative, all areas not drawn for clearance become the actual planes.

power board

A board that is designed to control power to other circuit boards.

power plane

The plane layer where power supplied to the printed circuit board is dispersed to the proper pins of each component requiring a power source.

SailWindpcb.ini

The SailWind Layout initialization file for default settings.

powerpcb.mdb

The SailWind Layout message file that contains error messages, prompts, and other miscellaneous text strings. This file must be located in the same folder as *SailWindPCB.exe*.

preferred routing direction

In the main GUI combo box, the Horizontal [H] or Vertical [V] designation next to a routing layer name. This designation indicates optimal direction for routing completion and can be set by the user or the system.

prepreg

A resin pre-impregnated sheet used to bond substrate laminate-pair layers together when a multilayer board is pressed together.

preset files

Library IQ files that enable you to save the preference settings you have established for a die design and use them in other designs. The Bond Pad Preferences files have a *.pre* file extension.

preview of CBP assignments

A preview that displays the substrate bond pads and wire bonds created when component bond pads are assigned to rings. This preview appears in the work area when the Assign CBPs to Rings dialog box is active.

preview of SBP guides

A real-time preview that displays any changes made to the number, geometry, or location of SBP guides. This preview appears in the work area when the Wire Bond Wizard dialog box is active.

well as in the design in which you place the reuse.

See also [private nets](#)

primary component side

The mounted side of the board when using through-hole components. See also [secondary component side](#)

primary objects

Primary object groups in the Object View tab of the Project Explorer contain non-removable design elements shown in a high-level object hierarchy. Primary objects are:

- layers
- components
- part decals
- net objects (including nets and pin pairs)
- via types

private nets

Nets that are contained completely within a physical design reuse.

See also [public nets](#)

probing

The testing of individual IC dice using very fine probes to temporarily connect each to a test computer to verify operation.

properties

A set of dialog boxes used to view or edit information about the selected object.

protect

Glues the routes and attached vias and prevents the autorouter from modifying them in any way.

protected routes

Traces that are placed in a protected state by Route Protection. This means that they cannot be moved or modified.

protected traces

Traces placed in a protected state (cannot be moved, or modified).

protected unroutes

Unrouted connections, or the unrouted portion of a partial route, that are placed in a protected state by the Route Protection feature. This means that they cannot be routed, moved, or modified.

public nets

Nets that are partially contained within a physical design reuse. Public nets exist in the reuse, as preferred routing direction

In the main GUI combo box, the Horizontal [H] or Vertical [V] designation next to a routing layer name. This designation indicates optimal direction for routing completion and can be set by the user or the system.

pulling an arc

Creating an arc from an existing line segment, where the diameter is derived from the line length.

QFP

An acronym for quad flat package - a surface mount IC with leads on each four sides.

quad

A square-shaped IC with pads on each of its four sides.

Quick Filter Settings

The shortcut menu selections available when no items are selected. These choices set the selection filter for commonly used tasks, enable quick access to the Find command, and Select All items as specified by the Selection Filter.

quick measure command

The Q *modeless command* which attaches a measurement line to the pointer and displays dx, dy and hypotenuse information, depending on pointer movement.

radial lead

A discrete part with pins that protrude straight down and do not extend beyond the perimeter of the component body. An example of this is a capacitor.

RAM

An acronym for Random Access Memory. The volatile (on chip rather than on disk) memory area available to the system for program operation.

range select

To select a series of geometric or route segments by first clicking on the start segment, then pressing and holding Shift and right-clicking on the end segment.

ratsnest

A term used to describe the display of all of the unrouted connections in a design. Also known as air lines or unroutes.

raw database

The raw database contains all components in the open database, regardless of assembly variants. When created, new assembly variants are based on the raw database, meaning that until you uninstall or substitute, a new assembly variant includes every component in the raw database.

read-only attribute

An attribute whose value cannot be changed in SailWind product dialog boxes. You can, however, modify attribute properties and the Attribute Dictionary entry, and can modify the attribute value in the library.

real width

To display traces at their specified width, as opposed to displaying them as one pixel centerlines.

real-time redraw

A feature that enables active regeneration of objects in the display any time the screen is redrawn. When you disable real-time redraw, regeneration occurs in the background, and the display is refreshed all at once after the background regeneration process is completed. Screen regeneration is quickest when real-time redraw is disabled.

record locking

Allowing two or more users to access the same library component at one time. However, only one user has access to save the component.

recover

Resolving an installation or operational issue, or salvaging a corrupt database by executing a specified series of steps.

redo

Repeats actions which have been undone.

redraw

Refreshes the display of the current screen image and the cursor.

reference designator

An identification assigned to each of a design's parts to distinguish them from other parts of the same type when placed on the printed circuit board. A reference designator is usually in the form of a letter that represents the part type, followed by a number. For example, C2 may represent the second capacitor in the design. SailWind Layout permits you to renumber the reference designators in one of several specific patterns enabling you to quickly find a part among thousands on the manufactured board. The reshuffled numbers that are rearranged in SailWind Layout are backward annotated to the schematic software to keep the designs synchronized.

relative coordinates

Coordinates that are based on a start point instead of the system origin.

rename

To assign a different name to a part or net.

reroute

Specifying that a trace, or a portion of a trace, follow a path different than the one currently being taken.

restricted layer

Layers that are either disabled for routing or have been disallowed by layer biasing rules. When a layer is restricted, routing is not permitted on the layer. Layers can be restricted for specific objects, such as a net or a pin pair.

restricted via

A via that is not permitted for use in the Routing Rules of SailWind Layout or Via Biasing properties in SailWind Router at any level of the rule hierarchy.

reuse

See [preview of CBP assignments](#).

reuse definition

The master copy of the physical design reuse that is saved to a file. The saved version of the physical design reuse is the version you should use in other designs. All resulting instances of the physical design reuse are based on this file.

reuse type

A name that identifies the type of reuse being created. A reuse type is equivalent to a library part type.

ring geometry

The shape of the die flag ring. The following shapes, or ring geometries, are supported: rectangle, rounded rectangle, chamfered rectangle, and arced shape.

romansim.fnt

The default file that contains definitions for the graphics for the SailWind stroke font, used to display text in SailWind products when system fonts are not in use.

rotate

The command that rotates by 90 degrees a component or object around its axis or selection point.

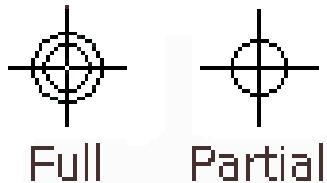
route

To create a metal etch trace of a specified width between pads.

route-completion target

This crosshair or bullseye symbol appears when routing from one pin of a pin pair to another pin or when rerouting a trace segment.

The partial target appears when you are overtop of an electrically compatible pin, but you have settings that are preventing you from routing to it - for example, the unroute of the pin pair you are routing is protected (you have selected the Protect Unroutes check box in the Pin Pair Properties).

**route loops**

A pin pair that contains a route that branches off the original route, then branches back into the same route to form a loop.

route pass

The autorouting pass that is the core pass that performs the majority of autorouting. During this pass, SailWind Router attempts to sequentially route each unroute until all connections are attempted. The Route pass contains serial, rip up and retry, push and shove, and touch and cross processes.

rounded length

The trace length monitor calculates routed length as the cumulative length of the trace. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned. If you start routing from the endpoint of a partially routed trace, the routed length includes the partially routed trace length. If the trace has branches, then the length is calculated from the branch point.

See also [estimated total length](#), [unrouted length](#)

routes

A series of traces that represents routed connectivity.

routing angle

The angle applied to adjacent segments as new corners are added to traces. For example, an orthogonal routing angle means adjacent segments will be created at 90-degree angles to each other.

routing order

The order in which the autorouter routes components, nets, and net classes.

routing pass types

There are several pass types, each of which is designed to complete a specific task. Each pass may use more than one algorithm and may also perform a number of subpasses.

- | | |
|--|--|
| <ul style="list-style-type: none">• <i>center pass</i>• <i>fanout</i>• <i>miters pass</i>• <i>optimize pass</i> | <ul style="list-style-type: none">• <i>patterns pass</i>• <i>route pass</i>• <i>test point pass</i>• <i>tune pass</i> |
|--|--|

routing strategy

The collective information SailWind Router uses to autoroute a design. This information includes which pass types SailWind Router should perform, whether to *protect* the resulting traces, and what *intensity* to assign to objects.

ru.cfg

A configuration file used by the nrus.exe program for Novell network security support.

rule values

The values of any item, regardless of its default rules or rules set assignments.

rules

An established set of conditions for a given net or design.

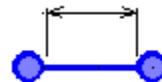
rules set

A specific set of user-assigned nondefault rules such as pin pair, groups, or classes.

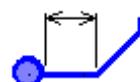
same net checking

Checks clearances between objects along the same net, as specified in the Clearance Rules dialog box. Object to object checking includes:

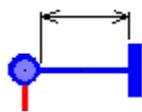
- Pad edge to pad edge.



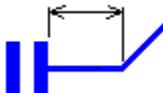
- Pad edge to inside corner of trace.



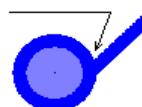
- SMD edge to pad edge.



- SMD edge to inside corner of trace.



This check prevents solder bridging during board manufacturing caused by acute angles between conductive objects such as the acute angle between pad and trace shown below.



same net rules

Specifying conditional settings, such as spacing, for connections belonging to the same signal name or net, rather than against other nets.

SBP

An acronym for Substrate Bond Pad.

SBP fanout

A single-segment fanout that connects SBPs to any-angle coupling traces.

SBP guide

The virtual snap line along which substrate bond pads are aligned during wire bond fanout generation. Each SBP guide determines the alignment of the substrate bond pads that are associated with the SBP ring aligned with this SBP guide.

SBP ring

A set of substrate bond pads aligned along an SBP guide. A substrate bond pad belongs to the ring on which it is aligned. In creating a wire bond fanout, you assign each component bond pad to a specific SBP ring.

schematic-driven design

The “standard” PCB design process, in which a netlist is first created in a schematic tool, and then passed to a layout tool, where the parts are laid out on the board and the connections routed. See also [layout-driven design](#).

scribe line or saw line

The separation between adjacent dies on the wafer. This path is used as the cutting area in sawing a wafer into the individual dies.

search

To locate specified information. One search method is to use the Find command.

secondary component side

The side opposite the mounted side for through-hole components. This side is typically wave soldered.

See also [*primary component side*](#)

secondary objects

Secondary object groups break primary objects into a more detailed hierarchy. You can add individual items to and remove individual items from secondary groups. Secondary objects include:

- net class
- pin pair group
- conditional rule
- matched length net group
- matched length pin pair group
- differential pair

seed

A part used by Cluster Placement, during cluster building, to search outward for other parts to add to the cluster.

segment

A single drafting line, path, or trace, defined by a beginning x/y coordinate and an ending x/y coordinate.

segmentation fault

The termination of a SailWind product due to a system crash or illegal instruction executed.

Select All

The Edit menu command that lets you select all items of a type specified in the Selection Filter. This option is also accessible from the shortcut menu when nothing is selected.

select mode

Point to the object and click the left mouse button. Select the command to perform on the object.

selecting

To highlight an object for editing, moving, viewing properties, or deleting.

selection filter

The dialog box inhibiting or enabling the selection of specific items.

serpentine route

A route that connects an any-angle coupling trace and a BGA pad, forming a snake-like pattern as it travels through the BGA.

session log

Information on the current session that appears in the Status tab of the Output window.

shape

The Selection Filter setting that enables or disables selection of an entire geometric object, not just its individual segments.

shared libraries

Libraries that can be accessed by more than one user across a network.

shielding

Specifying that one net be routed around another to provide protection from interference.

shortcut keys

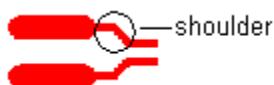
A key sequence that starts a command directly from the keyboard and without navigating through menus.

shortcut menu

A menu listing the possible actions to perform, based on the selected object.

shoulder

The part of the differential pair trace between the source pin and the gathering point, or between the split point and the destination pin.



See also [differential pairs](#), [gathering point](#), [split point](#).

signal

Voltage or current that is transferred between component pins by an electrical conductor.

signal pins

Pins that have a signal net, such as GND, assigned by the schematic capture program SailWind Logic during part type creation.

signal via

Via used to continue a signal from one layer to another.

silkscreen

An artwork layer containing the reference designator and component outline of all parts, used for the final board fabrication process.

single-sided board

A design where all pads, routing, and parts are placed on one side of the board.

single-sided die

A die that has substrate bond pads and a BGA grid array on the same side of the die.

See also [documentation layers](#)

sizing handles

Small, black squares that appear at the corners and along the sides of a rectangular area that surrounds a selected nontext object.

sketch route

A SailWind Layout command that reroutes existing traces by enabling you to draw a new route path using the pointer.

slice

Another term for wafer.

slotted holes

Oval holes in a printed circuit board, which may be plated or non-plated.

SMD

An acronym for Surface Mounted Device: the pin of a component that is attached to the PCB only on an outer surface and does not require drilled holes for component mounting.

smoothing

A command that automatically removes unneeded corners and segments and centers trace patterns between route obstacles.

SMT

An acronym for Surface Mount Technology.

snap modes

Various modes, available during dimensioning, that force the pointer to pick points based on the following parameters: intersection, any point on a line, any point in space, entire segments, the center point of an arc, and so on.

soft rule

A rule that is ignored if it alone prevents route completion. See also [hard rule](#), and Hard and Soft Rules, in the *SailWind Router User Guide*.

SOIC

An acronym for Small-Outline Integrated Circuit.

solder

A metal alloy used to attach each pin of a device to a printed circuit board.

solder dam

A small amount of solder mask used to limit molten solder from spreading further onto solderable conductors, in an area where solder mask is purposefully absent.

solder mask

The artwork layer for a nonconductive material that covers the entire board, except for pad locations. The solder mask provides a protective covering and prevents shorts during wave and reflow solder processes.

solder mask reliefs

Some components have large areas that need to dissipate heat. Others have large metallized areas (that are not pins) that need to be soldered to the board. To expose the copper area beneath these parts for soldering, the solder mask layer must have a cutout representing these areas. These cutouts are called solder mask reliefs. When the distance between pads of a fine pitch component is too small, the webs or fingers of solder mask between pads can break and wander on the board surface. To prevent this, a solder mask relief is applied to entire pad areas of a component. This is commonly called gang relief or a gang opening.

solder side

The back or bottom side of a printed circuit board. Solder side is named for the post assembly process, where the board is run through a special bath to solder all pins.

source

A pin type that indicates a signal radiating from the pin.

SPECCTRA

The product name for the Cadence Design Systems autorouter.

special symbols

Alternate decals that you specify as connectors. You can associate a logical pin type with each alternate to provide a graphical indication of the connector pin function in a schematic.

spider bonding

A method of connecting an integrated circuit die to its package leads. A lead frame is placed over the chip and all connections are made by just one operation of a bonding machine. [TAB](#) methods use this approach to interconnection.

spin

The command that rotates a component or object around its axis or selection point.

split

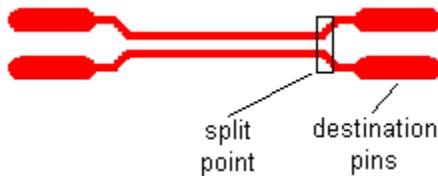
The command that creates a new corner at the pick point of the selected trace, enabling it to be rerouted.

split plane

A solid copper plane layer divided into two or more sections to isolate electrical signals from each other.

split point

The point near the destination pins where differential pair traces are no longer routed together and where the traces are routed individually to completion.



See also [differential pairs](#), [pair routing gap](#)

ssiact.exe

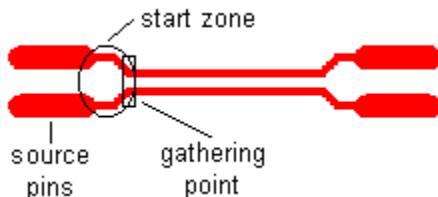
A program used to recommend set statement settings to properly adjust port access times for a security key.

stackup

The metal and dielectric layers used to implement the body of a printed circuit board. A signal metal layer carries signal traces. A plane metal layer is tied to a DC voltage. A dielectric layer is made from non-conducting material and separates two metal layers or coats the board surface.

start zone

The part of the differential pair between the source pins and gathering point.



See also [differential pairs](#), [gathering point](#)

starting layer

The first layer in a drill pair or via definition.

step-by-step mode

A mode in which the debugger runs a single line of code at a time.

stitching vias

Any SMD via, through-hole via, or partial via added to nets (on traces or within plane areas) in a repetitive manner. You can add these vias, also called free vias, for various purposes, including current and thermal needs. For example, you can place stitching vias in a plane area to provide conduction between two plane areas. You must assign stitching vias to a net, but they do not have traces attached to them.

strategy

A set of options that defines how a board should be autorouted.

strong

Places cluster members as close together as possible during placement operations. The minimum distance for placement is the same as the distance for part clearances in Design Rules.

structured attributes

Attributes that are related to each other by the prefix in their name. For example, the DFT attributes such as DFT.Nail Count Per Net, DFT.Nail Number, and DFT.Nail Diameter are structured attributes. Together, these structured attributes make an attribute group.

stub

A trace that enters another to create a T-junction. Stub lengths can be checked by the EDC program.

submicron

Dimensions smaller than one micron.

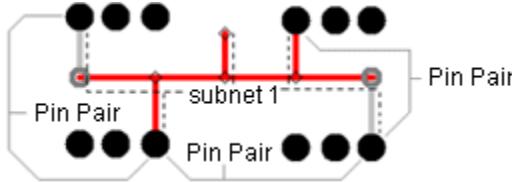
subnet

A collection of all traces and vias connecting two pins. Subnets are joined only through their common component pins and not through other nodes, such as a trace junctions, vias, or virtual points.

Subnets help to avoid errors or confusion caused when pin pairs of a net have unique, rather than common, design rules.

See also [node](#), [subnet](#), [connected islands](#), [virtual point](#).

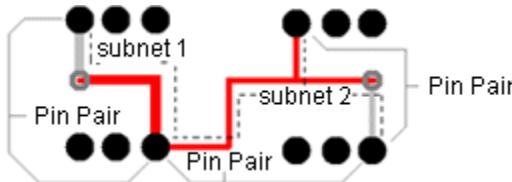
Figure 162. One subnet in a net

**subnets**

If a net has at least one pin pair with a unique design rule, such as a trace width difference, the net is automatically divided into subnets. If two pin pairs having the same rules are separated by at least one pin pair with different rules, the pin pairs are considered separate subnets. Therefore, subnets are islands of pin pairs that form an unbroken fragment within the net, where each fragment has uniform rules.

See also [differential pairs](#), [differential pairs](#).

Figure 163. Multiple subnets in a net



substrate

A material between copper laminate layers that comprise a laminate pair, or a laminate set in the case of completed multilayer boards.

substrate bond pads

Copper areas on the substrate to which a die's wire bonds are connected.

surface mount device

Pads are glued to the board rather than inserted.

swap file

The file created when a program runs out of RAM memory and writes memory to disk.

swapping

A placement optimization process that exchanges pins, gates, or entire parts.

The product *.ini* file entry that specifies the path for the SailWind product configuration files.

system attribute

An attribute that is set by, used by, and critical to a SailWind product, an external program, or Automation script (such as Sax Basic). You cannot modify the properties of a system attribute or modify the Attribute Dictionary entry for a system attribute.

system toolbars

System toolbars are specific to the SailWind programs. They feature several system toolbars, such as standard, routing, selection filter.

SystemDir

The product *.ini* file entry that specifies the path for the SailWind product configuration files.

T junction

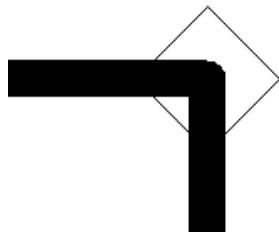
A trace that branches into another.

TAB

An acronym for *source*.

tacks

Small, diamond-shaped objects that anchor traces to their current location. Tacks are automatically generated under certain conditions and may also be manually added to a selected trace.



tandem traces

Traces on different layers that are checked for running parallel to each other.

The traces are subject to crosstalk if they run parallel to each other too long and the gap between them is too short.

Tape Automated Bonding (TAB)

A packaging method where silicon chips are joined to patterned metal traces, or leads, on polymer tape to form inner lead bonds which are attached to the next level of the assembly, typically a substrate or board.

Tape Ball Grid Array (TBGA)

A TAB packaging method in which tape automated bonding leads are replaced by a ball grid array.

TBGA

An acronym for *Tape Ball Grid Array (TBGA)*.

teardrop

A triangle shape that provides a smooth transition from a trace to a pad.

tented vias

Vias that are covered by solder mask on both sides of the board to seal the hole and protect it from wave solder.

terminal

The electrical center of a pin, as defined in the part decal.

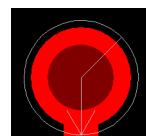
terminator

A pin type for high-speed circuit configurations that indicates a terminating resistor to match impedance of the trace. Terminators are used to reduce signal reflections that cause poor circuit performance.

test point

A test point is a group of objects that serve as a contact between the electrical element of the board and the probe of the testing device. A test point can also be a point on a node of a net, component pin, or via. Test points can also be a point on an unused component pin, such as a component pin that is not incorporated into any net.

When the via or pin is flagged as a test point, and Show Test Points is checked on the Routing tab of the Options dialog box, a down arrow symbol is drawn on the via or pad in the design:



test point pass

This autorouting pass analyzes the testability of the design, determines which nets require testing, adjusts the routes, and inserts test points to improve testability. You can select whether to add test points during routing or after routing.

testpnts.fmt

An ASCII file containing information about test points, including the test point name, the signal name, and the x/y coordinates. The report file generator creates this file.

thermal

A multi-spoke connection of a through hole pin pad, via, or surface mount pad to a copper plane.

thermal compression bonding

A method of wire bonding that does not use an intermediary metal or melting, but rather the flow of materials resulting from the combination of heat and pressure. It is also referred to as thermocompression bonding.

thermal relief

A spoke-shaped pattern that connects a via or pin, in the same net as the copper plane, to the surrounding copper. Thermal reliefs provide good pin soldering by preventing heat from dissipating throughout the plane layer.

thermal via

Via used to dissipate heat from an area or component.

thick-film process

A hybrid microelectronic process where conductors, insulators, and passive components are screened from special pastes onto the substrate.

thin-film process

The use of deposited films of conductive or insulating material, which may be patterned to form electronic components and conductors on a substrate or used as insulation material between successive layers of components.

three-state check box

A three-state check box helps to identify the state of the check box in a collection of objects.

Check box	Description
<input type="checkbox"/>	Cleared check box. The item or collection of items all have the check box in the cleared or unchecked state.
<input checked="" type="checkbox"/>	Selected check box. The item or collection of items all have the check box in the selected or checked state.
<input type="checkbox"/>	Indeterminate or mixed state check box. The collection of items have different states of the check box.

through holes

Although there are non-plated through holes, this term is used interchangeably with plated through holes. It indicates that the hole has internal plating. There are two basic types of components that can be placed on a circuit board: Surface Mount Technology (SMT) where the parts are soldered to the surface of the board, and through hole (TH) components, where the components have wire leads that are soldered into plated holes that go through the board (sometime written as thru-holes).

through-hole via

A via that passes through all electrical layers of the PCB design (as opposed to a partial via).

This is sometimes also called a through via.

tooling holes

Every board requires at least two tooling holes that the blank board manufacturer uses for layer alignment purposes during the manufacturing process. If you do not include them in the design, the manufacturer will add them to the board. Tooling holes are typically .125" non-plated holes with a tolerance of +/- .002". If the board is so small that the tooling holes will not fit, the manufacturer will add them to an area outside of the board outline. (These would typically get removed after final assembly.) There are two types of tooling holes: board tooling holes and panel tooling holes. Most boards are manufactured by stepping and repeating the single board image onto a larger panel so that multiple boards can be processed on a single panel. So, the board tooling holes are used for alignment purposes for individual boards, while the panel tooling holes are used for alignment of the entire panel during the manufacturing and assembly processes.

Use the Decal Editor and the Pad Stacks dialog box to create tooling holes. Save the single-terminal object as a part to the library for reuse.

See also Editing Pad Stacks

ToolTips

ToolTips appear below buttons and provide a command name or description for the buttons.

topology

The pattern of the trace and the order in which to connect pins in a net.

total length

The current routed length plus the total Manhattan length for remaining unroutes of the electrical net or pin pair. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.

Total length is reported for pin pairs when all the following are true: length rules are defined for the pin pair, the electrical net is a high-speed net, and copper sharing is disabled.

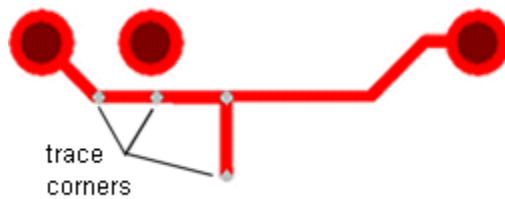
If pin pair rules are reported, the estimated total length of the pin pair is shown; otherwise, total length for nets is reported.

trace

A line segment that represents physical etch. A trace can appear as a single pixel line or as a double line to indicate its actual width.

trace corner

The vertex at which two trace segments are joined. A trace corner can also be the endpoint of a partially routed trace. The trace segments may be in line.

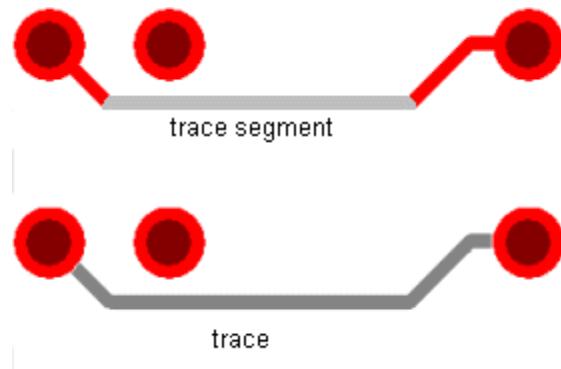


trace paths

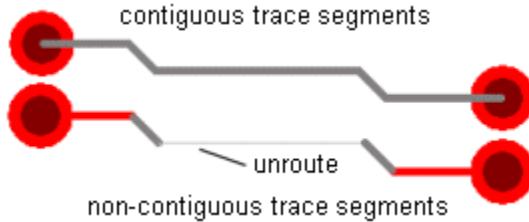
A continuous sequence of trace segments in the same trace on the same layer. Paths start and end at nodes, and cannot pass through a node.

trace segment

One section of a trace. A trace segment has one starting point and one ending point. A trace segment can be arced.



Trace segments are contiguous when they are joined end to end, in one continuous path, and belong to the same trace.



transparent layers

The mode that displays layers in a see-through mode so you can view multiple objects stacked upon each other. This is the modeless command T.

TrueLayer

The default mode of operation in SailWind Layout whereby an object on a documentation layer moves with a component if the component is moved from one side of the board to the other. For example, when you place a component on the top layer of the board, the reference designator of that component is visible on the Silkscreen Top layer (the documentation layer associated with the top layer of the board). Moving the component to the bottom side of the board automatically moves the reference designator for the component to the Silkscreen bottom layer.

TrueLayer also correctly plots paste masks of documentation-level pad shapes in CAM. The layer that the definitions move to is set in the Component Layer Associations dialog box.

By default, TrueLayer mode is enabled. To disable it, use the /NTL command-line switch. See Software Launch Options.

TTL

Acronym for Transistor-Transistor Logic.

tune pass

This autorouting pass adjusts the length of length-controlled traces. The pass examines trace lengths for only completely routed nets or pin pairs. The pass analyzes the current length of each net or pin pair if length rules and length control are enabled, based on the following conditions:

- If the cumulative length of the adjacent trace segments is within the range of minimum and maximum trace length, the tune pass skips the trace and does not adjust it.
- If the trace is longer than the maximum trace length, the tune pass rips it up and places it in a queue for routing.
- If the trace length is less than the minimum trace length, the tune pass changes the length by adding accordion patterns.

ultrasonic bonding

A wire bonding technique that uses ultrasonic energy and pressure to form the bond without heat.

underfill

Material injected under the die to ensure interconnect reliability against [#unique_1408](#) mismatch between the die and the substrate in a *flip chip* configuration.

undo

A command that enables you to remove the effects of the last command invoked.

undock

To isolate an application dataset from the main design project so it can be edited regardless of the network or the physical location of the dataset. The isolated dataset has no dependence on the main design project.

UndoMemorySize

The *SailWindpcb.ini* file entry that limits the maximum size of the buffer that is used to store ECO operations for Undo.

unions

Parts assigned to each other in fixed relative positions using Cluster Placement. These positions are maintained whenever a union is moved in Cluster Placement. A common example is the relationship between bypass capacitors and ICs.

units of measure

A commonly used set of measurements.

unroute

To convert a trace back into a connection.

unrouted length

The trace length monitor calculates unrouted length as the distance from the endpoint of the current trace segment (attached to the pointer) to its destination.

The unroute length calculation depends on the current routing angle:

Routing mode:	The calculation:
Orthogonal	Manhattan Length
Diagonal	The length of the shortest diagonal path between unroute ends
Any Angle	Point-to-point distance

The unrouted length is recalculated as the unroute dynamically reconnects to connection points. The routing angle also effects this calculation.

See also [routed length](#), [estimated total length](#)

unroutes

Thin, straight segments joining pins or coppers to indicate connectivity. Also called a link.

unused pins

Pins that are not connected to a net.

UserDir

An *.ini* file setting that specifies the path for SailWind product configuration files.

verb mode

Start a command by attaching a command to the pointer and then selecting objects to which you apply the command.

You can enter verb mode by selecting a command when no objects are selected. A small V attaches to the pointer to show that the selected command is active. The command remains attached to the pointer until you cancel verb mode.

See also [object mode](#)

vertex

A single point in the work area, defined by x and y coordinates.

via

A drilled and plated hole that passes conductivity from one layer to another.

via pair

A pair of vias used to change the routing layer for a differential pair when routing the controlled gap area.

See also [via](#), [differential pairs](#).

via type

A via or virtual pin padstack definition that is defined and named in SailWind Layout in the Pad Stacks Properties dialog box.

victim net

Nets that are interfered with by those tagged as aggressor nets during High-Speed or Electrodynamic Checking.

virtual memory

Writing memory areas to disk in the form of a swap file when RAM is filled. The size of the swap file is based on the free disk space or the limits imposed by the operating system.

virtual pin

A net object that, like a component pin, serves as a pin pair end, but uses the pad stack of a via. The pad stack can be through-hole or partial, or it can be a single-layer pad.

virtual point

A point along a trace segment that identifies a change in design rules, usually between trace rules and component rules. Virtual points are inserted into nets automatically when necessary, usually during autorouting operations. You cannot create, position, or otherwise edit a virtual point.

See also [subnets](#)

Visual Basic

Visual Basic is a scripting language developed by the Microsoft Corporation to enable users to customize applications using a standard scripting language.

visual editing

Visual Editing occurs when the source application for a linked or embedded OLE object opens within the [container application](#).

wafer

A thin disk of semiconductor material (usually silicon) on which many separate chips can be fabricated.

wafer sort

The electrical testing of each die on the [wafer](#) while still in wafer form.

WB

An acronym for [wire bond](#).

wedge bonding

A form of thermal compression wire bonding where the bond shapes the wire into a wedge shape.

width

The thickness of a trace or line.

wire bond

Fine wires, usually aluminum or gold, connecting the bonding pads on a die to the component package.

Wire Bond Editor

The Wire Bond Editor opens (explodes) a selected die part, so you can move, add, delete, and edit individual component bond pads and wire bonds in addition to substrate bond pads. You can also edit the die size.

wire bond fanout

A pattern of wires (typically gold) that arc out from component bond pads to substrate bond pads to provide connectivity between the die pins and the substrate package pins.

Wire Bond Wizard

A BGA toolbox feature that creates and places substrate bond pads and generates an automatic wire bond fanout between component bond pads and substrate bond pads.

wire bonder

The machine that connects wires between the chip bond pads and the substrate bond pads.

wire bonding

The process of electrically connecting a chip to the next level package with fine wires. The wires are either gold or aluminum.

workspace

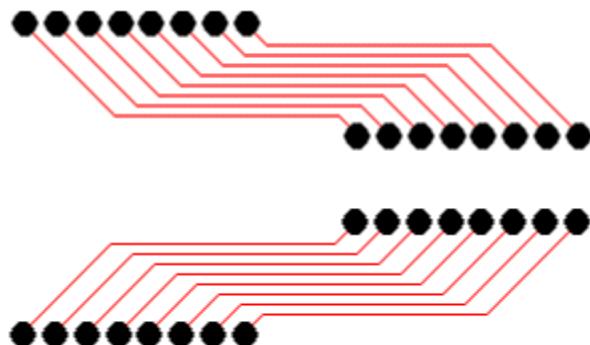
The actual work area where a design is created.

yield

The ratio of the number of acceptable units to the maximum number possible.

Z routing pattern

A collection of routes that form a pattern resembling the letter Z.



zoom

Modifying the view to make objects appear larger or smaller. Zooming in or out affects the amount of what can be viewed in the work area.

See also [*protect*](#).

Third-Party Information

Details on open source and third-party software that may be included with this product are available in the `<your_software_installation_location>/ThirdParty` directory.